DASH 8Xe®

DATA ACQUISITION RECORDER

Operations Manual

Part Number: 22834488

Astro-Med, Inc®

TEST & MEASUREMENT PRODUCT GROUP

Dash 8Xe
Operations Manual

Part Number: 22834488

Manual Version 2.2
Software Version 2.5

7/09

Specifications are subject to change without notice

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

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

Copyright

Trademarks

Dash 8Xe is a trademark 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.

FCC Compliance Statement

This device has been tested and complies with Part 15 of the FCC Rules for a Class A digital device. Operation is subject to the following two conditions: (1) this device may not cause harmful interference; and (2) this device must accept any interference received, including interference that may cause undesired operation.

Shielded cables must be used with this unit to ensure compliance with emissions limits. These limits are designed to provide reasonable protection against harmful interference when the equipment is operated in a commercial environment. This equipment generates, uses, and can radiate radio frequency energy; if not installed and used in accordance with the operations manual it may cause harmful interference to radio communications. Operation of this equipment in a residential area is likely to cause harmful interference in which case the purchaser will be required to correct the interference at its own expense.

Warning: Modifications to unit not expressly approved by the party responsible for compliance could void user authority to operate the equipment.

Canadian Emissions Requirements

This Class A digital apparatus complies with Canadian ICES-003.

Cet appareil numérique de la classe A est conforme à la norme NMB-003 du Canada.

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 devices for the Dash 8Xe recorder and Dash 8Xe Printer are the AC power inlet connectors. 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 warrant 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:

Declaration of Conformity
Declaration de Conformité
Ubereinstimmungserklärung
Dichiarazione di Conformità

ID	DoC-22834488
Manufacturer's name and address Nom et adresse du fabricant Hersteller Nome del costruttore	Astro-Med, Inc. 600 East Greenwich Avenue West Warwick, RI 02893 USA
Model No. Modele No. Model Nr. Modello No.	Dash 8Xe SCR-8 Dash 8XeR SCR-8R
Standards to which conformity is declared Standards auquel la conformité appartient Normen für welche Übereinstimmung erklärt wird Norme per le quali si dichiara la conformità	Safety: EN 61010-1 : 2001 EMC: EN 61326:1998, Class A EN 61000-3-2 : 1995
Application of Council Directives Application des Decisions du Conseil Anwendbar fur die Richtlinien Applicazione delle Direttive del Comitato	72/23/EEC 89/336/EEC 93/68/EEC

I, the undersigned, hereby declare that the equipment specified above conforms to the above Directive and Standard.

Je, Soussigné, déclare que l'équipment spécifié ci-dessus est en conformité avec la directive et le standard ci-dessus.

Ich, der unterzeichnende erkläre hiermit, daß das oben beschriebene Gerät den vorgenannten Richtlinien und Normen entspricht.

Il sottoscritto dichiara che l'apparecchio sopra specificato è conforme alle Direttive e Norme sopra specificate.

Steven Holbrook Quality Assurance Manager Astro-Med, Inc.	Steven G. Holbrook Date of issue: 07/31/2008 Place of issue: West Warwick, RI

European Contact: Your local Astro-Med, Inc. Sales and Service Office.

FRANCE - Astro-Med SNC, Parc d'Activities de Pissaloup, 1, rue Edouard Branly, 78190 Trappes (Tel. 331-3482-0900 Fax: 331-3482-0571 Email:AstroMedFrance@astromed.com)

GERMANY - Astro-Med GmbH, Senefelderstrasse 1/T6, D-63110 Rodgau (Tel. +49(0)6106-28368-0 Fax: 49-6106-771121 Email: AstroMedDeutschland@astromed.com)

UNITED KINGDOM - Astro-Med, Inc., Astro-Med House, 11 Whittle Parkway, Slough, SL1 6DQ. (Tel. 44-1628-668836 Fax: 44-1628-664994 Email: AstroMedUK@astromed.com)

Contents

Chapter 1: Getting started

This chapter provides a basic overview of the Dash 8Xe.

Introduction

The Dash 8Xe is a powerful and versatile data acquisition recording system that provides the capability to display, record, and review waveform data.

The system uses up to eight plug-in modules for signal input connections. Input modules are installed using the eight plug-in slots, which provide configuration flexibility for a variety of applications and future growth.

Data samples from up to eight signals can be acquired at rates up to 200,000 samples per second and streamed directly to a high-capacity hard drive.

Additionally, the Dash 8Xe has a scope card option that adds four channels with sample rates of up to ten million samples per second.

The Dash 8Xe utilizes a touch-screen as the main user interface. Many of the Dash 8Xe control buttons are customizable, providing the capability to modify the display based on the needs of the user or application.

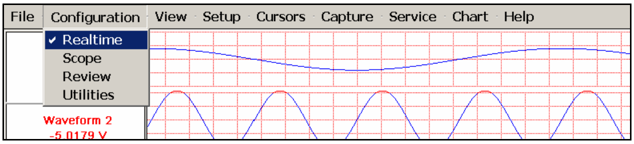
User interface

The menu bar and control panel displayed on the touch-screen provide access to the Dash 8Xe controls.

Menu bar

The menu bar is a group of drop-down menus located across the top of the display. All Dash 8Xe modes and features can be accessed from this menu, which behaves similar to menu bars in Windows-based applications. Options available from the menu bar 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. If the date and time are visible instead of the menu bar, choose the M button, which toggles the display of date/time and menu bar.

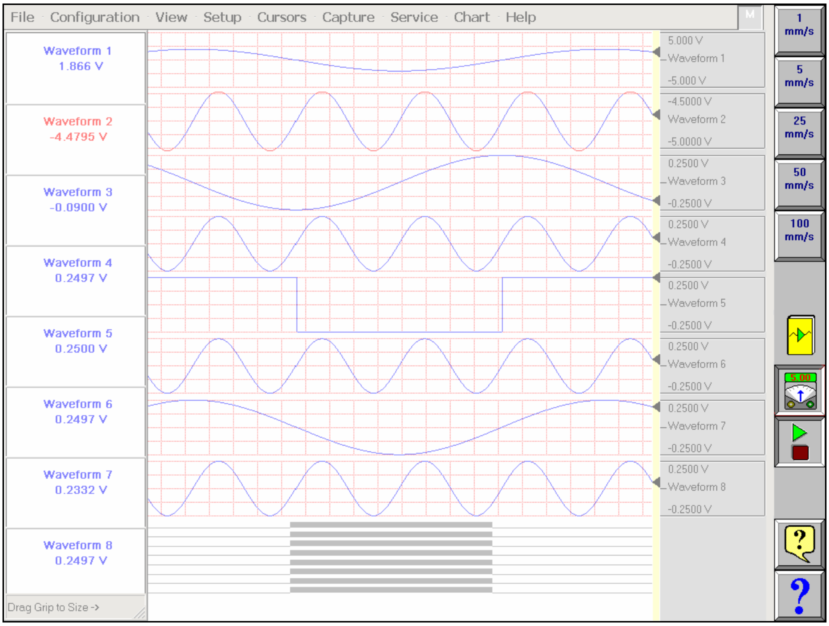


Note: The ">>" symbol in this manual indicates selections made using the menu bar. For example, "Configuration >> Realtime" indicates to choose Configuration, then Realtime from the menu bar.

Control panel

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

The following illustration displays the control panel.



Modes of operation

This section provides an introduction to Realtime, Scope, and Review modes.

Realtime mode

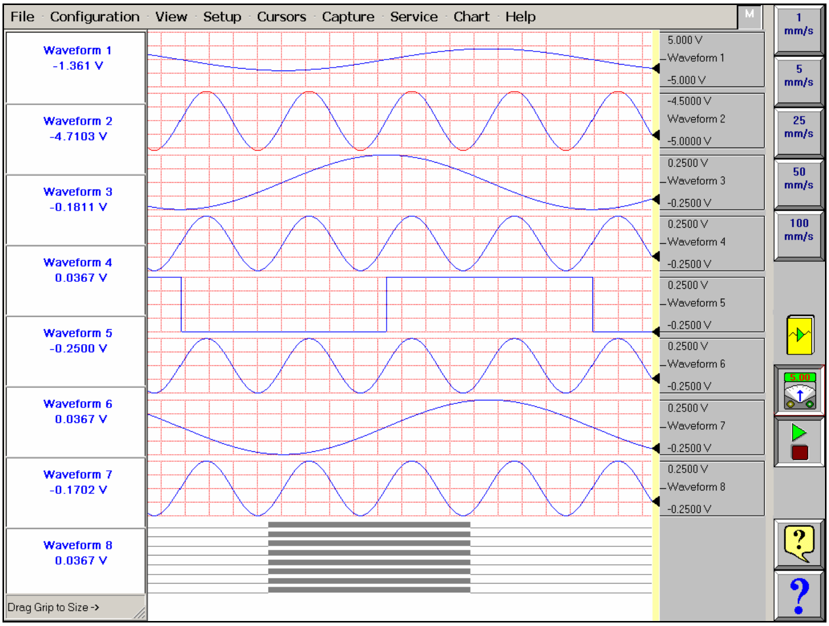
Realtime mode provides real-time waveform scrolling, monitoring, and data capture capabilities. Additionally, almost all system setup options are accessible from Realtime mode.

Use the following instructions to start Realtime mode from any other mode of operation.

To access 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.



Scope mode

Scope mode acts like a digital storage oscilloscope, providing high time-base resolution for viewing high-frequency signals.

If the optional DSO-8 scope card is installed, high-speed scope captures up to ten million samples per second can be acquired using Card mode.

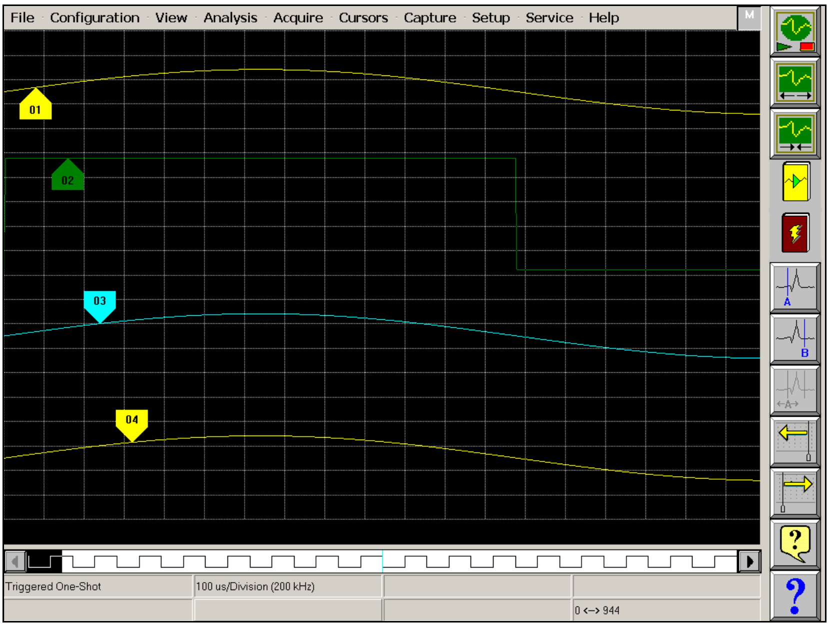
Use the following instructions to start Scope mode from any other mode of operation.

To access Scope mode:

1. Choose Configuration >> Scope from the menu bar.

   - If the DSO-8 option is not installed, choose Scope to enter scope mode.
   - If the DSO-8 option is installed, choose Standard to enter standard scope mode or Card to enter high-speed scope mode.

   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.



Review mode

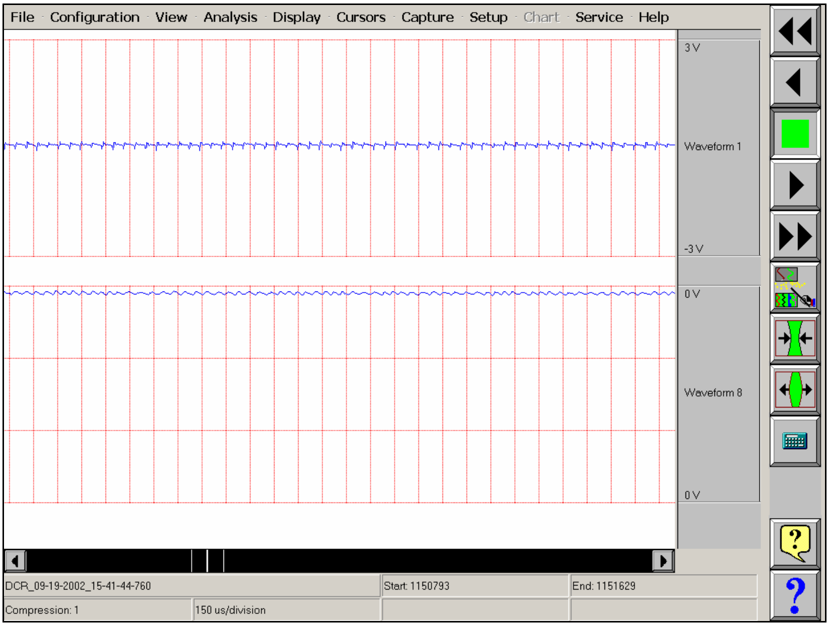
Review mode provides the capability to review and analyze saved data captures and scope captures.

Use the following instructions to start Review mode from any other mode of operation.

To access Review mode:

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



Help features

This section provides information about Dash 8Xe help resources.

Context help

The context help feature provides context sensitive, on-screen help for the Dash 8Xe. The help instructions provided vary based on the activity being performed.

To access context help from the menu bar:

1. Choose Help >> Context Help. Context help instructions for the current task will appear.

2. To close the help message, press any point on the screen.

To access context help from the control panel:

1. Choose the Context Help button.

   Context help instructions for the current task will appear.

2. To close the help message, press any point on the screen.

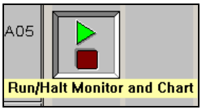
Icon help

The icon help feature provides on-screen help for Dash 8Xe icon buttons. This help feature provides brief descriptions of icon buttons.

To access icon help from the menu bar:

1. Choose Help >> Icon Help to activate the icon help function.

2. Choose any other icon button on the display. A short description of the button will appear.

   To view icon help for other buttons, choose them and read the descriptions.

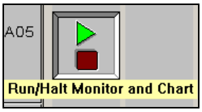
3. To exit icon help, choose Help >> Icon Help again.

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

To access icon help from the control panel:

1. Choose the Icon Help button to activate the icon help function.

2. Choose any other icon button on the display. A short description of the button will appear.



   To view icon help for other buttons, choose them and read the descriptions.

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

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

Online manual

The Adobe Portable Document Format (PDF) version of this manual is available on the Dash 8Xe for on-screen viewing.

To view the online manual:

1. Choose Help >> On-Line Help. 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 with the Dash 8Xe, contact Astro-Med, Inc. Technical Support via telephone at 877-867-9783 or e-mail at techserv@astromed.com.

Chapter 2: Hardware overview

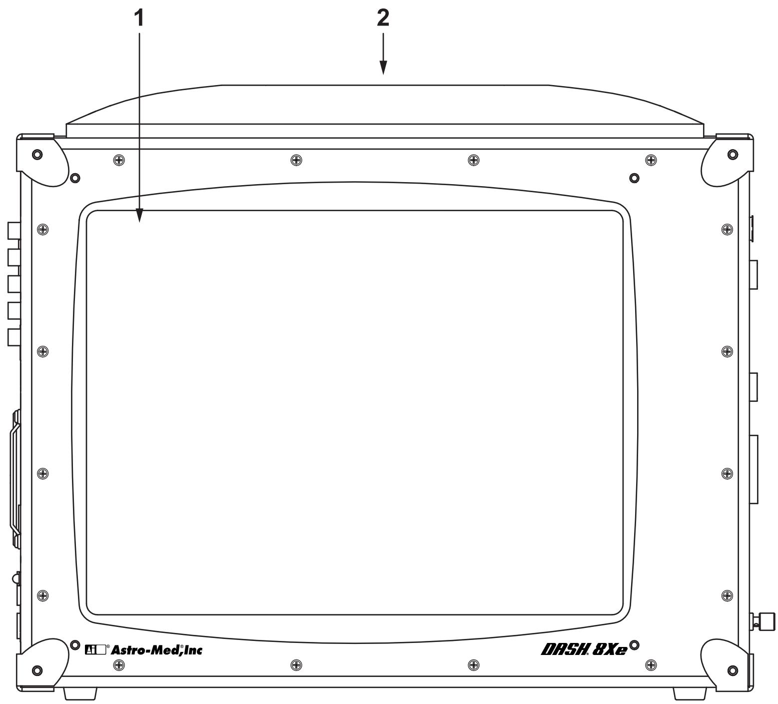
This chapter provides information about the physical components of the Dash 8Xe.

Dash 8Xe diagrams

This section provides a visual overview of the hardware characteristics of the Dash 8Xe. In addition to the diagrams, brief descriptions are included on various components.

Front view

The following diagram illustrates the front view of the Dash 8Xe.



1. Touch-screen
2. Carry handle

Touch-screen

The touch-screen serves as the user interface for the Dash 8Xe. On-screen menus and buttons are selected by touching the display. Alternatively, a stylus may be used 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.

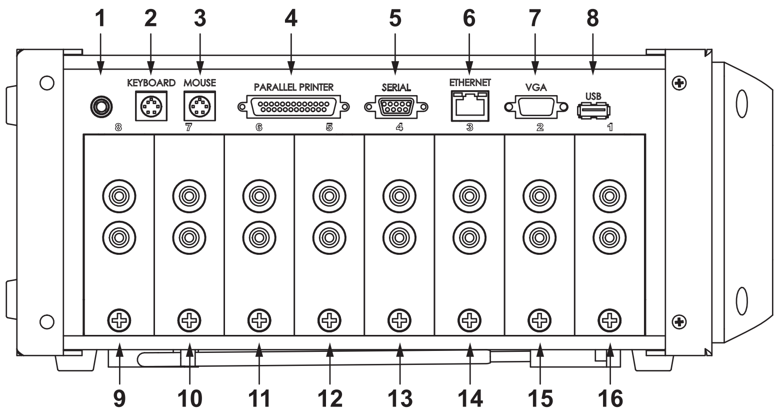
Cleaner should be sprayed on the cloth and not directly on the touch-screen. Spraying cleaner on the touch-screen could result in damage from liquid draining into the system.

Carry handle

The sturdy carry handle can be used while moving the Dash 8Xe.

Right side view

The following diagram illustrates the right side of the Dash 8Xe.



1. Ground connection
2. Keyboard port
3. Mouse port
4. Parallel port
5. Serial port
6. Ethernet port
7. VGA port
8. USB port
9. Input module 8
10. Input module 7
11. Input module 6
12. Input module 5
13. Input module 4
14. Input module 3
15. Input module 2
16. Input module 1

Ground connection

The ground connection is used to connect shields on input wiring, if needed.

Keyboard port

The keyboard port is used to connect a standard keyboard to the unit, if desired. Once connected, the keyboard can be used as an alphanumeric input device. Depending on the testing environment and needs, keyboard use may be more convenient than the touch-screen for some users or applications.

Mouse port
The mouse port is used to connect a standard mouse to the unit, if desired. Once connected, the mouse can be used as a pointing and selection device. Depending on the testing environment and needs, mouse use may be more convenient than the touch-screen for some users or applications.

Parallel port
The parallel port is used to connect a Microsoft ® Windows-compatible printer to the unit, if desired. This type of printer may be useful for printing screen captures.

Note: Using this type of printer may affect Realtime displays, and some printers will perform better than others. Astro-Med, Inc. will not guarantee full specifications while this type of printer is printing.

Serial port
The serial port is not currently used.

Ethernet port
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.

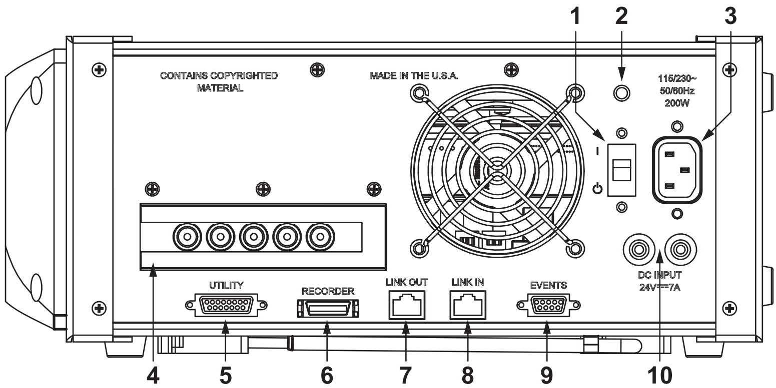
VGA port
The VGA port is used to connect the Dash 8Xe to an external monitor. The monitor must be capable of displaying a 1024 x 768 screen resolution.

USB port
The USB port is used to connect accessories to the unit. This port is typically used to connect portable memory storage devices.

Input modules
The analog input modules are used to connect signal inputs to the unit for measurement. Up to eight input modules can be installed at one time. The physical appearances of modules vary based on the input modules used. Several types of input modules are available to accommodate specific types of signal inputs.

Left side view

The following diagram illustrates the left side of the Dash 8Xe.



1. Power switch
2. Power indicator light
3. Power inlet
4. Scope card (optional)
5. Utility port
6. Recorder port
7. Link out port
8. Link in port
9. Event port
10. DC input

Power switch

The power switch is used to power-up and power-down the Dash 8Xe. The power-up sequence requires approximately two minutes.

Power indicator light

The power indicator light is used to visually indicate the power status of the Dash 8Xe. When the unit is operating properly with full power, this indicator will remain lit. During the power-down sequence, it will pulse for a few moments until the unit powers-down completely and turns off.

Power inlet

The power inlet is used to attach the power cable to the Dash 8Xe.

DSO-8 - Scope card (optional)

The optional scope card is used to provide a four-channel, high-speed scope mode. This scope mode is capable of sampling up to ten million samples per second.

Utility port
The utility port is used to provide external triggering and remote control of some functions.

Recorder port
The recorder port is used to connect the unit to the external thermal recorder printer.

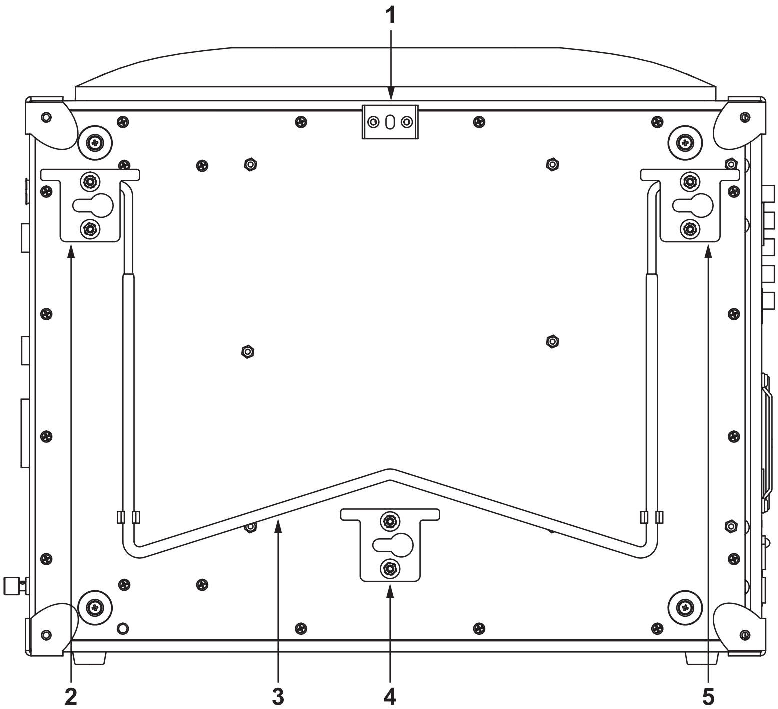
Link out/Link in ports
The link out and link in ports are used to connect multiple Dash 8Xe units. Connected units can sample and capture data simultaneously. Up to eight units can be linked in this manner.

Event port
The event port is used to connect binary event inputs. Up to eight event inputs can be connected via this port.

DC input
The DC input jacks are used to connect DC power to the unit.

Back view

The following diagram illustrates the back view of the Dash 8Xe.



1. Plate lock
2. Plate latch
3. Stand
4. Plate latch
5. Plate latch

Plate lock

The plate lock is used to lock accessories (cover/stand, thermal printer, etc.) into position on the back of the Dash 8Xe.

Plate latches

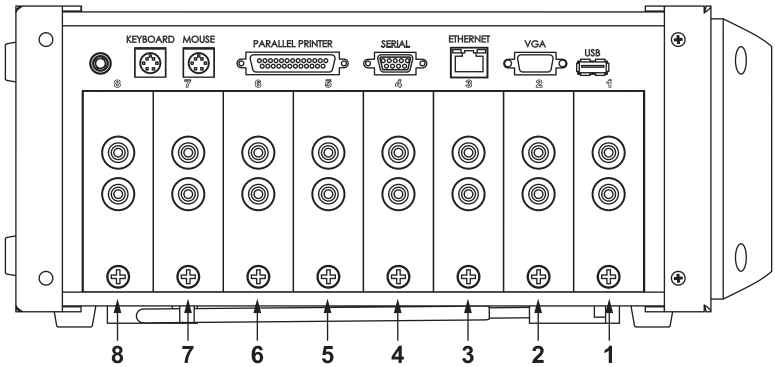
The three plate latches are used to connect accessories (cover/stand, thermal printer, etc.) to the back of the Dash 8Xe.

Stand

The built-in stand is used to position the Dash 8Xe on an angle.

Input module overview

The system uses up to eight plug-in modules for signal input connections. Input modules can be installed using the eight plug-in slots. The following diagram illustrates the layout of the Dash 8Xe signal input modules.



The input modules displayed in this diagram are used for illustration purposes. The appearances of input modules vary based on the module types installed.

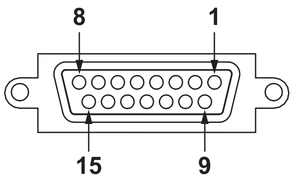
Note: For detailed information on input module types and connections, refer to *Chapter 3, Input modules*.

Pin configurations

This section describes pin configurations for the utility port and event port hardware components.

Utility port pin configuration

The following diagram illustrates the Dash 8Xe utility port pin configuration. All inputs are TTL or switch closure compatible.



1. External trigger input
2. Alarm output
3. Reserved (do not connect)
4. Reserved (do not connect)
5. External arm input
6. Security input
7. External abort input
8. Chart/page mark input
9. Trigger output
10. External motor clock input
11. External run/halt input
12. Common
13. Print demand, ID, or labels input
14. Five volt (100 Ω resistor)
15. External sample rate input

Pin 1 - 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.

Pin 2 - Alarm output
The alarm output can be used to signal when alarm conditions for selected signals occur. This is an active low output. The signal is TTL low during alarm signals. This pin can sink 16 mA.

Pin 3 - Reserved (do not connect)
No line should be connected to this pin.

Pin 4 - Reserved (do not connect)
No line should be connected to this pin.

Pin 5 - External arm input
The external arm input provides the capability to arm a data capture using an outside signal. A logic level low or a switch closure on this line for at least 100 ms will cause a data capture to be armed, provided there is storage available for the capture.

Pin 6 - 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.

Pin 7 - 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 Dash 8Xe to abort any data capture in progress.

Pin 8 - Chart/page mark input
The chart/page mark input provides the capability to print a full-page mark across the display or printed output.

Pin 9 - Trigger output
The trigger output pulses a TTL low in response to the Dash 8Xe recognizing a trigger. This line can be connected to the external trigger input line of another Dash 8Xe. In this case, the Dash 8Xe receiving the external trigger input signal will trigger in response to the same event as the unit sending the trigger output signal.

Pin 10 - External motor clock input
The external motor clock input provides the capability to use an external motor clock for controlling 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 11 - 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.

Pin 12 - Common
The common pin is used as ground.

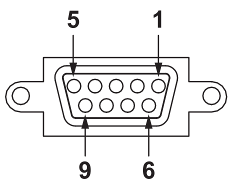
Pin 13 - Print demand, ID, or labels input
This input provides the capability to print the demand buffer, channel IDs, or channel labels on the external chart recorder. A logic level low or switch closure on this line for at least 100 ms will cause the selected item to be printed, provided the chart is running.

Pin 14 - Five volt (100 Ω resistor)
This pin provides five volts through a 100 Ω resistor.

Pin 15 - External sample rate input
This pin provides the capability to enter a sample rate for data captures via an external signal. External sample rates up to 100,000 Hz can be used. Rates approaching 200,000 Hz will work, but with sampling jitter of up to 5 μ sec. 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.

Event port pin configuration

The following diagram illustrates the Dash 8Xe event port pin configuration.



1. Event 1 input
2. Event 2 input
3. Event 3 input
4. Event 4 input
5. Event 5 input
6. Event 6 input
7. Event 7 input
8. Event 8 input
9. Ground

Event inputs (1-8)
Up to eight event inputs can be connected via the event port. Event signals have a high state (switch open) and a low state (switch closed), and are TTL and switch closure compatible.

Ground
This pin is used as ground.

Dash 8Xe cover

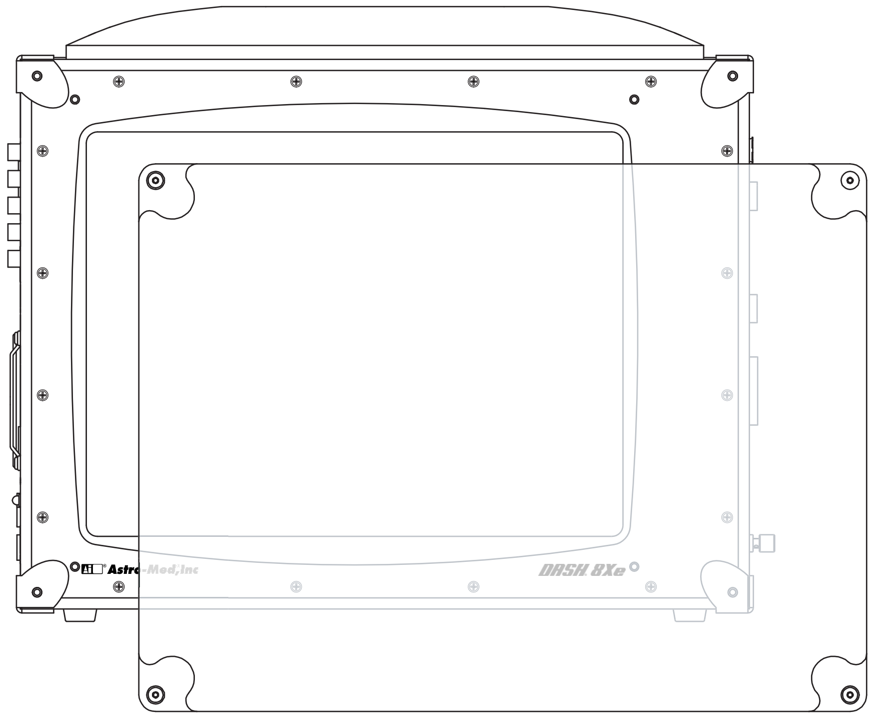
This section describes how to use the Dash 8Xe cover.

Using the cover

The clear plastic cover can be used to protect the touch-screen in rugged testing environments.

To use the cover:

1. Place the cover on the front of the Dash 8Xe and align the corner screw holes.

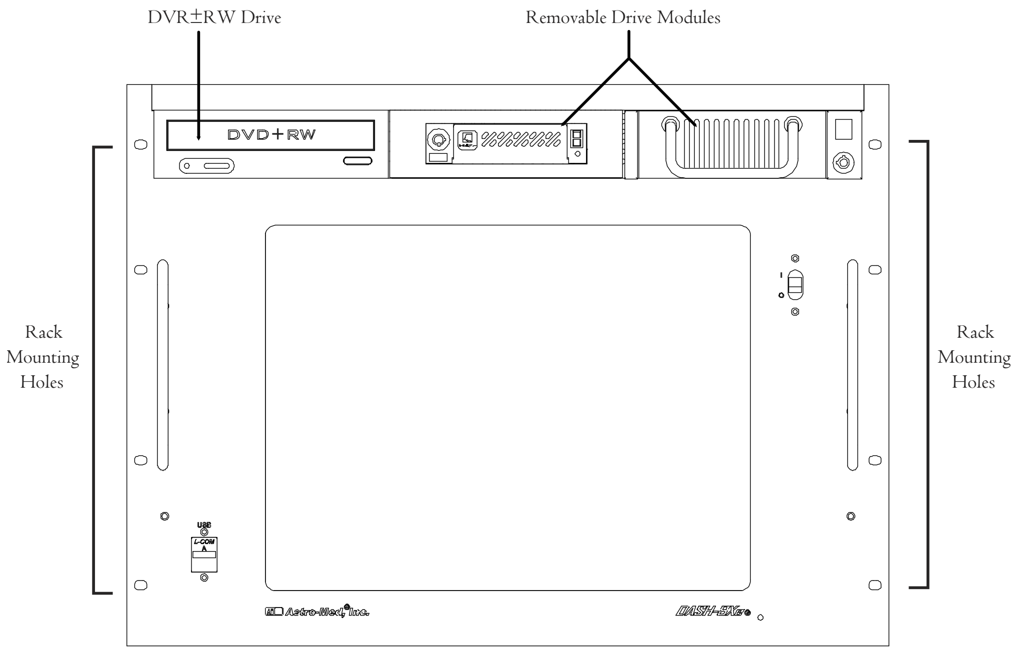


2. Tighten the corner screws to fasten the cover to the Dash 8Xe.

Dash 8XeR (rack mounted version)

The Dash 8XeR is a rack mounted version of the Dash 8Xe. This version of the device is easily mounted in a rack enclosure, and includes two removable hard drive modules and a DVD ± RW drive.

Note : For information on the rack mount version of the SCR-8 chart recorder, see Chapter 4 : Optional Hardware.



Mounting the Dash 8XeR

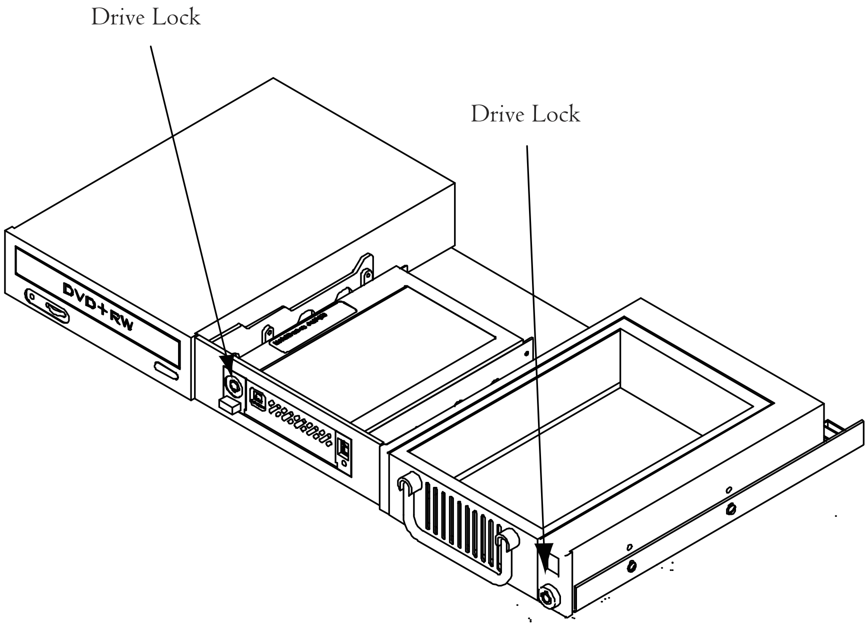
Use the following procedure to mount the Dash 8XeR in a rack enclosure.

To mount the Dash 8XeR:

1. Place the Dash 8XeR into a standard 19 inch rack frame.
2. With a total of eight bolts, attach the unit to the rack via the four rack mounting holes (pictured above) located on each side.

Using the Dash 8XeR Removable Drives

The drive modules included with the Dash 8XeR are easily removable, and are ideal for storing secure data. Use the following procedure to remove either of the two drives from the rack mounted unit.



To remove a drive module:

1. If you have not already done so, power down the Dash 8XeR unit completely. Never remove either of the drive modules while the unit is running.

2. Once the unit is off, insert the included drive key into the circular drive lock (pictured above), and turn the key clockwise to unlock the drive.

3. Gently pull the drive towards you (by the handle, if the module has one) to remove the drive.

4. When returning the drive to its slot, make sure to relock the drive by turning the key counter-clockwise -- the Dash 8XeR unit will not start until both drives are installed and locked.

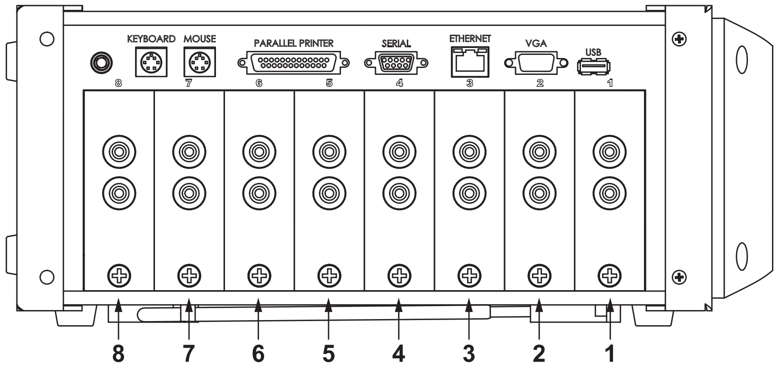
Chapter 3: Input modules

This chapter provides detailed information about the Dash 8Xe input modules.

Module locations

The system uses up to eight plug-in modules for signal input connections. Input modules can be installed using the eight plug-in slots located on the side of the Dash 8Xe.

The following diagram illustrates the layout of the Dash 8Xe signal input modules.



1. Input module 1
2. Input module 2
3. Input module 3
4. Input module 4
5. Input module 5
6. Input module 6
7. Input module 7
8. Input module 8

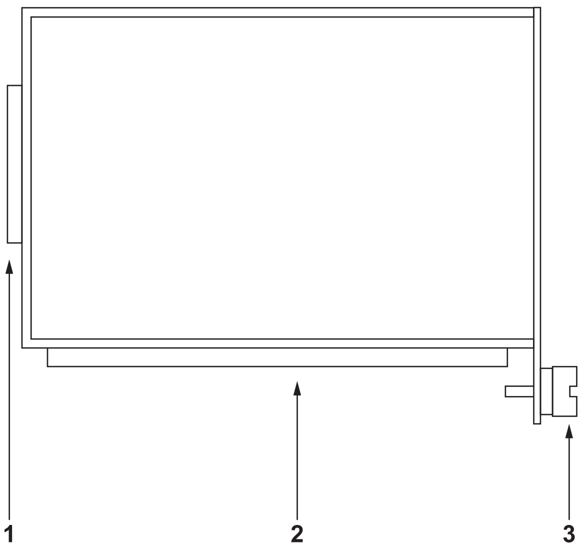
Note: The input modules displayed in this diagram are used for illustration purposes. The appearance of input modules will vary based on the module types installed.

Refer to *Appendix A: Specifications* for maximum allowable voltages on signal inputs.

Installing and removing modules

This section provides information on installing and removing Dash 8Xe input modules.

The following diagram illustrates the connection points on input modules.



1. Connector
2. Slot guide
3. Locking screw

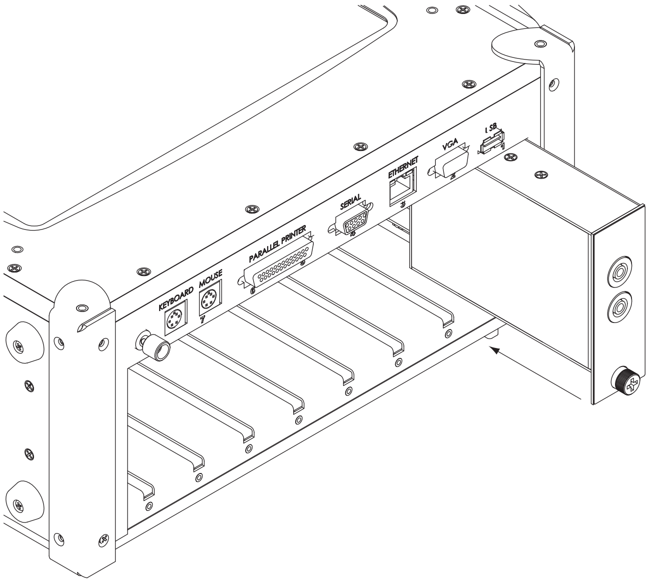
Connector
The connector is used to electronically connect the module to the Dash 8Xe.

Slot guide
The slot guide is used to align the module and guide it into position in the Dash 8Xe.

Locking screw
The locking screw is used to fasten the module into the Dash 8Xe.

To install a module:

1. Ensure that the Dash 8Xe power is off.
2. Select an empty module slot on the Dash 8Xe.
3. Insert the module into the slot. Ensure that the module slot guide is aligned with the opening in the Dash 8Xe slot. Then slide the module fully into the unit.



4. Tighten the module locking screw to fasten the module into position.
5. Turn on the Dash 8Xe power.
6. After the power-up process, use the input default utility to set all modules to a valid state.

   Choose Configuration >> Utilities to open the Utilities screen. Then choose Defaults >> Inputs from the menu bar.

Note: Defaulting the signal input settings restores the module settings to factory default settings. For modules that are not changed, you may want to note the input settings so you can conveniently change the settings back to the desired values.

To remove a module:

1. Ensure that the Dash 8Xe power is off.
2. Loosen the locking screw on the module being removed.
3. Slide the module out of the unit.

NDV1 - Non-isolated differential input module

This section provides information on the non-isolated differential input module (NDV1).

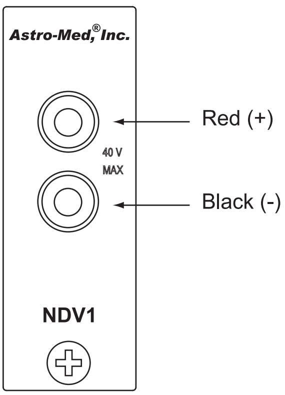
Note: The use of guarded banana test leads will eliminate the possibility of introducing a ground to the equipment under test if a regular banana test lead should come in contact with the recorder metal case.

Specifications

For detailed NDV1 module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the NDV1 module.



IHV1 - Isolated single ended high voltage input module

This section provides information on the isolated single ended high voltage input module (IHV1).

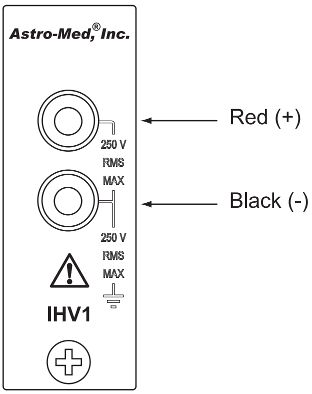
Note: The use of guarded banana test leads will eliminate the possibility of introducing a ground to the equipment under test if a regular banana test lead should come in contact with the recorder metal case.

Specifications

For detailed IHV1 module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the IHV1 module.



IBR1 - Isolated bridge input module

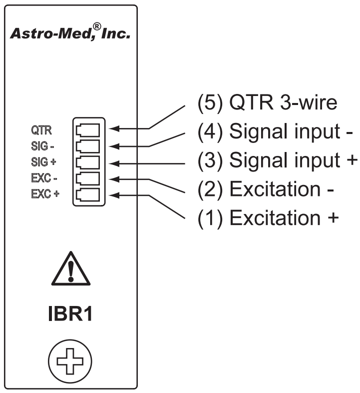
This section provides information on the isolated bridge input module (IBR1).

Specifications

For detailed IBR1 module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the IBR1 module.



Use the supplied mating connector (Astro-Med part number 25460-105) to attach 30-16 AWG wires to the IBR1.

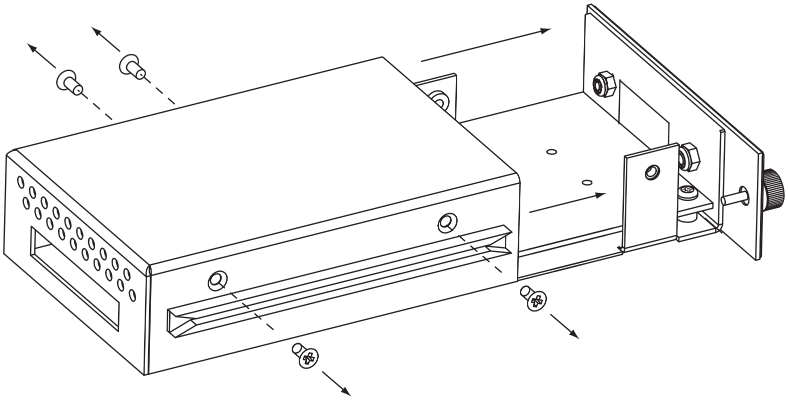
Internal bridge completion resistors

For some bridge connections, bridge completion resistors will be necessary. These resistors may be placed externally on fork headers internal to the IBR1. The internal board of the IBR1 provides four slots for connecting these resistors.

Use the following instructions to install internal bridge completion resistors.

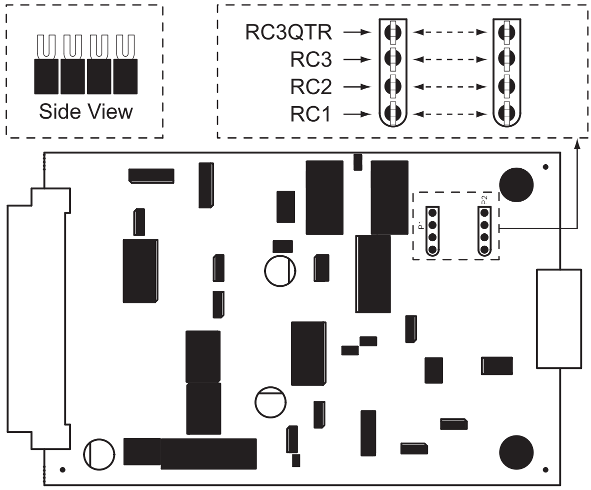
To install internal bridge completion resistors:

1. Remove the four screws from the top and bottom of the module. Then slide the face away from the module enclosure.



   The module board will be exposed.

2. Locate the internal bridge completion resistor slots, as indicated in the following diagram.



3. Position a resistor across each slot, connecting the two U-shaped prongs associated with each resistor. Solder the connection points to install and secure the resistors in place, then trim off all excessive lead lengths.

4. Replace the module enclosure.

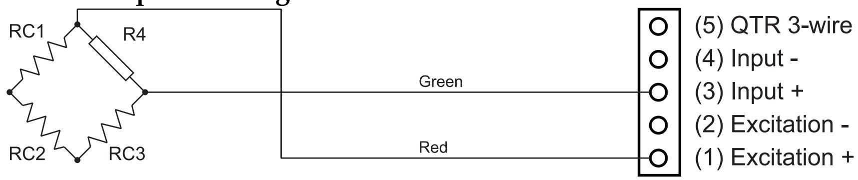
Bridge wiring diagrams

The following diagrams illustrate bridge wiring connections.

⚠ Dangerous voltages connected to an input pin may be present on other IBR1 pins. Care must be taken to prevent contact with IBR1 pin connections to prevent personal injury or damage to equipment.

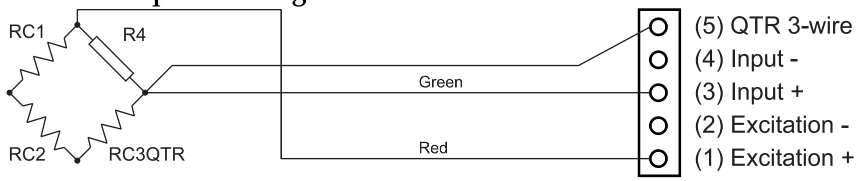
The colors specified in the following diagrams are used for reference only.

Two wire quarter bridge



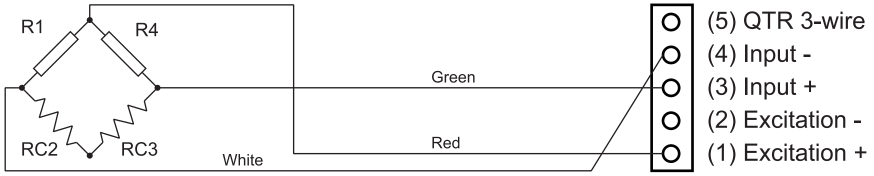
Internal bridge completion resistors RC1, RC2, and RC3 must be installed inside the IBR1 for this bridge application.

Three wire quarter bridge



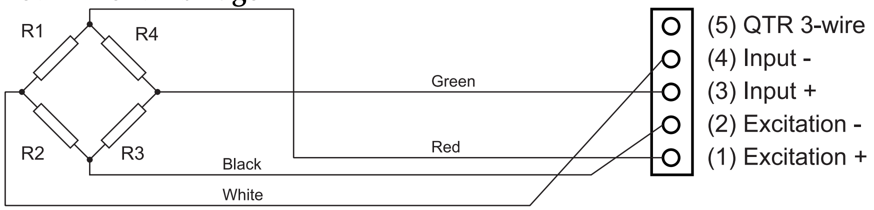
Internal bridge completion resistors RC1, RC2, and RC3QTR must be installed inside the IBR1 for this bridge application.

Three wire half (upper active) bridge



Internal bridge completion resistors RC2 and RC3 must be installed inside the IBR1 for this bridge application.

Four wire full bridge



No internal bridge completion resistors are installed for this bridge application.

IHV2 - Isolated differential very high voltage module

This section provides information on the isolated differential very high voltage module (IHV2).

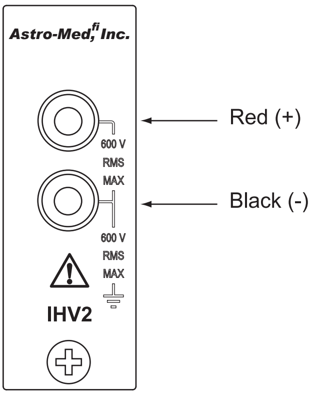
Note: The use of guarded banana test leads will eliminate the possibility of introducing a ground to the equipment under test if a regular banana test lead should come in contact with the recorder metal case.

Specifications

For detailed IHV2 module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the IHV2 module.



ITCU - Universal thermocouple input module

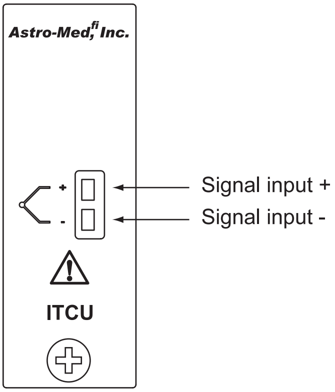
This section provides information on the universal thermocouple input module (ITCU).

Specifications

For detailed ITCU module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the ITCU module.



Temperature ranges

The following list describes temperature measurement ranges based on thermocouple type.

Type	Range (°C)
J	-210 to 1200
K	-200 to 1372
E	-200 to 1000
T	-200 to 400
N	-200 to 1300
B	250 to 1820
R	-20 to 1768
S	-20 to 1768

Application notes

The following notes provide background information about each thermocouple type.

Note: This information is intended primarily as an overview. For a more detailed explanation of thermocouple types and applications, refer to one of the many texts available on the subject of thermocouples.

Type J - This type is a popular thermocouple for general use. However, this type can rust in oxidizing environments and sub-zero temperatures.

Type K - This type is a popular thermocouple recommended for use in oxidizing environments.

Type E - This type has the highest voltage output of standard thermocouple types and does not corrode at sub-zero temperatures.

Type T - This type is often used in moist or sub-zero temperature environments. It provides excellent corrosion resistance.

Type N - This type is a newer, general purpose type which offers higher stability than standard types such as J, K, E, or T.

Type B - This type is used in high temperature environments. However, this type can be contaminated easily.

Type R - This type is a Japanese standard, often used in high temperature, oxidizing environments. However, this type can be contaminated easily.

Type S - This type is an international lab standard and resists oxidation and corrosion. However, this type can be contaminated easily.

Composition

The following list describes the composition based on thermocouple type.

Type	Composition
J	Fe versus Cu-Ni alloy
K	Ni-Cr alloy versus Ni-Al alloy
E	Ni-Cr alloy versus Cu-Ni alloy
T	Cu versus Cu-Ni alloy
N	Ni-Cr-Si alloy versus Ni-Si-Mg alloy
B	Pt-30% Rh versus Pt-6% Rh
R	Pt-13% Rh versus Pt
S	Pt-10% Rh versus Pt

Wire coloring standards

The following table describes the ANSI and IEC wire color standards based on thermocouple type.

Type	ANSI Wire Color Standard		IEC Wire Color Standard	
	Positive	Negative	Positive	Negative
J	White	Red	Black	White
K	Yellow	Red	Green	White
E	Purple	Red	Violet	White
T	Blue	Red	Brown	White
N	Orange	Red	Pink	White
B	Gray	Red	-	White
R	Black	Red	Orange	White
S	Black	Red	Orange	White

IRTD - Isolated resistance temperature detector input module

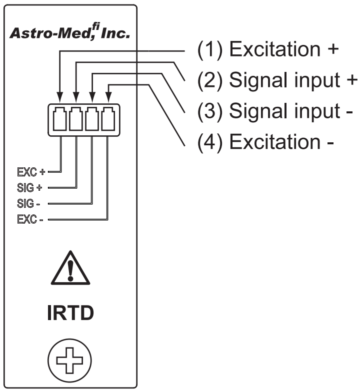
This section provides information on the isolated resistance temperature detector (IRTD) input module.

Specifications

For detailed IRTD module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the IRTD module.



Use the supplied mating connector (Astro-Med part number 25460-104) to attach 30-16 AWG wires to the IRTD.

Note: For two-wire and three-wire connections, a heavier wire gauge is recommended for longer cable runs to minimize measurement errors.

Internal resistors

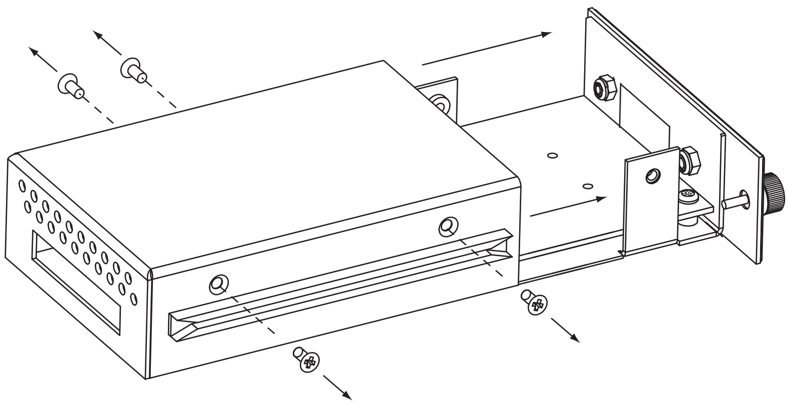
Two zero-ohm resistors are installed in the IRTD module to allow the use of two wire RTDs. Depending on the chosen wiring configuration, it may be necessary to remove one or both of these resistors.

Note: Refer to the following section for wiring diagrams and information about associated resistor removal.

Use the following instructions to remove internal resistors.

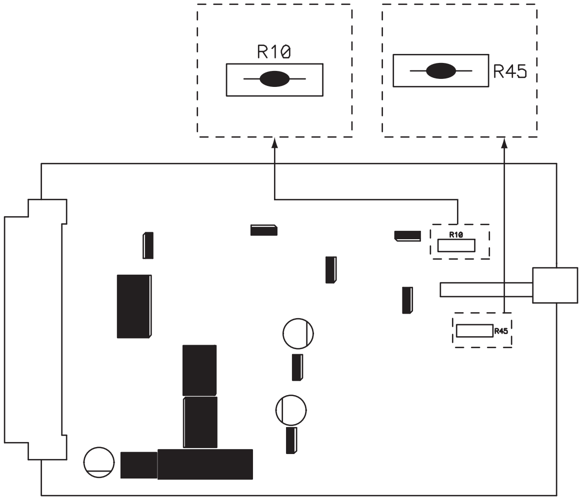
To remove internal resistors:

1. Remove the four screws from the top and bottom of the module. Then slide the face away from the module enclosure.



   The module board will be exposed.

2. Locate the internal resistors, as indicated in the following diagram.



3. Remove the resistors applicable to the desired wiring configuration.

Note: Refer to the following section for wiring diagrams and information about associated resistor removal.

---

4. Replace the module enclosure.

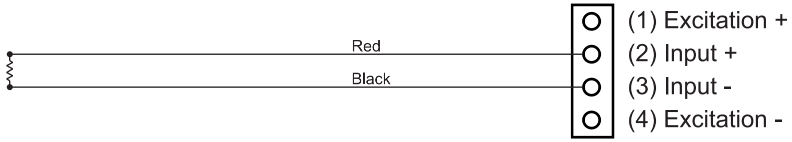
Wiring diagrams

The following diagrams illustrate typical wiring connections.

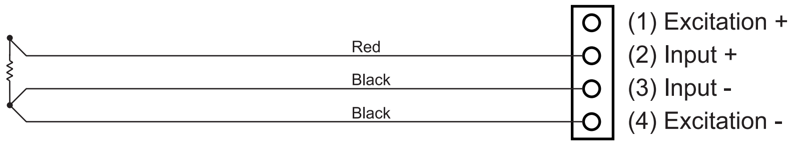
Dangerous voltages connected to an input pin may be present on other IRTD pins. Care must be taken to prevent contact with IRTD pin connections to prevent personal injury or damage to equipment.

The colors specified in the following diagrams are used for reference only.

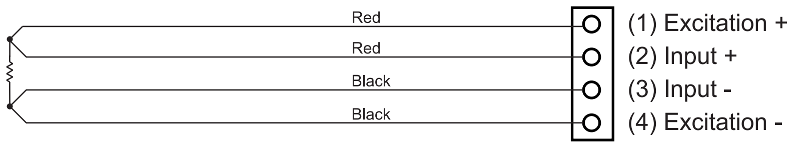
Two wire (R10 and R45 zero ohm resistors installed)



Three wire (R45 zero ohm resistor removed)



Four wire (R10 and R45 zero ohm resistors removed)



IHV3 - Isolated single ended high voltage module

This section provides information on the isolated single ended high voltage module (IHV3).

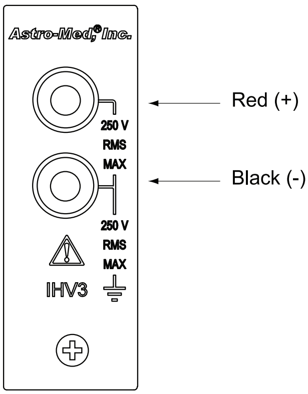
Note: The use of guarded banana test leads will eliminate the possibility of introducing a ground to the equipment under test if a regular banana test lead should come in contact with the recorder metal case.

Specifications

For detailed IHV3 module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the IHV3 module.



IDCV - Isolated high accuracy, wide dynamic range DC Voltmeter input module

This section provides information on the Isolated high accuracy, wide dynamic range DC Voltmeter input module.

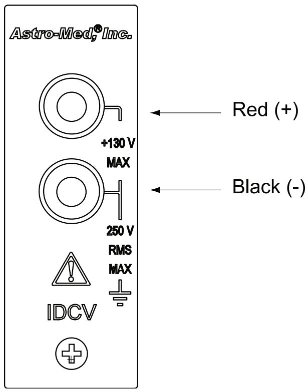
Note: The use of guarded banana test leads will eliminate the possibility of introducing a ground to the equipment under test if a regular banana test lead should come in contact with the recorder metal case.

Specifications

For detailed IDCV module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the IDCV module.



IDCV1 - Isolated high accuracy, wide dynamic range DC Voltmeter with 160V attenuator

This section provides information on the Isolated high accuracy, wide dynamic range DC Voltmeter with 160V attenuator input module.

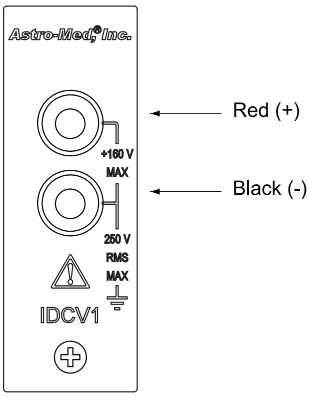
Note: The use of guarded banana test leads will eliminate the possibility of introducing a ground to the equipment under test if a regular banana test lead should come in contact with the recorder metal case.

Specifications

For detailed IDCV1 module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the IDCV1 module.



LIVM - Low Impedance Voltage Mode amplifier input module

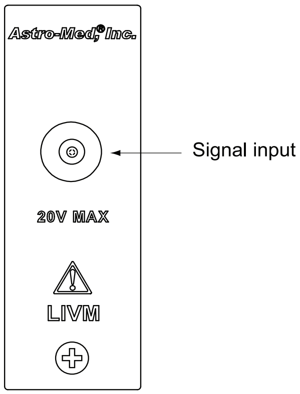
This section provides information on the Low Impedance Voltage Mode amplifier input module.

Specifications

For detailed LIVM module specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the Dash 8Xe via the LIVM module.



Application Notes

The LIVM module is designed to work with many IEPE (Integrated Electronics Piezo Electric) sensors designed to be used with constant current power supplies. These sensors are often called LIVM (Low Impedance Voltage Mode) sensors and are often sold under brand names such as ICP™ (trademark name of PCB Piezoelectronics / IMI Sensors), Piezotron™, Picotron™, PiezoBeam™, and K-Shear™ (trademark names of Kistler), and Isotron™ (trademark name of Endevco).

Chapter 4: Optional hardware

This chapter provides detailed information about optional hardware for the Dash 8Xe.

DSO-8 - High speed scope option

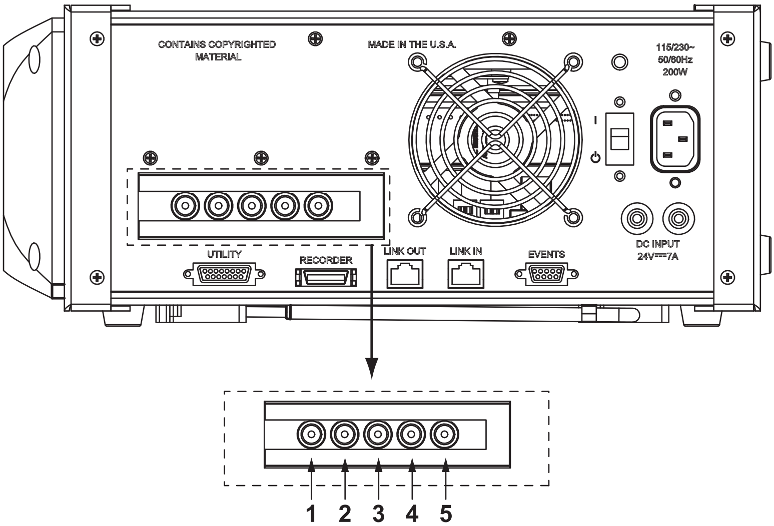
The DSO-8 high-speed scope option allows four additional input channels to be sampled at rates up to ten million samples per second each.

Specifications

For detailed DSO-8 specifications, refer to *Appendix A: Specifications*.

Input diagram

Use the following diagram to connect inputs to the DSO-8 scope card.



1. Signal input #1
2. Signal input #2
3. Signal input #3
4. Signal input #4
5. External trigger input

Signal inputs (1-4)

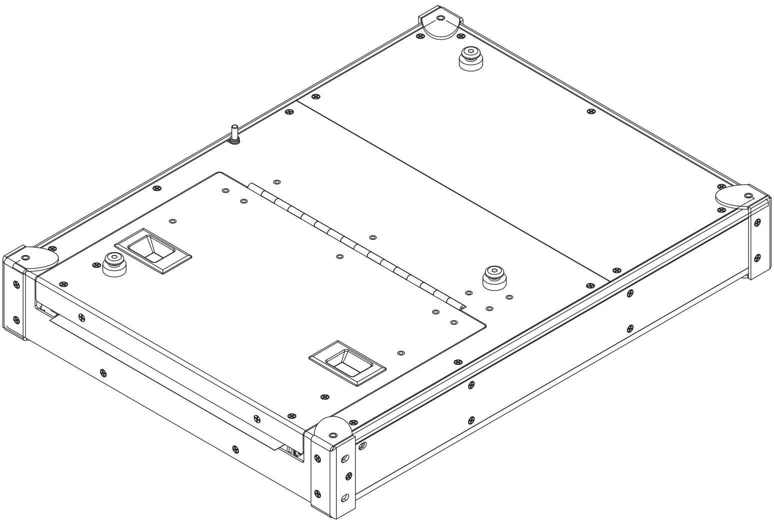
The signal inputs are used to connect signals for measurement. A BNC connection type is used to connect signals to the Dash 8Xe.

External trigger input

The external trigger input provides the capability to invoke a trigger using an outside signal. This input is TTL compatible.

SCR-8 - Thermal printer

The SCR-8 is an optional thermal chart recorder that produces printed output.



Installing the thermal printer

Use the following instructions to install the SCR-8 printer.

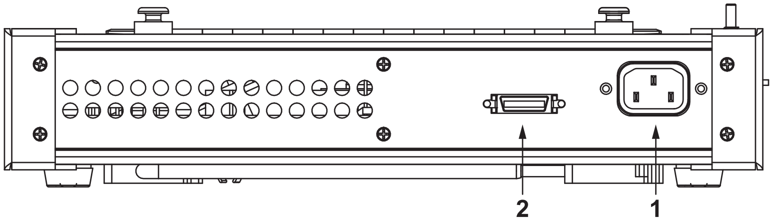
To install the thermal printer:

1. Ensure the power for the Dash 8Xe and SCR-8 is off.

Warning: Do not connect the printer when the Dash 8Xe is powered on.

2. Connect the power cable from the printer to an outlet. Then connect the printer cable from the Dash 8Xe recorder port to the printer's port.

   The power inlet (1) and recorder port (2) are noted in the following diagram.



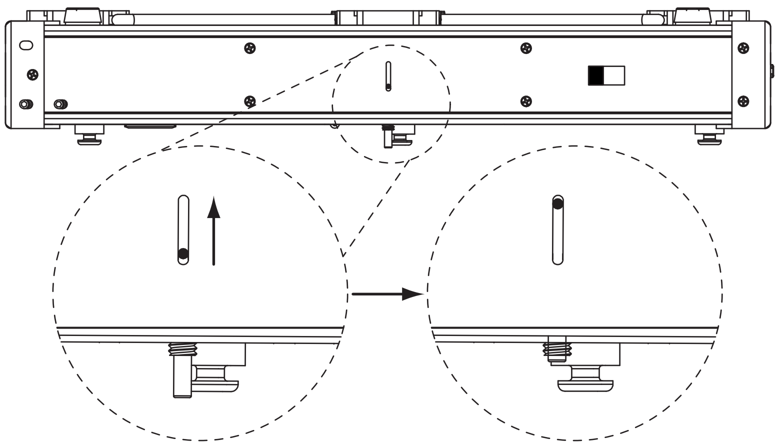
3. Turn on the SCR-8 power. Then turn on the Dash 8Xe power.

Attaching the thermal printer to the Dash 8Xe

For convenience and portability, the thermal printer can be attached to the back of the Dash 8Xe. This is an optional process, as the printer can be operated while detached from the Dash 8Xe.

To attach the thermal printer to the Dash 8Xe:

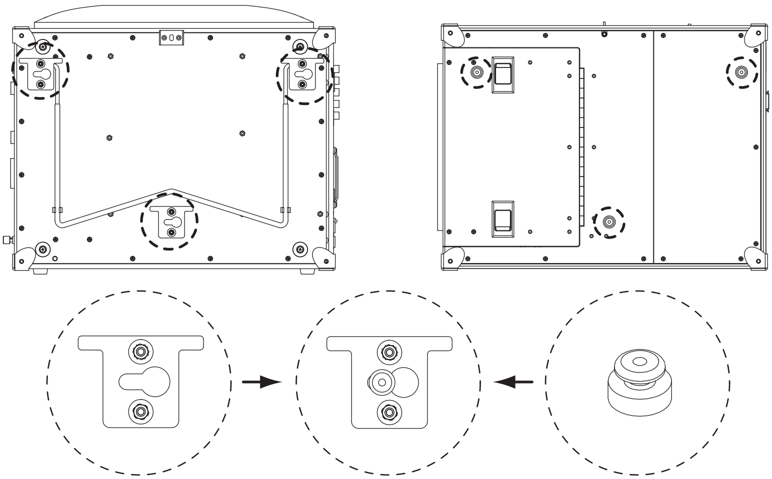
1. Press and hold the locking pin lever located on the top of the printer. When the pin lever is pressed, the locking pin on the front of the printer will retract, as indicated in the following diagram.



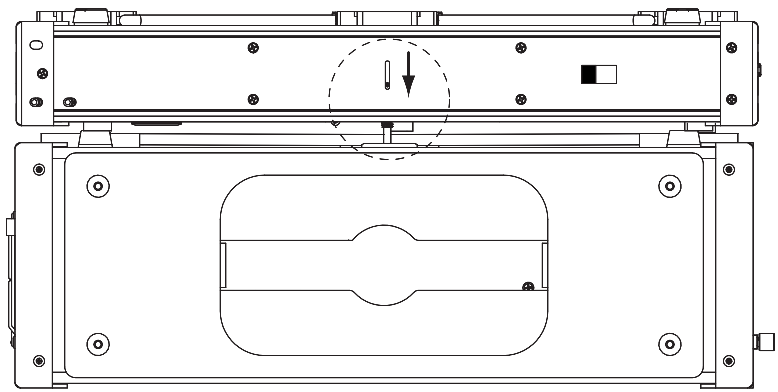
   This pin will need to remain retracted until the printer is placed into position on the back of the Dash 8Xe.

2. While facing the back of the Dash 8Xe, connect the printer to the back of the Dash 8Xe using the plate latches and latch buttons.

   Three plate latches are located on the back of the Dash 8Xe. Similarly, three latch buttons are located on the face of the printer. Insert the printer's latch buttons into the circular holes in the Dash 8Xe latch plates. Then slide the printer to the left to secure the printer into position.



3. Release the locking pin lever on the top of the printer. The locking pin will extend into the plate lock on the back of the Dash 8Xe. The printer will be locked into position.



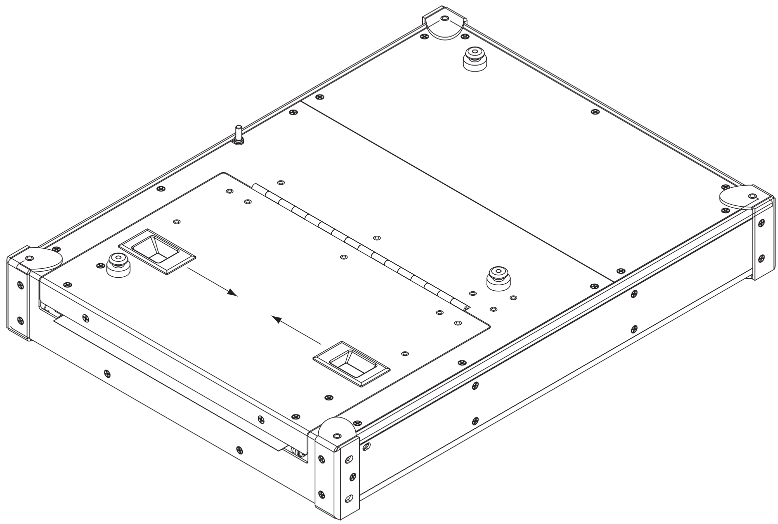
Loading paper

The SCR-8 utilizes Astro-Med Dash8 thermal paper, which is a z-fold format.

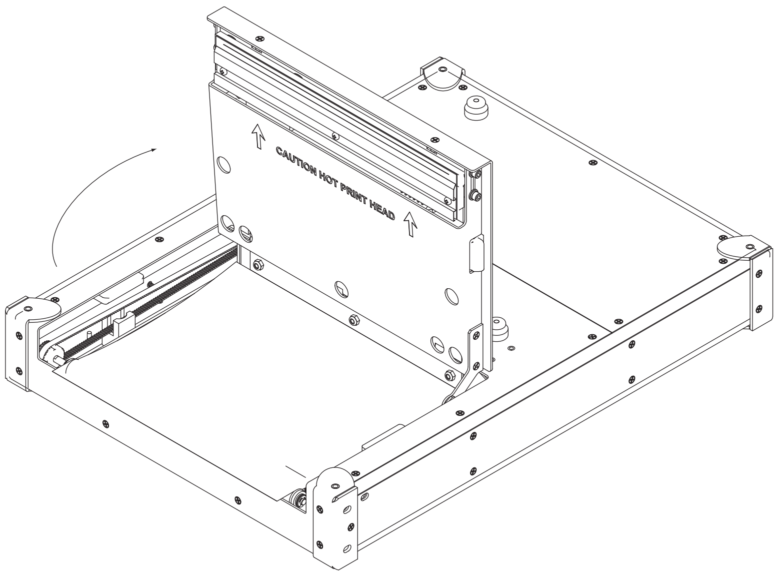
To load paper:

1. Detach the printer from the Dash 8Xe, if necessary.

2. Slide the two sliding latches toward the center of the printer.



3. While holding the sliding latches in position, pull outward to open the paper door.



The printhead may be hot. Do not touch any part of the printhead, as personal injury or printhead damage can occur.

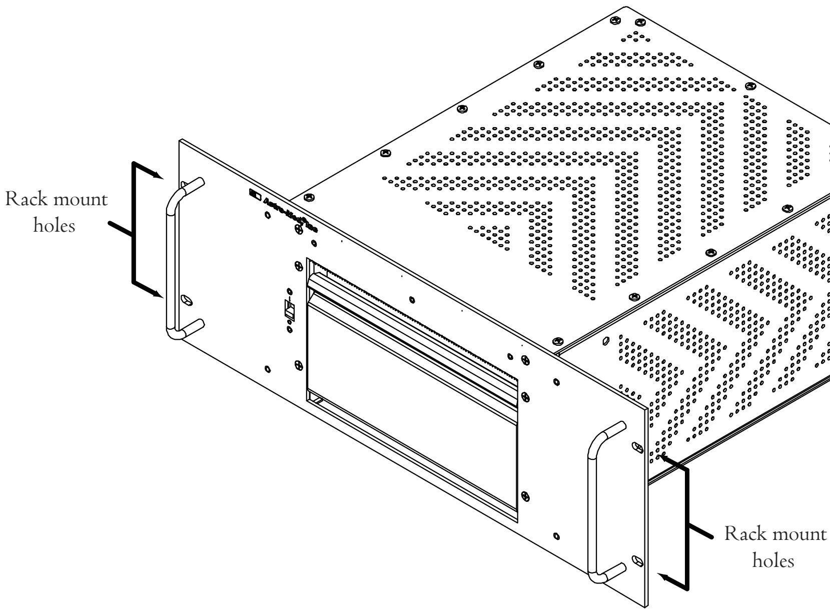
4. Load the Z-fold paper in the paper compartment. Then position the paper with a portion overlapping the roller near the paper exit.

5. Close the paper door. Then slide the latches outward to lock the door into position.

SCR-8R - Rack mount thermal printer

The SCR-8R is a rack mount version of the optional SCR-8 thermal chart recorder that combines the functionality of the SCR-8 with a convenient rack mount design.

Note : For more information on the rack mount version of the Dash 8XeR, see Chapter 2 : Hardware Overview.



Installing the thermal printer

Use the following procedures to connect the printer and install it in a rack enclosure.

To install the SCR-8R in a rack enclosure:

1. Place the SCR-8R into a standard 19 inch rack frame.

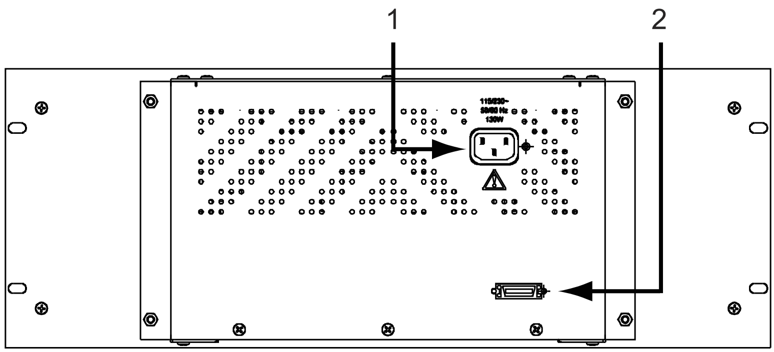
2. With a total of four bolts, attach the unit to the rack via the two rack mounting holes (pictured above) located on each side.

To connect the SCR-8R thermal printer to the Dash 8XeR:

1. Ensure the power for the Dash 8XeR and SCR-8R is off.

2. Connect the power cable from the printer to an outlet. Then connect the printer cable from the Dash 8XeR recorder port to the printer's port.

   The power inlet (1) and recorder port (2) are noted in the following diagram.



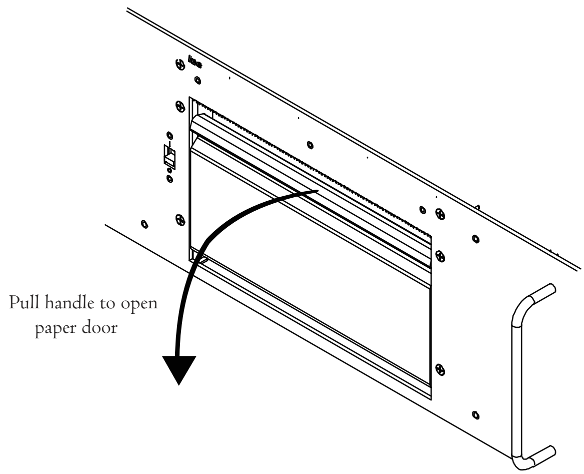
3. Locate the power switch on the front of the unit, near the paper door, and turn on the SCR-8R power. Then turn on the Dash 8XeR power.

Loading paper

The SCR-8R utilizes Astro-Med Dash8 thermal paper, which is a z-fold format.

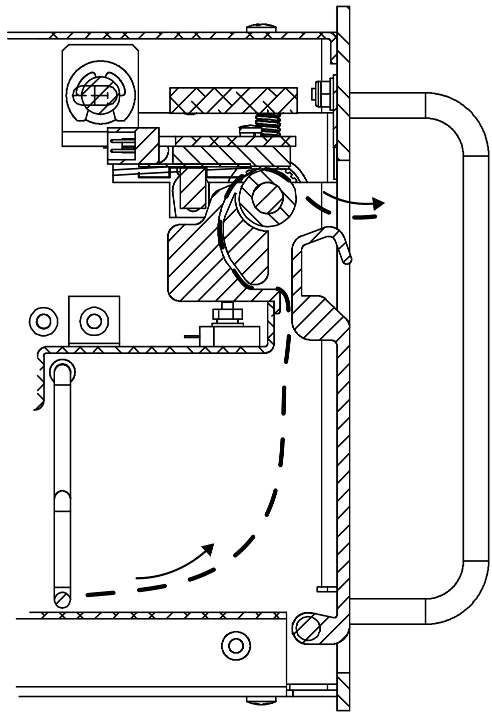
To load paper:

1. Pull the paper door handle forward to open the paper door.



⚠ The printhead may be hot. Do not touch any part of the printhead, as personal injury or printhead damage can occur.

2. Load the Z-fold paper in the paper compartment. Then position the paper with a portion overlapping the roller near the paper exit. The proper paper path is illustrated below.



(Side View)

3. Close the paper door.

EHD-USB - External hard drive

The EHD-USB is a USB-based external hard drive. The Dash 8Xe should be powered-down completely prior to installing or removing the EHD-USB. When the EHD-USB is connected, the Dash 8Xe will assign it a drive letter.

To install the EHD-USB:

1. Connect the EHD-USB to the Dash 8Xe via the USB port.

2. Turn the EHD-USB power on.

ECD-RW - External CD-RW drive

The ECD-RW is a USB-based external CD-RW drive. The Dash 8Xe should be powered-down completely prior to installing or removing the ECD-RW. When the ECD-RW is connected, the Dash 8Xe will assign it a drive letter.

To install the ECD-RW:

1. Ensure that the Dash 8Xe power is off.

2. Connect the ECD-RW to the Dash 8Xe via the USB port.

3. Turn the Dash 8Xe power on.

USB flash memory drive

USB flash memory drive accessories are compact, USB-based removable disks available in a variety of storage capacities.

To install a USB flash memory drive:

1. Connect the drive to the Dash 8Xe via the USB port.

   No drivers are necessary. When a USB flash drive is connected, the Dash 8Xe will assign it a drive letter.

External monitor

An external monitor can be connected to the Dash 8Xe via the VGA port. When installed, the monitor will display the contents of the Dash 8Xe touch-screen.

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

To install an external monitor:

1. Connect the external monitor to the VGA port located on the side of the Dash 8Xe.

   The drivers for standard monitors are pre-installed. No additional installation steps are necessary.

Mouse and keyboard

Depending on the testing environment and needs, mouse and/or keyboard use may be more convenient than the touch-screen for some users or applications.

Note: The mouse and keyboard provide an additional method for controlling the Dash 8Xe. The touch-screen will remain active, regardless of whether a mouse and keyboard are installed.

To install a mouse and keyboard:

1. Connect the desired mouse and/or keyboard to the Dash 8Xe.

   - To install a mouse, connect it to the mouse input located on the side of the Dash 8Xe.

- To install a keyboard, connect it to the keyboard input located on the side of the Dash 8Xe.

The Dash 8Xe provides built-in support for a variety of standard mice and keyboards. Installing drivers is not necessary.

2. Use the mouse and keyboard.

   - The mouse can be used as a pointing and selection device. By default, the mouse pointer is invisible. To display the mouse pointer, press the right mouse button.

   - The keyboard can be used as an alphanumeric input device while a keypad or number pad is visible on the display.

Windows-based printers

The Dash 8Xe supports Windows-based printers via the parallel port. 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 will. Astro-Med will not guarantee full specifications while a Windows-based printer is printing.

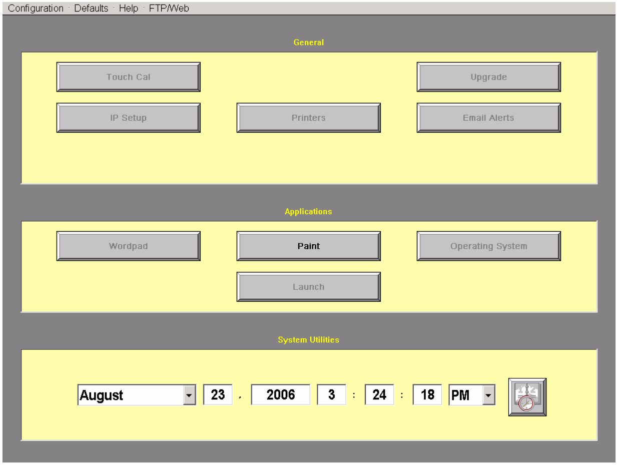
The installation process utilizes the Dash 8Xe parallel port and the printer installation and setup features of the Microsoft ® Windows operating system.

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 Dash 8Xe. A USB-based memory storage device can facilitate this process.

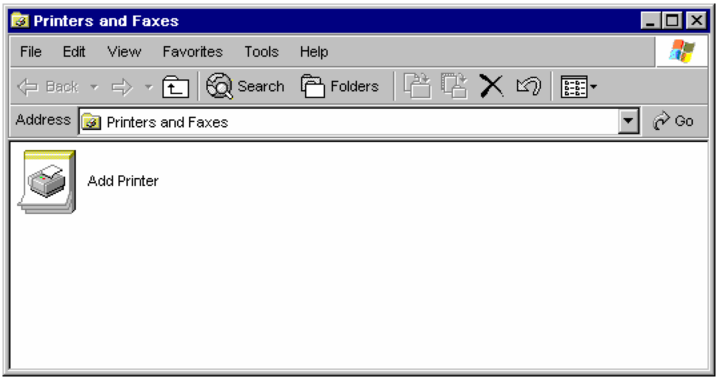
To install a Windows-based printer:

1. Connect the printer to the Dash 8Xe using a parallel 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 Microsoft ® Windows operating system.



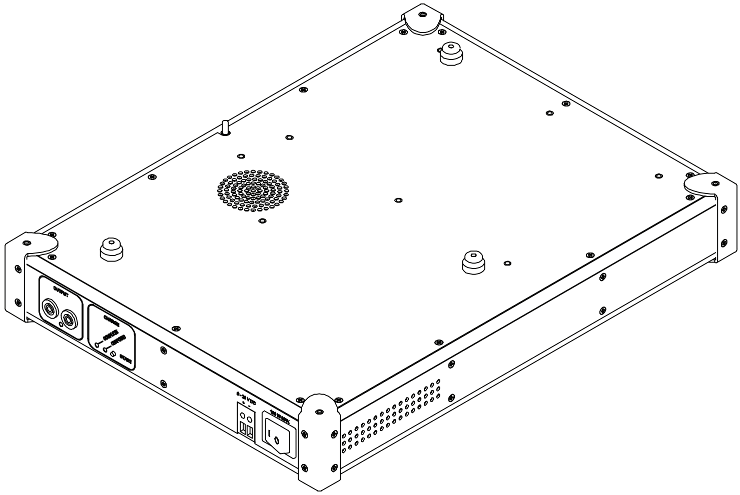
4. Double-click the Add Printer icon. The printer installation wizard will start.

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 Dash 8Xe via the USB port. When prompted by Windows to search for drivers, browse to the driver located on the USB storage device.

PIM-8 and PIM-8B DC Power Modules

The PIM-8 and 8B are power input modules that convert 9-20V DC power to 24V DC power usable by the Dash 8Xe.



PIM-8

The PIM-8 power input module is designed to attach to the Dash 8Xe through the same mounting cleat system used with the SCR-8 printer, although this process is not required for use. Once powered and connected to the Dash 8Xe, it will provide steady 24V DC power to the recorder.

To connect the PIM-8 :

1. Ensure that the power switches on both the Dash 8Xe and the PIM-8 are off.

2. Using AWG 12 stranded wire for short distances, or AWG 10 wire for longer distances, strip the wire's insulation back 13 mm.

3. Connect the wires to an external DC power source capable of at least 150 W. The appropriate power source current setting will vary (for example, 17A @ 9V and 7.5A @ 20V).

4. On the other end, insert the two wires into the DC input connector located next to the power switch on the PIM-8. Make sure there is no power applied to the wires when this is done.

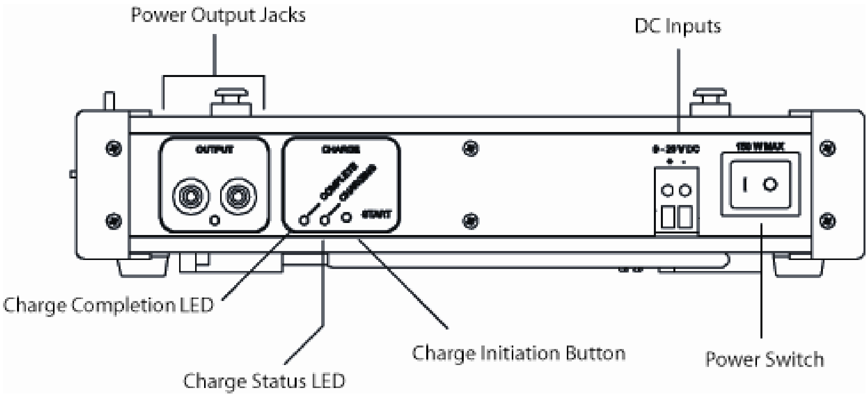
5. Tighten the connector to .5 NM (4.4 lb. in.) with a flat bladed screwdriver.

6. With the Dash 8Xe power switch off, connect the PIM-8 to the Dash 8Xe's two DC inputs jacks using the provided wires. You can now turn on the PIM-8 and the Dash 8Xe in any order to start the system.

As previously stated, the PIM-8 unit also attaches to the base of the Dash 8Xe. The PIM-8 has the same form factor as the SCR-8 Thermal Printer, and attaches in exactly the same way. Please see *Attaching the thermal printer to the Dash 8Xe* for step by step instructions detailing how to use the unit's mounting cleat system.

PIM-8B

The PIM-8B (pictured below) is extremely similar to the PIM-8; the only difference between the two is the PIM-8B's rechargeable battery, which enables the Dash 8Xe to run without an additional power source for over an hour. The connection and attachment procedure is exactly the same for both the PIM-8 and PIM-8B. The sole difference in operation lies in charging the PIM-8B.



To charge the PIM-8B :

1. With the Dash 8Xe and PIM-8B units attached, and the Dash 8Xe power switch turned off, turn on the PIM-8B power switch.

2. Press the charge initiation button. A full charge cycle takes up to seven hours, but the Dash 8Xe can be turned on as soon as the charge completion LED illuminates.

In order to maximize the lifespan of the battery, keep the following in mind when using the PIM-8B :

- Turn off the PIM-8B unit, or begin recharging it, as soon as possible once the battery has been depleted.
- During periods of non-use, remember to charge the battery every six months.
- In some cases, removing and restoring DC power can cause the PIM-8B to automatically restart the charging cycle. To prevent this (unnecessary charging can shorten battery lifespan), simply switch the PIM-8B unit off, and then back on again.

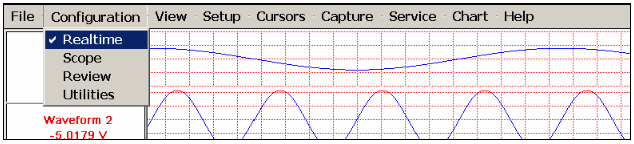
Chapter 5: Menus and buttons

This chapter provides information about the Dash 8Xe menus and buttons.

Menu bar

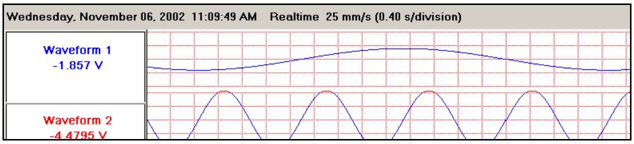
The menu bar is a group of drop-down menus located across the top of the display. All Dash 8Xe modes and features can be accessed from this menu, which behaves similar to menu bars in Windows-based applications. Options available from the menu bar will vary based on the mode of operation (Realtime, Scope, or Review) used.

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



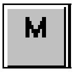
Displaying the time/date

The current time and date can be displayed across the top of the display instead of the menu bar. The following illustration displays the time/date display.



To switch between the menu bar and time/date display:

1. Choose the M button located on the right of the menu bar.



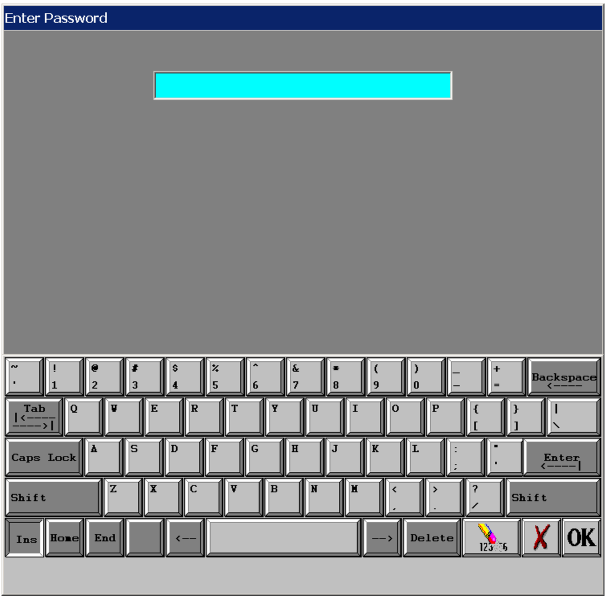
2. To switch back to the previous display, choose the M button again.

Locking the menu bar

The Dash 8Xe menu bar can be locked to prevent unauthorized users from accessing it. While locked, the menu bar cannot be used to access Dash 8Xe functions; the control panel must be used instead. The menu bar will remain locked, even if the Dash 8Xe power is cycled, until it is unlocked by a user.

To lock the menu bar:

1. Choose Service >> Security >> Menu Lock. A password entry window will open.



2. Enter the Dash 8Xe system password and choose OK.

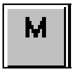
Note: The default password on the Dash 8Xe is "dash8x" (without the quotation marks). For information on changing the system password, refer to *Chapter 14: Service options*.

When the password has been correctly entered, the menu bar will be locked. When locked, the menu bar displays the time/date and does not allow access to menu bar options.

The M button will display as a lock, indicating that the menu bar is currently locked.

3. To unlock the menu bar, choose the lock icon. A password entry window will open. Enter the system password and choose OK. When the password has been correctly entered, the menu bar will be unlocked.

   The M button will be displayed, indicating that the menu bar is unlocked.



Commonly used system buttons

The following list describes buttons that are commonly used in Dash 8Xe windows.

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.

Copy copies the selected items (or characteristics of selected items) to the clipboard.

Paste pastes the clipboard contents to the selected location.

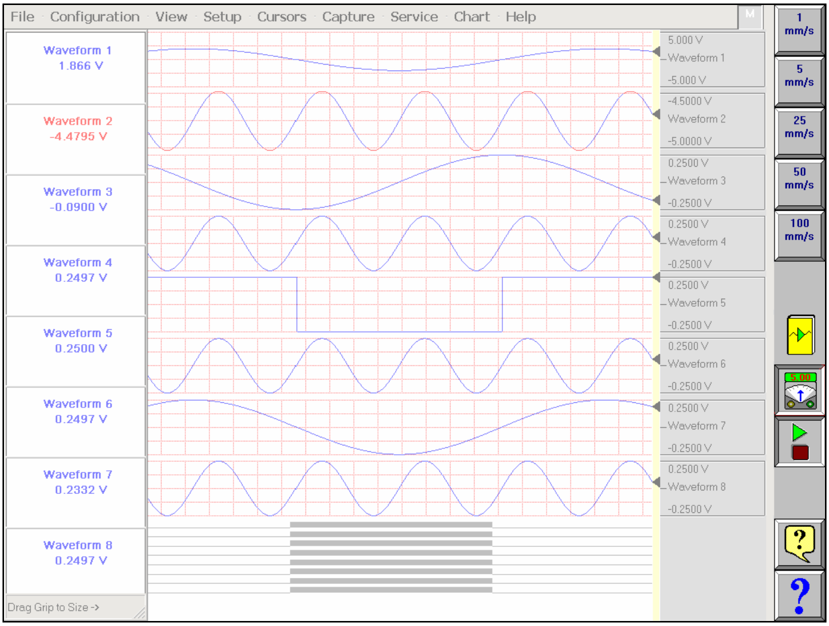
Save saves files.

Load loads files.

Control panel

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

The following illustration displays the control panel.



Customizing the control panel

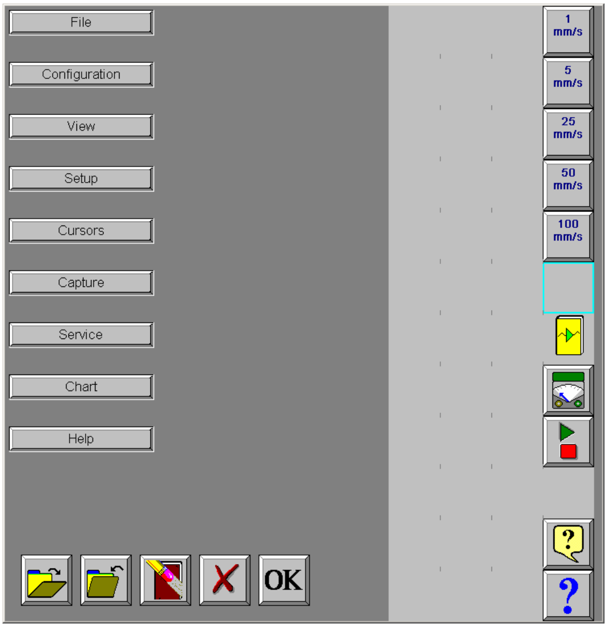
Most of the control panel buttons are one-touch equivalents of multi-step functions involving the menu bar and/or specific windows. Adding frequently used functions and removing unused functions can save time and effort while using the Dash 8Xe.

For example, if the “full page mark” function is used frequently, it might be wise to add the full page mark icon button to the control panel. Instead of using the menu bar to choose View >> Full Page Mark, the function can be utilized by pressing one control panel button.

Use the following instructions to customize the Realtime, Scope, or Review mode control panel.

To customize the control panel:

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



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.

Note: The size of the control panel will vary based on the mode used. Realtime mode provides 48 button slots while Scope and Review modes provide 40 button slots.

---

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

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

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

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

5. Choose OK to complete the control panel customization process.

Control panel setup files

Control panel setup files contain information about the icon buttons 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 Dash 8Xe units.

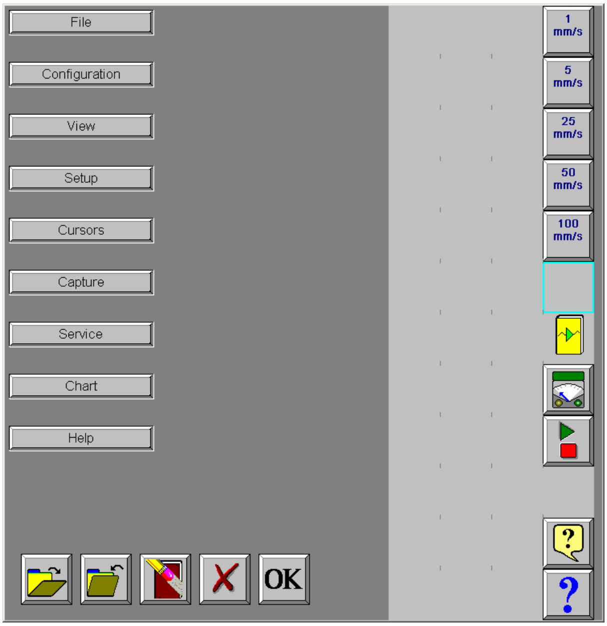
Saving control panel setup files

Once control panels are customized, it may be helpful to save them to a file on a disk. This process creates a control panel setup file that can be used to load control panel layouts at a later time.

Use the following instructions to save control panel setup files. Control panel setup files can be saved from the Control Panel Setup window. In addition to this method, the menu bar can be used as a shortcut for saving control panel setup files.

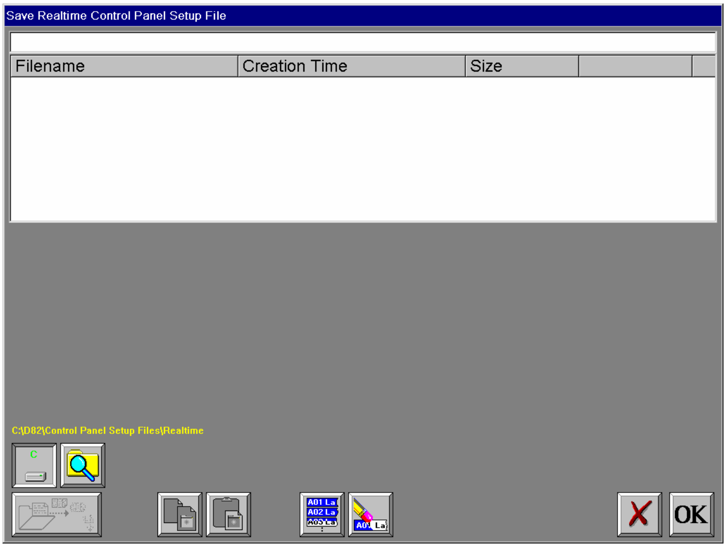
To save control panel setup files:

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



2. Choose the Save button.

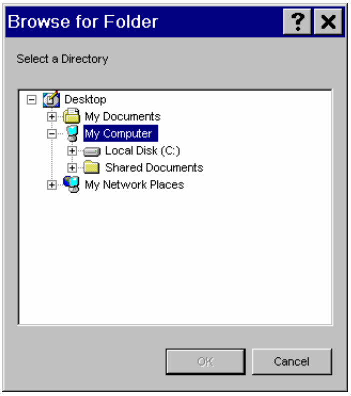
The Save Control Panel Setup File window will open.



3. By default, the control panel setup file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

   If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

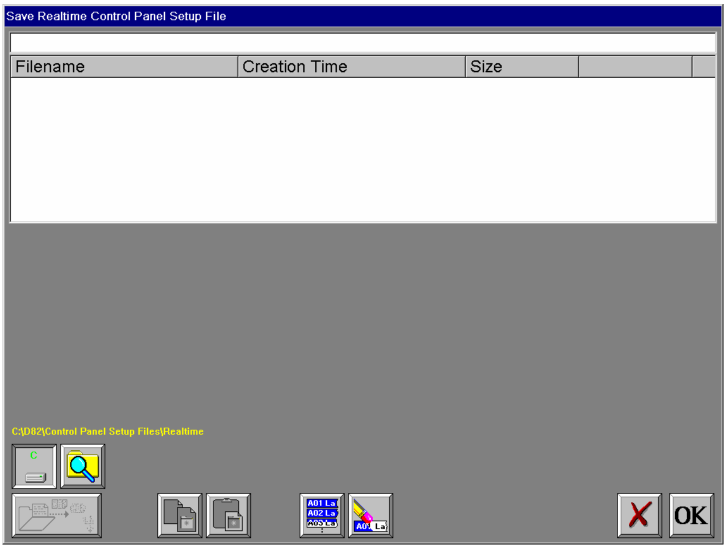
4. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the control panel setup file and choose the OK button. The specified file name will appear in the field.

5. Choose the OK button in the Save Control Panel Setup File window. The control panel setup file will be saved.

To save control panel setup files using the menu bar:

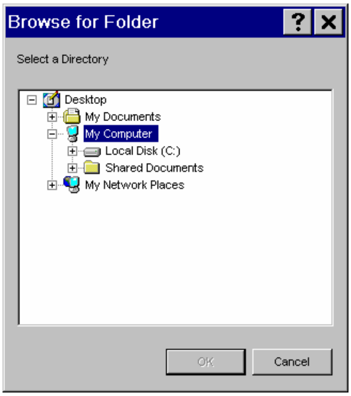
1. Choose File >> Save >> Control Panel. The Save Control Panel Setup File window will open.



2. By default, the control panel setup file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

   If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

3. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the control panel setup file and choose the OK button. The specified file name will appear in the field.

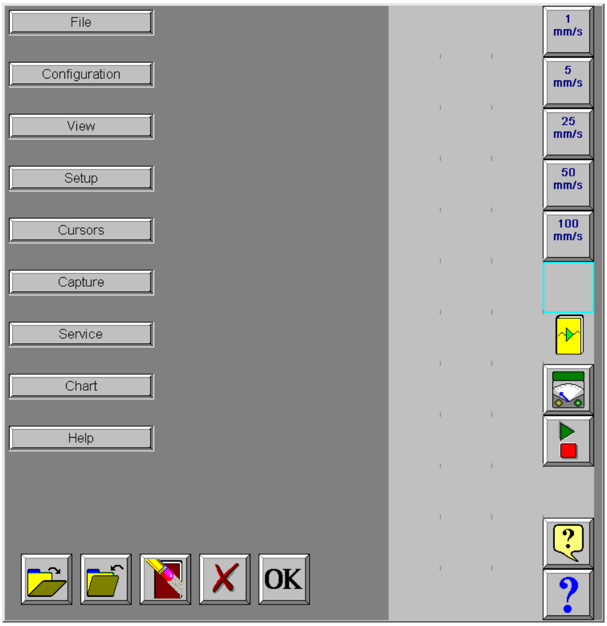
4. Choose the OK button in the Save Control Panel Setup File window. The control panel setup file will be saved.

Loading control panel setup files

Use the following instructions to load control panel setup files. Control panel setup files can be loaded from the Control Panel Setup window. In addition to this method, the menu bar or control panel can be used as a shortcut for loading control panel setup files.

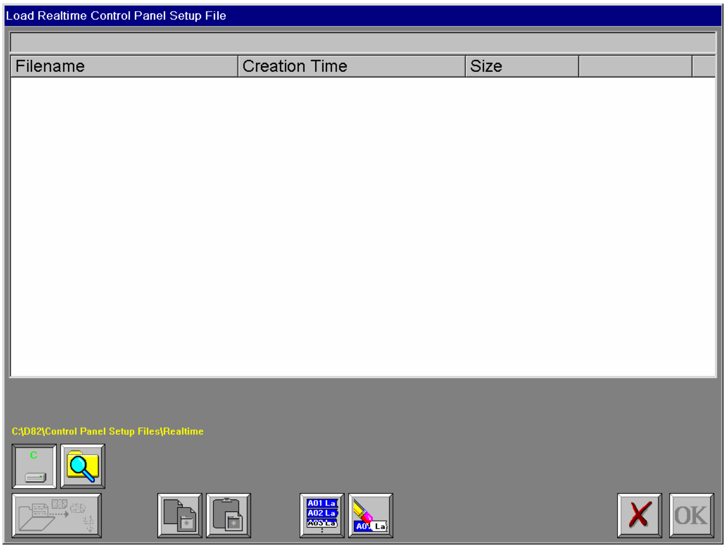
To load control panel setup files:

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



2. Choose the Load button.

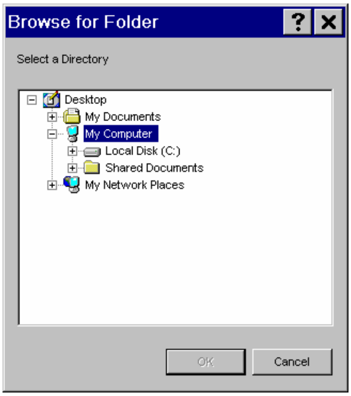
The Load Control Panel Setup File window will open.



3. All saved control panel setup files in the default location on the system drive (C) will be displayed. Select a control panel setup file from this list.

   If necessary, a file can be retrieved from a different location on the system drive. To select a file from an alternate location, choose the Browse Folders button.

The Browse for Folder window will appear.



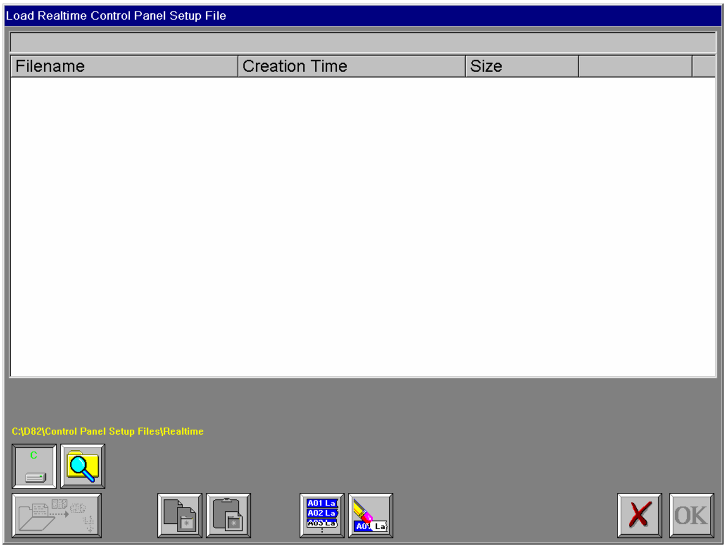
Select a file and choose the OK button. The path displayed in yellow text will be updated to indicate the new file selected.

4. Choose the OK button. The selected control panel setup file will be loaded, and the control panel will be modified accordingly.

   Then choose the OK button in the Control Panel Setup window.

To load control panel setup files using the menu bar:

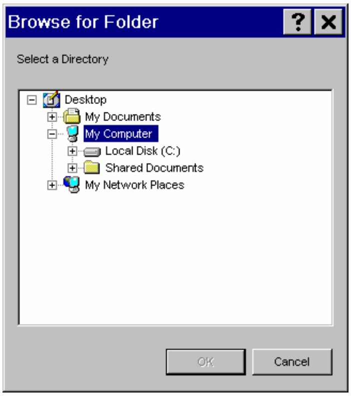
1. Choose File >> Load >> Control Panel. The Load Control Panel Setup File window will open.



2. All saved control panel setup files in the default location on the system drive (C) will be displayed. Select a control panel setup file from this list.

   If necessary, a file can be retrieved from a different location on the system drive. To select a file from an alternate location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file and choose the OK button. The path displayed in yellow text will be updated to indicate the new file selected.

3. Choose the OK button. The selected control panel setup file will be loaded, and the control panel will be modified accordingly.

To load control panel setup files using the control panel:

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

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

   Choose the File button. A sub menu will appear. Then choose Control Panel. The Load Control Panel Setup File window will open.

   Select a control panel setup file and choose OK.

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

The name displayed in this icon will correspond to the name of the control panel setup file.

Chapter 6: Channel setup

This chapter provides information about setting up channel inputs and amplifiers.

Channel setup concepts

The following concepts are commonly used during channel setup-related 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 operations performed on a signal input.

Signals
Signals are voltage levels transmitted to the Dash 8Xe from the measurement source. Signals enter the Dash 8Xe via signal input modules located on the side of the unit. The input modules used will vary based on the types of signals measured.

Slot numbers
Slot numbers indicate the physical location of input modules. Eight slots are available.

Result labels
By default, waveforms are identified by result labels corresponding to signal input modules. These labels can be modified to assign meaningful names to waveforms.

For example, instead of using default labels:

Waveform 1
Waveform 2
Waveform 3
etc . . .

Descriptive labels can be assigned:

Power
Power (after filter)
Pressure
etc . . .

Spans
The span indicates the size of the channel from top edge to bottom edge. Allowable span ranges vary based on the input module used to connect the signal.

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 the module unit (voltage).

All signal information enters the Dash 8Xe as a module unit. However, converting the module unit to an alternative unit of measure may be desirable in applications that measure pressure, strain, or any other non-module unit.

Note: The relationship between the module unit 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 port provides an alarm output pin that can be used to signal when alarm conditions for selected signals occur.

User-selectable filters
User-selectable filters can be set up to filter out specific ranges of frequencies while allowing others to pass through for measurement.

Attenuators
Each input module provides a built-in attenuator. Attenuators limit the maximum signal input to the Dash 8Xe. The attenuator settings available vary based on the input module.

Hardware lock
The hardware lock, when active, limits span values 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.

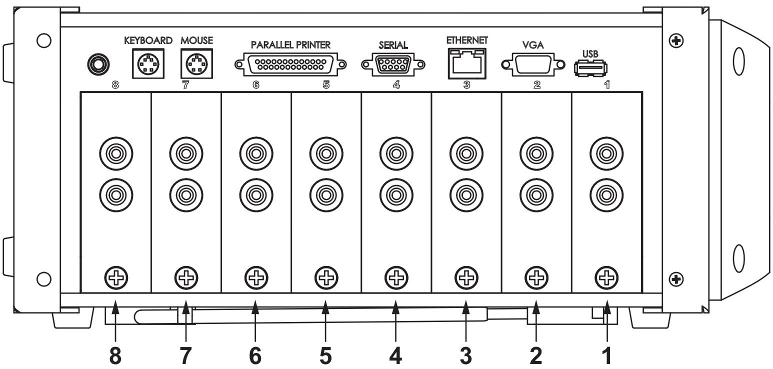
Source Common is a reference point at the voltage source.

Signal input modules

Before setting up channels, signal inputs must be connected to the Dash 8Xe input modules. The input modules used for connecting signals will vary based on the types of signals being measured.

Note: This section provides a brief overview of signal inputs as they relate to signal setup. For more detailed information on signal inputs, refer to *Chapter 3: Input modules*.

The following diagram illustrates the layout of the Dash 8Xe signal input modules.



In the Dash 8Xe software, waveforms are identified by result labels corresponding to signal input modules. For example, the signal connected to module 1 would be labeled "Waveform 1" by default. These labels can be changed as part of the channel setup process.

Channel amplifier setup

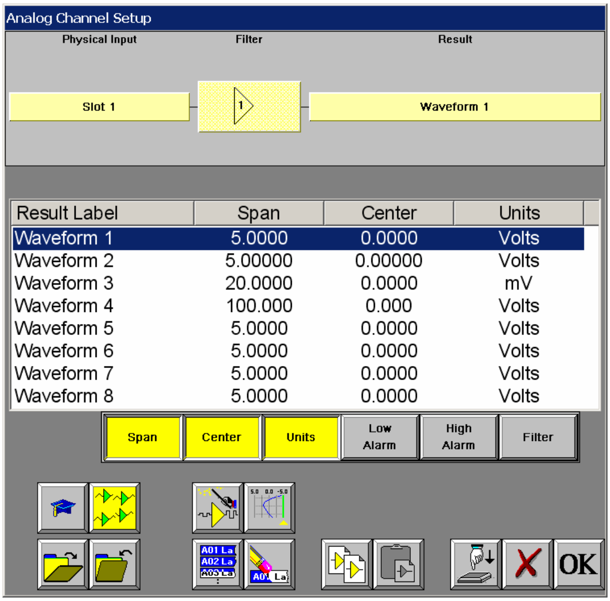
Use the following instructions to set up channel amplifiers. The channel amplifier setup process involves configuring a variety of options.

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 signal input modules.

To set up channel amplifiers:

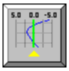
1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.



2. Assign result labels by selecting a waveform and choosing the Result Label column heading. A keypad will appear. Enter a new name for the waveform and choose OK.

3. Define spans by selecting a waveform and choosing the Span column heading. A number pad will appear. Enter a span value and choose OK.

4. Choose whether to specify the center or bottom values of channels using the Bottom/Center button. The icon on the button changes based on the current selection.



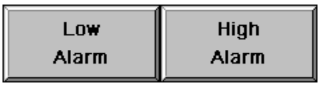
Choose the Center icon to specify center values of channels.

Choose the Bottom icon to specify bottom values of channels.

Select a channel and choose the Center or Bottom column heading. A number pad will appear. Enter a new value for the center or bottom and choose OK.

5. If necessary, set up user engineering units. For details about setting up engineering units, refer to *Engineering unit setup* later in this chapter..

6. Display the Low Alarm and High Alarm options by choosing High Alarm on the scroll buttons.

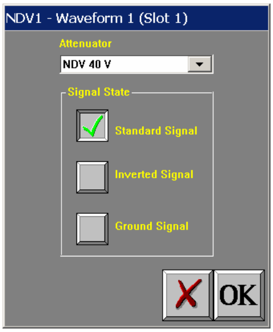


7. To use high and/or low alarms, select a channel and choose the Low Alarm or High Alarm column heading. A sub menu will appear.

Choose New Value. A keypad will appear. Then enter the new high or low alarm level and choose OK.

8. To specify attenuator and signal state settings, select a channel and choose the Advanced Setup button.

The advanced setup window for the selected channel will open.



Select an attenuator setting from the drop-down list. The attenuator setting limits the input signal to the selected maximum voltage. Attenuator options will vary based on the type of signal input module used.

Select a signal state option. The signal state determines how the signal from the input module is initially processed.

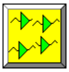
Standard Signal - If the Standard Signal option is checked, the signal from the input module will be used.

Inverted Signal - If the Inverted Signal option is checked, the inverse of the signal from the input module will be used.

Ground Signal - If the Ground Signal option is checked, the signal from the input module will be grounded.

Choose the OK button. The advanced setup window will close.

9. Use the Hardware Lock button 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 to complete the channel amplifier setup process.

Setting channel ranges automatically

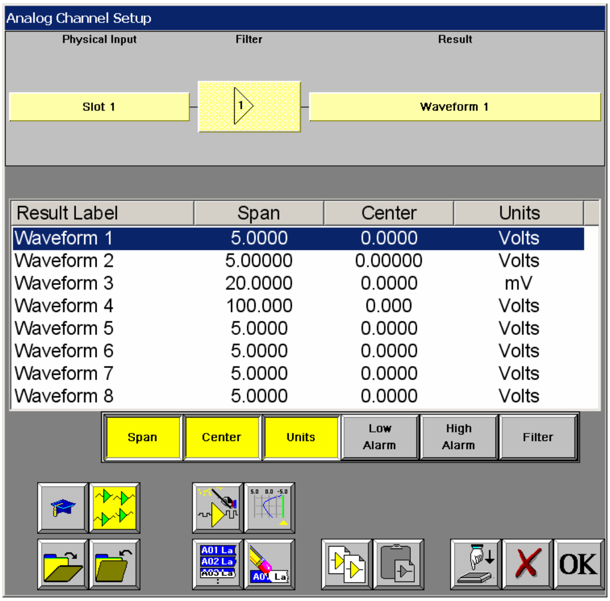
The Analog Channel Setup window provides the capability to enter channel spans manually using a number pad. However, it may be more convenient to calculate channel spans based on the content of input signals.

Channels will be sampled for a selected time period. Then a span and bottom will be assigned to each channel based on highest and lowest signal points for each channel during the sampling process.

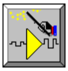
Use the following instructions to automatically derive channel spans based on a time period of input sampling.

To set channel ranges automatically:

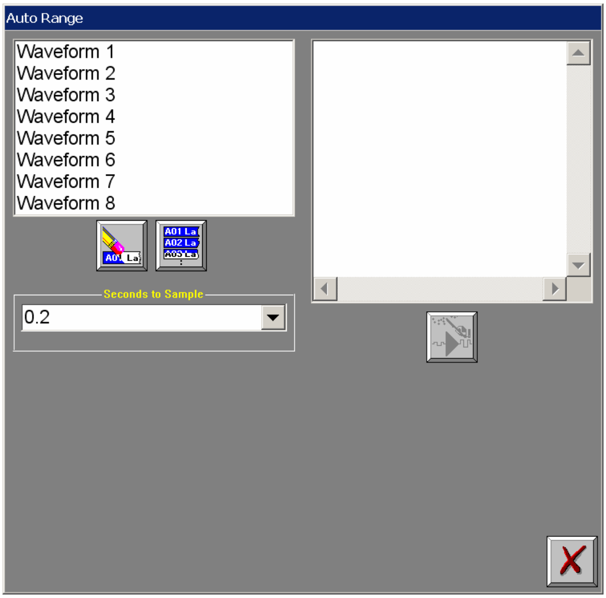
1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.



2. Choose the Auto Range button.

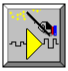


The Auto Range window will open.



3. Select the waveform(s), and then select a sample time duration with the Seconds to Sample drop-down list.

4. Choose the Auto Range button.



Channels will be sampled for a selected time period. Then a span and bottom will be assigned to each channel based on highest and lowest signal points for each channel during the sampling process.

5. Choose the X button in the Auto Range window to return to the Analog Channel Setup window.

Engineering unit setup

Engineering units provide the capability to display user-selected units instead of the module unit (voltage).

All signal information enters the Dash 8Xe 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 non-voltage 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).

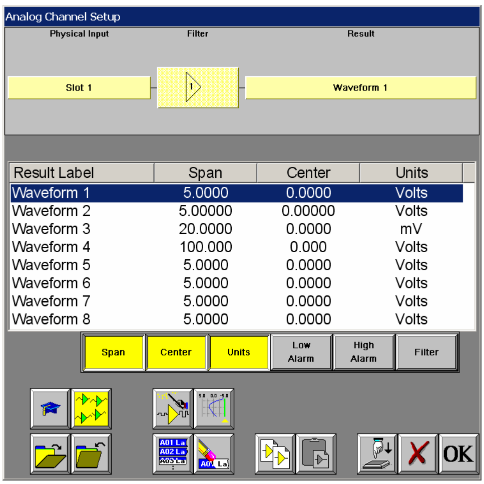
After engineering units are defined and enabled, all appropriate menus will be displayed in the designated engineering unit values. For example, if pounds per square inch (PSI) are used as engineering units, the PSI label and value will be displayed instead of voltage.

Setting up engineering units by entering a scale and offset

Use the following instructions to set up engineering units by entering a scale and offset for the units.

To set up engineering units by entering a scale and offset:

1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.



2. Set up user engineering units by selecting a waveform and choosing the Units column heading, then choose Enter User Units.

Check the Enable User Units option. This option activates engineering units.

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.

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

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

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 module units
- The number of module units equivalent to zero engineering units

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

Choose OK in the User Unit Setup window to complete the engineering unit setup.

3. Choose OK in the Analog Channel Setup window.

Setting up engineering units based on signal input

Use the following instructions to set up engineering units based on a signal input. With this alternative method of setting up engineering units, you do not enter the scale and offset directly. Instead, you specify high and low engineering units based on high and low signal activity.

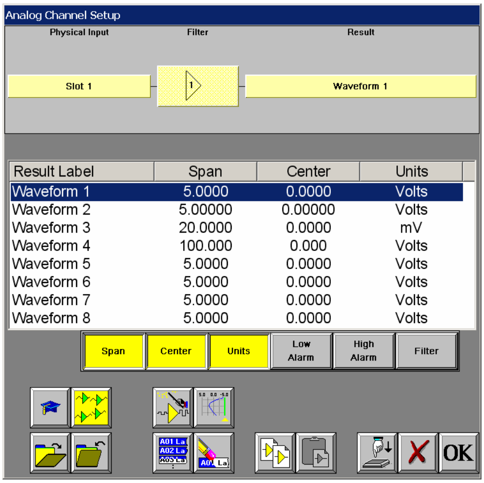
This process will vary based on the type of signal you are using. The following options are available:

- Scale to Level (DC)
- Scale to Signal (AC)

Note: This method of setting up engineering units is available only in Realtime mode. It does not apply to Scope mode.

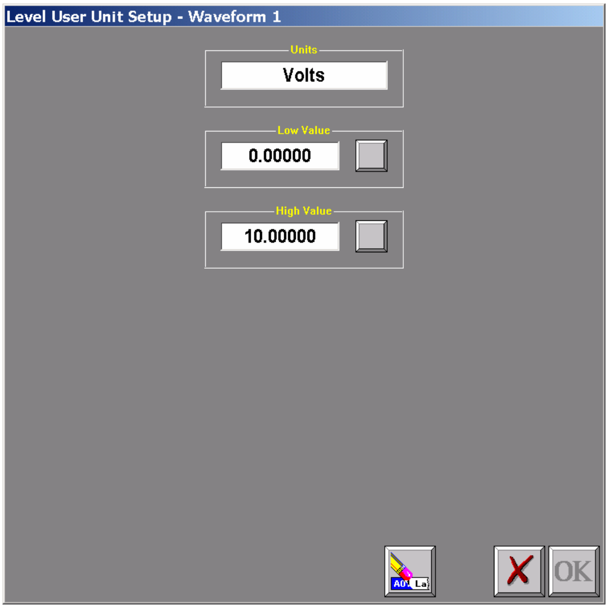
To scale to levels (DC):

1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.



2. Set up amplifier options of the waveform that will use engineering units. Then choose the Apply button.

3. Set up user engineering units by selecting the waveform and choosing the Units column heading, then choose Scale to Levels (DC).



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

Enter low and high values for engineering units. This process assumes you know the engineering unit values associated with low and high signal levels.

- **Low Value** - Adjust the signal to the level associated with the lowest engineering unit value that will be used. Then choose the Low Value field. A number pad will appear.

  Enter the engineering unit value associated with the current signal and choose OK. Then choose the button to the right of the Low Value field. The low value will be calculated.

- **High Value** - Adjust the signal to the level associated with the highest engineering unit value that will be used. Then choose the High Value field. A number pad will appear.

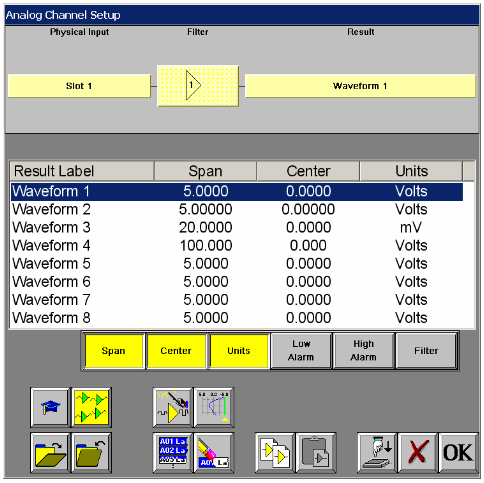
Enter the engineering unit value associated with the current signal and choose OK. Then choose the button to the right of the High Value field. The high value will be calculated.

Choose OK in the Level User Unit Setup window to complete the engineering unit setup.

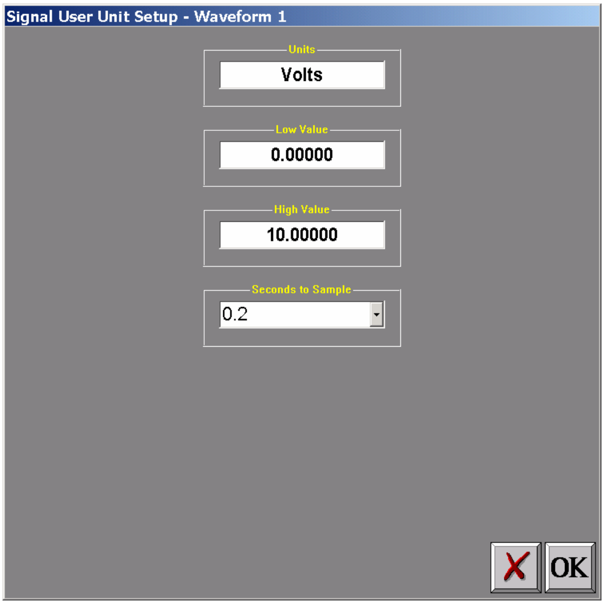
4. Choose OK in the Analog Channel Setup window.

To scale to signal (AC):

1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.



2. Set up user engineering units by selecting the waveform and choosing the Units column heading, then choose Scale to Signal (AC).



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

Enter low and high values for engineering units. This process assumes you know the engineering unit values associated with low and high signal levels.

- **Low Value** - Choose the Low Value field. A number pad will appear. Enter the engineering low unit value associated with the current signal and choose OK.
- **High Value** - Choose the High Value field. A number pad will appear. Enter the high engineering unit value associated with the current signal and choose OK.

Select the length of time to sample the signal. Then choose OK. The signal will be sampled and the low and high values will be calculated.

The signal will be adjusted to full scale and centered on the grid.

3. Choose OK in the Analog Channel Setup window.

Filters

Filters limit waveform amplitude based on frequency and thereby restrict or allow the waveforms to pass.

There are four aspects to the Dash 8Xe filtering system.

Anti-aliasing filter - This filter limits input frequencies and prevents aliasing.

Implementation:	Hardware
Type:	Bessel lowpass
Cutoff (-3dB):	45 KHz
Order:	4
Roll-off:	-80 dB/decade

Fixed lowpass filter - This filter limits high-frequency noise.

Implementation:	Digital signal processing
Topology:	IIR Biquad
Type:	Bessel lowpass
Cutoff (-3dB):	46 KHz
Order:	2
Roll-off:	-40 dB/decade

Built-in user-selectable filters - These filters provide user-selectable filtering options.

Implementation:	Digital signal processing
Topology:	IIR Biquad
Cutoff (-3dB):	User selectable
Type:	Bessel Lowpass, Highpass, Bandpass, Notch
Order:	2 (100 Hz or higher), 1 (below 100 Hz)
Roll-off:	-40 dB/decade (100 Hz or higher), -20 dB/decade (below 100 Hz)

Note: User-selectable filters are described in the following section.

File-based filter - This filter limits frequencies based on a customized set of coefficients stored in a file. The following file-based filters are included with the Dash 8Xe.

Chebyschev - This four-pole filter is available for specialized filtering needs. Chebyschev low-pass (LP) filter values include 10, 50, 100, 250, 500, and 1000 Hz.

Butterworth - This four-pole filter is available for specialized filtering needs. Butterworth low-pass (LP) values include 10, 50, and 500 Hz. High-pass (HP) values include 1, 10, 100, and 1000 Hz.

Bessel - This four-pole filter is available for specialized filtering needs. Bessel low-pass (LP) values include 10, 50, 100, 250, 500, 1000 Hz.

Note: For more information on custom file-based filters, please contact technical support.

User-selectable and file-based filter setup

User-selectable and file-based filters can be set up to filter out specific ranges of frequencies while allowing others to pass through for measurement. This section provides information about setting up user-selectable and file-based filters.

Filter types

The following filters are available. Available filter ranges may vary based on the input modules used.

Low-pass (LP)

This type filters out frequencies above a specified cutoff point. Low-pass filter values range from the following:

NDV1 (1 - 35,000 Hz)
IHV1 (1 - 40,000 Hz)
IBR1 (1 - 40,000 Hz)
IHV2 (1 - 40,000)
ITCU (not available)
IRTD (not available)
IHV3 (1 - 40,000 Hz)
IDCV (not available)
IDCV1 (not available)
LIVM (1 - 40,000 Hz)

High-pass (HP)
This type filters out frequencies below a specified cutoff point. High-pass filter values range from the following:

NDV1 (10 - 35,000 Hz)
IHV1 (10 - 40,000 Hz)
IBR1 (10 - 40,000 Hz)
IHV2 (10 - 40,000)
ITCU (not available)
IRTD (not available)
IHV3 (10 - 30,000 Hz)
IDCV (not available)
IDCV1 (not available)
LIVM (1 - 40,000 Hz)

Band-pass (BP)
This type filters out frequencies above and below specified cutoff points, allowing only a specific band (or range) of frequencies to pass. The two band-pass cutoff points are derived from the possible Low-pass and High-pass filter values.

Root-means-square (RMS)
This type 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.

Response Time	**Cutoff Frequency**
Slow (2 Seconds)	0.12 Hz
Medium (.2 Seconds)	1.2 Hz
Fast (.02 Seconds)	12 Hz
Very Fast (.002 Seconds)	120 Hz

The RMS filter is available in the following modules:

NDV1	IHV2
IHV1	IHV3
IBR1	LIVM

Chebyschev
This four-pole filter is available for specialized filtering needs. Chebyschev low-pass (LP) filter values include 10, 50, 100, 250, 500, and 1000 Hz. The Chebyschev filter is available in the following modules:

NDV1	IHV2
IHV1	IHV3
IBR1	LIVM

Butterworth
This four-pole filter is available for specialized filtering needs. Butterworth low-pass (LP) values include 10, 50, and 500 Hz. High-pass (HP) values include 1, 10, 100, and 1000 Hz. Butterworth filters are available in the following modules:

NDV1	IHV2
IHV1	IHV3
IBR1	LIVM

Bessel
This four-pole filter is available for specialized filtering needs. Bessel low-pass (LP) values include 10, 50, 100, 250, 500, 1000 Hz. The Bessel filter is available in the following modules:

NDV1	IHV2
IHV1	IHV3
IBR1	LIVM

Notch
Filters out a specific frequency (50, 60, and 400 Hz). The notch filter is available in the following modules:

NDV1	IHV2
IHV1	IHV3

IBR1	LIVM

Auto-tracking (Automatic)
This type is a low-pass filter with a value of the sample rate divided by four. The automatic filter is available in the following modules:

NDV1	IHV2
IHV1	IHV3
IBR1	LIVM

Frequency Counter
The frequency counter can be used to count the frequency of zero-crossing signals that are at least 25% of the attenuator. The frequency counter range will vary based on the input module.

The frequency counter is available in the following modules:

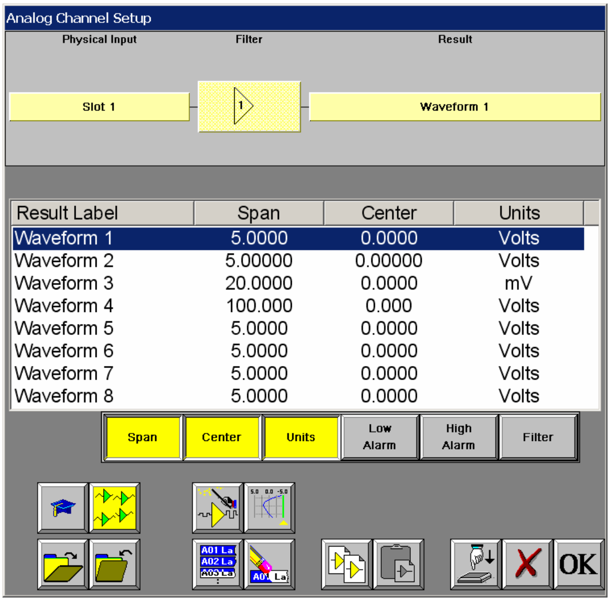
IHV1	Frequency counter range 2 – 60 KHz
IBR1	Frequency counter range 2 – 60 KHz (200 mV and 2V attenuators) 40 KHz (20 mV and 50 mV attenuators)
IHV2	Frequency counter range 2 – 40 KHz
IHV3	Frequency counter range 2 – Attenuator bandwidth + 5 KHz

Setting up filters

Filters are configured using the same window used to set up channel amplifiers. Use the following instructions to set up filters.

To set up filters:

1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.



2. Select a waveform from the list. A graphic will appear on the top of the window. This graphic visually indicates the filtering formula used to calculate the selected waveform.

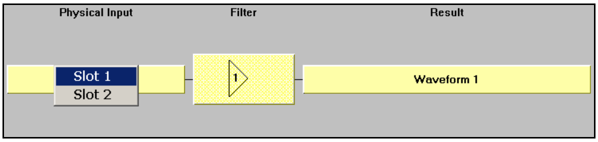
   The following components are displayed in the filtering formula:

   Physical Input - The Physical Input indicates the slot location of the input module used to connect the signal to the Dash 8Xe.

   Filter - The Filter indicates the type and numeric settings for the filtering, RMS calculation, or frequency counter conversion operation performed on the signal from the physical input.

Result - The Result indicates the waveform generated as a result of the filtering operation performed on the signal from the physical input.

3. To select a physical input slot, choose the Physical Input graphic. A sub menu will appear.



Select a physical input slot. The filter will be applied to the signal entering the Dash 8Xe via the module in the selected input slot.

Note: In many cases, the physical input will numerically correspond to the resulting waveform. For example, the signal from Slot 1 may be filtered and displayed as Waveform 1.

In some situations, it may be useful to filter a signal from a different physical input. For example, the signal from Slot 2 may be filtered and displayed as Waveform 1. This filtering method can be used only between the following physical input groups: 1-2, 3-4, 5-6, and 7-8. For example, the signal from Slot 5 cannot be displayed as Waveform 1, but it can be displayed as Waveform 5 or 6.

In instances where two waveforms are derived from the same physical input, the frequency counter operation can be performed on only one of the waveforms.

---

4. Apply a filter by choosing the Filter graphic. A sub menu will appear. Choose a filter type from the list, then select the numeric filter values, if necessary.

Note: Available filter configurations will vary depending on the input modules used.

In addition to the filter graphic, the Filter column heading can also be used to apply filters to channels.

---

5. Choose OK to complete the filter set up process.

Signal setup files

Signal setup files contain channel setup information such as signal input, amplifier, and filter configurations. 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 Dash 8Xe units.

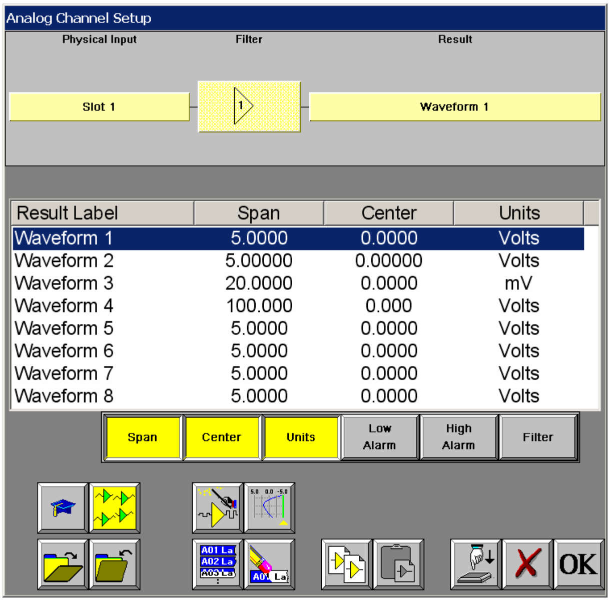
Saving signal setup files

Once channel amplifier settings are configured, it may be helpful to save them to a file on disk. This process creates a signal setup file that can be used to load signal configurations at a later time.

Use the following instructions to save signal setup files. Signal setup files can be saved from the Analog Channel Setup window. In addition to this method, the menu bar can be used as a shortcut for saving signal setup files.

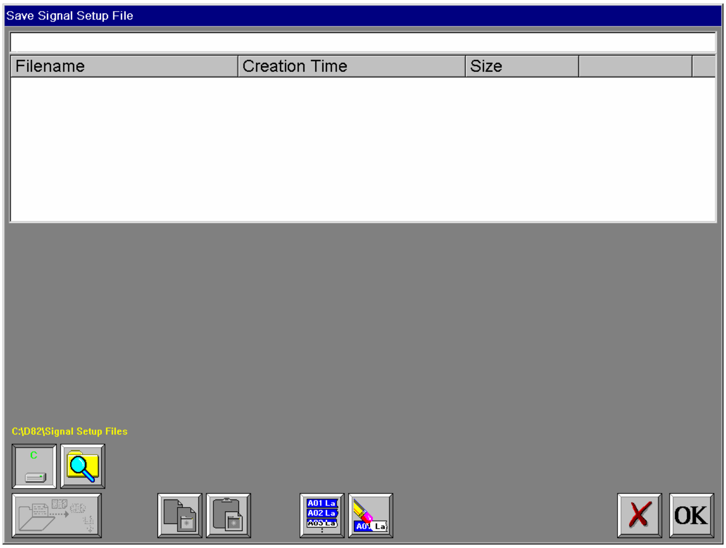
To save signal setup files:

1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.



2. Choose the Save button.

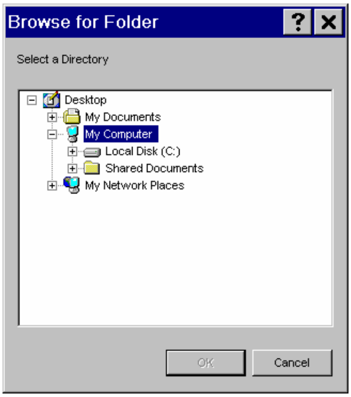
The Save Signal Setup File window will open.



3. By default, the signal setup file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

   If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

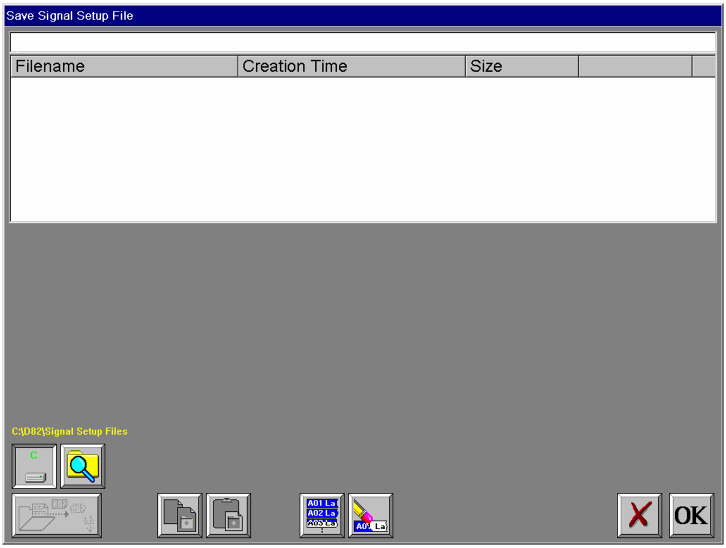
4. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the signal setup file and choose the OK button. The specified file name will appear in the field.

5. Choose the OK button in the Save Signal Setup File window. The signal setup file will be saved.

To save signal setup files using the menu bar:

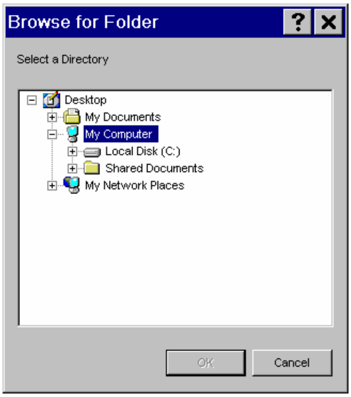
1. Choose File >> Save >> Signals to open the Save Signal Setup File window.



2. By default, the signal setup file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

   If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

3. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the signal setup file and choose the OK button. The specified file name will appear in the field.

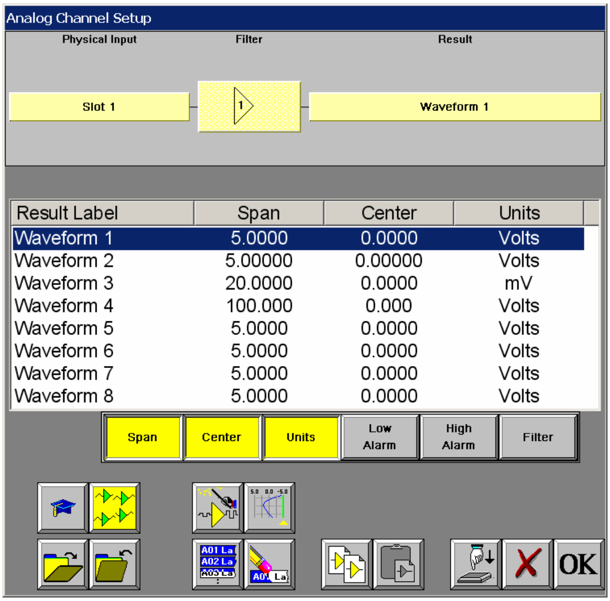
4. Choose the OK button in the Save Signal Setup File window. The signal setup file will be saved.

Loading signal setup files

Use the following instructions to load signal setup files. Signal setup files can be loaded from the Analog Channel Setup window. In addition to this method, the menu bar or control panel can be used as shortcuts for loading signal setup files.

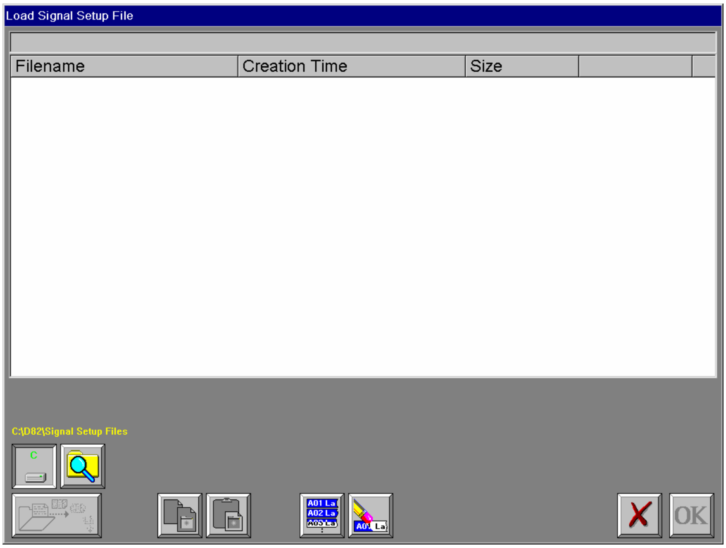
To load signal setup files:

1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.



2. Choose the Load button.

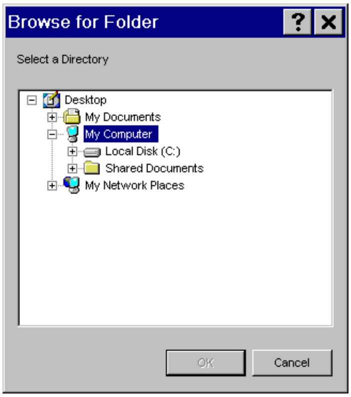
The Load Signal Setup File window will open.



3. All saved signal setup files in the default location on the system drive (C) will be displayed. Select a signal setup file from this list.

   If necessary, a file can be retrieved from a different location on the system drive. To select a file from an alternate location, choose the Browse Folders button.

The Browse for Folder window will appear.

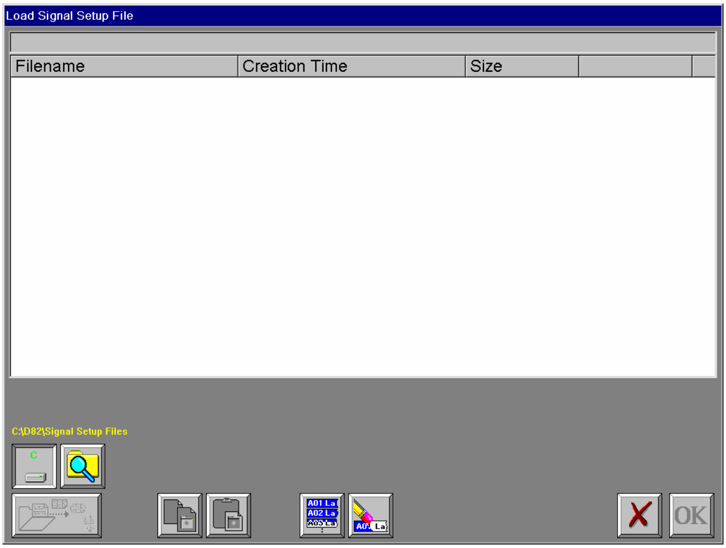


Select a file and choose the OK button. The path displayed in yellow text will be updated to indicate the new file selected.

4. Choose the OK button. The selected signal setup file will be loaded, and the channel amplifiers will be modified accordingly.

To load signal setup files using the menu bar:

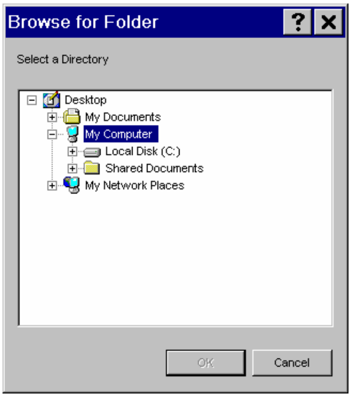
1. Choose File >> Load >> Signals to open the Load Signal Setup File window.



2. All saved signal setup files in the default location on the system drive (C) will be displayed. Select a signal setup file from this list.

   If necessary, a file can be retrieved from a different location on the system drive. To select a file from an alternate location, choose the Browse Folders button.

   The Browse for Folder window will appear.



Select a file and choose the OK button. The path displayed in yellow text will be updated to indicate the new file selected.

3. Choose the OK button. The selected signal setup file will be loaded, and the channel amplifiers will be modified accordingly.

To load signal setup files using the control panel:

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

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

   Choose the File button. A sub menu will appear. Then choose Signals. The Load Signal Setup File window will open.

   Select a signal setup file and choose OK.

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

   The name displayed in this icon will correspond to the name of the signal setup file.

Chapter 7: Display setup

This chapter provides information about setting up the Dash 8Xe display.

Channel views

The Dash 8Xe provides flexible viewing options for channels. This section explains how to set up basic and custom channel views.

Basic channel views

The Dash 8Xe provides basic views that can be quickly accessed from the menu bar. These pre-set views display a fixed number of waveforms in a basic layout.

To display a basic channel view:

1. Choose View from the menu bar. A sub menu will appear.

2. Select the desired type of basic view.

    - In Realtime mode, choose from Basic 2, 4, or All Channels.

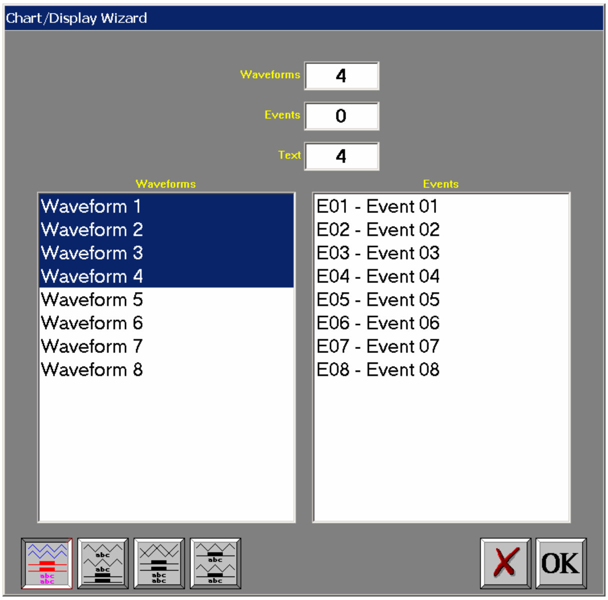
    - In Scope or Review mode, choose from Basic 1, 2, 4, or All Channels.

Custom channel views

The Chart/Display Wizard window is used to set up customized channel views. It provides the capability to select only the waveforms, events, and text labels needed. Additionally, a chart format can be selected to control the layout.

To set up a custom channel view:

1. Choose View >> Chart/Display Wizard to open the Chart/Display Wizard window.



2. Select the waveforms to display by choosing them from the Waveform list box.

   For an alternative method of selecting waveforms, choose the Waveform number field. A number pad will appear. Enter the number of waveforms to display and choose OK. The specified number of waveforms will be selected from the Waveform list box, starting at the first waveform.

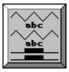
3. Select the events to display by choosing them from the Event list box.

   For an alternative method of selecting events, choose the Events number field. A number pad will appear. Enter the number of events to display and choose OK. The specified number of events will be selected from the Events list box, starting at the first event.

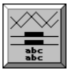
4. If the optional thermal printer is installed, a Text field will be available. Use this option to specify the number of channel text buffers to print on the chart.

5. Select a chart layout using the buttons in the lower-left corner of the window.

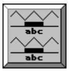
Format 1 shows the waves for all channels, followed by events. Text information appears on the bottom of the chart.



Format 2 shows the waves for a channel, followed by the channel text. This pattern is repeated for each channel. Event information appears on the bottom of the display.



Format 3 shows overlapping waves for all channels, followed by events. Text information appears on the bottom of the chart.

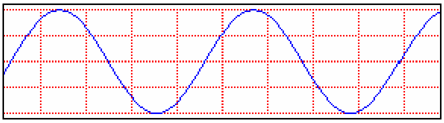


Format 4 shows the wave for a channel, followed by the event, then text. This pattern is repeated for each channel.

6. Choose the OK button. The customized view will be displayed.

Grids

The Dash 8Xe draws waveforms on customizable grids. These grids provide a visual aid that can be used for a variety of measurement purposes. Grid placement, size, and color can be modified to fit the needs of the application.



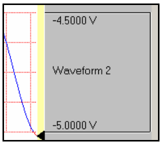
This section describes how to manage grids.

Editing grids

Default grids are displayed with four divisions. If desired, custom grids can be set up with a specific number of divisions. The channel grid can also be removed, displaying the waveform with no grid divisions. Use the following instructions to edit a grid.

To edit a grid:

1. Choose the channel label area on the right side of a waveform.



A sub menu will appear.

2. Choose Edit Grid. A sub menu will appear.

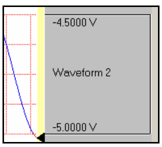
   - To remove the grid, choose Off.
   - To edit the grid, choose Set Divisions. A number pad will appear. Enter a grid division number and choose OK. The new grid will be displayed.

Moving grids

Each grid starts in a default position based on the number of grids displayed. Grids can be moved vertically to a new position on the display. Use the following instructions to move a grid to a different position.

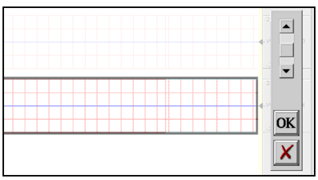
To move a grid:

1. Choose the channel label area on the right side of a waveform.



A sub menu will appear.

2. Choose Move Grid. A grid position slider bar will appear.



3. Move the grid vertically using the slider bar, then choose OK.

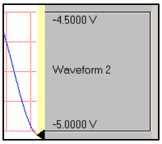
Note: If a grid is moved on top of an existing grid, the waveforms from both grids will be combined into one grid and waveforms will be overlapped.

Resizing grids

Each grid starts with a default size based on the number of grids displayed. Grids can be resized vertically to occupy a larger or smaller display area. Use the following instructions to change the size of a grid.

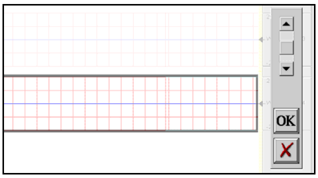
To resize a grid:

1. Choose the channel label area on the right side of a waveform.



   A sub menu will appear.

2. Choose Size Grid. A grid size slider bar will appear.



3. Resize the grid vertically using the slider bar, then choose OK.

Selecting a grid color

Grid start with a default color. A different grid color can be selected and applied to all grids on the display. By experimenting with different grid, waveform, and background colors, an optimal color scheme can be created. Use the following instructions to change the color of a grid.

To select a grid color:

1. Choose View >> Grid Color. A sub menu will appear.

2. Choose a color. The selected color will be applied to all grids on the display.

Selecting a background color

The default background color varies based on the mode of operation. A different background color can be selected and applied. By experimenting with different grid, waveform, and background colors, an optimal color scheme can be created. Use the following instructions to change the background color.

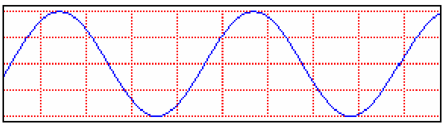
To select a background color:

1. Choose View >> Background Color. A sub menu will appear.

2. Choose a color. The selected color will be applied to the background.

Waveforms

Waveforms are drawn on the display to visually represent signal activity. They can result directly from a signal input or from a filtering operation performed on a signal input. A variety of options are available to control the location and appearance of waveforms.



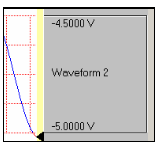
This section describes how to manage waveforms.

Overlapping waveforms

Multiple waveforms can be overlapped and displayed in one grid. This feature is helpful for comparing more than one waveform. Use the following instructions to overlap waveforms.

To overlap waveforms:

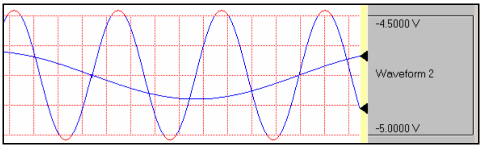
1. Choose the channel label area on the right side of a waveform.



A sub menu will appear.

2. Choose Add. A list of available waveforms will appear.

3. Select the waveform to add. The waveform will be overlapped and displayed simultaneously with the other waveforms in the grid.

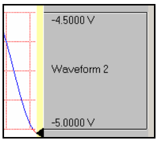


Replacing waveforms

A waveform in a grid can be directly replaced with a different waveform. Use the following instructions to replace waveforms.

To replace waveforms:

1. Choose the channel label area on the right side of a waveform.



A sub menu will appear.

2. Choose Replace. The next step will vary based on the number of waveforms displayed in the grid.

   - If the grid contains one waveform, a list of replacement waveforms will appear. Select a replacement waveform from the list.

   - If the grid contains more than one waveform, a list of waveforms in the grid will appear. Select the waveform to replace.

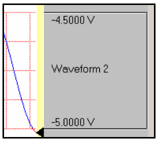
     A list of replacement waveforms will appear. Select a replacement waveform from the list.

Removing waveforms

Individual waveforms can be removed from grids. Use the following instructions to remove waveforms.

To remove waveforms:

1. Choose the channel label area on the right side of a waveform.



   A sub menu will appear.

2. Choose Remove. The next step will vary based on the number of waveforms displayed in the grid.

   - If the grid contains one waveform, the waveform and grid will be removed.

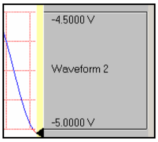
   - If the grid contains more than one waveform, a list of waveforms in the grid will appear. Select the waveform to remove.

Selecting an active waveform

When there are two or more overlapped waveforms in a grid, it may be necessary to select one as the active waveform. The active waveform values will be displayed in the channel meter and channel label area. Use the following instructions to select an active waveform.

To select an active waveform:

1. Choose the channel label area on the right side of a waveform.



   A sub menu will appear.

2. Select Choose Active Channel. A list of waveforms will appear.

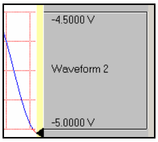
3. Select a waveform. The selected waveform will become the active waveform.

Selecting waveform colors

The display color of each waveform can be modified. This feature is helpful to distinguish multiple waveforms displayed in one grid. By experimenting with different grid, waveform, and background colors, an optimal color scheme can be created. Use the following instructions to select waveform colors.

To select waveform colors:

1. Choose the channel label area on the right side of a waveform.



   A sub menu will appear.

2. Choose Waveform Color. The next step will vary based on the number of waveforms displayed in the grid.

   - If the grid contains one waveform, a list of waveform colors will appear. Select a color from the list.

   - If the grid contains more than one waveform, a list of waveforms in the grid will appear. Select the waveform to color.

     A list of colors will appear. Select a color from the list.

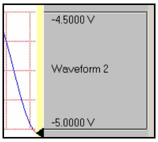
Alarm colors

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.

An alarm color is used to indicate portions of a waveform that extend below or above specified boundaries. Use the following instructions to modify alarm colors.

To select alarm colors:

1. Choose the channel label area on the right side of a waveform.



A sub menu will appear.

2. Choose Alarm Color. The next step will vary based on the number of waveforms displayed in the grid.

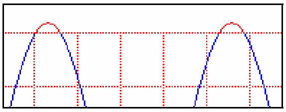
    - If the grid contains one waveform, a list of alarm colors will appear. Select a color from the list.

    - If the grid contains more than one waveform, a list of waveforms in the grid will appear. Select the waveform to color.

      A list of alarm colors will appear. Select a color from the list.

Overrange colors

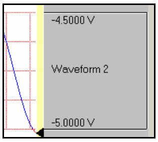
An overrange color is used to indicate portions of a waveform that extend below or above the grid boundaries. The following illustration displays a waveform extending above the grid boundaries. The portions of the waveform outside the grid are drawn in the specified overrange color.



Use the following instructions to modify overrange colors.

To select overrange colors:

1. Choose the channel label area on the right side of a waveform.



A sub menu will appear.

2. Choose Overrange Color. The next step will vary based on the number of waveforms displayed in the grid.

   - If the grid contains one waveform, a list of overrange colors will appear. Select a color from the list.
   - If the grid contains more than one waveform, a list of waveforms in the grid will appear. Select the waveform to color.

     A list of overrange colors will appear. Select a color from the list.

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. This section provides information about the available event features.

Event inputs

Event inputs are connected via the event port, which is located on the side of the Dash 8Xe.

Note: For detailed information on the event port pin configuration, refer to *Chapter 2: Hardware overview*.

Changes in event inputs will cause the state of an event to change.

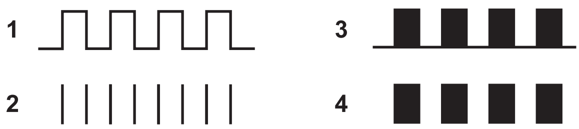
Physical switch operation - An event is high when the switch is open, and low when the switch is closed.

TTL logic level changes - The event is high when the input level is a TTL high (greater than 2 volts), and low when the input level is a TTL low (less than .8 volts).

Caution: The maximum DC voltage is 5.5 volts. Higher voltages will damage the event input circuit.

Selecting an event style

Event styles determine how event signals are drawn on the display. Four styles are available. The following diagram illustrates the available event styles.



1. Standard
2. Tick
3. Bar/line
4. Bar/off

To select an event style:

1. Choose View >> Event Style. A list of event styles will appear.

2. Select an event style.

Adding events

Up to eight event signals can be drawn on the display. Use the following instructions to add an event to the display.

Note: The Chart/Display Wizard window can also be used to add events to the display. Refer to the *Channel views* section of this chapter for more information on this process.

To add an event:

1. Choose Setup >> Events. A list of events will appear.

2. Choose an event to add. A list of options will appear.

3. Choose Add. The event will be added to the display. The Add option will be available only if the event is not currently displayed.

Removing events

Use the following instructions to remove an event from the display.

Note: The Chart/Display Wizard window can also be used to remove events from the display. Refer to the *Channel views* section of this chapter for more information on this process.

To remove an event:

1. Choose Setup >> Events. A list of events will appear.

2. Choose an event to remove. A list of options will appear.

3. Choose Remove. The event will be removed from the display. The Remove option will be available only if the event is currently displayed.

Moving events

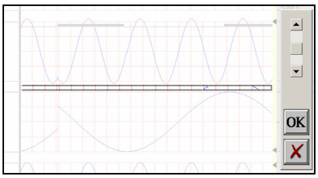
By default, event display locations are determined by the layout chosen in the Chart/Display Wizard window. Events can be moved vertically to a new position on the display. Use the following instructions to move events.

To move events:

1. Choose Setup >> Events. A list of events will appear.

2. Choose an event to move. A list of options will appear.

3. Choose Move. An event position slider bar will appear.



4. Move the event vertically using the slider bar, then choose OK.

Selecting an event color

The display color of each event can be modified. This feature is helpful to distinguish between multiple events on the display. Use the following instructions to select event colors.

To select an event color:

1. Choose Setup >> Events. A list of events will appear.

2. Choose an event to color. A list of options will appear.

3. Choose Color. A list of colors will appear.

4. Select a color from the list. The event color will be changed to the selected color.

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 display setup files to accommodate multiple tasks and/or users. Additionally, view setup files are portable, so they can be shared with other Dash 8Xe units.

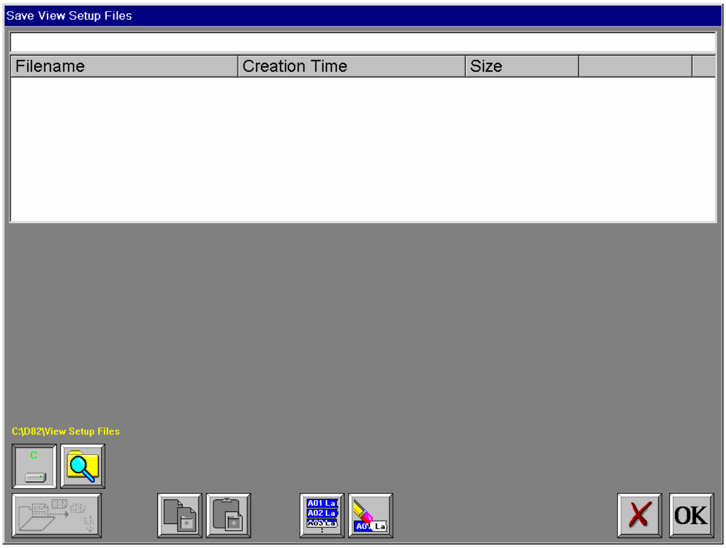
Saving view setup files

Once a display is customized, it may be helpful to save it to a file on disk. This process creates a view setup file that can be used to load display settings at a later time.

Use the following instructions to save view setup files.

To save view setup files:

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

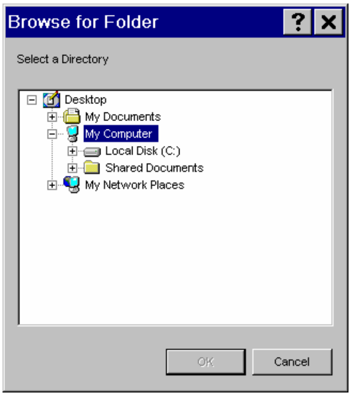


2. By default, the view setup file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

   If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

   The Browse for Folder window will appear.

   Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

3. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the view setup file and choose the OK button. The specified file name will appear in the field.

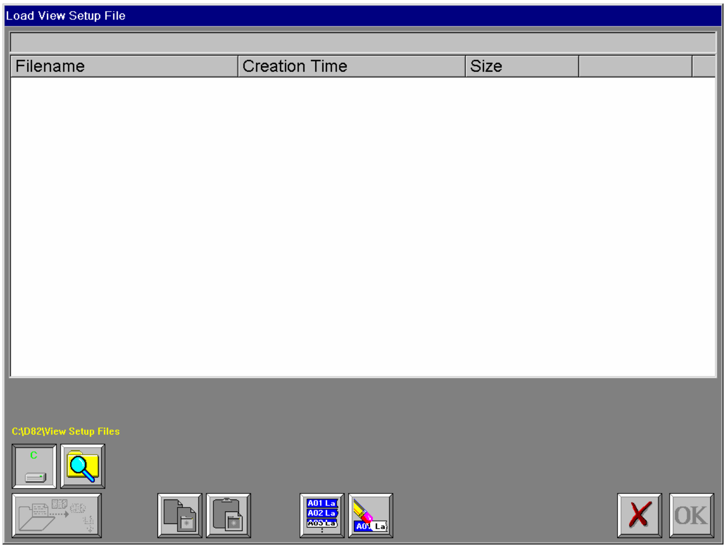
4. Choose the OK button in the Save View Setup File window. The view setup file will be saved.

Loading view setup files

Use the following instructions to load view setup files. View setup files can be loaded from the menu bar or control panel.

To load view setup files using the menu bar:

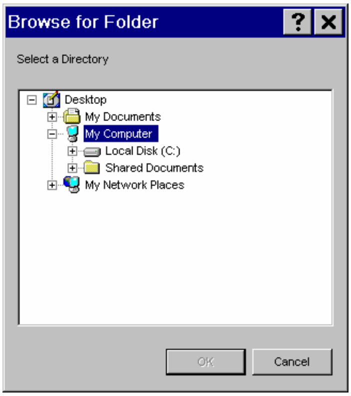
1. Choose File >> Load >> View. The Load View Setup File window will open.



2. All saved view setup files in the default location on the system drive (C) will be displayed. Select a view setup file from this list.

   If necessary, a file can be retrieved from a different location on the system drive. To select a file from an alternate location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file and choose the OK button. The path displayed in yellow text will be updated to indicate the new file selected.

3. Choose the OK button. The selected view setup file will be loaded, and the view will be modified accordingly.

To load view setup files using the control panel:

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

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

   Choose the File button. A sub menu will appear. Then choose View. The Load View Setup File window will open.

   Select a view setup file and choose OK.

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

The name displayed in this icon will correspond to the name of the view setup file.

Chapter 8: Realtime mode overview

This chapter provides an overview of the Dash 8Xe Realtime mode.

Introduction

Realtime mode provides real-time waveform scrolling, monitoring, and data capture capabilities. Additionally, almost all system setup options are accessible from Realtime mode.

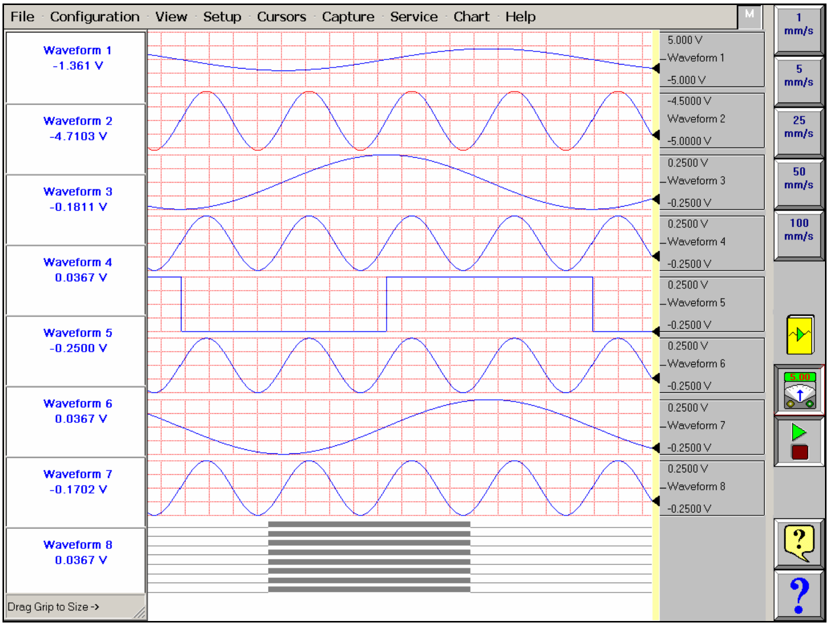
Accessing Realtime mode

When the Dash 8Xe is powered-on, Realtime mode is automatically started. Use the following instructions to start Realtime mode from any other mode of operation.

To access 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

The following instructions provide information on freezing/running the monitor. This task can be accomplished using the menu bar or control panel.

To freeze/run the monitor using the menu bar:

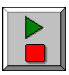
1. Choose View >> Freeze Display.

   The on-screen monitor will halt.

2. Repeat this process to toggle between frozen and running monitor states.

To freeze/run the monitor using the control panel:

1. Choose the Run/Halt Monitor and Chart button.

   The on-screen monitor will halt.

2. Repeat this process to toggle between frozen and running chart and monitor states.

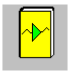
Default control panel

The following buttons are provided in the default Realtime control panel.

Note: The Realtime mode control panel can be customized to meet the needs of specific application or users. Refer to *Chapter 5: Menus and buttons* for information on customizing the control panel.

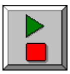
Set Chart Speed changes the speed of the on-screen and printed charts. In this example, the chart speed will be changed to 25 mm/s when the button is pressed.

Custom speed icons can be configured during the control panel setup process.



Analog Channel Setup opens the Analog Channel Setup window, which is used to set up channel amplifiers and filters.

Show/Hide Meter displays and hides the channel meter, which provides a numeric data logger of signal values updated once per second.



Run/Halt Monitor and Chart starts and stops the on-screen display from running.

Chart Run/Halt starts and stops the printed chart from running. This function is available only when the optional printer is installed.

Context Help provides on-screen help based on the mode or window in use.

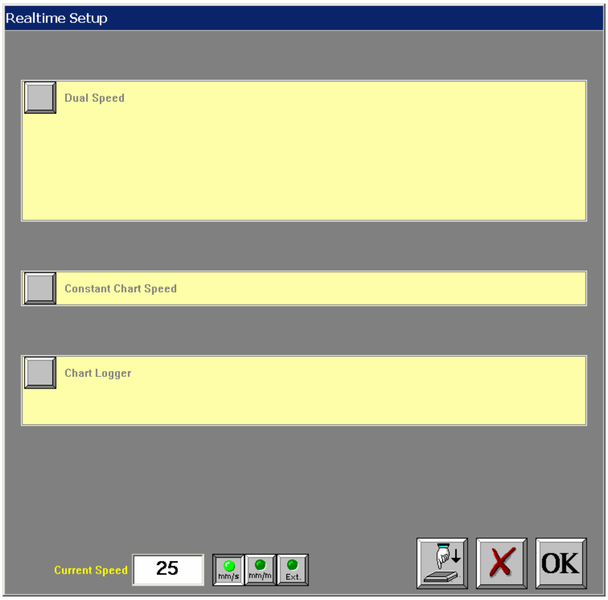
Icon Help provides on-screen help for the purpose of identifying icon buttons.

Realtime mode setup

The Realtime mode setup process involves setting up chart/monitor speeds. Use the following instructions to set up Realtime mode.

To set up realtime mode:

1. Choose Setup >> Realtime Settings from the menu bar. The Realtime Setup window will open.



Four main setup options are available in this window: Dual Speed, Constant Chart Speed, Chart Logger, and External Motor Clock Divider. These options cannot be used simultaneously.

Dual Speed - The Dual Speed option provides the capability to set up two different chart/monitor speeds. Each speed can be used for a specific amount of time or until a trigger or abort occurs.

This option is commonly used to increase the chart/monitor speed immediately after a trigger.

Constant Chart Speed - The Constant Chart Speed option provides the capability to run the printed chart at a specified speed. The monitor speed can then be adjusted without affecting the chart speed. When this option is used, only speeds that are a multiple of the chart speed can be used as the display speed.

Note: The optional thermal printer is required to use this option.

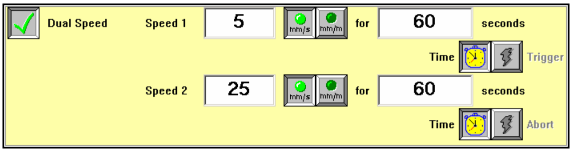
Chart Logger - The Chart Logger option provides the capability to print data in a numeric format instead of a graphical waveform format.

Note: The optional thermal printer is required to use this option.

External Motor Clock Divider - The Clock Divider allows more precise control of the speed provided to the Dash 8Xe by an externally connected device. For this reason, the Clock Divider is only available when the "Ext." button is selected under Current Speed.

The Clock Divider can be set to any value between 1 and 10. The input rate will then be reduced by this factor.

2. To set up a dual chart speed, check the Dual Speed option. A group of dual chart speed options will appear.



Set up chart speed 1:

- Enter a chart speed in millimeters per second or millimeters per minute. Then select a duration for the speed using a specific amount of time, or until a trigger occurs.

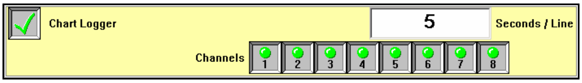
Set up chart speed 2:

- Enter a chart speed in millimeters per second or millimeters per minute. Then select a duration for the speed using a specific amount of time, or until an abort occurs.

3. To set up a constant chart speed, check the Constant Chart Speed option. Chart speed options will appear.

Specify the constant chart speed in millimeters per second or millimeters per minute.

4. To set up the chart logger, check the Chart Logger option. Chart logger options will appear.



Select the channels that will be logged and specify a value for seconds/line.

5. If necessary, modify the current chart/monitor speed on the bottom of the window. The current speed can be specified in millimeters per second or minute, or as external.

   External speeds are specified using an input via the utility port. When the external chart/monitor speeds are used, the Dual Speed, Constant Speed, and Chart Logger options will be disabled.

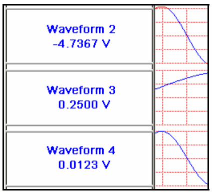
6. Choose the OK button to finish setting up Realtime mode.

Measurement and analysis tools

This section provides information about the various measurement tools available in Realtime mode.

Channel meter

The channel meter displays real-time values for signals. When active, the meter is located on the left side of the waveform display area. The meter can be configured to display information about viewed signals or all signals.



To display the channel meter:

1. Access the channel meter from the menu bar or control panel.

   - To access the channel meter from the menu bar, choose View >> Meter.

   - To access the channel meter from the control panel, choose the Meter icon button.

   Repeat this step to toggle between showing and hiding the meter.

2. If necessary, modify the meter display by choosing Setup >> Meter Channels. A sub menu will appear.

   - To display only the signals viewed in the waveform display area, choose View Channels.

   - To display all signals connected to the Dash 8Xe, choose All Channels.

   Alternately, touching the channel meter will display a sub-menu with similar view options.

3. If necessary, modify the meter font size by choosing Setup >> Meter Channels >> Font. A sub menu will appear. Choose whether to display small, medium, or large fonts.

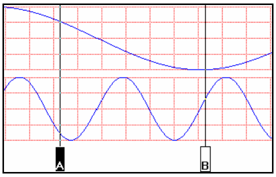
Cursors

Cursors are a valuable tool that can be used for measuring signals. They are used in conjunction with the Channel Information window to view a variety of measurements.

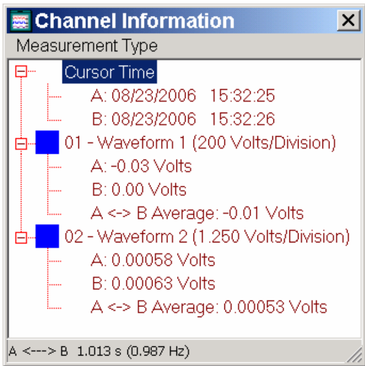
Note: To save time and gain more control over cursors, add cursor buttons to the control panel. Refer to *Chapter 5: Menus and buttons* for more information on customizing the control panel.

To measure signal information 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 View >> Show 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. The following measurement types are available: Average, Max-Min, Peak-Peak, Slope, and RMS.

Average - Average displays the midpoint value of the data represented by the cursor. When using two cursors, this measurement displays the average of the midpoint values of the data between cursors A and B.

Max-Min - Max-Min displays the maximum and minimum signal values of the data represented by the cursor. When using two cursors, this measurement displays the maximum and minimum signal values represented by the data between cursors A and B.

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

Slope - Slope displays the slope (delta V per unit time) of the signal between cursors A and B. This measurement is available only when both cursors are displayed.

RMS - RMS displays the root-means-square of the midpoints of the max-min pairs (line segments). This measurement is available only when both cursors are displayed.

3. To change the measurement type, choose the Measurement Type option on the menu bar of the Channel Information window. A sub menu will appear. Choose a measurement type from the list.

4. If necessary, move one or both of the cursors. Cursors must be activated before they can be moved.

   - **Move cursor A or B individually** by choosing Cursors >> Active Cursor >> Cursor A or Cursor B to activate it. Then press the active cursor and drag it to a new location.

   - **Move cursor A and B together** by choosing Cursors >> Active Cursor >> Cursor A + B to activate both cursors. Then press one of the cursors and drag it to a new location. The other cursor will move as well.

Note: As a shortcut, activate a cursor by pressing it once on the touch-screen. To move it, press it again, but do not release it. While pressing, drag it to a new location, and then release it.

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

5. If necessary, modify the color and type of the cursors.

   - To modify the cursor color, choose Cursors >> Color. Then select a color from the options that appear.

   - To modify the cursor type, choose Cursors >> Type. Then select a type from the options that appear.

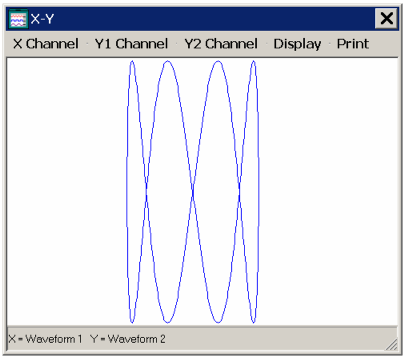
Select the Color option to use color cursors. Select the Invert option to use pseudo-transparent cursors.

XY Plot

Use the following instructions to display the XY Plot.

To display the XY plot:

1. Choose View >> XY Plot to open the XY plot window.



2. Select the X, Y1, and Y2 channels using the X Channel, Y1 Channel, and Y2 Channel menu options.

3. If necessary, select the Display menu to use the following display options:

   Refresh - Use the Refresh option to clear the window and restart the XY plot drawing.

   Connect Points - Use the Connect Points option to draw lines to connect the sample points on the XY plot. When this option is active, segments will be drawn.

   Background Color - Use the Background Color option to select a background color for the XY plot window.

   Foreground Color - Use the Foreground Color option to select a color for the points and segments drawn on the XY plot.

   Grid - Use the Grid option to display a grid. Grid color and division settings can also be specified.

Point Buffer - Use the Point Buffer option to turn on the point buffer. A point buffer size can also be specified.

Open Paint After Save - Use the Open Paint After Save option to open a plot in Microsoft Paint after saving the plot. When this option is active, plots will automatically be opened in Paint.

Save Plot - Use the Save Plot option to save the XY plot as a bitmap (BMP) image file.

4. If the printer is installed, use the Print >> Print Screen option to print the XY Plot window contents.

XY plot templates

XY plot templates allow you to display a customized background template in the XY plot window. This background is a visual aid you can use for comparing plot results against a standard template you define.

This section describes how to create and use XY plot templates.

To create XY plot templates:

1. Use a plain text editor, such as Notepad, on a personal computer to create and save a coordinates file. Save the file with the extension CRD.

2. Enter curve information in the coordinates file. You can enter up to ten curves. Refer to the following example when creating curves:

```
[Curve 1]
Thickness=2
Color=1
Pt1=100,490
Pt2=100,510
Pt3=100,500

[Curve 2]
Thickness=2
Color=1
Pt1=490,100
Pt2=510,100
Pt3=500,100
```

- **[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 following colors are available:

Value	**Color**
0	Black
1	Bright Red
2	Red
3	Bright Green
4	Green
5	Bright Blue
6	Blue
7	Yellow
8	Brown
9	Bright Magenta
10	Magenta
11	Bright Cyan
12	Cyan
13	Gray
14	Bright Gray
15	White

- **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

  n = 1 to 300
  x = 0 to 1000
  y= 0 to 1000

  The points are to be scaled from 0 - 1000 and are contained in the first quadrant (both X and Y positive).

3. Enter text fields in the coordinates file. You can enter up to 100 text fields. Refer to the following example when creating text fields:

```
[Text 1]
Color=1
Horizontal=0
Vertical=-1
Pt=100,512
String=4.0

[Text 2]
Color=1
Horizontal=0
Vertical=-1
Pt=200,512
String=3.0
```

- **[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 curve. The following colors are available:

Value	**Color**
0	Black
1	Bright Red
2	Red
3	Bright Green
4	Green
5	Bright Blue
6	Blue
7	Yellow
8	Brown
9	Bright Magenta
10	Magenta
11	Bright Cyan
12	Cyan
13	Gray
14	Bright Gray
15	White

- **Horizontal** - Specify the horizontal justification of the text: (-1) left justified, (0) centered, (1) right justified.

- **Vertical** - Specify the vertical justification of the text: (-1) bottom justified, (0) centered, (1) top justified.

- **Point** - Enter the point at which to position the text field. Points are specified in the following format: Pt=x,y

  x = 0 to 1000
  y= 0 to 1000

  The points are to be scaled from 0 - 1000 and are contained in the first quadrant (both X and Y positive).

- **String** - Enter the text string to display in the field. You can enter up to 100 characters.

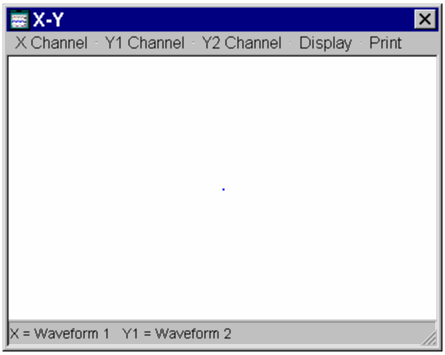
4. Save the coordinates file. Ensure you save it with a CRD extension. Then copy it to a USB-based memory storage device.

Note: Windows may display the CRD file as a "card file" icon. This only affects how the icon appears in Windows, and will not affect file contents or how the file is used.

To use XY plot templates:

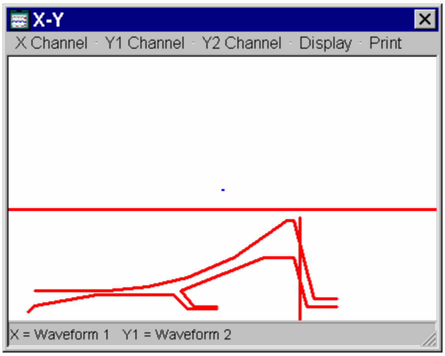
1. Insert the USB-based memory storage device that contains the coordinates (CRD) file into the Dash 8Xe USB port.

2. Choose View >> XY Plot to open the XY plot window.

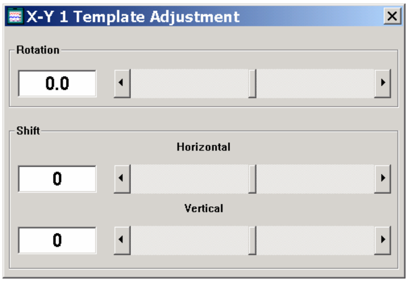


3. To load an XY template, choose Display >> Template >> Load Template. A file selection window will open.

   Browse to the coordinates file (CRD) and choose OK. The XY template will be loaded in the XY plot window.



4. To adjust the XY template, choose Display >> Template >> Adjust Template. The Template Adjustment window will open.



   - Adjust the rotation of the XY template by entering a specific rotation in degrees, or by moving the rotation slider bar.

   Note: If a mouse is connected, you can right-click and drag in the XY plot window to adjust rotation. After rotating the XY template, press the right mouse button to re-display the mouse pointer.

   - Adjust the horizontal and vertical position of the XY template by entering shift values in pixels, or by moving the horizontal and vertical position slider bars.

Note: You can also adjust the horizontal and vertical position by touching and dragging in the XY plot window. If a mouse is connected, you can left-click and drag to adjust position.

5. To remove the XY template from the XY plot window, choose Display >> Template >> Remove Template. The XY template will be removed.

View options

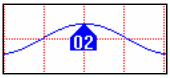
This section describes how to view signal IDs, full-page marks, and on-screen pen options.

Printing signal IDs

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

To print signal IDs:

1. Choose View >> Print IDs from the menu bar.



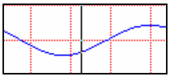
A signal ID indicator will be displayed and printed for each waveform and event.

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.

To print a full-page mark:

1. Choose View >> Full Page Mark from the menu bar.



A mark will be printed vertically across all waveforms.

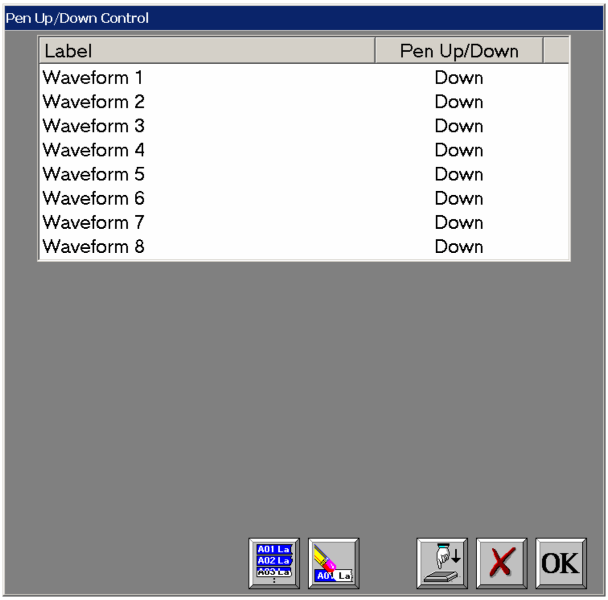
Adjusting pens

Waveforms are drawn by on-screen pens that can be raised and lowered. When pens are in the up position, waveforms will not be drawn. When they are in the down position, waveforms will be drawn. The pen appearance, or style can also be modified.

Use the following instructions to raise/lower pens and modify the pen style.

To raise and lower pens:

1. Choose Setup >> Pen Lift to open the Pen Up/Down Control window.



   This window displays a list of waveforms and the current pen position for each waveform.

2. Raise or lower pens as needed.

   - To raise pens, select the waveform(s) and choose the Pen Up/Down column heading. A sub menu will appear. Choose the Up option. Pens for the selected waveforms will be raised.

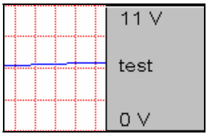
   - To lower pens, select the waveform(s) and choose the Pen Up/Down column heading. A sub menu will appear. Choose the Down option. Pens for the selected waveforms will be lowered.

3. To complete the pen raise and lower process, choose OK.

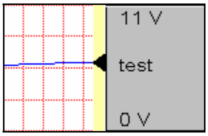
To select a pen style:

1. Choose View >> Pen Style from the menu bar. A list of pen style options will appear.

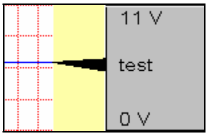
2. Select a pen style.



None - The None pen style does not display an on-screen pen.



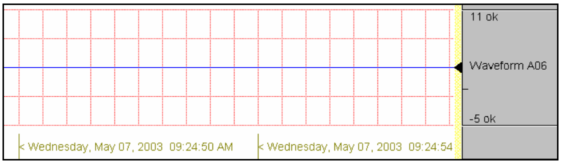
Moving Origin - The Moving Origin pen style displays a pen that moves vertically as it draws waveforms.



Fixed Origin - The Fixed Origin pen style displays a pen with a tip that tilts vertically on a fixed axis as it draws waveforms.

Viewing the time log

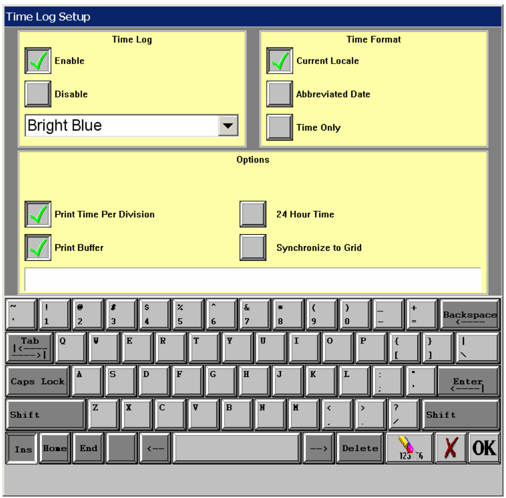
The time log appears along the bottom of the display and marks points along the time axis.



Use the following instructions to display and customize the time log.

To view the time log:

1. Choose View >> Time Log from the menu bar. The Time Log Setup window will open.



2. Choose whether to enable or disable the time log. Then select a color for the time log text.

3. Select a time format. The following formats are available:

   Current Locale displays the day, date, and time as indicated in the following example:

      Thursday, May 15, 2003 02:03:11 PM

Abbreviated Date displays a shorter version of the date and time, as indicated in the following example:

05/15/03 02:03:11 PM

Time Only displays only the time, as indicated in the following example:

02:03:11 PM

4. Specify your time log option preferences.

   - To print the amount of time per division, check the Print Time Per Division option.
   - To print a text buffer as part of the time log, check the Print Buffer option. Then enter a text string in the field.
   - To print the time in 24-hour time instead of the standard time, check the 24 Hour Time option.
   - To synchronize time log entries to grid lines, check the Synchronize to Grid option.

5. When you are finished setting up the time log, choose OK.

Global setup files

Global setup files contain a variety of 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.

The following items can be saved as part of a global setup file.

Signals - Saves channel amplifier settings.

Note: For information on creating signal setup files individually, refer to *Chapter 6: Channel setup*.

Realtime mode control panel - Saves the Realtime mode control panel.

Note: For information on creating control panel setup files individually, refer to *Chapter 5: Menus and buttons*.

Review mode control panel - Saves the Review mode control panel.

Note: For information on creating control panel setup files individually, refer to *Chapter 5: Menus and buttons*.

Scope mode control panel - Saves the Scope mode control panel.

Note: For information on creating control panel setup files individually, refer to *Chapter 5: Menus and buttons*.

View - Saves the view and display setup.

Note: For information on creating view setup files individually, refer to *Chapter 7: Display setup*.

Miscellaneous - Saves system settings not included in the other files.

Global files can greatly decrease the amount of time spent on setting up channels, control panels, and views before measurement.

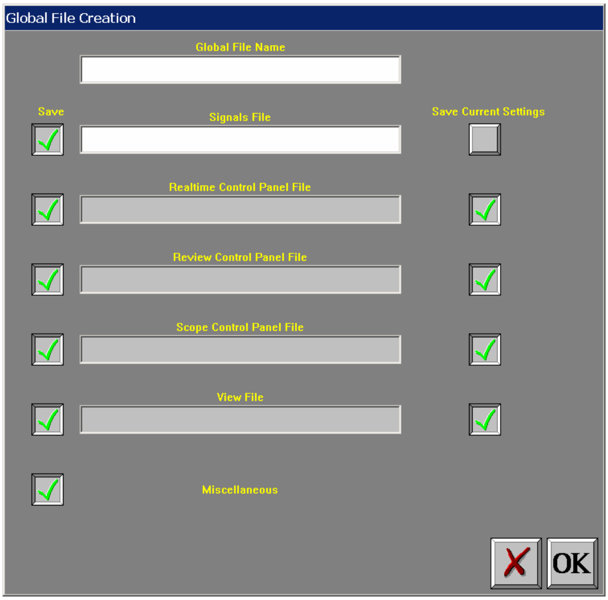
It may be helpful to create a library of global setup files for commonly used measurement configurations. Additionally, global setup files are portable, so they can be shared with other Dash 8Xe units.

Saving global setup files

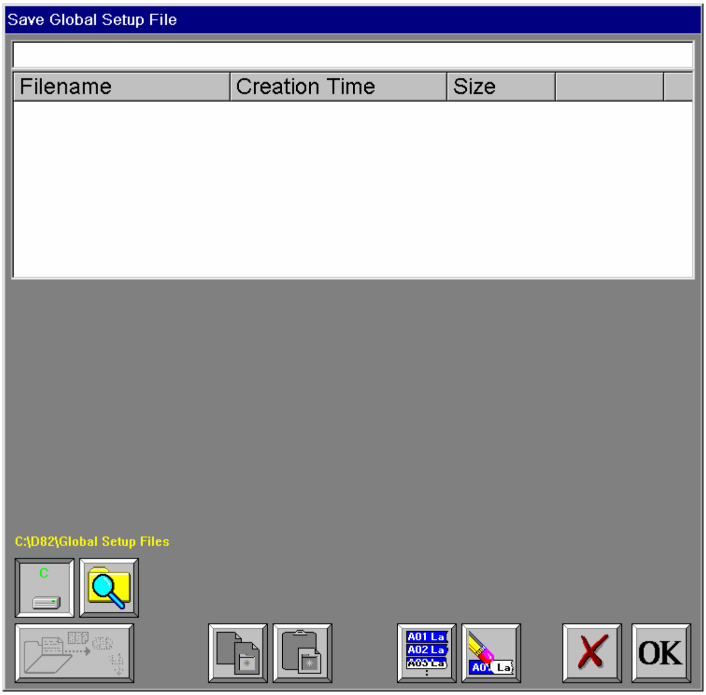
Use the following instructions to save global setup files. Global setup files are saved using the Global File Creation window, which is available from the menu bar.

To save global setup files:

1. Choose File >> Save >> Global to open the Global File Creation window.



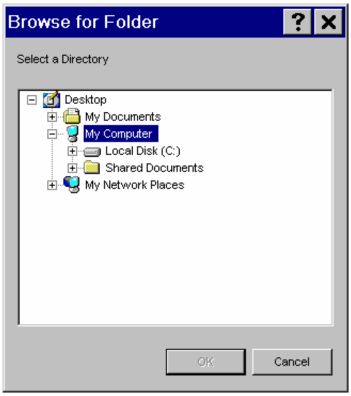
2. Choose the Global File Name field. The Save Global Setup File window will open.



By default, the global setup file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

3. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the global setup file and choose the OK button. The Global File Creation window will re-appear, with the specified file name displayed in the Global File Name field.

4. For each item available in the global file, choose whether to save the item in the global file, and choose whether to save the current settings or specify an existing setup file for each item.

   Specify whether to save each item in the global file.

   - To save the item as part of the global file, check the Save checkbox.

   Specify whether to save the current settings or select an existing setup file for each item.

   - To save the current settings, check the Save Current Settings checkbox.

   - To specify a setup file instead of using the current settings, uncheck the Save Current Settings checkbox. Then choose the associated file name field for the item. A file load window for the item will appear. Select a file and choose the OK button.

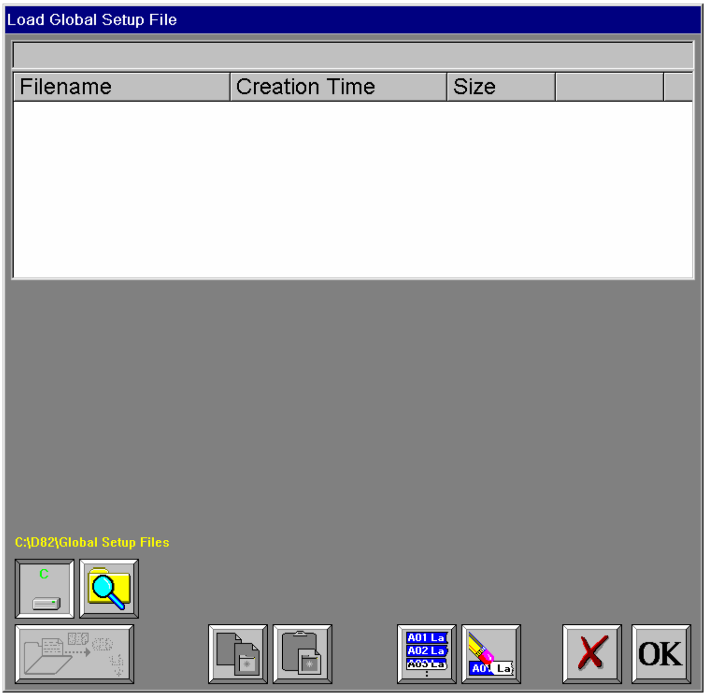
5. Choose the OK button in the Global File Creation window. The global setup file will be saved.

Loading global setup files

Use the following instructions to load global setup files. Global setup files can be loaded from the menu bar or control panel.

To load global setup files using the menu bar:

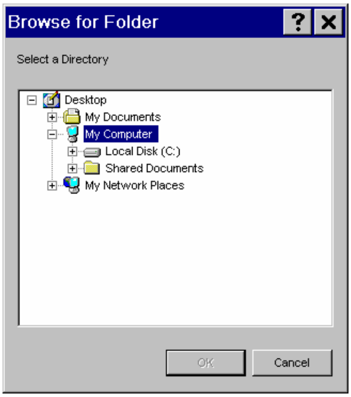
1. Choose File >> Load >> Global. The Load Global Setup File window will open.



2. All saved global setup files in the default location on the system drive (C) will be displayed. Select a global setup file from this list.

   If necessary, a file can be retrieved from a different location on the system drive. To select a file from an alternate location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file and choose the OK button. The path displayed in yellow text will be updated to indicate the new file selected.

3. Choose the OK button. The selected global setup file will be loaded, and the settings will be modified accordingly.

Note : To delete a global setup file, select the file and press the delete file button. The software will ask you to confirm your choice, and will then delete the file.

To load global setup files using the control panel:

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

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

   Choose the File button. A sub menu will appear. Then choose Global. The Load Global Setup File window will open.

   Select a global setup file and choose OK.

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

The name displayed in this icon will correspond to the name of the global setup file.

Chapter 9: SCR-8 thermal printer use

This chapter provides information about the optional thermal printer.

Freezing and running the chart

The printed chart can be stopped and started using two different methods:

- Freezing/running the chart and monitor simultaneously
- Freezing/running the chart independently from the monitor

The following instructions provide information on freezing/running the chart and monitor, and freezing/running the chart independently. Each of these tasks can be accomplished using the menu bar or control panel.

Note: The chart run and halt function can also be controlled via an input pin on the utility port. Refer to *Chapter 2: Hardware overview* for information on the utility port pin configuration.

To freeze/run both the chart and monitor using the menu bar:

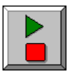
1. Choose View >> Freeze Display.

   The on-screen monitor and the printed chart will halt.

2. Repeat this process to toggle between frozen and running chart and monitor states.

To freeze/run both the monitor and chart using the control panel:

1. Choose the Run/Halt Monitor and Chart button.

   The on-screen monitor and the printed chart will halt.

2. Repeat this process to toggle between frozen and running chart and monitor states.

To freeze/run the chart using the menu bar:

1. Choose Chart >> Halt.

   The printed chart will halt, but the monitor will continue running.

2. Repeat this process to toggle between frozen and running chart states.

To freeze/run the chart using the control panel:

1. Add the Chart Run/Halt button to the control panel.

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

   Choose the Chart button. A sub menu will appear. Then choose Chart Run/Halt. The Chart Run/Halt button will be added to the control panel.

   Then choose OK to close the Control Panel Setup window.

2. Choose the Chart Run/Halt button.

   The printed chart will halt, but the monitor will continue running. Choose the Chart Run/Halt button again to toggle between frozen and running chart states.

Adjusting the chart speed

The chart printing rate is controlled by the Chart Speed buttons on the control panel. Five default chart speed options are provided (1, 5, 25, 50, and 100 mm/s).

The control panel can be customized to add other chart speeds in units of millimeters per second or minute. Refer to *Chapter 5: Menus and buttons* for more information on customizing the control panel.

Note: The chart speed can also be controlled via an input pin on the utility port. Refer to *Chapter 2: Hardware overview* for information on the utility port pin configuration.

To adjust the chart speed:

1. Choose a Chart Speed button.

The speed of the on-screen monitor and printed chart will be adjusted based on the selection.

Printing the screen

Use the following instructions to print the contents of the Dash8X display.

To print the screen:

1. Choose File >> Print Screen from the menu bar. The screen will be printed.

Printing a chart mark

Use the following instructions to print a full-page mark across the width of the chart.

To print a chart mark:

1. Choose Chart >> Mark from the menu bar. A mark will be printed across the chart.

Adjusting trilevel printing rates

Trilevel marks are printed along the bottom of the chart to indicate time measurements. Small, medium, and large marks are printed based on a time interval set.

To adjust trilevel printing rates:

1. Choose Setup >> Trilevel Rate from the menu bar. A list of trilevel rate options will appear.

    0.01, 0.10, 1.00
    0.02, 0.20, 2.00
    0.04, 0.40, 4.00
    0.10, 1.00, 10.00
    0.20, 2.00, 20.00
    0.40, 4.00, 40.00
    1.00, 10.00, 100.00
    3.00, 30.00, 300.00
    6.00, 60.00, 600.00

Each trilevel set represents a series of three time intervals (in seconds).

The first number in each series indicates how often the small marks will be printed. The second number in each series indicates how often the medium marks will be printed. Similarly, the last number in each series indicates how often the large marks will be printed.

2. Select a trilevel rate from the list. The rate will be applied.

Demand buffer

The demand buffer is a text string of 128 characters stored in memory that can be printed on the chart when necessary. This text prints parallel to the time axis. This section describes how to print, edit, and move the demand buffer.

Printing the demand buffer

Use the following instructions to print the demand buffer on the chart.

To print the demand buffer:

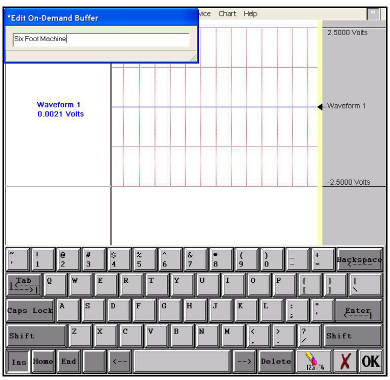
1. Choose Chart >> Print Demand Buffer. The demand buffer will be printed on the chart.

Editing the demand buffer

Use the following instructions to edit the text stored in the demand buffer.

To edit the demand buffer:

1. Choose Chart >> Edit Demand Buffer. A keypad will appear with the current demand buffer displayed in the text field.



2. Edit the demand buffer text and choose OK. The demand buffer will be updated based on the edits.

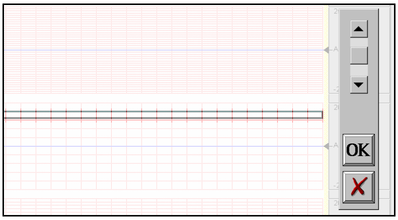
3. The demand buffer text field will remain visible, and can be changed on-the-fly while the chart is running.

Moving the demand buffer

Use the following instructions to move the demand buffer to a new location on the chart.

To move the demand buffer:

1. Choose Chart >> Move Demand Buffer. A slider bar will appear with the current position of the demand buffer highlighted.



2. Move the demand buffer vertically using the slider bar. Then choose OK. The demand buffer will be moved to the new position.

Printing channel limits

Use the following instructions to print the upper and lower limit values for each grid.

To print channel limits:

1. Choose Chart >> Print Limits from the menu bar. The upper and lower limits for each grid will be printed on the chart.

Editing the system log

The system log is printed along the top of the chart. It contains information such as time, date, and chart speed as well as the inverse of the chart speed for ease of periodic measurement. In addition, it can hold 30 characters of text that is printed as well.

Use the following instructions to edit the system log.

To edit the system log:

1. Choose Chart >> Edit System Log from the menu bar. A keypad will appear with the current system log displayed in the text field.

2. Edit the system log text and choose OK. The system log will be updated based on the edits.

Printing channel reports

When the channel report option is enabled, channel scaling information will be printed at the end of each text buffer. Use the following instructions to print channel reports.

To print channel reports:

1. Choose Chart >> Channel Report from the menu bar. The channel report option will be enabled.

2. To disable the channel report option, repeat this process.

Printing channel IDs next to waveforms

When the Auto ID option is enabled, channel numbers will be printed next to waveform traces approximately once per two sheets of paper.

To print channel IDs next to waveforms:

1. Choose Chart >> Auto ID from the menu bar. The Auto ID option will be enabled.

2. To disable the Auto ID option, repeat this process.

Adjusting the waveform trace thickness

The thickness of waveforms can be changed for better contrast with the grid. Use the following instructions to adjust the waveform trace thickness.

To adjust the waveform trace thickness:

1. Choose Chart >> Trace Thickness from the menu bar. A list of twelve thickness options will appear (1 dot – 12 dots).

2. Select a waveform thickness. The selected size will be used to print waveforms.

Using text buffers

The Dash 8xe provides a total of 18 text buffers, which can be added and remove as needed. Text buffers are periodically printed parallel to the time axis. This section describes how to add, remove, edit, and move text buffers.

Adding text buffers

Use the following instructions to add text buffers to the chart.

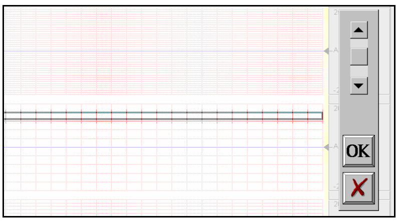
To add text buffers:

1. Choose Chart >> Text Buffers >> Add from the menu bar. A list of text buffers will appear. Then select the text buffer to add.

   Note: Text buffers that have already been added will be "grayed out," and cannot be selected.

   A keypad will appear.

2. Enter the buffer text and choose OK. A buffer position slider bar will appear.

3. Move the text buffer vertically using the slider bar and choose OK. The text buffer will be added in the selected position.

Removing text buffers

Use the following instructions to remove text buffers from the chart.

To remove text buffers:

1. Choose Chart >> Text Buffers >> Remove from the menu bar. A list of text buffers will appear. Then select the text buffer to remove.

   Note: Text buffers that do not exist will be "grayed out," and cannot be selected.

   The text buffer will be removed from the chart.

Editing text buffers

Use the following instructions to edit the text stored in text buffers.

To edit text buffers:

1. Choose Chart >> Text Buffers >> Edit from the menu bar. A list of text buffers will appear. Then select the text buffer to edit.

   Note: Text buffers that do not exist will be "grayed out," and cannot be selected.

   A keypad will appear.

2. Edit the buffer text and choose OK. The text buffer will be updated based on the edits.

Moving text buffers

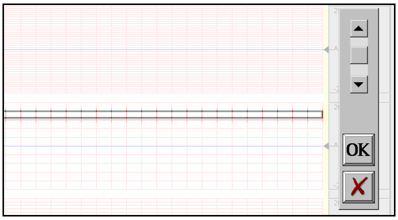
Use the following instructions to move text buffers to different locations on the chart.

To move text buffers:

1. Choose Chart >> Text Buffers >> Move from the menu bar. A list of text buffers will appear. Then select the text buffer to move.

   Note: Text buffers that do not exist will be "grayed out," and cannot be selected.

A buffer position slider bar will appear.



2. Move the text buffer vertically using the slider bar and choose OK. The text buffer will be moved to the selected position.

Motor adjustments

The Potentiometer window (Chart >> Motor Adjustments) is used to adjust the SCR-8 motor. This function is for Astro-Med trained personnel only. Please contact Technical Support for further information.

Chapter 10: Data capture

This chapter provides information about the data capture capability of the Dash 8Xe.

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 a dedicated hard drive.

Data capture record (DCR) file
The Dash 8Xe saves data capture information in a type of file format known as a DCR 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. The size can be defined in units of time (hours, minutes, or seconds) or number of samples (KS/channel).

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 storage allocation

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

This allocation amount is defined by a percentage of the whole data capture size. For example, a pre-trigger storage allocation 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 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.

Auto archive

The Dash 8Xe 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 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 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. It also details the setup process for the IRIG time code, which will be helpful for users requiring IRIG time code input.

Data capture setup

Use the following instructions to set up a data capture. The data capture setup process involves entering a file name for the data capture, specifying storage allocation options, defining and selecting channel sample rates, and activating the desired automation options.

To set up a data capture:

1. Choose Capture >> Capture Settings to open the Capture Setup window.

In the lower-left corner of the window, a hard drive status bar is displayed. This bar provides a graphical representation of data capture hard drive space. The following color key is used:

White - The white-colored area indicates empty space that is available for data captures.

Blue - The blue-colored area indicates used space that already contains data captures.

Green - The green-colored area indicates the anticipated space the next valid data capture will occupy.

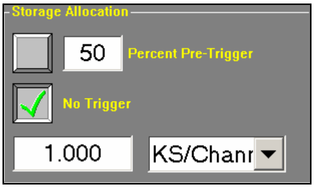
Red - The red-colored area indicates that the next data capture will exceed the amount of space available on the drive.

The Capture Setup window also provides a format button, which can be used to format the data capture hard drive. Formatting the drive is useful for deleting all records from the drive. Individual records can be deleted using Review mode.

2. Specify a DCR file name by choosing the Base Filename field. A keypad will appear. Enter a 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 specified file name.

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

   - If the data capture will contain pre-trigger data, check the Percent Pre-Trigger option. Then choose the Percent Pre-Trigger 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.

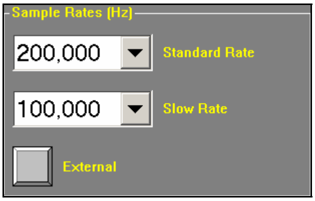
   - If the data capture will not contain pre-trigger data, check the No Trigger option.

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 Storage Allocation 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.

An estimate of needed disk space for the DCR file is derived based on the data capture storage allocation and sample rate specified. This estimate is displayed in the hard drive status bar.

5. Specify sample rates to define the speeds at which signals are sampled. Sample rates are defined in units of Hz (hertz), the number of samples per second.



- For data captures recording pre-trigger data, each channel can be sampled at a standard rate ranging from .l01667 Hz (one sample per minute) to 200,000 Hz. Select a sample rate from the Standard Rate drop-down list.

- For data captures not recording pre-trigger data, each channel can be sampled using one of two available sample rates (standard or low).

  The standard rate can be set to a value ranging from .01667 Hz (one sample per minute) to 200,000 Hz. The slow rate can be set to a value up to half the standard rate. Select samples rates from the Standard Rate and Slow Rate drop-down lists.

- As an alternative, an external sample clock can be used to define a sample rate. The external clock provides the capability for sample rates up to 40,000 Hz. Rates approaching 200,000 Hz will work, but with sampling jitter of up to 5 μ sec.

Note: To avoid aliasing, the sample rate must be at least two times the highest signal frequency being captured. User-selectable filters can be set up to limit signals if possible frequencies are unpredictable.

Aliasing can be checked by performing a test capture at a 200,000 Hz sample rate. At this rate, the Dash 8Xe hardware filters will protect against aliasing.

---

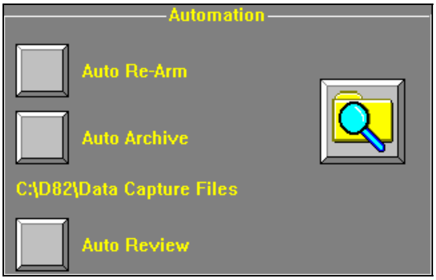
6. Assign sample rates to channels by selecting channels from the list box. Then choose the Rate column heading. Select a rate from the options that appear.

   Off - Data will not be recorded for the selected channels.

   Standard - Data will be recorded at the standard rate for the selected channels.

   Slow - Data will be recorded at the slow rate for the selected channels. This rate is unavailable when capturing pre-trigger data.

7. Specify automation options by choosing whether to enable automatic re-arming, archiving, and auto-review.



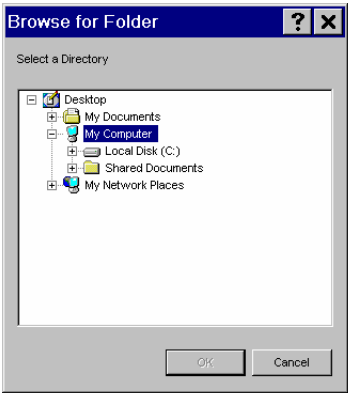
- To automatically re-arm the capture immediately after the current capture is complete, check the Auto Re-Arm option.

  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 archive the DCR file to the system or other drive immediately after the capture is complete, check the Auto Archive option. The location the file will be saved in will be displayed in yellow text beneath the Auto Archive button.

  If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

Note : In addition to accessing other folders on the system drive, the Browse for Folder window will also provide access to any connected USB storage devices.

- To automatically open the DCR file in Review mode immediately after the data capture is complete, check the Auto Review option.

8. Choose OK to complete the data capture setup process.

IRIG time code setup

If desired, an IRIG B time code can be captured during the data capture process.

The IRIG B time code can be captured using an event channel that is decoded in the review process to allow measurement in absolute time. A minimum sample rate of 10 KHz must be used, and the capture duration must be at least 2 seconds.

All measurements are accurate to one millisecond. Use the following instructions to set up the IRIG B time code.

To set up the IRIG time code:

1. Choose Capture >> Capture Settings to open the Capture Setup window.

2. Ensure that the sample rate is 10 KHz or greater.

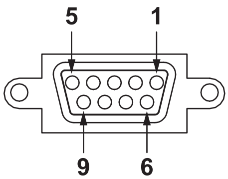
3. Ensure that the capture duration is at least 2 seconds.

4. In the channel list, ensure that events are captured at the Standard rate.

5. Select the event port pin that will be used to input the IRIG B signal. The selected IRIG Event Channel list value will be used as the IRIG input.

   For example, if E02 - Event 02 is selected, event port pin number 2 will be used for IRIG B input.

   The following event port diagram illustrates the layout of event channel pins. Connect the IRIG B TTL signal between the specified pin and ground (pin 9).



Note: For more details on the event port pin configuration, refer to *Chapter 2: Hardware overview*.

6. Choose OK to complete the IRIG time code setup process.

   Once a data capture has been saved using these settings, the IRIG B time code information can be viewed in Review mode. To view IRIG B time choose View >> Status Text Format >> Absolute Time from the menu bar in Review mode.

Trigger setup

Triggers are user-defined events that initiate the post-trigger data capture process. This section provides information about the types of triggers available and the setup process for each trigger.

Trigger types

The following list describes each trigger type available in the Dash 8Xe.

Note: If multiple types are set up, a trigger will occur when trigger conditions are met for *any* trigger type.

Amplitude Level OR

Amplitude Level OR is a commonly used method that produces a trigger when a signal exceeds a certain level. This type of trigger cannot be used simultaneously with the Amplitude Window OR trigger.

This trigger utilizes OR logic. A trigger will occur when *any* of the specified signals exceed a certain level.

Amplitude Window OR

Amplitude Window OR is a commonly used method that produces a trigger when signals move outside a predetermined window. This type of trigger cannot be used simultaneously with the Amplitude Level OR trigger.

This trigger utilizes OR logic. A trigger will occur when *any* of the specified signals move outside of a predetermined window.

Slew OR

Slew OR produces a trigger when the specified change in signal slew rate, or frequency occurs.

This trigger utilizes OR logic. A trigger will occur when *any* of the specified signals meet the slew rate conditions.

Slew AND
Slew AND produces a trigger when more than one channel must meet its respective slew rate conditions.

This trigger utilizes AND logic. A trigger will occur when *all* of the specified signals meet the slew rate conditions.

Events
Event triggers produce a trigger when event inputs meet their change in logic state. This is an event pattern trigger, as all events must meet the selected logic state to cause a trigger.

Manual
Manual is a commonly used method that allows the operator to produce a trigger manually via the menu bar or control panel.

External
External uses an external signal via the utility port to produce a trigger. The external signal is a switch closure to ground or TTL low to produce a trigger. Process control signals can be used to produce this trigger.

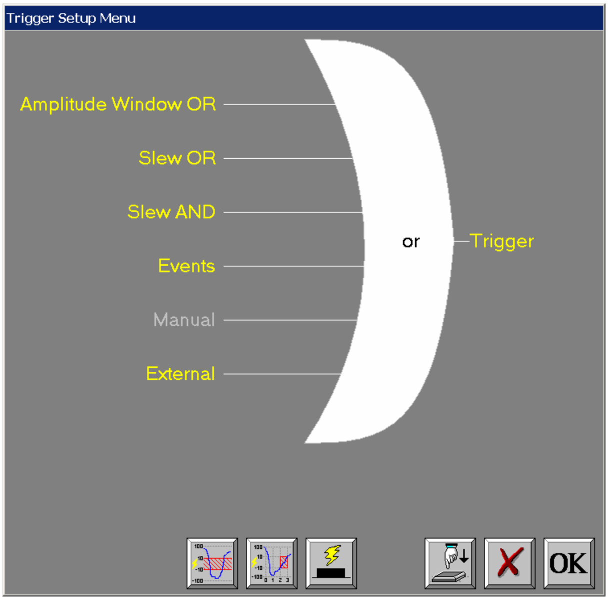
Enabling and disabling triggers

The Trigger Setup Menu window is used to enable and disable triggers. It is also the starting point of the trigger setup process. Use the following instructions to enable and disable triggers.

To enable and disable triggers:

1. Choose Setup >> Trigger Settings to open the Trigger Setup Menu window.

Note: To save time, add a control panel icon button for the Trigger Setup Menu window. For information on adding buttons to the control panel, refer to *Chapter 5: Menus and buttons*.



2. Enable and disable triggers by selecting the trigger labels. The color of the label will indicate whether the trigger is enabled.

   - If the trigger label text is yellow, the trigger is enabled.
   - If the trigger label text is gray, the trigger is disabled.

3. Choose OK to confirm the trigger enable/disable settings.

Setting up an Amplitude Level OR trigger

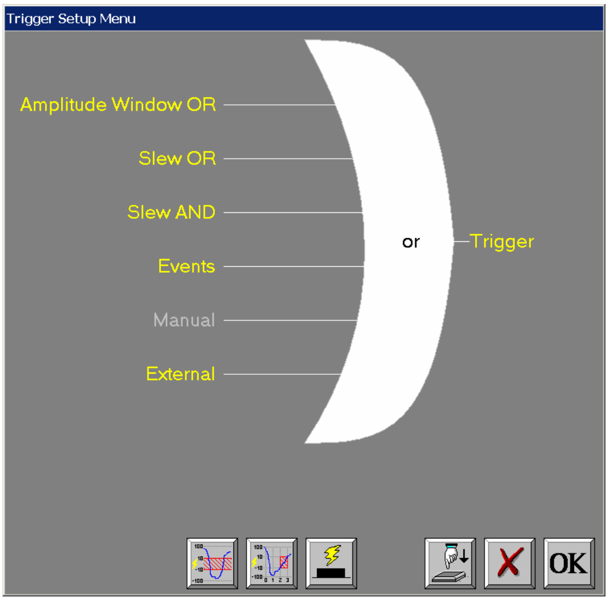
Amplitude Level OR is a commonly used method that produces a trigger when a signal exceeds a certain level. This type of trigger cannot be used simultaneously with the Amplitude Window OR trigger.

This trigger utilizes OR logic. A trigger will occur when *any* of the specified signals exceed a certain level.

Use the following instructions to set up an Amplitude Level OR trigger.

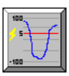
To setup an Amplitude Level OR trigger:

1. Choose Setup >> Trigger Settings to open the Trigger Setup Menu window.

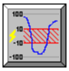


2. Open the Trigger Level window. The process for opening this window will vary based on which amplitude trigger option is displayed (Amplitude Level OR *or* Amplitude Window OR).

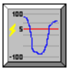
   - If the Amplitude Level OR trigger label is visible, open the Trigger Level window by choosing the Trigger Level button.



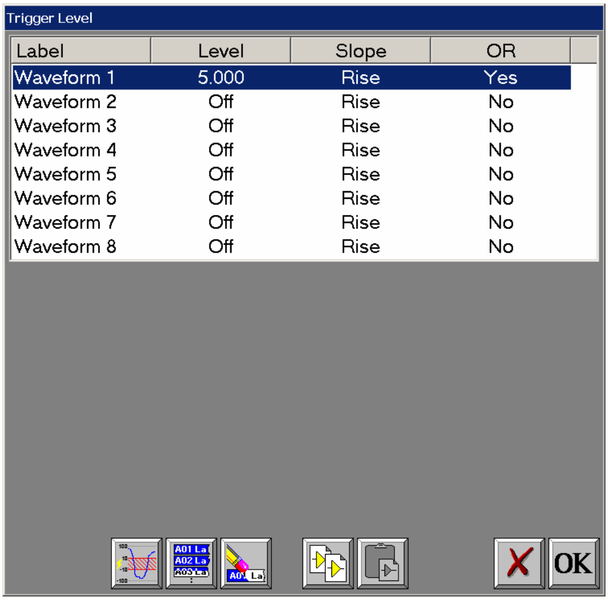
- If the Amplitude Window OR trigger label is visible instead, open the Trigger Window screen by choosing the Trigger Window button.

  From the Trigger Window screen, choose the Trigger Level button.

The Trigger Level window will open.



3. Define trigger levels by selecting the waveform(s) and choosing the Level column heading. A sub menu will appear.

   Choose the New Value option. A number pad will appear.

   Enter the trigger amplitude level and choose OK.

4. Define slope types by selecting the waveform(s) and choosing the Slope column heading. A sub menu will appear.

   Choose the type of slope required to initiate a trigger. Available options are Rise, Fall, or Rise or Fall.

5. Select whether to enable trigger conditions for each waveform by selecting the waveform(s) and choosing the OR column heading. A sub menu will appear.

   - Choose Yes to enable trigger conditions for the selected waveforms.

   - Choose No to disable trigger conditions for the selected waveforms.

6. Choose OK in the Trigger Level window. The Trigger Setup Menu window will re-appear.

   In the Trigger Setup Menu window, enable the Amplitude Level OR trigger by choosing the Amplitude Level OR label. The label text color will change to yellow when the trigger is enabled.

   Choose OK in the Trigger Setup Menu window to complete the Amplitude Level OR trigger setup process.

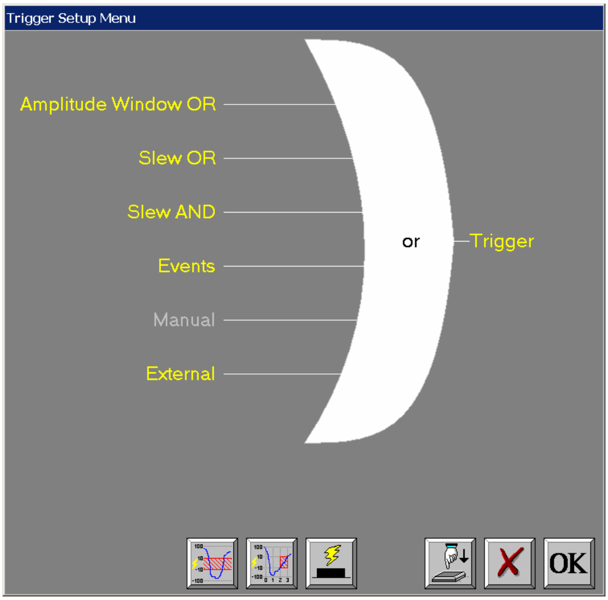
Setting up an Amplitude Window OR trigger

Amplitude Window OR is a commonly used method that produces a trigger when signals move outside a predetermined window. This type of trigger cannot be used simultaneously with the Amplitude Level OR trigger.

This trigger utilizes OR logic. A trigger will occur when *any* of the specified signals move outside of a predetermined window.

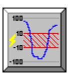
To set up an Amplitude Window OR trigger:

1. Choose Setup >> Trigger Settings to open the Trigger Setup Menu window.

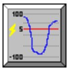


2. Open the Trigger Window screen. The process for opening this screen will vary based on which amplitude trigger option is displayed (Amplitude Window OR *or* Amplitude Level OR).

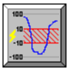
   - If the Amplitude Window OR trigger label is visible, open the Trigger Window screen by choosing the Trigger Window button.



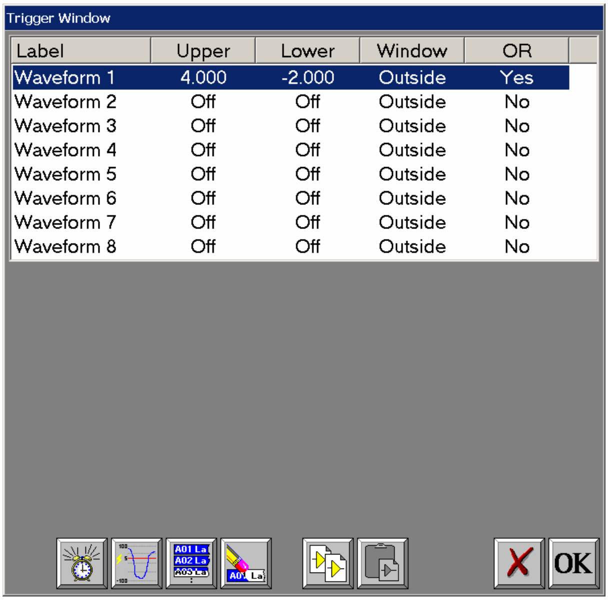
- If the Amplitude Level OR trigger label is visible instead, open the Trigger Level window by choosing the Trigger Level button.

  From the Trigger Level window, choose the Trigger Window button.

The Trigger Window screen will open.



3. Define upper and lower trigger window boundaries by entering specific values or assigning values used for alarm levels, which are defined during the channel setup process.

   - To enter specific upper and lower trigger window boundaries, select the waveform(s) and choose the appropriate column heading (Upper/Lower). A sub menu will appear.

     Choose the New Value option. A number pad will appear. Then enter the trigger level and choose OK.

   - To assign alarm level values, select the waveform(s) and choose the Alarm Levels button.

     The current upper and lower alarm levels, as defined in the Analog Channel Setup window, will be assigned as the upper and lower levels for selected waveforms.

Note: Assigning alarm levels is a one-time step that applies the current alarm values to the upper and lower trigger window boundaries. Using the Alarm Levels button does not link the two values.

For example, if alarm levels for the selected waveforms are later changed in the Analog Channel setup window, the lower and upper trigger window boundaries will *not* be automatically updated based on the new alarm values.

---

4. Specify the type of window by selecting the waveform(s) and choosing the Window column heading. A sub menu will appear.

   - To cause a trigger when signals move outside of the window limits, select the Outside option.

   - To cause a trigger when signals move inside the window limits, choose the Inside option.

5. Select whether to enable trigger conditions for each waveform by selecting the waveform(s) and choosing the OR column heading. A sub menu will appear.

   - Choose Yes to enable trigger conditions for the selected waveforms.
   - Choose No to disable trigger conditions for the selected waveforms.

6. Choose OK in the Trigger Window screen. The Trigger Setup Menu window will re-appear.

   In the Trigger Setup Menu window, enable the Amplitude Window OR trigger by choosing the Amplitude Window OR label. The label text color will change to yellow when the trigger is enabled.

   Choose OK in the Trigger Setup Menu window to complete the Amplitude Window OR trigger setup process.

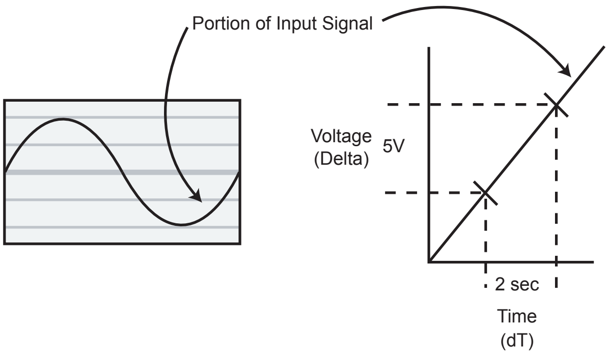
Setting up a Slew OR trigger

Slew OR produces a trigger when the specified change in signal slew rate, or frequency occurs.

This trigger utilizes OR logic. A trigger will occur when *any* of the specified signals meet the slew rate conditions.

Example

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



Delta - The Delta indicates the slew voltage span. The slope of this voltage span with respect to “dT” and “Window” will set the trigger point.

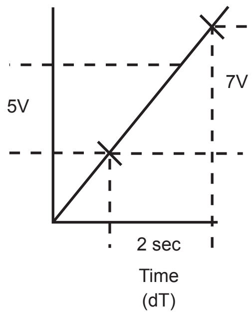
For this example, Delta is set to 5 V.

dT - The dT, or delta time, indicates the time to monitor the slope change of the “Delta” voltage span. This can be set from 400 uS to 20 seconds. This is done by choosing a number that when multiplied with 0.0004 (400 uS, the shortest time you can set) results in the “dT” time.

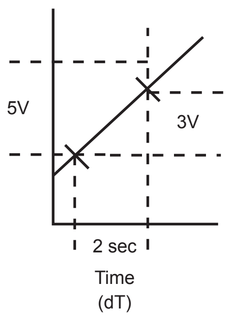
For this example, dT is set to 5000, resulting in a dT of 2 seconds.

Window - The Window indicates whether to trigger when the slew rate of the “Delta” with respect to “dT” rises above or below the specified trigger parameters.

- If using an “above window,” the following scenario would result in a trigger: Signal Slew Rate = 7V / 2 sec = 3.5 V/s

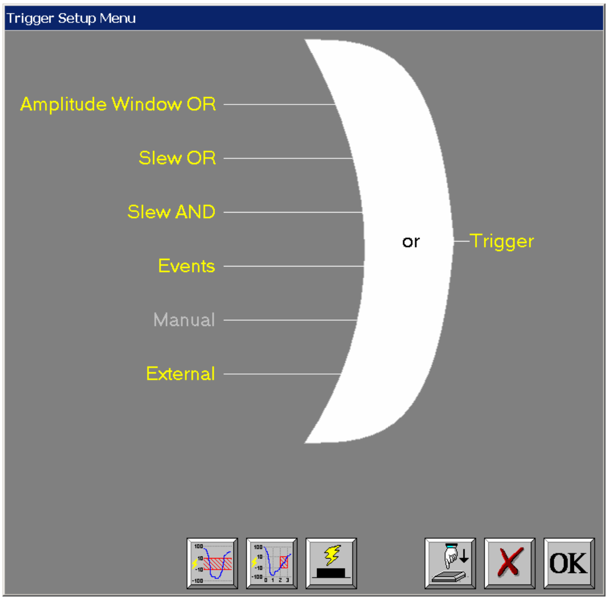


- If using a “below window,” the following scenario would result in a trigger: Signal Slew Rate = 3V / 2 sec = 1.5 V/s

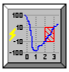


To set up a Slew OR trigger:

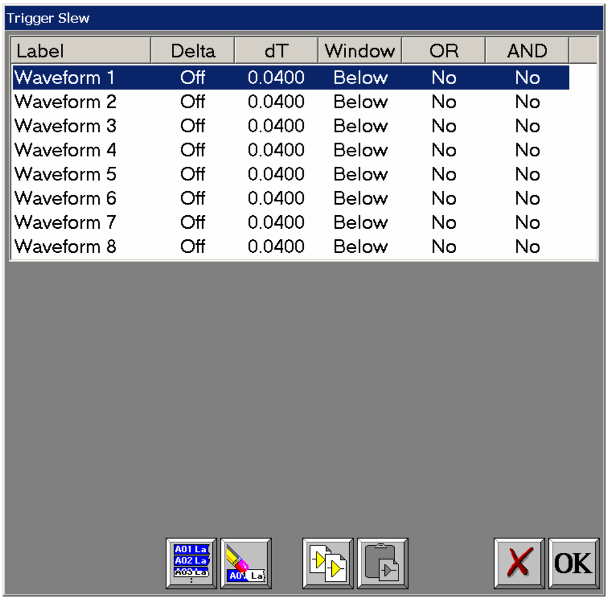
1. Choose Setup >> Trigger Settings to open the Trigger Setup Menu window.



2. Choose the Trigger Slew button.



The Trigger Slew window will open.



3. Define a slew delta by selecting the waveform(s) and choosing the Delta column heading. A sub menu will appear.

   Choose the New Value option. A number pad will appear.

   Enter the slew delta and choose OK.

4. Define a delta time by selecting the waveform(s) and choosing the dT column heading. A sub menu will appear.

   Choose the New Value option. A number pad will appear.

   Enter the delta time in seconds and choose OK.

5. Specify the type of window by selecting the waveform(s) and choosing the Window column heading. A sub menu will appear.

   - To cause a trigger when the signal rises above the chosen slew rate, select the Above option.

   - To cause a trigger when the signal falls below the chosen slew rate, select the Below option.

6. Select whether to enable trigger conditions for each waveform by selecting the waveform(s) and choosing the OR column heading. A sub menu will appear.

   - Choose Yes to enable trigger conditions for the selected waveforms.

   - Choose No to disable trigger conditions for the selected waveforms.

Note: The AND column is not used when setting up Slew OR triggers. This option is used during the Slew AND trigger setup process.

7. Choose OK in the Trigger Slew window. The Trigger Setup Menu window will re-appear.

   In the Trigger Setup Menu window, enable the Slew OR trigger by choosing the Slew OR label. The label text color will change to yellow when the trigger is enabled.

   Choose OK in the Trigger Setup Menu window to complete the Slew OR trigger setup process.

Setting up a Slew AND trigger

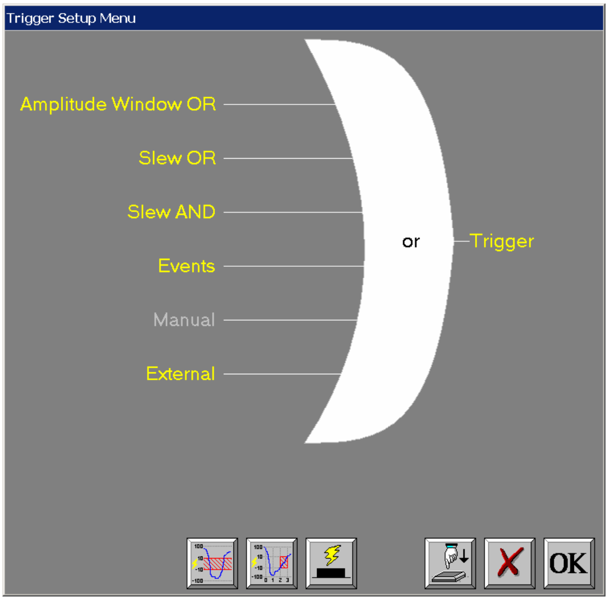
Slew AND produces a trigger when more than one channel must meet its respective slew rate conditions.

Note: The slew concepts for this trigger are identical to the concepts described in the *Setting up a Slew OR trigger* section.

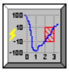
This trigger utilizes AND logic. A trigger will occur when *all* of the specified signals meet the slew rate conditions.

To set up a Slew AND trigger:

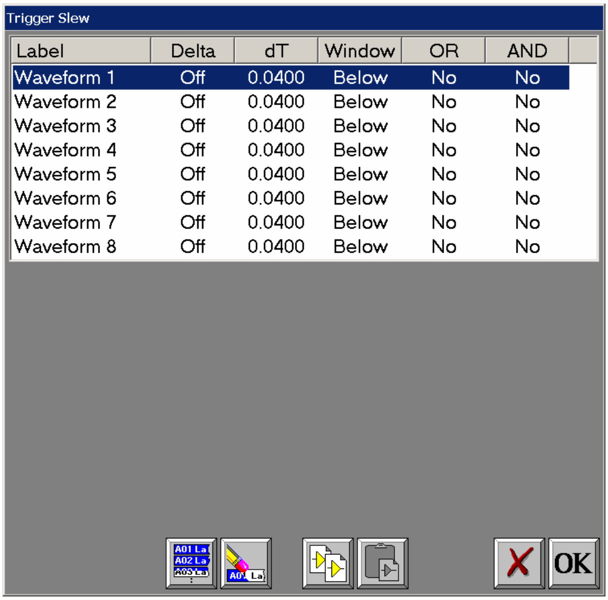
1. Choose Setup >> Trigger Settings to open the Trigger Setup Menu window.



2. Choose the Trigger Slew button.



The Trigger Slew window will open.



3. Define a slew delta by selecting the waveform(s) and choosing the Delta column heading. A sub menu will appear.

   Choose the New Value option. A number pad will appear.

   Enter the slew delta and choose OK.

4. Define a delta time by selecting the waveform(s) and choosing the dT column heading. A sub menu will appear.

   Choose the New Value option. A number pad will appear.

   Enter the delta time in seconds and choose OK.

5. Specify the type of window by selecting the waveform(s) and choosing the Window column heading. A sub menu will appear.

   - To cause a trigger when the signal rises above the chosen slew rate, select the Above option.

   - To cause a trigger when the signal falls below the chosen slew rate, select the Below option.

6. Select whether to enable trigger conditions for each waveform by selecting the waveform(s) and choosing the AND column heading. A sub menu will appear.

   - Choose Yes to enable trigger conditions for the selected waveforms.
   - Choose No to disable trigger conditions for the selected waveforms.

   Note: The OR column is not used when setting up Slew AND triggers. This option is used during the Slew OR trigger setup process.

   ---

7. Choose OK in the Trigger Slew window. The Trigger Setup Menu window will re-appear.

   In the Trigger Setup Menu window, enable the Slew AND trigger by choosing the Slew AND label. The label text color will change to yellow when the trigger is enabled.

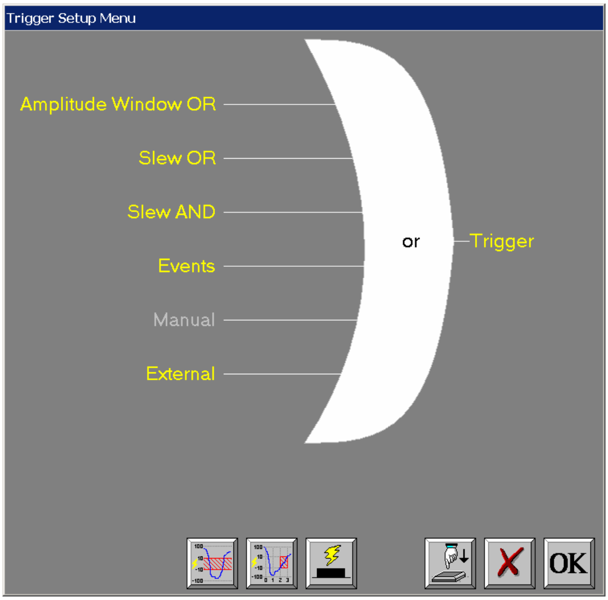
   Choose OK in the Trigger Setup Menu window to complete the Slew AND trigger setup process.

Setting up an Event trigger

Event triggers produce a trigger when event inputs meet their change in logic state. This is an event pattern trigger, as all events must meet the selected logic state to cause a trigger.

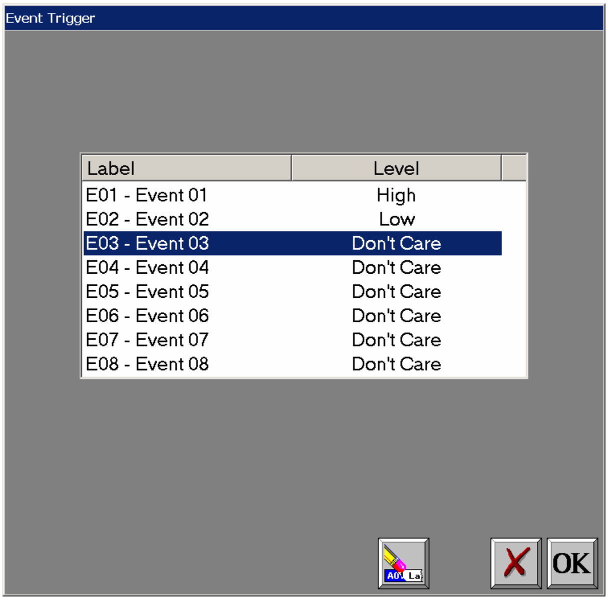
To set up an Event trigger:

1. Choose Setup >> Trigger Settings to open the Trigger Setup Menu window.



2. Choose the Event Trigger button.

The Event Trigger window will open.



3. If necessary, assign event labels by selecting an event and choosing the Label column heading. A keypad will appear. Enter a new name for the event and choose OK.

4. Specify an event level combination that will cause a trigger by selecting the event(s) and choosing the Level column heading. A sub menu will appear.

   Don't Care - If the Don't Care option is chosen, changes in the selected event will not be considered in the trigger pattern.

   High - If the High option is chosen, a trigger will occur when the selected event changes to its high state (assuming all other event state trigger conditions are met).

   Low - If the Low option is chosen, a trigger will occur when the selected event changes to its low state (assuming all other event state trigger conditions are met).

Note: A trigger will occur only when *all* of the selected event state conditions are met.

---

5. Choose OK in the Event Trigger window. The Trigger Setup Menu window will re-appear.

   In the Trigger Setup Menu window, enable the Event trigger by choosing the Events label. The label text color will change to yellow when the trigger is enabled.

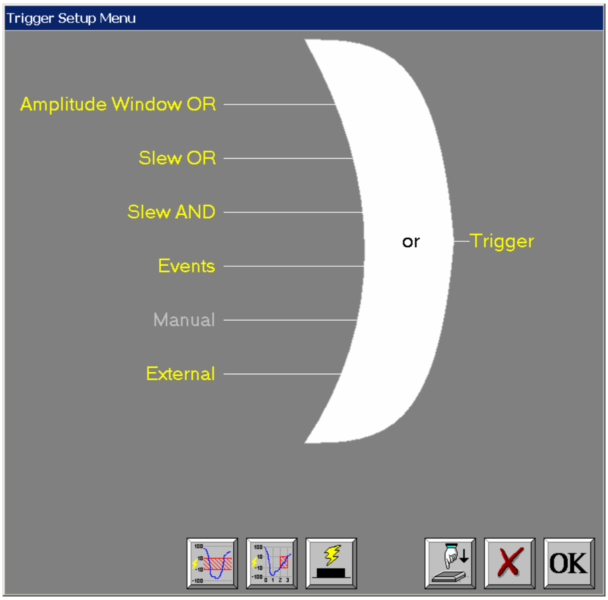
   Choose OK in the Trigger Setup Menu window to complete the Event trigger setup process.

Setting up a Manual trigger

Manual is a commonly used trigger method that allows the operator to produce a trigger manually via the menu bar or control panel.

To set up a Manual trigger:

1. Choose Setup >> Trigger Settings to open the Trigger Setup Menu window.



2. Enable the Manual trigger by choosing the Manual label. The label text color will change to yellow when the trigger is enabled.

   Choose OK in the Trigger Setup Menu window to complete the Manual trigger setup process.

   Once a data capture is armed, Manual triggers are initiated by choosing Capture >> Manual Trigger from the menu bar.

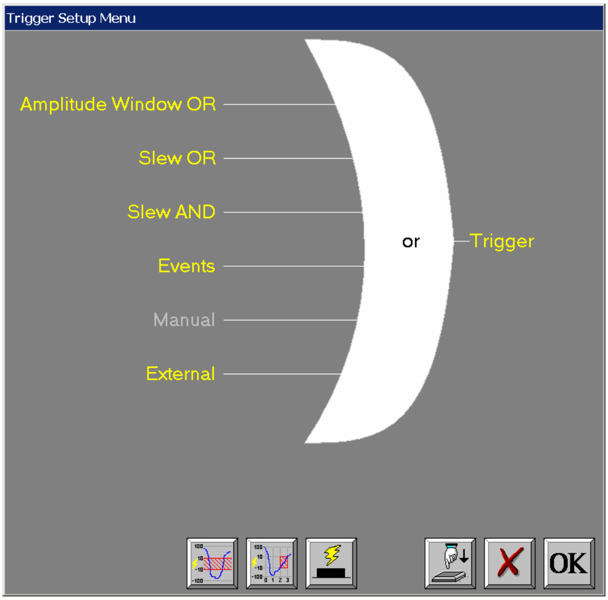
Note: To save time, add a Manual Trigger button to the control panel. Refer to *Chapter 5: Menus and buttons* for more information on customizing the control panel.

Setting up an External trigger

External triggers use an external signal via the utility port to produce a trigger. The external signal is a switch closure to ground or TTL low to produce a trigger. Process control signals can be used to produce this trigger.

To set up an External trigger:

1. Choose Setup >> Trigger Settings to open the Trigger Setup Menu window.



2. Enable the External trigger by choosing the External label. The label text color will change to yellow when the trigger is enabled.

   Choose OK in the Trigger Setup Menu window to complete the External trigger setup process.

   Once a data capture is armed, External triggers are initiated via a utility port input signal.

Note: Refer to *Chapter 2: Hardware overview* for more information on the utility port.

Data capture abort setup

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.

This section provides information about the types of aborts available and the setup process for each abort.

Data capture abort types

The following list describes each abort type available in the Dash 8Xe.

Note: If multiple types are set up, an abort will occur when abort conditions are met for *any* abort type.

Slew OR

Slew OR produces an abort when the specified change in signal slew rate, or frequency occurs.

This abort utilizes OR logic. An abort will occur when *any* of the specified signals meet the slew rate conditions.

Slew AND

Slew AND produces an abort when more than one channel must meet its respective slew rate conditions.

This abort utilizes AND logic. An abort will occur when *all* of the specified signals meet the slew rate conditions.

Events

Event aborts produce an abort when event inputs meet their change in logic state. This is an event pattern abort, as all events must meet the selected logic state to cause an abort.

Manual

Manual is a commonly used method that allows the operator to produce an abort manually via the menu bar or control panel.

External

External uses an external signal via the utility port to produce an abort. The external signal is a switch closure to ground or TTL low to produce an abort. Process control signals can be used to produce this abort.

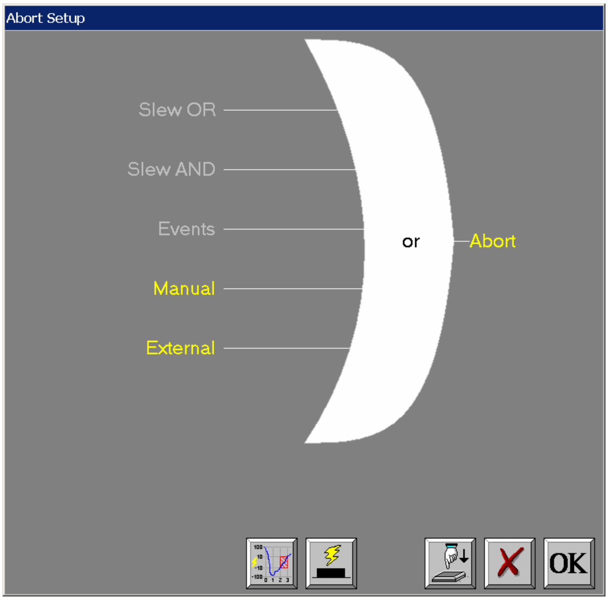
Enabling and disabling data capture aborts

The Abort Setup Menu window is used to enable and disable aborts. It is also the starting point of the abort setup process. Use the following instructions to enable and disable aborts.

To enable and disable aborts:

1. Choose Setup >> Abort Settings to open the Abort Setup window.

Note: To save time, add a control panel icon button for the Abort Setup window. For information on adding buttons to the control panel, refer to *Chapter 5: Menus and buttons*.



2. Enable and disable aborts by selecting the abort labels. The color of the label will indicate whether the abort is enabled.

   - If the abort label text is yellow, the abort is enabled.
   - If the abort label text is gray, the abort is disabled.

3. Choose OK to confirm the abort enable/disable settings.

Setting up a Slew OR abort

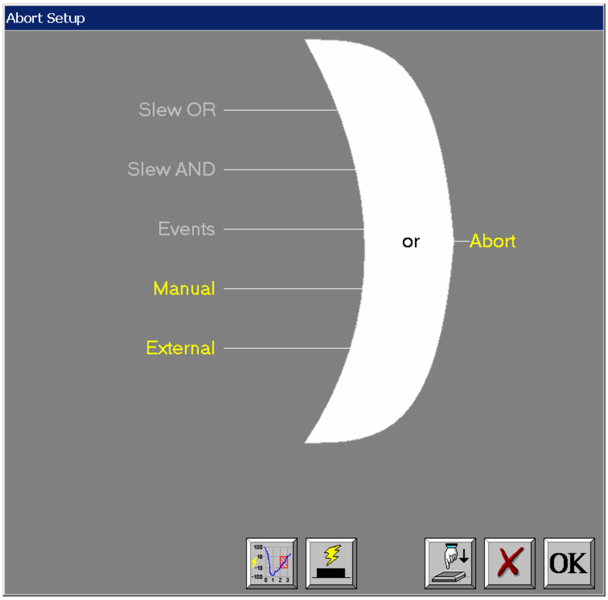
Slew OR produces an abort when the specified change in signal slew rate, or frequency occurs.

Note: The slew concepts for this trigger are similar to the concepts described in the *Setting up a Slew OR trigger* section. However, instead of producing a trigger, an abort will occur under the specified conditions.

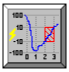
This abort utilizes OR logic. An abort will occur when *any* of the specified signals meet the slew rate conditions.

To set up a Slew OR abort:

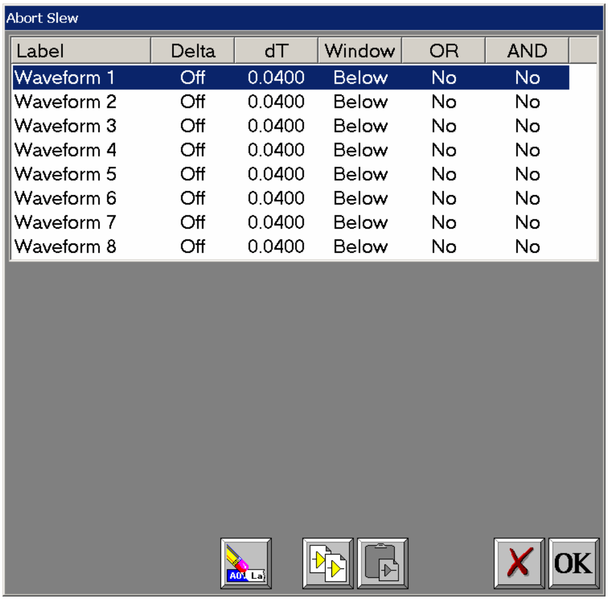
1. Choose Setup >> Abort Settings to open the Abort Setup window.



2. Choose the Abort Slew button.



The Abort Slew window will open.



3. Define a slew delta by selecting the waveform(s) and choosing the Delta column heading. A sub menu will appear.

   Choose the New Value option. A number pad will appear.

   Enter the slew delta and choose OK.

4. Define a delta time by selecting the waveform(s) and choosing the dT column heading. A sub menu will appear.

   Choose the New Value option. A number pad will appear.

   Enter the delta time in seconds and choose OK.

5. Specify the type of window by selecting the waveform(s) and choosing the Window column heading. A sub menu will appear.

   - To cause an abort when the signal rises above the chosen slew rate, select the Above option.

   - To cause an abort when the signal falls below the chosen slew rate, select the Below option.

6. Select whether to enable abort conditions for each waveform by selecting the waveform(s) and choosing the OR column heading. A sub menu will appear.

   - Choose Yes to enable abort conditions for the selected waveforms.
   - Choose No to disable abort conditions for the selected waveforms.

Note: The AND column is not used when setting up Slew OR aborts. This option is used during the Slew AND abort setup process.

---

7. Choose OK in the Abort Slew window. The Abort Setup window will re-appear.

   In the Abort Setup window, enable the Slew OR abort by choosing the Slew OR label. The label text color will change to yellow when the abort is enabled.

   Choose OK in the Abort Setup window to complete the Slew OR abort setup process.

Setting up a Slew AND abort

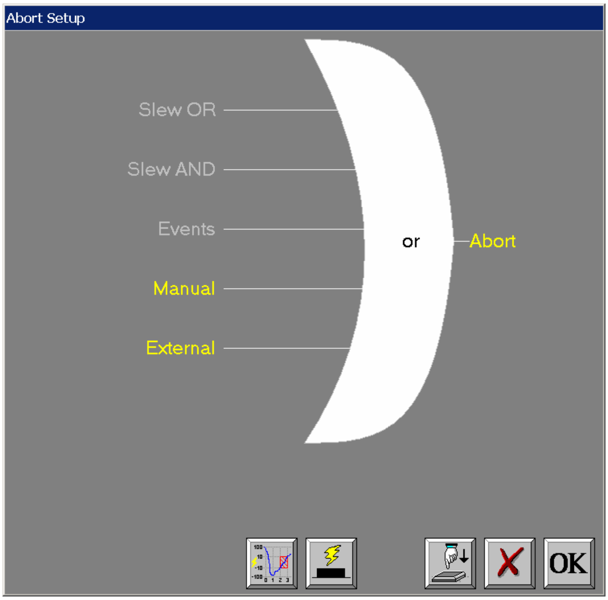
Slew AND produces an abort when more than one channel must meet its respective slew rate conditions.

Note: The slew concepts for this trigger are similar to the concepts described in the *Setting up a Slew OR trigger* section. However, instead of producing a trigger, an abort will occur under the specified conditions.

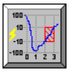
This abort utilizes AND logic. An abort will occur when *all* of the specified signals meet the slew rate conditions.

To set up a Slew AND abort:

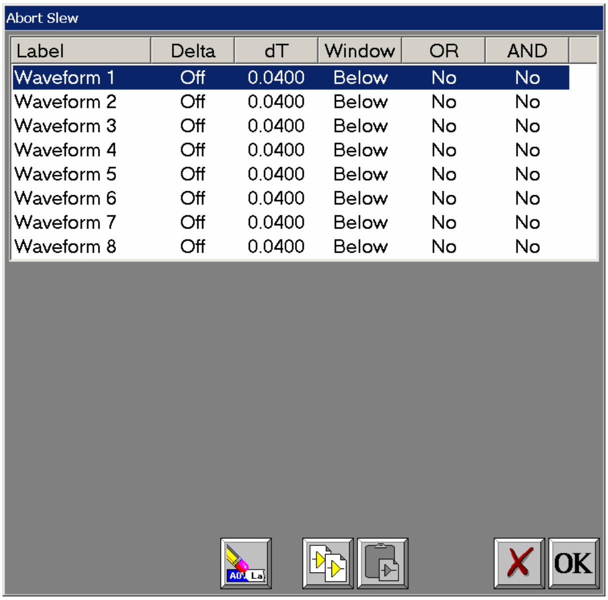
1. Choose Setup >> Abort Settings to open the Abort Setup window.



2. Choose the Abort Slew button.



The Abort Slew window will open.



3. Define a slew delta by selecting the waveform(s) and choosing the Delta column heading. A sub menu will appear.

   Choose the New Value option. A number pad will appear.

   Enter the slew delta and choose OK.

4. Define a delta time by selecting the waveform(s) and choosing the dT column heading. A sub menu will appear.

   Choose the New Value option. A number pad will appear.

   Enter the delta time in seconds and choose OK.

5. Specify the type of window by selecting the waveform(s) and choosing the Window column heading. A sub menu will appear.

   - To cause an abort when the signal rises above the chosen slew rate, select the Above option.

   - To cause an abort when the signal falls below the chosen slew rate, select the Below option.

6. Select whether to enable abort conditions for each waveform by selecting the waveform(s) and choosing the AND column heading. A sub menu will appear.

   - Choose Yes to enable abort conditions for the selected waveforms.

   - Choose No to disable abort conditions for the selected waveforms.

   Note: The OR column is not used when setting up Slew AND aborts. This option is used during the Slew OR abort setup process.

7. Choose OK in the Abort Slew window. The Abort Setup window will re-appear.

   In the Abort Setup window, enable the Slew AND abort by choosing the Slew AND label. The label text color will change to yellow when the abort is enabled.

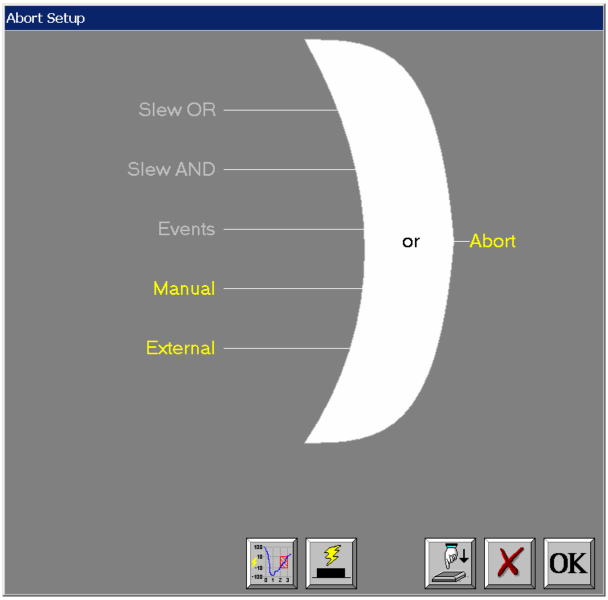
   Choose OK in the Abort Setup window to complete the Slew AND abort setup process.

Setting up an Event abort

Event aborts produce an abort when event inputs meet their change in logic state. This is an event pattern abort, as all events must meet the selected logic state to cause an abort.

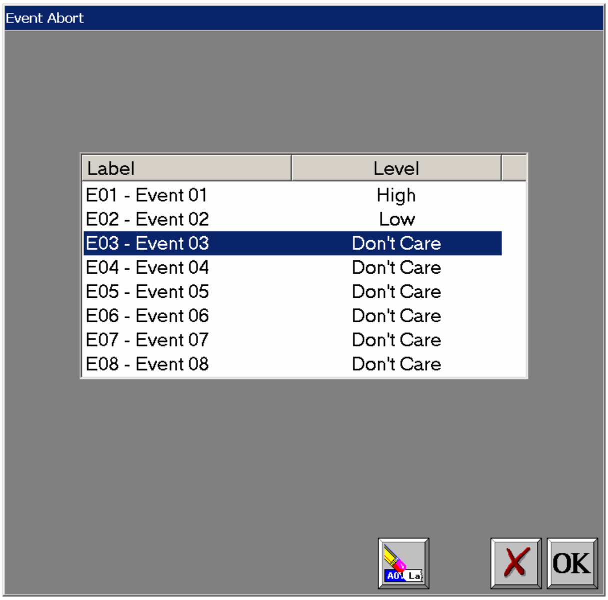
To set up an Event abort:

1. Choose Setup >> Abort Settings to open the Abort Setup window.



2. Choose the Event Abort button.

The Event Abort window will open.



3. If necessary, assign event labels by selecting an event and choosing the Label column heading. A keypad will appear. Enter a new name for the event and choose OK.

4. Specify an event level combination that will cause an abort by selecting the event(s) and choosing the Level column heading. A sub menu will appear.

   Don't Care - If the Don't Care option is chosen, changes in the selected event will not be considered in the abort pattern.

   High - If the High option is chosen, an abort will occur when the selected event changes to its high state (assuming all other event state abort conditions are met).

   Low - If the Low option is chosen, an abort will occur when the selected event changes to its low state (assuming all other event state abort conditions are met).

Note: An abort will occur only when *all* of the selected event state conditions are met.

---

5. Choose OK in the Event Abort window. The Abort Setup window will re-appear.

   In the Abort Setup window, enable the Event abort by choosing the Events label. The label text color will change to yellow when the abort is enabled.

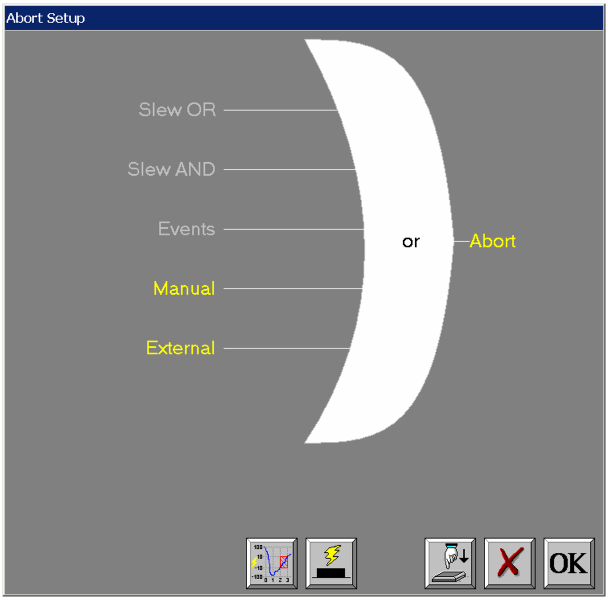
   Choose OK in the Abort Setup window to complete the Event abort setup process.

Setting up a Manual abort

Manual is a commonly used abort method that allows the operator to produce an abort manually via the menu bar or control panel.

To set up a Manual abort:

1. Choose Setup >> Abort Settings to open the Abort Setup window.



2. Enable the Manual abort by choosing the Manual label. The label text color will change to yellow when the abort is enabled.

   Choose OK in the Abort Setup window to complete the Manual abort setup process.

   Once a data capture is armed, Manual aborts are initiated by choosing Capture >> Manual Abort from the menu bar.

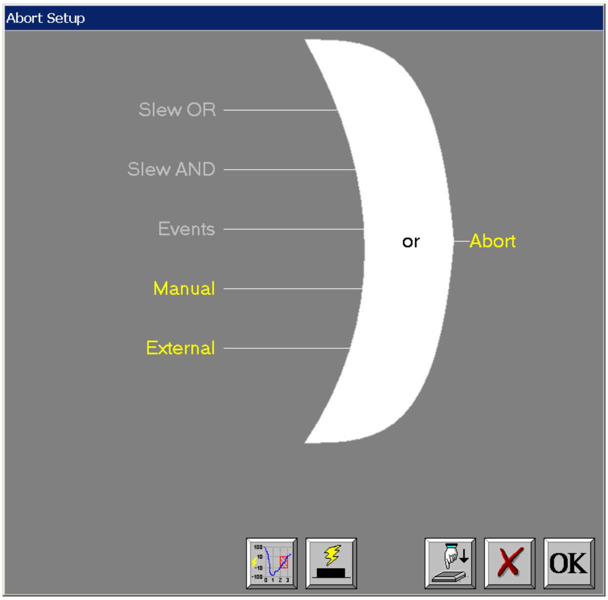
Note: To save time, add a Manual Abort button to the control panel. Refer to *Chapter 5: Menus and buttons* for more information on customizing the control panel.

Setting up an External abort

External aborts use an external signal via the utility port to produce an abort. The external signal is a switch closure to ground or TTL low to produce an abort. Process control signals can be used to produce this abort.

To set up an External abort:

1. Choose Setup >> Abort Settings to open the Abort Setup window.



2. Enable the External abort by choosing the External label. The label text color will change to yellow when the abort is enabled.

   Choose OK in the Abort Setup window to complete the External abort setup process.

   Once a data capture is armed, External aborts are initiated via a utility port input signal.

Note: Refer to *Chapter 2: Hardware overview* for more information on the utility port.

Data capture

This section provides information on arming data captures. It also provides instructions for capturing text notes.

Arming 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.

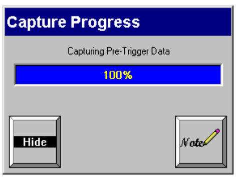
- If a pre-trigger recording percentage is used, the pre-trigger recording phase begins when the system is armed. Pre-trigger samples will be acquired and stored up to the specified amount. When the specified amount of pre-trigger data has been stored, the oldest sample will be replaced by most recent,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

Note: To save time, add data capture-related buttons to the control panel. Refer to *Chapter 5: Menus and buttons* for more information on customizing the control panel.

To arm a data capture:

1. Choose Capture >> Arm from the menu bar. The Capture Progress window will open.



If the data capture includes pre-trigger data, the Dash 8Xe 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 >> Manual Trigger from the menu bar.

   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.

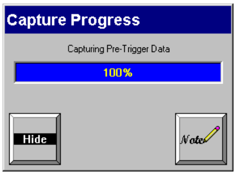
   - If manual aborts are used, manually abort the data capture when appropriate by choosing Capture >> Manual Abort from the menu bar.

Adding notes to data captures

Text notes can be recorded as part of a data capture. Use the following instructions to add text notes to a data capture in progress.

To add notes to data captures:

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



2. Choose the Note button 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.

Note: If any programmable note buttons are configured, they will appear along the bottom of the keypad.

When you choose a programmable note button, the associated note text will be entered in the text field above the keypad.

To enter another note, repeat this process.

Setting up programmable note buttons

Programmable note buttons allow you to set up one-touch annotations with pre-configured text strings. You can later enter the entire note string during a data capture by choosing one button from the Edit Notetag window. This feature saves time and keystrokes in the Edit Notetag window.

You can use up to five programmable note buttons. Note buttons are controlled by the "notelist.ini" file, which is located in the C:\D82 folder. You will need to modify this file using a text editor such as Notepad.

Note: A keyboard is required to edit the text in the "notelist.ini" file.

An excerpt from this file is included below.

```
[Note1]
NoteValid=1
ButtonText=Note Button 1
NoteText=This is the note for button 1

[Note2]
NoteValid=1
ButtonText=Note Button 2
NoteText=This is the note for button 2

. . .
```

Five note button entries are present in the file. You will need to set up the parameters for each note button.

NoteValid - This option controls whether the note is active. Enter "1" to enable the note button, or "0" to disable it.

ButtonText - This option controls the text label for the note button. Enter a meaningful label for the button. Approximately 12 characters will fit in the button label.

NoteText - This option controls the text that will be entered when you choose the note button. The note text can be a maximum of 300 characters in length.

Chapter 11: Scope captures

This chapter provides information about the scope capture capability of the Dash 8Xe.

Scope modes

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.

Scope captures can be embedded into Realtime data captures to allow capturing high-speed and trend data simultaneously.

The following two scope mode configurations are available:

Scope mode - Standard
This is a standard feature on all Dash 8Xe units.

Scope mode - Card
This is a purchased option for the Dash 8Xe. If the optional DSO-8 scope card is installed, high-speed scope captures up to ten million samples per second can be acquired using Card mode.

Scope mode overview

This section provides an overview of scope mode.

Accessing Scope mode

Use the following instructions to start Scope mode from any other mode of operation.

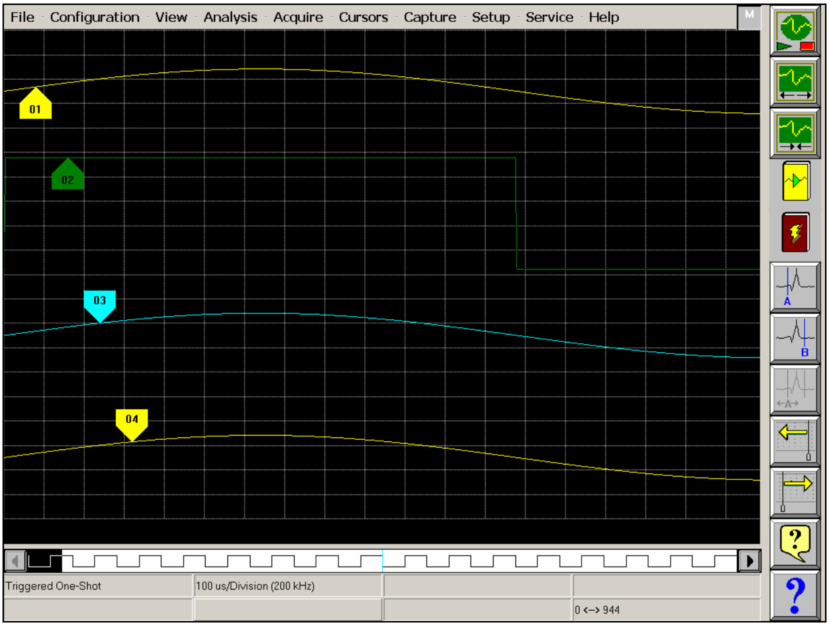
To access Scope mode:

1. Choose Configuration >> Scope from the menu bar.

    - If the DSO-8 option is not installed, choose Scope to enter scope mode.

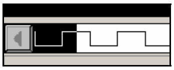
    - If the DSO-8 option is installed, choose Standard to enter standard scope mode or Card to enter high-speed scope mode.

    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 Scope mode display. It can be used to navigate to other areas of the scope 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.

To navigate though a file using the track view:

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

2. 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.

3. To move directly to a new location in a file, press a non-shaded area of the track view. The display will move to the selected location.

To change the signal displayed in the track view:

1. To change the track view signal using the menu bar, choose View >> Track View Channel. A list of available signals and events will appear.

   Select a signal or event. The track view will be updated to display the chosen signal or event.

2. To change the track view signal using channel IDs, View >> Show Channel IDs. Channel ID labels will be displayed.

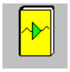
   Choose an ID label. The track view will be updated to display the chosen signal or event.

Default control panel

The following buttons are provided in the default Scope control panel.

Note: The Scope mode control panel can be customized to meet the needs of specific application or users. Refer to *Chapter 5: Menus and buttons* for information on customizing the control panel.

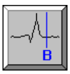
Scope Arm/Abort arms and aborts scope captures.



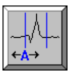
Amp Setup opens the Analog Channel Setup window, which is used to set up channel amplifiers and filters.

Trigger Setup opens the Trigger Setup Menu window, which is used to set up data capture triggers.

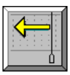
Show/Hide Cursor A displays and hides cursor A.



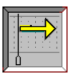
Show/Hide Cursor B displays and hides cursor B.



Set Active Cursor changes the active cursor between A, B, or A & B.



Move Cursor Left moves the active cursor(s) to the left each time the button is pressed.



Move Cursor Right moves the active cursor(s) to the right each time the button is pressed.

Context Help provides on-screen help based on the mode or window in use.

Icon Help provides on-screen help for the purpose of identifying icon buttons.

View options

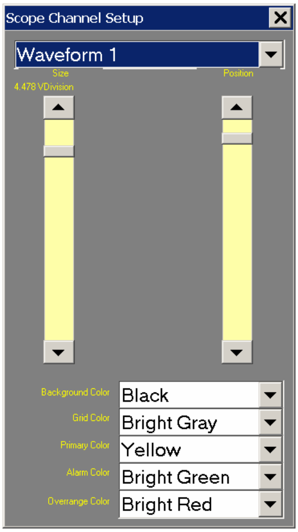
This section describes how to set up signal display options, display trigger lines, and display signal IDs.

Setting up signal display options

You will use the Scope Channel Setup window to modify the appearance of signals in Scope mode.

To set up scope signal display options:

1. Choose View >> Channel Setup from the menu bar. The Scope Channel Setup window will open.



2. Select a signal using the drop-down list.

3. Adjust the signal size and vertical position using the slider bars.

4. Select colors for the following global settings:

   Background Color - The color of the waveform display area background.

   Grid Color - The color of the grid, if used.

5. Select colors for the following signal-specific settings:

   Primary Color - The color of the signal waveforms.

   Alarm Color - The color of the sections of waveforms that are in the alarm range.

   Overrange Color – The color of the sections of waveforms that are over the grid range.

6. Set up other channels, if necessary.

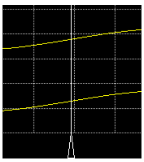
7. Complete the channel display setup process by choosing the X in the upper-right corner of the window.

Displaying the trigger line

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

To display the trigger line:

1. Choose View >> Show Trigger Line from the menu bar. The trigger line will be displayed.



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

Displaying signal IDs

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

Note: Signal IDs can be used to change the signal displayed in the track bar. To change the track bar signal, choose a signal ID label.

To display signal IDs:

1. Choose View >> Show Channel IDs from the menu bar.

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

2. To hide the signal IDs, repeat this process.

Measurement and analysis tools

This section provides information about the measurement tools available in Scope mode.

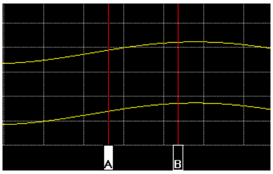
Cursors

Cursors are a valuable tool that can be used for measuring signals. They are used in conjunction with the Channel Information window to view a variety of measurements.

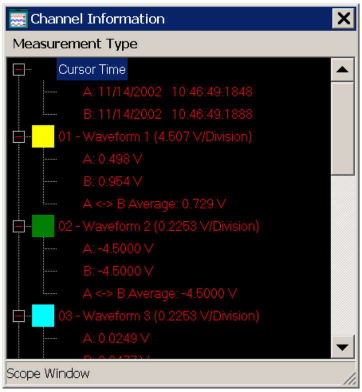
Note: To save time and gain more control over cursors, add cursor buttons to the control panel. Refer to *Chapter 5: Menus and buttons* for more information on customizing the control panel.

To measure signal information 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 View >> Show 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. The following measurement types are available: Average, Max-Min, Peak-Peak, Slope, and RMS.

Average - Average displays the midpoint value of the data represented by the cursor. When using two cursors, this measurement displays the average of the midpoint values of the data between cursors A and B.

Max-Min - Max-Min displays the maximum and minimum signal values of the data represented by the cursor. When using two cursors, this measurement displays the maximum and minimum signal values represented by the data between cursors A and B.

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

Slope - Slope displays the slope (delta V per unit time) of the signal between cursors A and B. This measurement is available only when both cursors are displayed.

RMS - RMS displays the root-means-square of the midpoints of the max-min pairs (line segments). This measurement is available only when both cursors are displayed.

3. To change the measurement type, choose the Measurement Type option on the menu bar of the Channel Information window.

4. If necessary, move one or both of the cursors. Cursors must be activated before they can be moved.

   - **Move cursor A or B individually** by choosing Cursors >> Active Cursor >> Cursor A or Cursor B to activate it. Then press the active cursor and drag it to a new location.

   - **Move cursor A and B together** by choosing Cursors >> Active Cursor >> Cursor A + B to activate both cursors. Then press one of the cursors and drag it to a new location. The other cursor will move as well.

Note: As a shortcut, activate a cursor by pressing it once on the touch-screen. To move it, press it again, but do not release it. While pressing, drag it to a new location, and then release it.

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

5. If necessary, modify the color and type of the cursors.

   - To modify the cursor color, choose Cursors >> Color. Then select a color from the options that appear.

   - To modify the cursor type, choose Cursors >> Type. Then select a type from the options that appear.

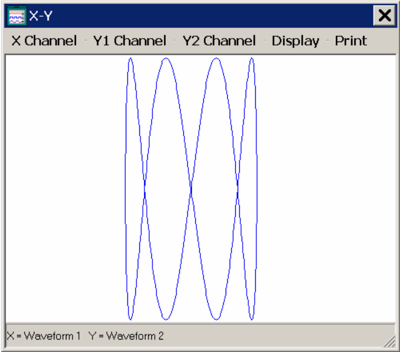
     Select the Color option to use color cursors. Select the Invert option to use pseudo-transparent cursors.

XY Plot

Use the following instructions to display the XY Plot.

To display the XY plot:

1. Choose Analysis >> XY Plot Window to open the XY plot window.



2. Select the X, Y1, and Y2 channels using the X Channel, Y1 Channel, and Y2 Channel menu options.

3. If necessary, select the Display menu to use the following display options:

   Refresh - Use the Refresh option to clear the window and restart the XY plot drawing.

   Connect Points - Use the Connect Points option to draw lines to connect the sample points on the XY plot. When this option is active, segments will be drawn.

   Background Color - Use the Background Color option to select a background color for the XY plot window.

   Foreground Color - Use the Foreground Color option to select a color for the points and segments drawn on the XY plot.

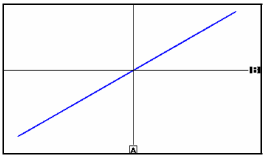
   Grid - Use the Grid option to display a grid. Grid color and division settings can also be specified.

   Open Paint After Save - Use the Open Paint After Save option to open a plot in Microsoft Paint after saving the plot. When this option is active, plots will automatically be opened in Paint.

Save Plot - Use the Save Plot option to save the XY plot as a bitmap (BMP) image file.

4. If necessary, choose the Cursors menu option to use cursors. The cursors in this window function like cursors in other Dash 8Xe functions, with the exception of the following features.

   - Cursor A is a vertical cursor and cursor B is a horizontal cursor.

   - To restrict cursor movement to the data points, choose Cursors >> Lock On Data. The starting and ending data points will be highlighted automatically. The Lock On Data option is unavailable when the Y2 plot is used.

     To move the cursors to the start or end point on the plot, choose Cursors >> Goto Start or Goto End.

     To move the cursors among data points, use the Move Cursor Left and Move Cursor Right control panel icon buttons.

5. If the printer is installed, use the Print >> Print Screen option to print the XY Plot window contents.

XY plot templates

XY plot templates allow you to display a customized background template in the XY plot window. This background is a visual aid you can use for comparing plot results against a standard template you define.

For detailed information about creating and using XY plot templates, refer to *XY plot templates* in *Chapter 8: Realtime mode overview*.

Fourier Transform window

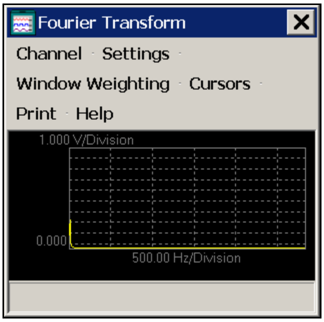
The FFT is created from the data displayed in the main scope screen. This data can consist of up to 1024 segments. Each segment consists of two points, a minimum and maximum, which represent the minimum and maximum signal values for each period.

FFT calculations require a single point for each period. The point used for the FFT calculation is the midpoint of the minimum and maximum. Therefore, a signal overlay compressed may not yield accurate FFT results.

Use the following instructions to measure signals with the Fourier Transform window.

To use the Fourier Transform window:

1. Choose Analysis >> Fourier Transform from the menu bar. The Fourier Transform window will open.



2. Choose the Channel option. A list of signals will appear. Select the signal to analyze.

3. If necessary, select the Settings menu to modify the following options:

   Background Color - The Background Color option is used to select a color for the background of the window.

   Channel Color - The Channel Color option is used to select a color for the channel in the window.

   Grid - The Grid option is used to select whether to display a grid. Grid color, amplitude axis, and frequency axis are also specified using this option.

   Plot - The plot option is used to choose between linear magnitude, logrithmic magnitude, or magnitude^2.

Size - The size option is used to adjust the size of the window. Available window sizes include small, medium, and large.

Frequency Expansion - The Frequency Expansion option is used to expand the frequency axis to provide a more detailed view.

4. If necessary, choose a window weight option. The following options are available: Hanning, Hamming, Blackman, Barlett, Triangle, Kaiser, Bman-Harris, and Rectangle.

5. If necessary, choose the Cursors menu option to use cursors. The cursors in this window function similar to cursors in other Dash 8Xe functions.

6. Close the Fourier Transform window by choosing the X in the upper-right corner of the window.

Scope captures

This section provides information on scope captures as well as instructions for setting up and initiating scope captures.

Scope capture overview

A scope capture is a high-speed snapshot of signal information. Scope captures are similar to Realtime 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 Realtime data captures.

Scope captures can be saved independently or as an embedded part of a Realtime data capture.

Scope capture setup

Use the following instructions to set up a scope capture. The scope capture setup process involves selecting a capture type, time base, size, pre-trigger storage allocation, and choosing whether to embed the scope capture into a Realtime data capture.

To set up scope captures:

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

Note: The trigger/abort setup process for scope captures is identical to the trigger/abort setup process for data captures. For more information on setting up triggers and aborts, refer to *Chapter 10: Data capture*.

2. Select the type of scope capture to set up by choosing Acquire from the menu bar. A list of options will appear.

   One-Shot - Perform a one-time scope capture without using a trigger.

   Continuous - Perform multiple scope captures without using a trigger.

   Triggered One-Shot - Perform a one-time scope capture that is initiated by a trigger.

   Triggered Continuous - Perform multiple scope captures that are initiated by a trigger.

3. Select a time base for the scope capture by choosing Acquire from the menu bar. A list of options will appear in units of ms/Division or us/Division. Select a time base from the list.

4. Select a size for the scope capture by choosing Acquire from the menu bar. Then choose Scope Size from the options that appear. A list of scope sizes will appear.

   Select a scope capture size based on number of samples. For example, select 2 kSamples for a 2,000 sample scope capture. If the scope capture exceeds the display size, a scrolling track view bar will appear on the bottom of the screen.

5. If triggers will be used in the scope capture, specify a pre-trigger percentage by choosing Acquire >> Pre-Trigger Percent. A number pad will appear. Enter the percentage of the scope capture to record before the trigger and choose OK.

6. To embed the scope capture into a data capture, choose Capture >> Auto Save Scope. When a data capture is armed and triggered, a scope capture will be saved as part of the previously set up Realtime data capture.

Note: The Auto Save Scope with Capture option saves high-speed scope captures while collecting realtime data to the hard drive. Care should be used with this option, however, as a large number of scope captures will be taken if the unit is constantly being triggered.

Initiating scope captures

Scope captures are initiated using a similar process as Realtime 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.

To initiate a scope capture:

1. Set up the scope capture as described in the previous section.

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

   - To use the menu bar, choose Acquire >> Arm/Abort.

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

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.

   Note: Triggers are ignored until all of the pre-trigger data is filled.

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.

Note: Triggers are ignored until all of the pre-trigger data is filled.

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

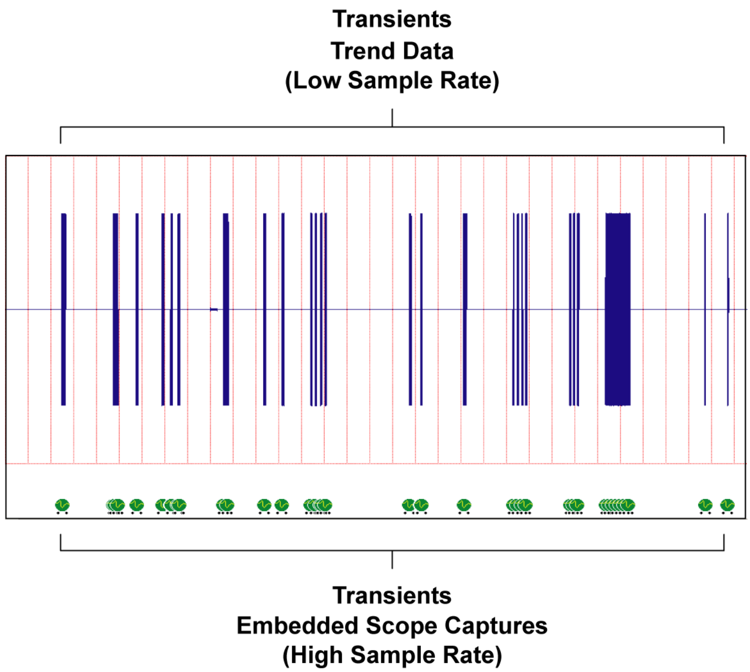
- To use the menu bar, choose Acquire >> Arm/Abort.
- To use the control panel, choose the Arm/Abort Scope Capture button.

The currently running scope capture will be aborted.

Embedded scope captures

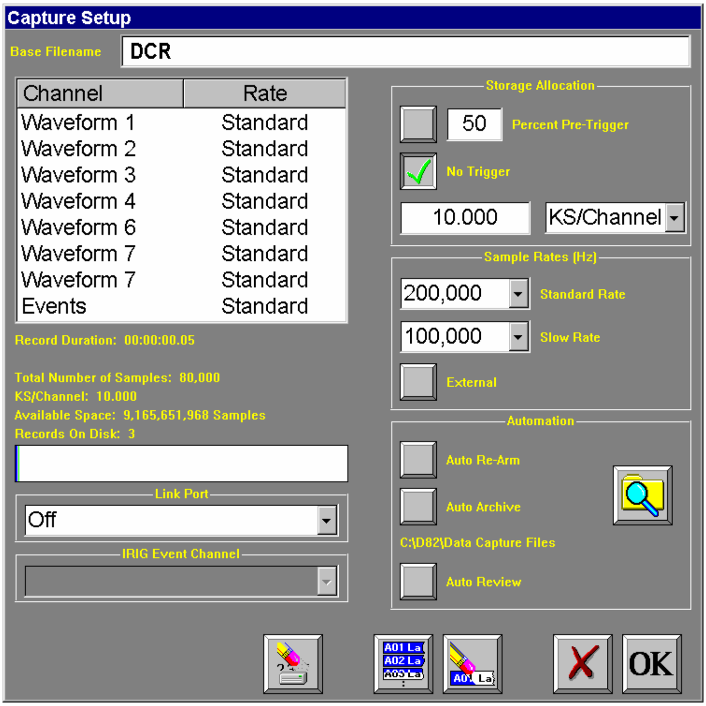
Embedded scope captures allow you to save trend data at low sample rates while capturing transients at high sample rates. When you open the trend data capture in review mode, the embedded scope captures will be viewable.

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.



To set up embedded scope captures:

1. In Scope mode, choose Capture >> Capture Settings to open the Capture Setup window. You will use this window to set up the trend data capture.



Note: More information about using the Capture Setup window is located in *Chapter 10: Data capture*.

2. Select the channels to capture and select the Standard rate.

3. 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.

4. 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).

   Select a unit of measure from the Storage Allocation 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.

An estimate of needed disk space for the DCR file is derived based on the data capture storage allocation and sample rate specified. This estimate is displayed in the hard drive status bar.

5. Specify a standard sample rate to define the speed at which the trend signals are sampled. Sample rates are defined in units of Hz (hertz), the number of samples per second per channel.

6. Ensure that Auto Re-Arm and Auto Archive are not checked.

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

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

9. Set up the triggers (and aborts if necessary) using the Trigger Setup menu (Setup >> Trigger Settings) and Abort Setup menu (Setup >> 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.

Note: The trigger/abort setup process for scope captures is identical to the trigger/abort setup process for data captures. For more information on setting up triggers and aborts, refer to *Chapter 10: Data capture*.

10. Select Acquire >> Triggered Continuous from the menu bar.

11. Select a time base for the scope capture by choosing Acquire from the menu bar. A list of options will appear in units of ms/Division or us/Division. Select a time base from the list.

12. Select a size for the scope capture by choosing Acquire from the menu bar. Then choose Scope Size from the options that appear. A list of scope sizes will appear.

    Select a scope capture size based on number of samples. For example, select 2 kSamples for a 2,000 sample scope capture. If the scope capture exceeds the display size, a scrolling track view bar will appear on the bottom of the screen.

13. Specify a pre-trigger percentage by choosing Acquire >> Pre-Trigger Percent. A number pad will appear. Enter the percentage of the scope capture to record before the trigger and choose OK.

14. Arm the scope capture using the menu bar or control panel.

    - To use the menu bar, choose Acquire >> Arm/Abort.

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

15. 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.

Archiving scope captures

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

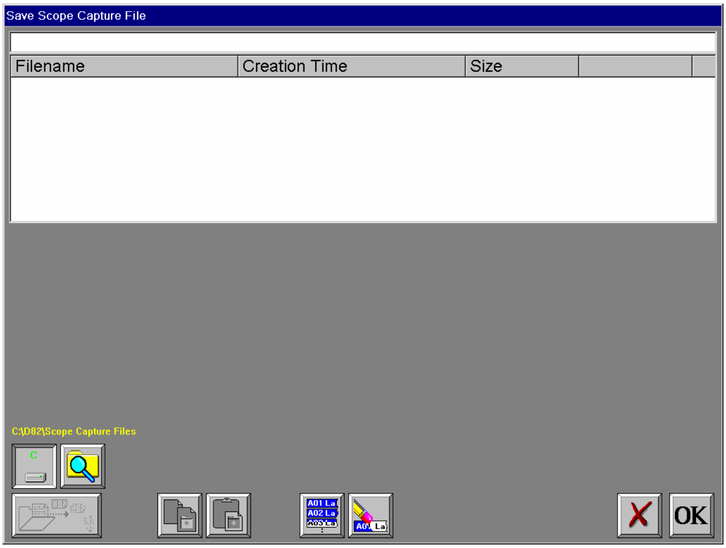
Archiving as data capture records

Use the following information to archive as data capture records. This binary format is preferable when using Dash 8Xe Offline or AstroVIEW on a PC.

To archive as data capture records:

1. With a scope capture on the display, choose File >> Archive as Data Capture Record. A sub menu will appear.

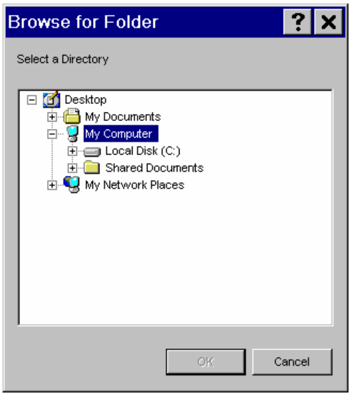
2. Choose whether to archive the entire file, current page, or the area between cursors. The Save Scope Capture File window will open.



3. By default, the file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

The Browse for Folder window will appear.



Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

4. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the file and choose the OK button. The specified file name will appear in the field.

5. Choose the OK button in the Save Data Capture File window. The file will be archived.

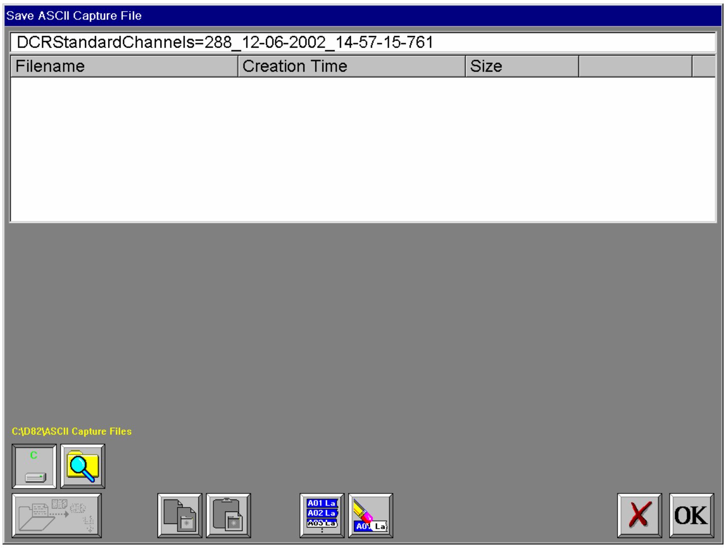
Archiving as ASCII

Use the following information to archive as ASCII files. ASCII format is helpful for archiving and analysis in other programs.

To archive as ASCII:

1. With a file open in Review mode, choose File >> Archive as ASCII from the menu bar. Then select whether to archive the entire file, current page, or the area between cursors.

   The Save ASCII Capture File window will open.

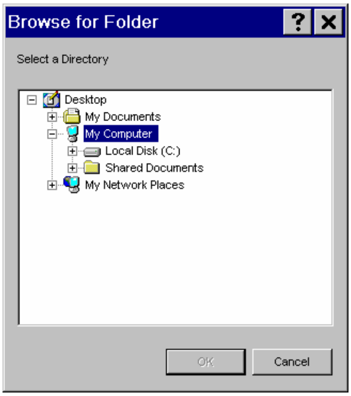


2. By default, the file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

   If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

   The Browse for Folder window will appear.

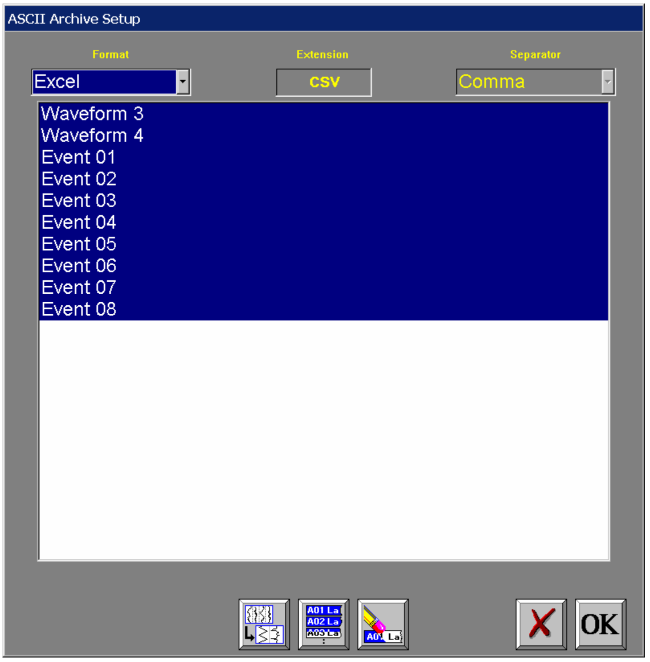
   

   Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

3. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the file and choose the OK button. The specified file name will appear in the field.

4. Choose the OK button in the Save Data Capture File window. The ASCII Archive Setup window will appear.



5. Specify the file setup options.

   - Choose Excel format to save the file with a CSV extension and comma delimiters.
   - Choose MatLab to save the file in a format that can be used in MatLab 6.0 and above.
   - Choose Other format to save the file with a custom extension and delimiters.

   Select the channels and events to include in the archive. By default, all channels and events are selected.

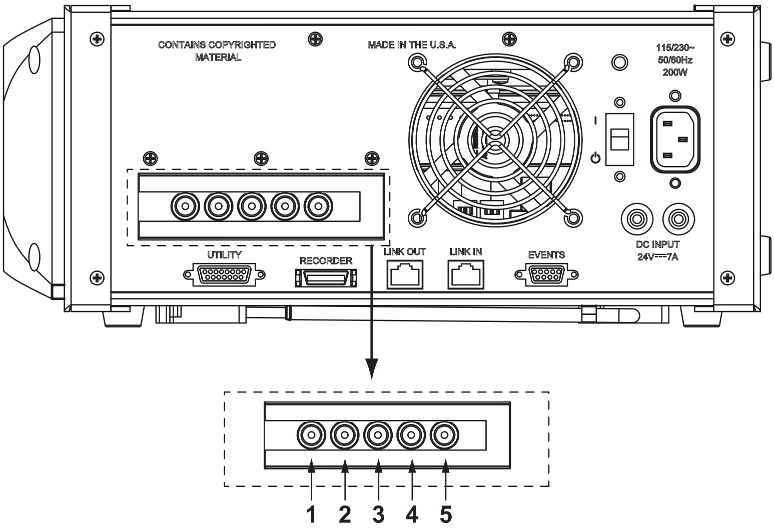
6. Choose OK. The archive file will be saved.

Scope card setup

This section describes the scope card-specific setup instructions necessary for scope card operation.

Signal input diagram

Use the following diagram to connect inputs to the DSO-8 scope card.



1. Signal input #1
2. Signal input #2
3. Signal input #3
4. Signal input #4
5. External trigger input

Signal inputs (1-4)

The signal inputs are used to connect signals for measurement. A BNC connection type is used to connect signals to the Dash 8Xe.

External trigger input

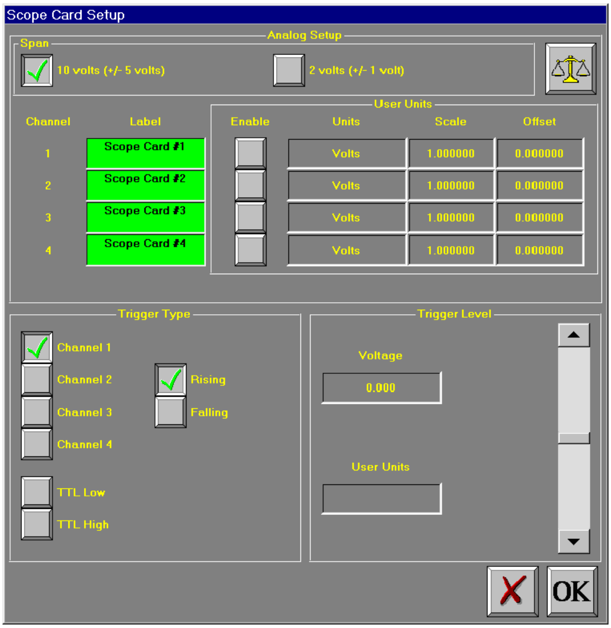
The external trigger input provides the capability to invoke a trigger using an outside signal. This input is TTL compatible.

Channel setup

Once the signal inputs are connected, use the following instructions to set up scope card channels.

To set up channels:

1. Choose Setup >> Scope Card Settings to open the Scope Card Setup window.



This window provides a variety of scope card setup options. A calibrate button is also available for the calibration of scope card channels.

2. Select a channel span by checking one of the following options.

   10 volts (+/- 5 volts)
   2 volts (+/- 1 volt)

3. If necessary, modify channel labels by choosing a channel label field. A keypad will appear.

   Enter a label for the signal and choose OK.

4. If necessary, set up user engineering units by checking the Enable option for the associated channel.

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

   Choose the Scale field. A number pad will appear. Enter the waveform change in user engineering units that is equal to one module unit. Then choose OK.

   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 module units
   - The number of module units equivalent to zero engineering units

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

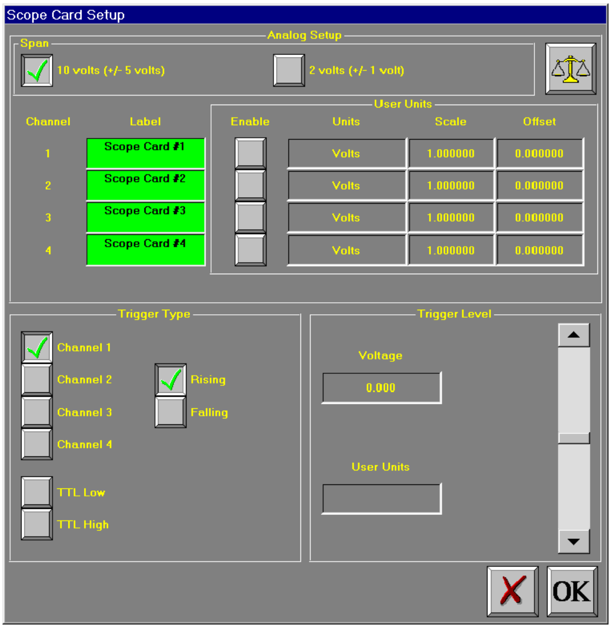
5. Choose the OK button to finish setting up scope card channels.

Trigger setup

Use the following instructions to set up scope card trigger conditions.

To set up triggers:

1. Choose Setup >> Scope Card Settings to open the Scope Card Setup window.



2. Select the channel that will be monitored for a trigger. Then select whether a trigger should occur when the signal rises above or falls below a specified level.

   Specify the trigger level by choosing the Voltage field. A number pad will appear. Enter the level that will be used as the trigger condition, then choose OK.

3. To use an external trigger, select one of the TTL Low or TTL High options. When the selected TTL signal type is detected by the external trigger input on the scope card, a trigger will occur.

4. Choose the OK button to finish setting up triggers.

Chapter 12: Data capture review & analysis

This chapter provides information about the Dash 8Xe review and analysis capabilities.

Review mode overview

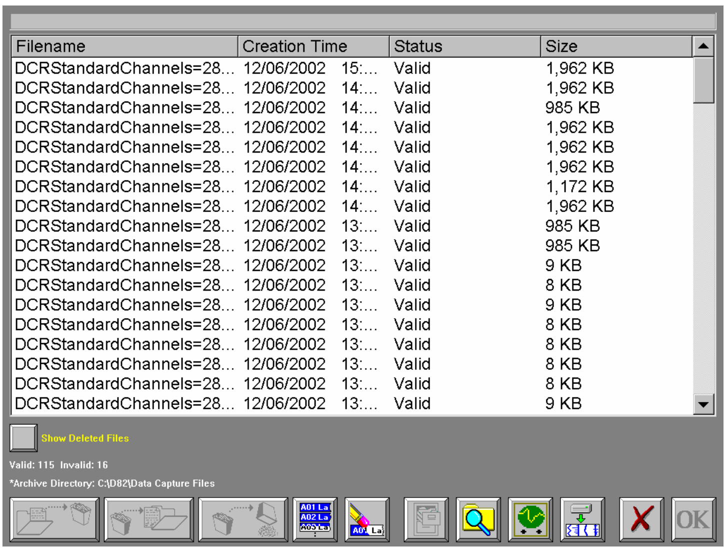
Review mode provides the capability to review and analyze saved data capture and scope capture files. It also provides file management features.

Accessing Review mode

Use the following instructions to start Review mode from any other mode of operation. Ensure that a file is available for review prior to accessing Review mode.

To access Review mode:

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

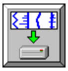


This window provides a list of all files stored on the data capture drive.

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 the OK button.

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

      Select an archived data capture file and choose the OK button.

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

      Select a scope capture file and choose the OK button.

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

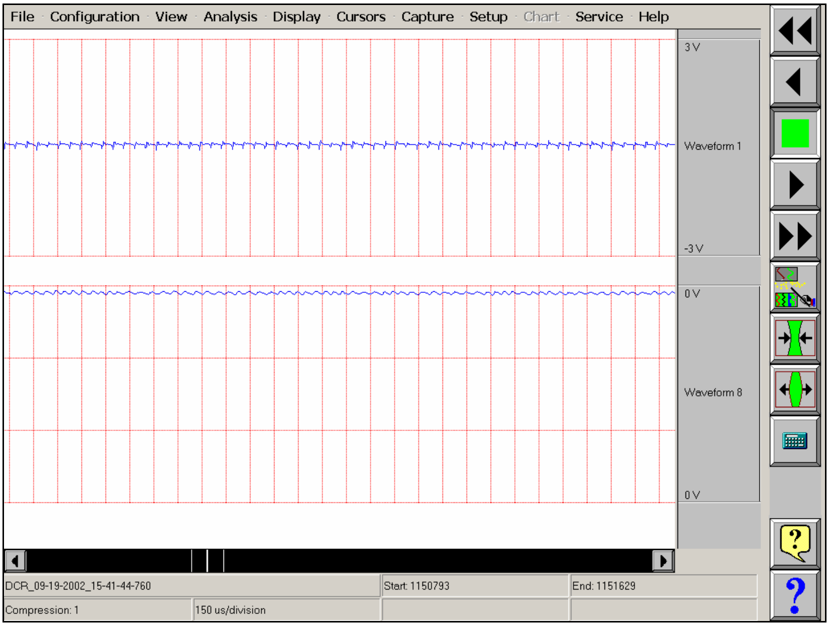
      

      Navigate to a specific file and choose the OK button.

      Note : In addition to accessing other folders on the system drive, the Browse for Folder window will also provide access to any connected USB storage devices.

   The selected file will be opened in Review mode.

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.

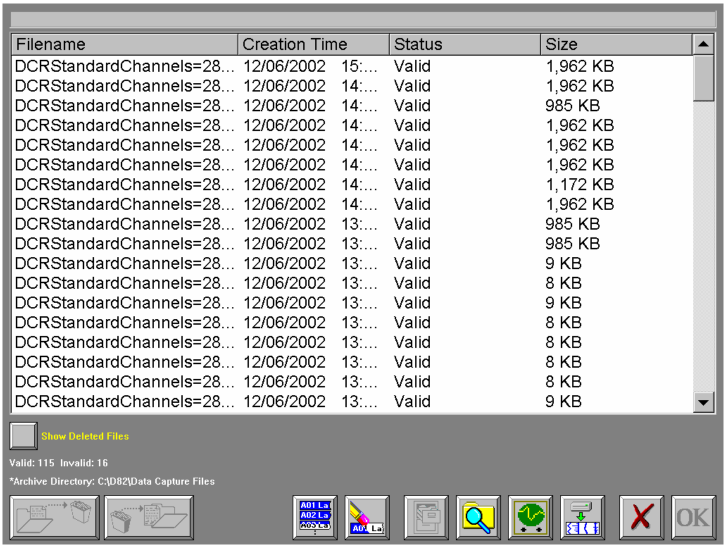


Loading new capture files

Use the following instructions to open a new capture file while in Review mode.

To load new capture files:

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



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

2. Select a file to open.

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

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

     Select an archived data capture file and choose the OK button.

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

  Select a scope capture file and choose the OK button.

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

  Navigate to a specific file and choose the OK button.

Note : In addition to accessing other folders on the system drive, the Browse for Folder window will also provide access to any connected USB storage devices.

The selected file will be opened.

Opening next/previous files

Use the following instructions to quickly open the next or previous file on a specific drive during data capture review.

To open the next/previous file:

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

   - From Review mode, choose File >> Load New Capture 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 >> Show Next File or Show 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.

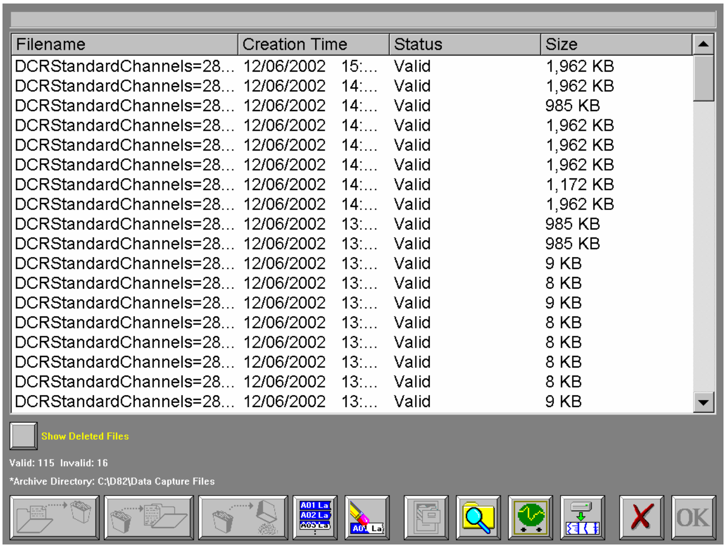
Note: To save time, add the Show Next File and Show Previous File buttons to the control panel. Refer to *Chapter 5: Menus and buttons* for more information on customizing the control panel.

Deleting files

Use the following instructions to delete files. This process is initiated by launching Review mode from any other mode of operation.

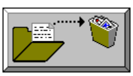
To delete files:

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



Filename	Creation Time		Status	Size
DCRStandardChannels=28...	12/06/2002	15:...	Valid	1,962 KB
DCRStandardChannels=28...	12/06/2002	14:...	Valid	1,962 KB
DCRStandardChannels=28...	12/06/2002	14:...	Valid	985 KB
DCRStandardChannels=28...	12/06/2002	14:...	Valid	1,962 KB
DCRStandardChannels=28...	12/06/2002	14:...	Valid	1,962 KB
DCRStandardChannels=28...	12/06/2002	14:...	Valid	1,962 KB
DCRStandardChannels=28...	12/06/2002	14:...	Valid	1,172 KB
DCRStandardChannels=28...	12/06/2002	14:...	Valid	1,962 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	985 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	985 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	9 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	8 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	9 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	8 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	8 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	8 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	8 KB
DCRStandardChannels=28...	12/06/2002	13:...	Valid	9 KB

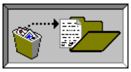
2. To send files to the trash, select the files and choose the Send to Trash button.



The selected files will be transferred to the trash.

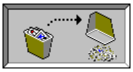
3. To retrieve files from the trash, check the Show Deleted Files checkbox. The file list will be refreshed to display deleted files. Then select the files from the list.

   Choose the Retrieve from Trash button.

   The selected files will be retrieved from the trash.

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

   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.

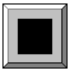
Default control panel

The following buttons are provided in the default Review mode control panel.

Note: The Review mode control panel can be customized to meet the needs of specific application or users. Refer to *Chapter 5: Menus and buttons* for information on customizing the control panel.

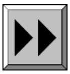
Scroll Backward scrolls the chart backward.

Rewind scrolls the chart backward quickly.



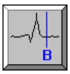
Scroll Stop stops the chart from scrolling.

Scroll Forward scrolls the chart forward.

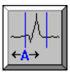


Fast Forward scrolls the chart forward quickly.

Show/Hide Cursor A displays and hides cursor A.



Show/Hide Cursor B displays and hides cursor B.



Set Active Cursor changes the active cursor between A, B, or A & B.

Context Help provides on-screen help based on the mode or window in use.

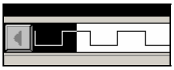
Icon Help provides on-screen help for the purpose of identifying icon buttons.

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 Review mode display. It can be used to navigate to other areas of the file.



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.

To navigate though a file using the track view:

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

2. 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.

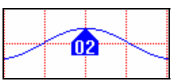
3. To move directly to a new location in a file, press a non-shaded area of the track view. The display will move to the selected location.

To change the signal displayed in the track view:

1. To change the track view signal using the menu bar, choose View >> Track View Channel. A list of available signals and events will appear.

   Select a signal or event. The track view will be updated to display the chosen signal or event.

2. To change the track view signal using channel IDs, View >> Show Channel IDs. Channel ID labels will be displayed.



Choose an ID label. The track view will be updated to display the chosen signal or event.

Using the control panel

The default control panel for Review mode provides a variety of buttons to scroll through the file. Refer to the *Default control panel* section in this chapter for more information on scroll icons.

Menu bar

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

Note: Many of the navigational features found on the menu bar are also included in the default control panel for Review mode. For more information on customizing the control panel, refer to *Chapter 5: Menus and buttons*.

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.

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.

To show all of the file:

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. 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.

The maximum amount of compression allowed is that which will cause the file to fit on one page.

To set a compression:

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. 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.

To set up an expansion:

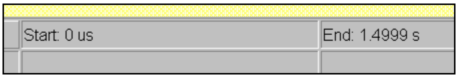
1. Choose Display >> Expansion. A number pad will appear.
2. Enter the desired expansion ratio and choose OK. The file will be expanded.

View and display 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.

To select a status text format:

1. Choose View >> Status Text. A sub menu will appear.

2. Select a status text display option. The following options are available.

   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.

   The selected status text will be displayed.

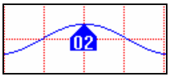
Displaying signal IDs

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

Note: Signal IDs can be used to change the signal displayed in the track bar. To change the track bar signal, choose a signal ID label.

To display signal IDs:

1. Choose View >> Show Channel IDs from the menu bar.



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

2. To hide the signal IDs, repeat this process.

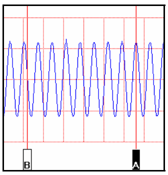
Auto scaling between cursors

Use the following instructions to apply a full scale to channels based on the area between cursors.

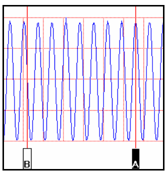
To auto scale between cursors:

1. Show both of the cursors by selecting Cursors >> Show Cursor A and Cursors >> Show Cursor B from the menu bar.

2. Position the cursors at the beginning and end of the data you want to auto scale.



3. Choose Display >> Autoscale Between Cursors. Amplitude scaling will be applied to all channels based on the area between the cursors.



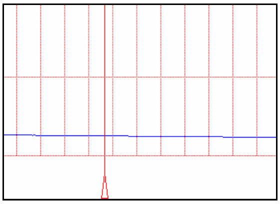
4. To remove the auto scaling, choose Display >> Clear Autoscale.

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.

To display the trigger line:

1. Choose Display >> Show Trigger Line from the menu bar. The trigger line will be displayed.



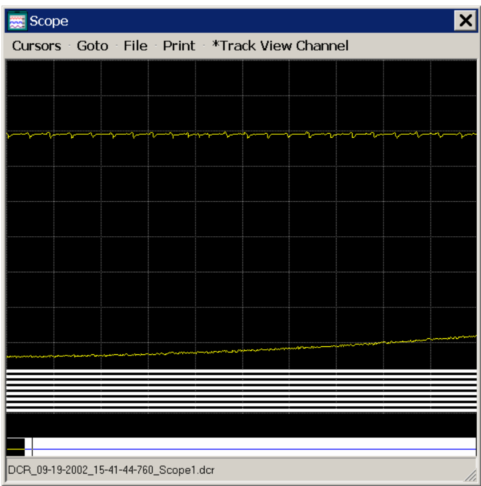
2. To hide the trigger line, repeat this process.

Viewing scope captures

If embedded scope captures are included in a capture, they can be viewed in Review mode.

To view scope captures:

1. Choose View >> Scope Viewer from the menu bar. The Scope window will open.



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

2. If necessary, choose the Cursors menu option to use cursors. The cursors in this window function similar to cursors in other Dash 8Xe functions.

3. If necessary, choose Goto >> Scope Location from the menu to scroll to the scope location in the file.

4. If necessary, move to the next or previous scope capture in the file by choosing File >> Next *or* Previous.

5. If necessary, change the signal displayed in the track view by choosing Track View Channel. A sub menu will appear. Select a new track view channel from the list.

6. Choose the X in the upper-right corner to close the Scope window.

Viewing and editing notes

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

To view and edit notes:

1. Chose View >> Note >> Note Viewer from the menu bar. The Edit Notetag window will open.

> **Note:** As an alternate method of opening the Edit Notetag window, scroll to and choose a pencil icon in the file.
>
> 

Use the Find button to scroll to other note locations in the file, if necessary.

2. If necessary, edit the note using the keypad.

3. Choose the OK button to close the Edit Notetag window. Any note edits will be saved.

Measurement and analysis tools

This section provides information about the various measurement tools available in Review mode.

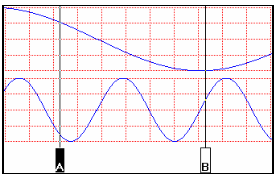
Cursors

Cursors are a valuable tool that can be used for measuring signals. They are used in conjunction with the Channel Information window to view a variety of measurements.

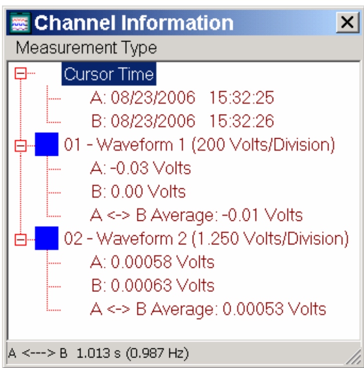
Note: To save time and gain more control over cursors, add cursor buttons to the control panel. Refer to *Chapter 5: Menus and buttons* for more information on customizing the control panel.

To measure signal information 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 View >> Show 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. The following measurement types are available: Average, Max-Min, Peak-Peak, Slope, and RMS.

Average - Average displays the midpoint value of the data represented by the cursor. When using two cursors, this measurement displays the average of the midpoint values of the data between cursors A and B.

Max-Min - Max-Min displays the maximum and minimum signal values of the data represented by the cursor. When using two cursors, this measurement displays the maximum and minimum signal values represented by the data between cursors A and B.

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

Slope - Slope displays the slope (delta V per unit time) of the signal between cursors A and B. This measurement is available only when both cursors are displayed.

RMS - RMS displays the root-means-square of the midpoints of the max-min pairs (line segments). This measurement is available only when both cursors are displayed.

3. To change the measurement type, choose the Measurement Type option on the menu bar of the Channel Information window. A sub menu will appear. Choose a measurement type from the list.

4. If necessary, move one or both of the cursors. Cursors must be activated before they can be moved.

   - **Move cursor A or B individually** by choosing Cursors >> Active Cursor >> Cursor A or Cursor B to activate it. Then press the active cursor and drag it to a new location.

   - **Move cursor A and B together** by choosing Cursors >> Active Cursor >> Cursor A + B to activate both cursors. Then press one of the cursors and drag it to a new location. The other cursor will move as well.

Note: As a shortcut, activate a cursor by pressing it once on the touch-screen. To move it, press it again, but do not release it. While pressing, drag it to a new location, and then release it.

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

5. If necessary, modify the color and type of the cursors.

    - To modify the cursor color, choose Cursors >> Color. Then select a color from the options that appear.

    - To modify the cursor type, choose Cursors >> Type. Then select a type from the options that appear.

      Select the Color option to use color cursors. Select the Invert option to use pseudo-transparent cursors.

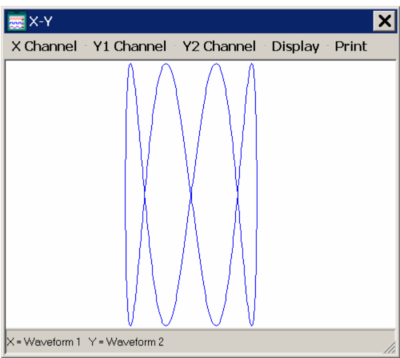
XY Plot

Use the following instructions to display the XY Plot.

Note: The review mode XY plot is generated based upon points in the DCR file being reviewed. When cursors are enabled, however, the XY plot will be generated based upon review mode screen points.

To display the XY plot:

1. Choose Analysis >> XY Plot Window to open the XY plot window.



2. Select the X, Y1, and Y2 channels using the X Channel, Y1 Channel, and Y2 Channel menu options.

3. If necessary, select the Display menu to use the following display options:

   Refresh - Use the Refresh option to clear the window and restart the XY plot drawing.

   Connect Points - Use the Connect Points option to draw lines to connect the sample points on the XY plot. When this option is active, segments will be drawn.

   Background Color - Use the Background Color option to select a background color for the XY plot window.

   Foreground Color - Use the Foreground Color option to select a color for the points and segments drawn on the XY plot.

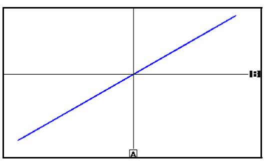
   Grid - Use the Grid option to display a grid. Grid color and division settings can also be specified.

   Open Paint After Save - Use the Open Paint After Save option to open a plot in Microsoft Paint after saving the plot. When this option is active, plots will automatically be opened in Paint.

   Save Plot - Use the Save Plot option to save the XY plot as a bitmap (BMP) image file.

4. If necessary, choose the Cursors menu option to use cursors. The cursors in this window function like cursors in other Dash 8Xe functions, with the exception of the following features.

   - Cursor A is a vertical cursor and cursor B is a horizontal cursor.



   - To restrict cursor movement to the data points, choose Cursors >> Lock On Data. The starting and ending data points will be highlighted automatically. The Lock On Data option is unavailable when the Y2 plot is used.

To move the cursors to the start or end point on the plot, choose Cursors >> Goto Start or Goto End.

To move the cursors among data points, use the Move Cursor Left and Move Cursor Right control panel icon buttons.

5. If the printer is installed, use the Print >> Print Screen option to print the XY Plot window contents.

XY plot templates

XY plot templates allow you to display a customized background template in the XY plot window. This background is a visual aid you can use for comparing plot results against a standard template you define.

For detailed information about creating and using XY plot templates, refer to *XY plot templates* in *Chapter 8: Realtime mode overview*.

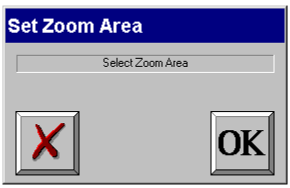
Zoom window

The zoom window magnifies a portion of a capture for detailed analysis.

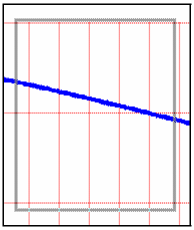
Note: Data displayed in the Zoom window is generated based upon points in the DCR file being reviewed.

To use the zoom window:

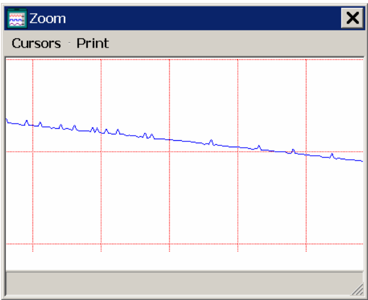
1. Choose Analysis >> Zoom Window from the menu bar. The Set Zoom Area window will open.



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 Set Zoom Area window. The Zoom window will appear.



The Zoom window displays the rectangular area selected. Cursors are available via the Cursors menu. This area can be scrolled by pressing and dragging the outlined area in the waveform display.

If the optional thermal printer is installed, the contents of the Zoom window can be printed using the Print menu.

4. To close the window, choose the X in the upper-right corner of the window.

Fourier Transform window

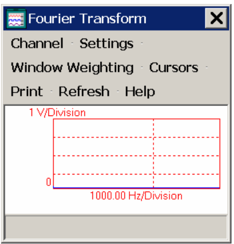
The FFT is created from the data displayed in the main review screen. This data can consist of up to 1024 segments. Each segment consists of two points, a minimum and maximum, which represent the minimum and maximum signal values for each period.

FFT calculations require a single point for each period. The point used for the FFT calculation is the midpoint of the minimum and maximum. Therefore, a signal overlay compressed may not yield accurate FFT results.

Use the following instructions to measure signals with the Fourier Transform window.

To use the Fourier Transform window:

1. Choose Analysis >> Fourier Transform from the menu bar. The Fourier Transform window will open.



2. Choose the Channel option. A list of signals will appear. Select the signal to analyze.

3. If necessary, select the Settings menu to modify the following options:

   Background Color - The Background Color option is used to select a color for the background of the window.

   Channel Color - The Channel Color option is used to select a color for the channel in the window.

   Grid - The Grid option is used to select whether to display a grid. Grid color, amplitude axis, and frequency axis are also specified using this option.

   Plot - The plot option is used to choose between linear magnitude, logrithmic magnitude, or magnitude^2.

Size - The size option is used to adjust the size of the window. Available window sizes include small, medium, and large.

Frequency Expansion - The Frequency Expansion option is used to expand the frequency axis to provide a more detailed view.

4. If necessary, choose a window weight option. The following options are available: Hanning, Hamming, Blackman, Barlett, Triangle, Kaiser, Bman-Harris, and Rectangle.

5. If necessary, choose the Cursors menu option to use cursors. The cursors in this window function similar to cursors in other Dash 8Xe functions.

6. Close the Fourier Transform window by choosing the X in the upper-right corner of the window.

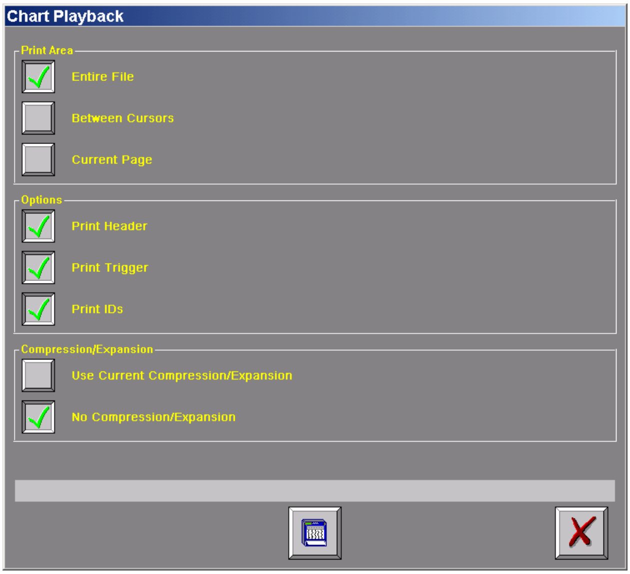
Printing data capture files

If you are using the optional SCR-8 thermal printer, you can print data capture files from Review mode.

Note: For information about the SCR-8 printer, refer to *Chapter 2: Hardware overview.*

To print data capture files:

1. Choose Chart >> Print. The Chart Playback window will open.



2. Choose the print area that will be printed on the SCR-8.

   You can print the entire file, the portion of the file between cursors, or the page currently visible on the display.

3. Choose the print options you want to include in the printed output.

   Print Header - Choose this option to print information related to the data capture file at the start of playback. The header contains information including the file name, channel labels, capture time, compression, and sample counts.

   Print Trigger - Choose this option to print a full page mark during playback to indicate the trigger point.

   Print IDs - Choose this option to print channel IDs periodically during playback. Channel numbers will be printed next to waveform traces approximately once per two sheets of paper.

4. Choose the compression/expansion option you want to use in the printed output.

   Use Current Compression/Expansion - Choose this option to print the file using the compression/expansion settings currently configured in Review mode.

   No Compression/Expansion - Choose this option to print the file without using any compression/expansion.

5. Choose the print button. The chart will be printed from the SCR-8 and configured with the selected options.

Archiving files

Archiving a file copies the file from the data capture drive to the system (Windows) 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.

This section describes how to archive files in data capture record and ASCII formats.

Archiving as data capture records

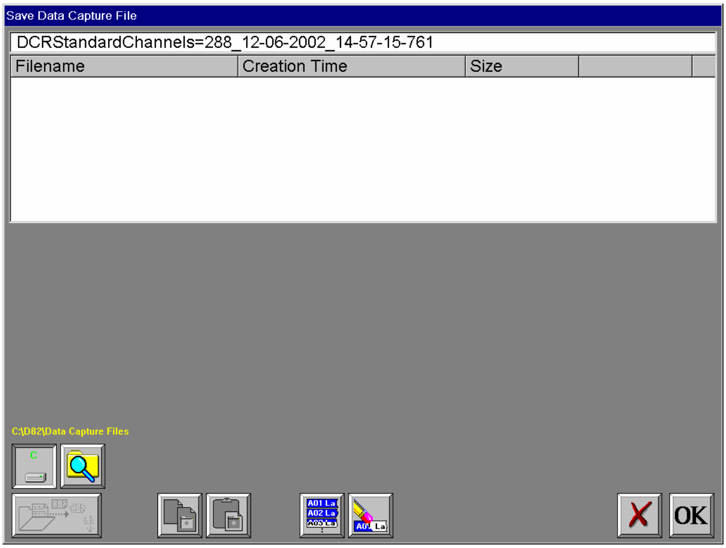
Use the following information to archive as data capture records. This binary format is preferable when using Dash 8Xe Offline or AstroVIEW on a PC.

Note: Files can also be archived as data capture records in the file selection window prior to accessing Review mode. Select the files to archive and choose the file cabinet icon button. This method is helpful for archiving multiple files at the same time.

To archive as data capture records:

1. With a file open in Review mode, choose File >> Archive from the menu bar. Then select whether to archive the entire file, current page, or the area between cursors.

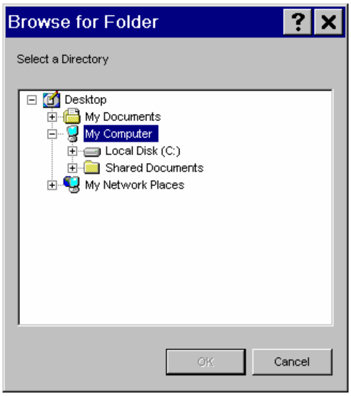
The Save Data Capture File window will open.



2. By default, the file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

   If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

   The Browse for Folder window will appear.



Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

3. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the file and choose the OK button. The specified file name will appear in the field.

4. Choose the OK button in the Save Data Capture File window. The file will be archived.

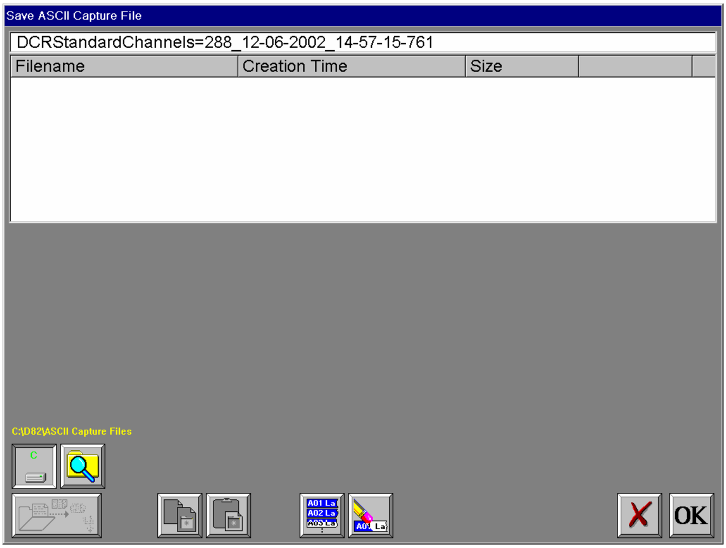
Archiving as ASCII

Use the following information to archive as ASCII files. ASCII format is helpful for archiving and analysis in other programs. Because the ASCII format results in large file sizes, this format is not recommended for larger data captures. However, it is ideal for archiving small parts of data captures.

To archive as ASCII:

1. With a file open in Review mode, choose File >> Archive as ASCII from the menu bar. Then select whether to archive the entire file, current page, or the area between cursors.

   The Save ASCII Capture File window will open.

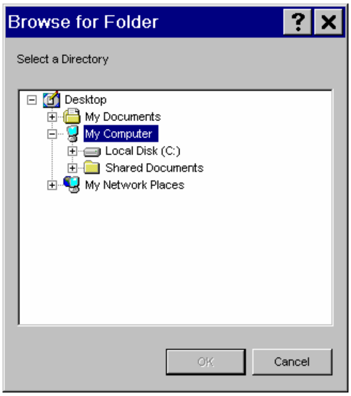


2. By default, the file will be saved on the system drive (C) in the noted location. The path to this location is displayed in yellow text.

   If necessary, the file can be saved in a different location. To select an alternate file location, choose the Browse Folders button.

   The Browse for Folder window will appear.

   Select a file location and choose the OK button. The path displayed in yellow text will be updated to indicate the new file destination.

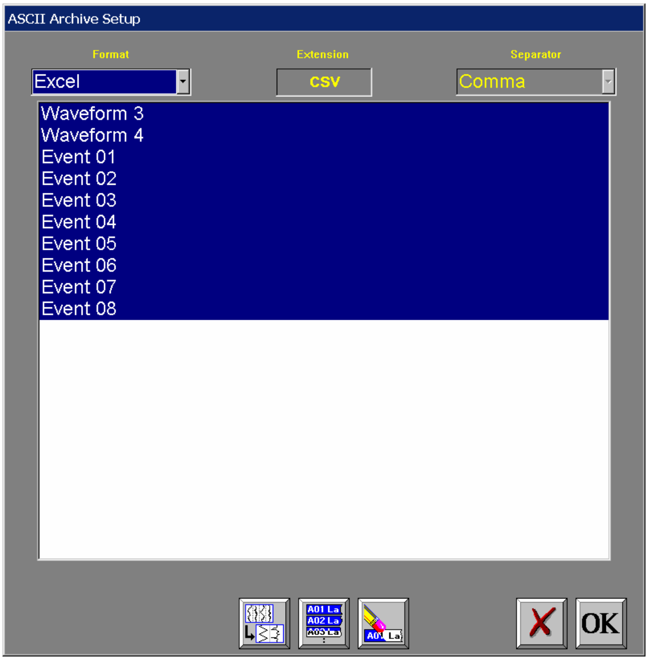
Note : In addition to accessing other folders on the system drive, the Browse for Folder window will also provide access to any connected USB storage devices.

---

3. Choose the file name field above the column headings. A keypad will appear.

   Enter a name for the file and choose the OK button. The specified file name will appear in the field.

4. Choose the OK button in the Save Data Capture File window. The ASCII Archive Setup window will appear.



5. Specify the file setup options.

   - Choose Excel format to save the file with a CSV extension and comma delimiters.
   - Choose MatLab to save the file in a format that can be used in MatLab 6.0 and above.
   - Choose Other format to save the file with a custom extension and delimiters.

   Select the channels and events to include in the archive. By default, all channels and events are selected.

6. Choose OK. The archive file will be saved.

Archiving files in a compressed format

You can archive files in a compressed format to save disk space.

A compression factor is used to reduce the number of data points in the archived file. The compression factor is determined by the display compression (Display >> Compression).

For example, if the compression is set at 50, the following will occur: For every 50 data points, one average point will be created to replace the 50 points. The compressed archive file will contain all of the averaged points. This results in an archive file size that is a fraction of the original.

Use the following instructions to archive files in a compressed format.

To archive files in a compressed format:

1. If necessary, modify the compression factor of the file.

   While reviewing a file, choose Display >> Compression. A number pad will appear. Enter a compression factor and choose the OK button.

2. Choose to archive the file in the desired format.

   - To archive in data capture record format, choose File >> Archive File >> Compressed.

   - To archive in ASCII format, choose File >> Archive as ASCII >> Compressed.

   A sub menu will appear. Select whether to archive the entire file, current page, or the area between cursors.

   Note: Once the Compressed format is chosen, the process for archiving the file is identical to the non-compressed version. Refer to the previous two sections for information on archiving non-compressed files.

Chapter 13: Networking & communications

This chapter provides information about Dash 8Xe networking and communications options.

Link port

Link ports provide a method of directly connecting multiple Dash 8Xe units. When units are connected with a link port, they can sample and capture data simultaneously. Up to eight units can be linked in this manner.

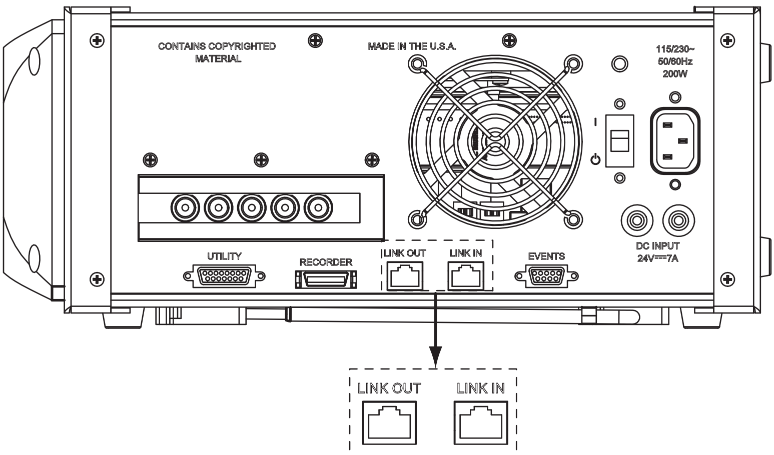
This type of arrangement allows multiple Dash 8Xe units to share data capture-related features. Link port connections cannot be used to transfer files between multiple Dash 8Xe units.

Connecting link ports

Link ports are connected using standard ethernet category 5 cables with RJ-45 connectors. Use the following instructions to connect multiple Dash 8Xe units via link port.

To connect link ports:

1. Locate the Link Out and Link In ports on the side of the Dash 8Xe.



2. Connect the Dash 8Xe units in a circular configuration.

   Connect the Link Out from one Dash 8Xe to the Link In on the next Dash 8Xe in the circle. Repeat this process until the circular configuration is complete and all units are linked together.

   Note: Each link cable should not be longer than 400 feet.

Setting up link port data captures

A link port data capture differs from a standard data capture using one unit. In a link port data capture, multiple Dash 8Xe units are connected together.

Note: This section assumes knowledge of data capture concepts and processes. For detailed data capture information, refer to *Chapter 10: Data capture*.

Each unit in the configuration can be set up with identical or different data capture settings. When a trigger occurs on one of the linked units, all other units in the configuration will also be triggered.

The Dash 8Xe uses a master and slave configuration to control link port data captures. Use the following guidelines to understand how master and slave units can operate during a link port data capture.

Master

One Dash 8Xe in the configuration is designated as the master unit. The master unit:

- Provides the sample clock to all slave units to ensure sample synchronization
- Can arm, abort, and trigger data captures

Slave

All other Dash 8Xe units in the configuration are designated as slaves. Slave units:

- Can only trigger data captures

Use the following instructions to set up a link port data capture.

To set up a link port data capture:

1. Ensure that all of the Dash 8Xe units are properly connected via the link ports.

2. Set up the master unit.

   On the Dash 8Xe that will be the master unit, choose Capture >> Capture Settings to open the Capture Setup window.

   Choose Master from the Link Port drop-down list.

   Set up the data capture, trigger, and abort settings for the master unit.

3. Set up slave units.

   On each Dash 8Xe that will be a slave unit, choose Capture >> Capture Settings to open the Capture Setup window.

Choose Slave from the Link Port drop-down list.

Note: Dash 8Xe units should never be set to Slave without first being connected via link port to other units.

Set up the data capture and trigger settings for each slave unit.

4. Start the data capture.

   Arm the data capture using the master unit.

   If a trigger occurs on any linked Dash 8Xe, all connected units will be triggered.

   If necessary, the capture can be aborted by the master unit.

IP information

The Dash 8Xe can be connected to an existing network and assigned an IP address for TCP/IP communication.

Default values

This section notes the factory default IP values for the Dash 8Xe.

IP Address	192.168.255.1
Subnet Mask	255.255.255.0
Gateway	None

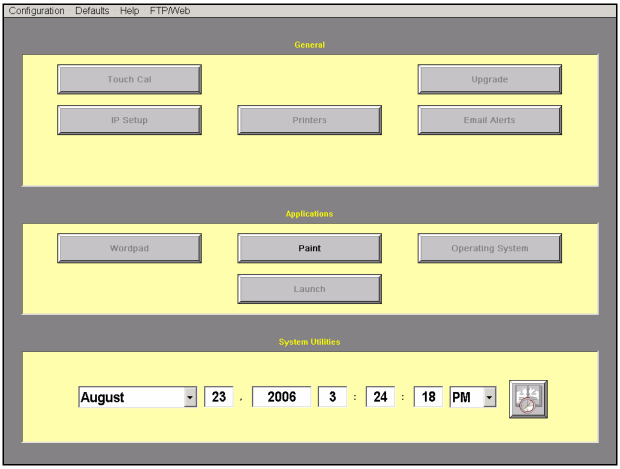
Modifying IP information

The Dash 8Xe can be configured to obtain an IP address automatically using Dynamic Host Configuration Protocol (DHCP). Alternately, a specific IP address can be assigned.

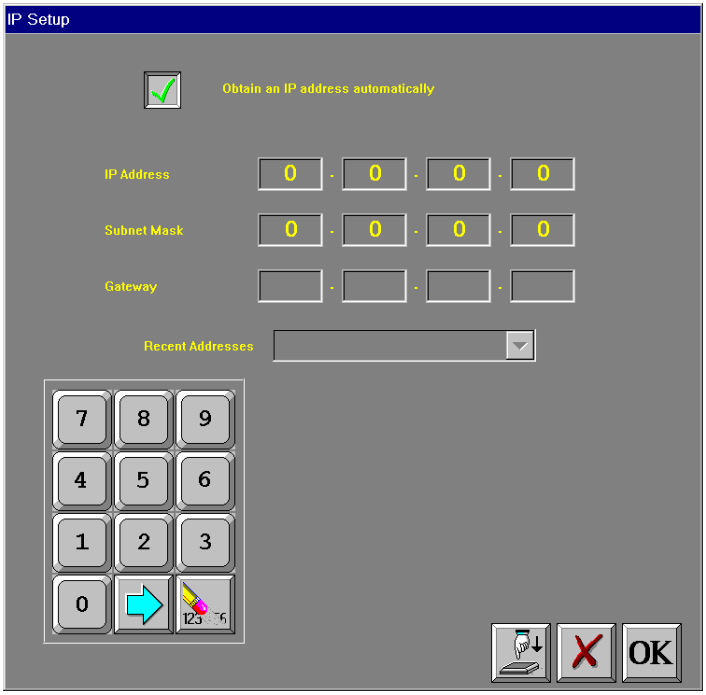
Use the following instructions to modify the Dash 8Xe IP information.

To modify IP information:

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



2. Choose the IP Setup button to open the IP Setup window.



3. To use DHCP, check the Obtain an IP address automatically checkbox.

4. To specify specific values for the IP address, ensure that the Obtain and IP address automatically option is unchecked.

   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.

Note: When an IP address is used, it will be added to the Recent Addresses drop-down list. To select this address in the future, select it from the list instead of entering it again.

---

5. Choose the OK button to finish setting up the IP information.

File transfer protocol (FTP)

This section provides information on using FTP applications to transfer files to and from the Dash 8Xe. It also provides a detailed list of the Dash 8Xe folder structure and file types.

Using FTP applications

Any FTP program can be used to log into the Dash 8Xe and transfer files. Some web browsers also provide FTP functionality. When using a browser, enter the FTP address in the address bar, and then enter the username and password when prompted.

Note: FTP applications will provide access to all files on the Dash 8Xe, including important system files. Use caution to ensure that system files are not accidentally deleted.

This feature should be used only by individuals familiar with FTP applications and concepts. For a more direct and user-friendly method of downloading data capture files, refer to the *Web browser interface* section of this chapter.

To use FTP Applications:

1. Ensure the Dash 8Xe is on and properly connected to a network.

2. On the Dash 8Xe unit, choose Configuration >> Utilities to open the Utilities screen. Then choose FTP/Web >> Enabled to activate the Dash 8Xe FTP/Web function.

   A check mark will be displayed next to the Enabled option on the FTP/Web menu to indicate the FTP/Web functions are active.

3. Launch a file transfer protocol (FTP) application using a computer connected to the same network.

   Log into the Dash 8Xe with the following information:

Username:	dash8x
Password:	dash8x (or other system password if changed)
FTP Address:	ftp:// 192.168.255.1/ (or other IP address if changed)

Dash 8Xe folders and files

All Dash 8Xe files are stored in a folder located in the root directory. The path to this folder is: C:\D82.

Within this folder, the following sub folders are present:

ASCII Capture Files - This folder contains all data capture files that have been saved in ASCII format. These files have a CSV extension.

Control Panel Setup Files - This folder contains three folders (Realtime, Review, and Scope). These folders contain control panel setup files for Realtime, Review, and Scope mode. The file extensions for these files are as follows:

Realtime control panel file	.DCP
Review control panel file	.RCP
Scope control panel file	.SCP

Data Capture Files - This folder contains sub folders that represent data capture files. To transfer data capture files, copy the entire folder. The data capture folders contain a data capture record (DCR) file, and an IDX file, which contains a listing of external items used in a data capture (notes and scope captures). Additionally, these external items (TXT and SCR) are also stored in this folder.

Note: Data capture record (DCR) files can be opened and analyzed with AstroVIEWX or Dash 8Xe offline software.

Global Setup Files - This folder contains all file groups associated with global setup files. Global file groups consist of up to eight files with the same name, but different extensions. To transfer a global setup file, copy *all* of the files within the global file group.

Scope Capture Files - This folder contains scope capture record files. These files have an extension of SCR.

Signal Setup Files - This folder contains signal setup files. These files have an extension of DSS.

System Capture Files - This folder contains files for system use only.

Upgrade - This folder is used to upgrade the Dash 8Xe software. It is also used to automatically load control panel, signal, and view setup files. When a setup file is transferred to this folder, the file will automatically be loaded, and then deleted from the system.

Upload - This folder is used to automatically load global setup files. Use the following process to transfer a global setup file:

> First, copy all of the files in the global setup file group to this folder *except* the DGS file, which contains a list of all files in the global setup file group. Next, copy the DGS file to the folder. The files will automatically be loaded, and then deleted from the system.
>
> The DGS file initiates the loading of the global setup file, so it is important that this file is the last file copied to the folder.

View Setup Files - This folder contains view setup files. These files have an extension of DVS.

Web browser interface

The Dash 8Xe provides a web browser-based interface that can be accessed from any computer connected to the same network. The web browser interface provides basic functionality to support remote use.

Accessing the web browser interface

Use the following instructions to access the Dash 8Xe via a web browser.

To access the web browser interface:

1. Ensure the Dash 8Xe is on and properly connected to a network.
2. On the Dash 8Xe unit, choose Configuration >> Utilities to open the Utilities screen. Then choose FTP/Web >> Enabled to activate the Dash 8Xe FTP/Web function.

   A check mark will be displayed next to the Enabled option on the FTP/Web menu to indicate the FTP/Web functions are active.
3. Launch a web browser using a computer connected to the same network.
4. Enter the following address in the address bar:

   http:// 192.168.255.1/dash8x

   If the IP address of the Dash 8Xe has been changed from the default value, enter the new IP address instead.

The web browser interface will open and display the Status page.

The status page displays current system settings.

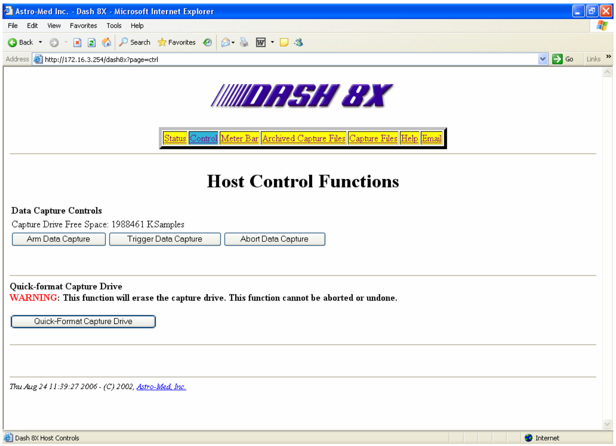
Note: A help page that provides descriptions of other page functions is also available. To access the help page, choose the Help link.

Using the control functions

The web interface provides basic host control functions to control data captures from a computer on the network. A data capture can be armed, triggered, and manually aborted from this page. Note that slow rate captures cannot be made in this manner, and that captures must be manually aborted.

To use control functions:

1. With the browser interface open, choose the Control link. The Host Control Functions page will open.



2. Arm, trigger, or manually abort a data capture from this page by choosing the appropriate button.

   The data capture in this case uses capture settings that are currently configured on the Dash 8Xe.

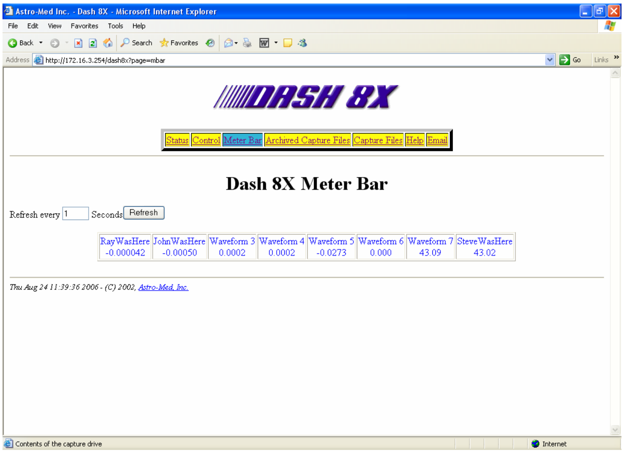
3. Quick-format the data capture drive by clicking the appropriate button in the bottom left corner of the screen. Note that this action cannot be undone.

Viewing meter bar values

The current meter bar values can be viewed remotely using the web interface. The meter bar page functions exactly as the meter bar on the Dash 8Xe; it shows realtime numerical values of the channels.

To view meter bar values:

1. With the browser interface open, choose the Meter Bar link. The Dash 8Xe Meter Bar page will open.



2. To see the values change in realtime, specify a refresh rate for the meter bar and choose the Refresh button.

   The meter bar values will be updated based on the selected rate.

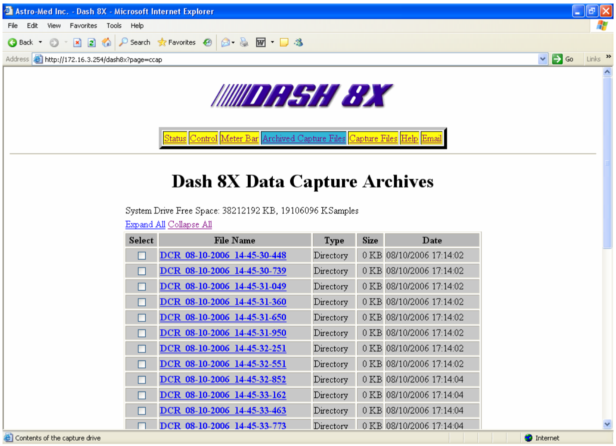
3. To stop refreshing the meter bar values, choose the Stop Refreshing link.

Accessing archived data capture files

Use the following instructions to download data capture files to a computer connected to the network. Files can also be deleted using this feature.

To access archived data capture files:

1. With the browser interface open, choose the Archived Capture Files link. The Dash 8Xe Data Capture Archives page will open.



2. Choose a viewing preference by choosing the Expand All or Collapse All option. The expanded view provides a listing of all files within a data capture record. The collapsed view provides a listing of data capture record folders.

   To expand or collapse individual data capture record folders, choose them individually.

3. To download a file, choose the link for the file. A download prompt will open. Choose to save the file and save it to a location on the computer.

4. To delete a data capture record, select the checkbox next to the record name. Then choose the Delete Selected Files button.

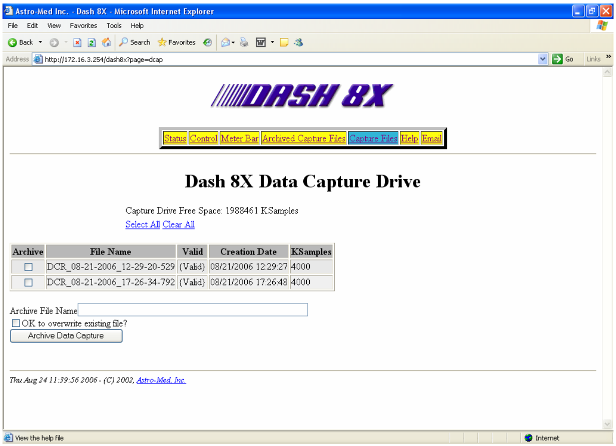
Archiving data capture files

Use the following instructions to archive data capture files to the Dash 8Xe system drive from a computer connected to the network.

When the data capture files are archived, they can be downloaded using the Archived Capture Files page.

To archive data capture files:

1. With the browser interface open, choose the Capture Files link. The Dash 8Xe Data Capture Drive page will open.



This page displays the contents of the data capture drive.

2. To archive a file, choose the Archive checkbox next to the file to archive. Repeat this procedure for all files you wish to archive. If necessary, you can use the Select All option to choose all files displayed in the window.

   Select whether to overwrite an existing file, if one exists, using the checkbox. Checking this option allows overwriting of an existing file if it has the same name as the file being archived.

   Then choose the Archive Data Capture button. The file(s) will be archived.

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

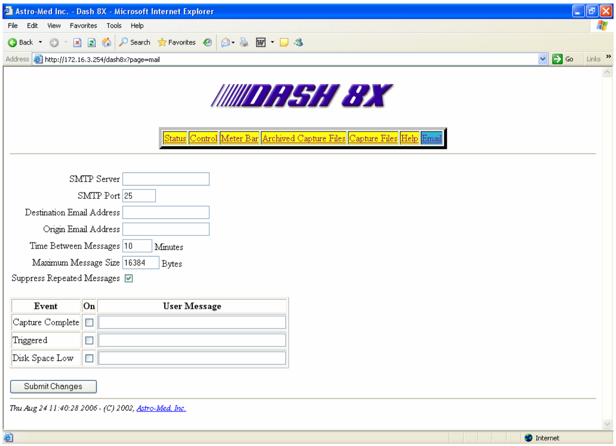
Setting up email alerts

Email alerts can be sent from Dash 8Xe units connected to a network. These messages inform email recipients of events including data capture completion, triggers, and low disk space conditions.

Use the following instructions to set up email alerts remotely.

To set up email alerts:

1. With the browser interface open, choose the Email link. The Email page will open.



2. Enter the name of the SMTP mail server that will be used to send email messages. Then enter the SMTP port.

3. Enter destination and source email addresses.

   - Email addresses in the Destination Email Address field will receive an email message when the specified conditions occur.

     Multiple email addresses can be entered by placing a semicolon between each address. The maximum number of characters this field can hold is 256.

   - The email address in the Origin Email Address field will be listed as the sender in all email messages the Dash 8Xe generates.

4. Specify the email setup options.

- The value in the Time Between Messages field indicates the amount of time allowed for messages to accumulate before they are sent in one email.

  Enter the amount of time for messages to “build up” using this field.

- The value in the Maximum Message Size field indicates the maximum allowable size of the email messages that can be set up.

- To suppress repeated messages, check the Suppress Repeated Messages checkbox. When this option is enabled, subsequent messages of the same type will not be sent.

5. Select the types of events that will result in a message, and enter the associated email messages.

    Capture Complete - The capture complete message occurs when a data capture completes successfully.

    Trigger - The trigger message occurs when a trigger condition occurs during a data capture.

    Disk Space Low - The disk space low message occurs when the space on the Dash 8Xe hard drive is limited.

   To enable a message for a particular event, check the appropriate check box. Then enter a text message in the associated field.

6. To complete the email setup process, choose the Submit Changes button.

Chapter 14: Service options

This chapter provides information about the Dash 8Xe service options.

Calibrating channels

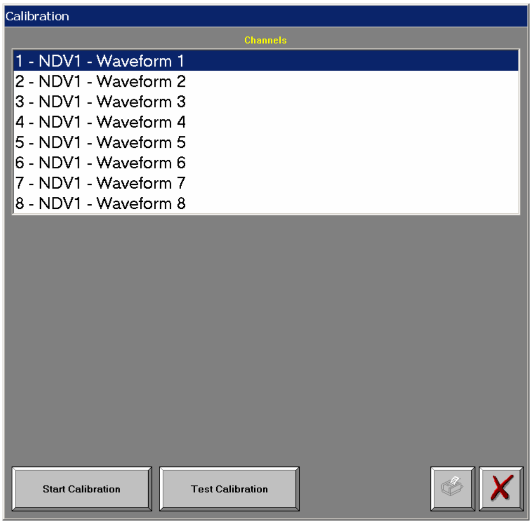
The Dash 8Xe provides a calibration function that adjusts the calibration of the signal input modules based on known standards. Calibration should be performed approximately once per year. New signal modules are calibrated at the factory.

Note: Known voltages must be applied during the calibration process. The set of needed voltages will vary based on the input module. Ensure that all required voltages are available prior to starting the calibration process.

Use the following instructions to calibrate channels.

To calibrate channels:

1. Choose Service >> Calibration to open the Calibration window.



2. Select a channel, then choose the Start Calibration button. The calibration routine will begin.

The Dash 8Xe will prompt for specific voltages to be placed at the selected input channel. These voltages will differ based on the input module being calibrated.

Module	**Required Voltages**
NDV1	0, 2.5, 15 V
IHV1	0, 2.5, 10, 15, 100 V
IBR1	20, 100 mV 1 V
IHV2	100 V
ITCU	50 mV (accurate to 5 uV or better) *Copper shorting plug - type U miniature thermocouple connector & copper wire* - 100° C (accurate to .5° C or better) *Thermocouple type N - type N miniature thermocouple connector & type N wire*
IRTD	100 Ω (accurate to .03 Ω or better) 300 Ω (accurate to .05 Ω or better)
IHV3	50, 500 mV 5, 15, 100 V
IDCV, IDCV1	100 V (accurate to +/- 6 mV or better) *BNC cables and adapters should be avoided. Double banana connectors must be of the same construction.*
LIVM	25, 100, 250 mV 2.5 V

The selected channel will be calibrated based on the voltages.

3. Calibrate other channels, if necessary.

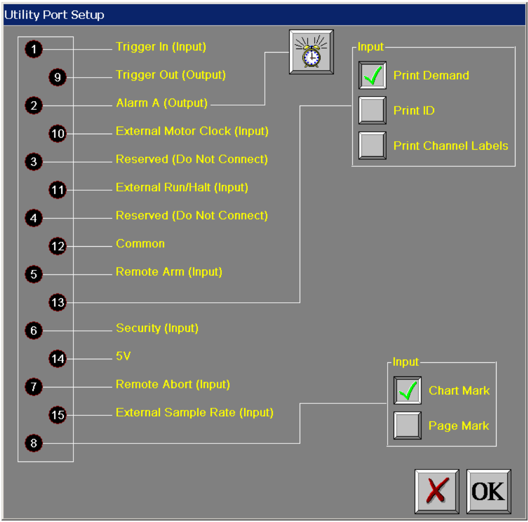
4. Choose the X button to close the Calibration window.

Viewing and modifying utility port settings

Use the following instructions to view and modify utility port settings.

To view and modify utility port settings:

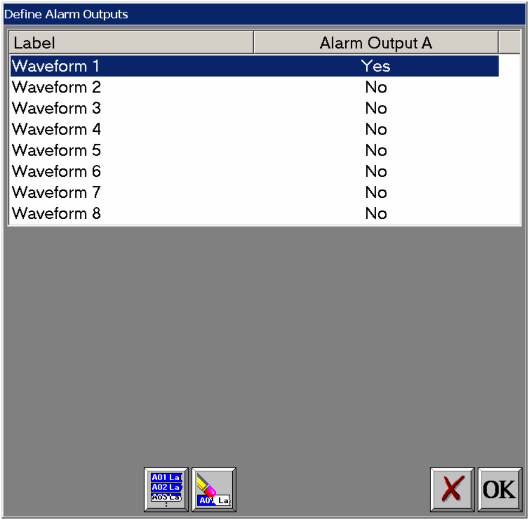
1. Choose Service >> Utility Port to open the Utility Port Setup window.



This window provides a graphical representation of the utility port pins and setup options.

2. To modify alarm output settings, choose the Alarm button.

The Define Alarm Outputs window will open.



Select a channel (or multiple channels with "or" logic) and choose the Alarm Output A column heading. A sub menu will appear. Choose Yes or No to indicate whether an alarm output should be used for the selected signal.

Choose the OK button to finish setting up alarm outputs.

3. To specify the type of print input, check a Port 13 Input option.

   Print Demand - The Print Demand option will print the demand buffer.

   Print ID - The Print ID option will print the signal IDs.

   Print Channel Labels - The Print Channel Labels will print the labels assigned to each channel.

4. To specify a chart/page mark input, check a Port 8 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 scrolling waveform display area.

5. Choose the OK button to save the utility port setup options.

Changing the system password

The Dash 8Xe system password is a security tool that prevents unauthorized access to the menu bar, operating system, and other Dash 8Xe functions. Use the following instructions to change the system password.

To change the system password:

1. Choose Service >> Security >> Set Password to open the Set Password window.

2. Enter the old password and choose the OK button.

Note: The factory default system password is “dash8x” (without quotation marks).

When the password has been correctly entered, the New Password field will appear.

3. Enter the new password and choose the OK button. The Verify New Password field will appear.

Confirm the new password by re-entering it and choosing the OK button.

The system password will be changed.

Changing the Calibration Password

You are required to enter a password in order to access the Calibration window; by default, this is the same as the system password. It is possible, however, to assign a unique password to the Calibration window. Use the following instructions to assign a Calibration password.

To change the system password:

1. Choose Service >> Security >> Set Cal Password to open the Set Cal Password window.

2. Enter the old password and choose the OK button.

Note: The factory default system password is “dash8x” (without quotation marks).

When the password has been correctly entered, the New Password field will appear.

3. Enter the new password and choose the OK button. The Verify New Password field will appear.

Confirm the new password by re-entering it and choosing the OK button.

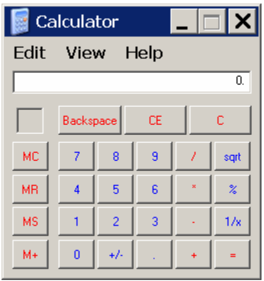
The calibration password will be changed.

Using the calculator

Use the following instructions to access the calculator program, which is part of the Windows operating system.

To use the calculator:

1. Choose Service >> Calculator to open the Calculator window.



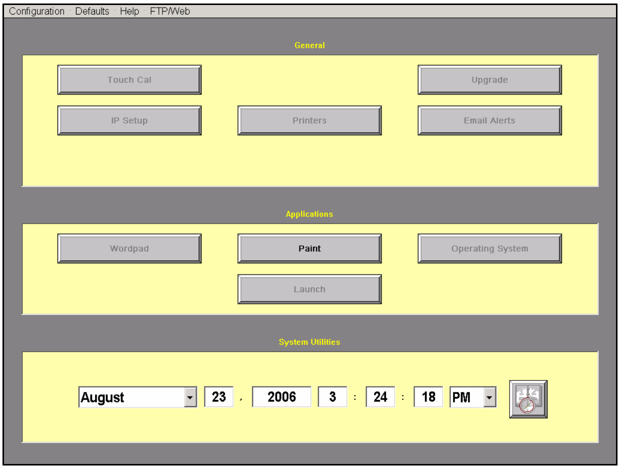
2. Choose the X button to close the Calculator window.

Chapter 15: Utilities

This chapter provides information about Dash 8Xe utilities.

Accessing utilities

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



Use the utilities screen to perform the tasks detailed in the following sections.

Restoring default settings

It may be helpful to restore some settings to their original state. Use the following instructions to return the Dash 8Xe settings to the factory-default configurations.

Note: Defaulting the signal input settings restores the module settings to factory default settings. For modules that are not changed, you may want to note the input settings so you can conveniently change the settings back to the desired values.

To restore default settings:

1. Choose Defaults from the menu bar. A sub menu will appear.

2. Choose from the following options:

   Inputs - The Inputs option restores default settings for signals.

   Realtime View - The Realtime View option restores the default view settings for Realtime mode.

   Realtime Panel - The Realtime Panel option restores the default Realtime mode control panel.

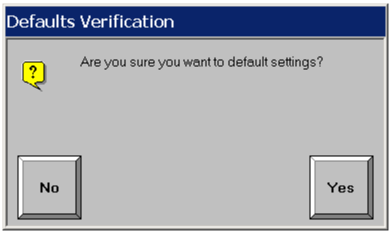
   Scope View - The Scope View option restores the default view settings for Scope mode.

   Scope Panel - The Scope Panel option restores the default Scope mode control panel.

   Review Panel - The Review Panel option restores the default Review mode control panel.

   System - The System option restores all of the above options to their default state.

A default confirmation message will appear.



3. To confirm the default settings, choose Yes. To cancel the default settings, choose No.

Selecting a language

The language setting of the Dash 8Xe can be changed to meet the needs of users.

To select a language:

1. Choose Help >> Language from the menu bar. A sub menu will appear. The following language options are available:

    American
    British
    French
    Italian
    German

2. Select a language. The selected language will be used for all Dash 8Xe text.

General utilities

The following general utilities are available in the utilities screen.

Touch Cal
The Touch Cal utility is used to set up the touch screen calibration and other display options.

To start the touch screen calibration process, choose the Touch Cal button.

IP Setup
The IP Setup utility is used to specify IP address information for the Dash 8Xe in networked environments. This process is described in *Chapter 13: Networking and communications*.

Printers
The Printers utility is used to install an external printer connected to the parallel port. This process is described in *Chapter 4: Optional hardware*.

Upgrade
The Upgrade utility is used to upgrade the Dash 8Xe system software.

Email Alerts
The Email Alerts utility is used to set up email notifications from the Dash 8Xe in networked environments.

Upgrading the Dash 8Xe software

Software upgrades may be released in the future for the Dash 8Xe. Use the following instructions to upgrade the Dash 8Xe software.

Note: Upgrading the Dash 8Xe software may require other changes to the system. Be sure to check with Astro-Med Technical Support prior to installing any software upgrades.

To upgrade the Dash 8Xe software:

1. Insert a USB flash memory drive containing the software upgrade.
2. From the Utilities screen, choose the Upgrade button. The Dash 8Xe software will be upgraded.

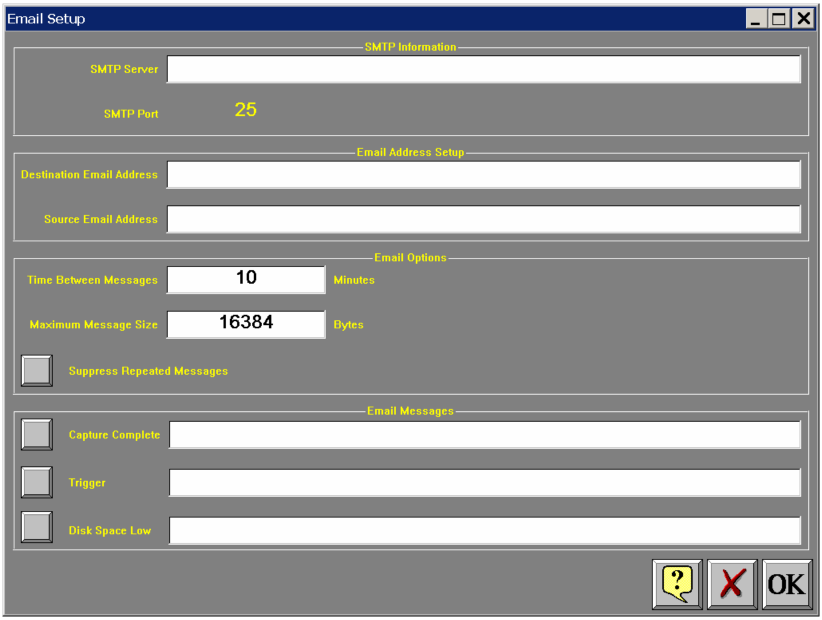
Setting up email alerts

Email alerts can be sent from Dash 8Xe units connected to a network. These messages inform email recipients of events including data capture completion, triggers, and low disk space conditions.

Use the following instructions to set up email alerts.

To set up email alerts:

1. From the Utilities screen, choose the Email Alerts button. The Email Setup window will open.



2. Choose the SMTP Server field. A keypad will appear.

   Enter the name of the SMTP mail server that will be used to send email messages. Then choose OK.

3. Set up destination and source email addresses.

   - Email addresses in the Destination Email Address field will receive an email message when the specified conditions occur.

     To add a destination address, choose the Destination Email Address field. A keypad will appear. Enter the email address of the intended recipient and choose OK.

     Multiple email addresses can be entered by placing a semicolon between each address. The maximum number of characters this field can hold is 256.

- The email address in the Source Email Address field will be listed as the sender in all email messages the Dash 8Xe generates.

  To add a source address, choose the Source Email Address field. A keypad will appear. Enter the source email address and choose OK.

4. Specify the email setup options.

   - The value in the Time Between Messages field indicates the amount of time allowed for messages to accumulate before they are sent in one email.

     To change the time between messages, choose the Time Between Messages field. A number pad will appear. Enter the amount of time for messages to "build up" and choose OK.

   - The value in the Maximum Message Size field indicates the maximum allowable size of the email messages that can be set up.

     To change the maximum size, choose the Maximum Message Size field. A number pad will appear. Enter the maximum message size and choose OK.

   - To suppress repeated messages, check the Suppress Repeated Messages checkbox. When this option is enabled, subsequent messages of the same type will not be sent.

5. Select the types of events that will result in a message, and enter the associated email messages.

   Capture Complete - The capture complete message occurs when a data capture completes successfully.

   Trigger - The trigger message occurs when a trigger condition occurs during a data capture.

   Disk Space Low - The disk space low message occurs when the space on the Dash 8Xe hard drive is limited.

   To enable a message for a particular event, check the appropriate check box. Then choose the associated text field. A keypad will appear.

   Enter the text message and choose OK.

6. To complete the email setup process, choose the OK button in the Email Setup window.

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 Dash 8Xe software is running.

Operating System
The Operating System utility is used to exit the Dash 8Xe software and access the Windows operating system.

Launch utility

The Launch utility is used to launch other applications while the Dash 8Xe software is running. Use the following instructions to access the launch utility.

Note: Astro-Med, Inc. does not guarantee that any application can be used while running the Dash 8Xe.

To use the launch utility:

1. From the Utilities screen, choose the Launch button. The Launch Application window will open.

2. Choose the Application field. Then enter the full path of a program's ".exe" file using the keypad and choose the OK button. 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 the OK button instead of entering the path again.

   Paths can be removed from this list by selecting the path and choosing the erase button directly below the application list.

Operating system

The Operating System utility is used to exit the Dash 8Xe software and access the operating system. Use the following instructions to access the Microsoft ® Windows operating system.

Note: As a shortcut, you can also access the operating system from Realtime, Scope, and Review mode by choosing File >> Exit from the menu bar.

To access the operating system:

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

2. Enter the system password and choose OK. The Dash 8Xe software will shut down, providing access to the operating system.

Note: The default password on the Dash 8Xe is “dash8x” (without the quotation marks). For information on changing the system password, refer to *Chapter 14: Service options*.

System utilities

The following system utilities are available in the utilities screen.

System Time

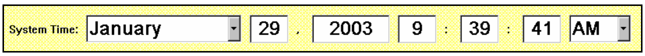
The System Time utility is used to set the clock on the operating system.

Setting the system time

Use the following instructions to set up the system time.

To set the system time:

1. Use the System Time fields and drop-down lists to enter the current date and time.



Chapter 16: Dash 8Xe offline software

This chapter provides information about the optional Dash 8Xe offline software.

Dash 8Xe offline overview

Dash 8Xe offline software is the Dash 8Xe program modified to run on a personal computer running Windows 98, 2000, ME, or XP. It provides most of the functionality of the standard Dash 8Xe program and can be used for a variety of purposes.

Some common uses of the Dash 8Xe offline software include:

Setup file creation - The offline software can be used to create view, control panel, signal, and global setup files. Once created, these files can be transferred to a Dash 8Xe using a USB storage device or network connection.

Data review - The offline software can be used to view data files captured using the Dash 8Xe. Data files can be reviewed with Dash 8Xe offline Review mode.

Note: Do not confuse Dash 8Xe offline software with AstroVIEWX.

Dash 8Xe offline is an optional program that emulates the Dash 8Xe environment on a personal computer. AstroVIEWX is a program that provides the capability to review data capture files on a personal computer. These are two distinct software products.

Installation

A wizard-based installation is used to install the Dash 8Xe offline software. The installation program should automatically run when the CD is inserted in a CD drive.

Note: If the program does not automatically run, launch the “setup.exe” file from the CD using Windows Explorer or the Run option from the Start menu (d:/setup.exe - where d: is the CD drive).

File transfer

Transferring data files from the Dash 8Xe to a computer running Dash 8Xe offline software can be accomplished with a USB storage device or the offline software FTP function.

Using a USB storage device

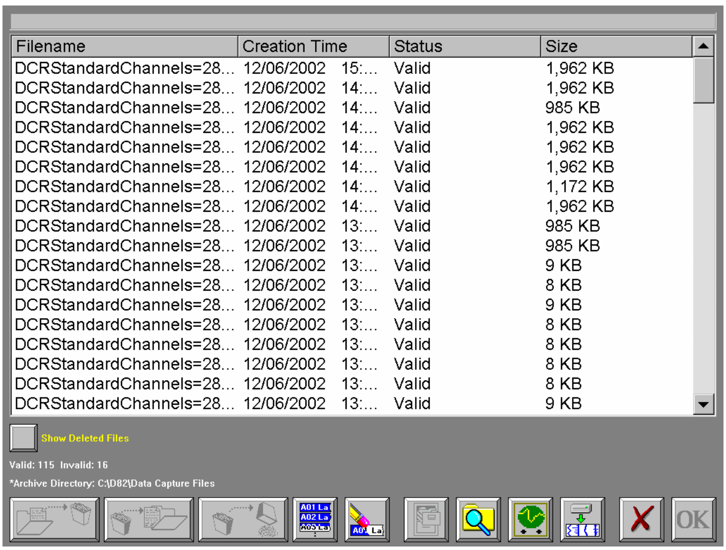
The typical transfer process involves copying files to the Dash 8Xe clipboard, pasting the files to a USB storage device, and retrieving files from the USB storage device with the Dash 8Xe offline software. These instructions can also be used to create backup copies of files for archive purposes.

To use a USB storage device:

1. Ensure that a USB storage device is properly connected to the Dash 8Xe.

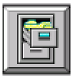
Note: For information on USB storage devices, refer to *Chapter 4: Optional hardware*.

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



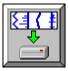
This window provides a list of all files stored on the data capture drive.

3. If the file to be transferred to the USB storage device has not been archived to the system drive already, select the file and click the archive file button.



4. Select the type of files to transfer to the USB storage device.

   - To copy archived data captures files from the system drive, choose the Archived Data Capture button.

   - To copy scope capture files from the system drive, choose the Scope Capture button.

   The Load Data Capture File or Load Scope Capture File will open.

5. Select the files to transfer and choose the Copy button.

   The selected files will be copied to the Dash 8Xe clipboard.

6. Choose the Disk Drive button corresponding to the drive used for the USB storage device. The contents of the drive will be displayed in the list.

7. Choose the Paste button.

   The files in the Dash 8Xe clipboard will be pasted to the selected drive location.

8. Remove the USB storage device from the Dash 8Xe and connect it to the computer running Dash 8Xe offline software.

The files on the USB storage device can be accessed using the Dash 8Xe offline software. To open a file from the device, choose the corresponding drive letter when using a load file window.

Using the Dash 8Xe offline FTP function

The Dash 8Xe offline software provides a built-in FTP feature that can be used to transfer files to/from a Dash 8Xe connected to the same network.

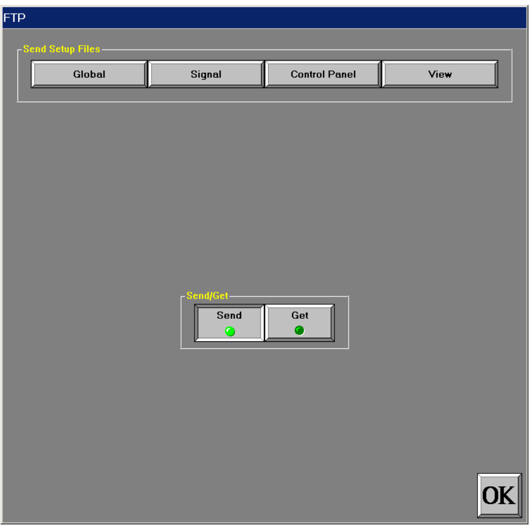
To send setup files to a Dash 8Xe:

1. Ensure that a Dash 8Xe and a computer running Dash 8Xe offline software are on and connected to the same network.

2. On the Dash 8Xe unit, choose Configuration >> Utilities to open the Utilities screen. Then choose FTP/Web >> Enabled to activate the Dash 8Xe FTP/Web function.

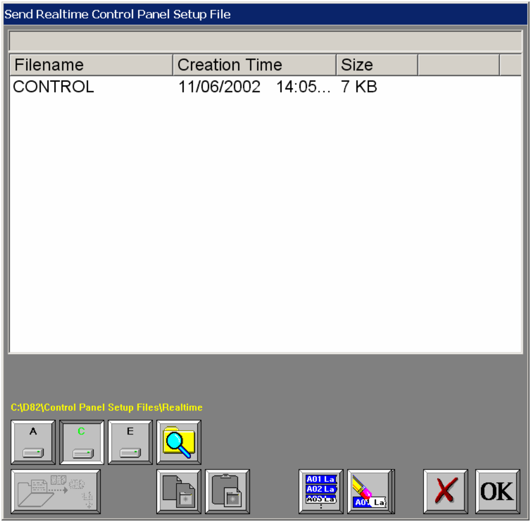
   A check mark will be displayed next to the Enabled option on the FTP/Web menu to indicate the FTP/Web functions are active.

3. In the Dash 8Xe offline software, choose Service >> Network >> FTP to open the FTP window.



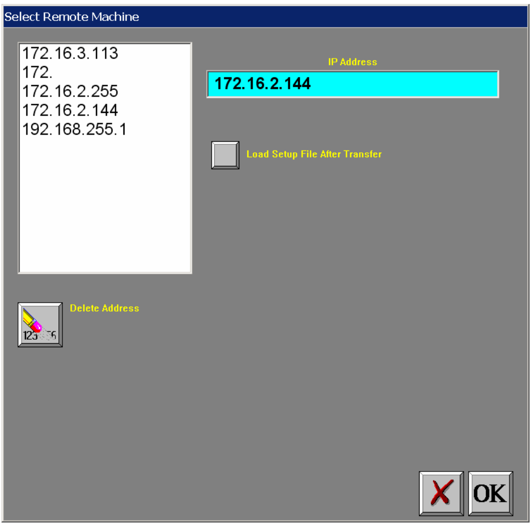
4. Choose the type of file to send to the Dash 8Xe and press the corresponding button. Global, signal, control panel, and view setup files can be transferred.

The Send File window will open.



5. Select the file to send, then choose the FTP button.

The Select Remote Machine window will open.

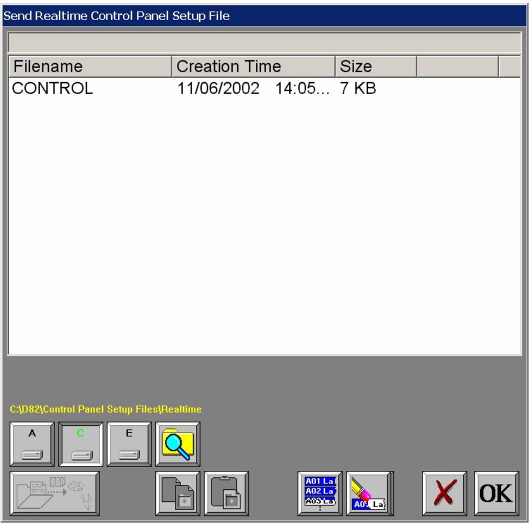


6. Choose the IP Address field to specify an IP address. A keypad will appear. Use the keypad to enter the IP address of the Dash 8Xe that will receive the transferred file, then choose the OK button.

   When an IP address is entered, it will be added to the list in this window. To choose the IP address in the future, select it from this list instead of entering it again. To remove an IP address from this list, select it and choose the Delete Address button.

7. Choose whether to load the setup file automatically after transfer. To load the file after transfer, check the “Load Setup File After Transfer” option.

8. Choose the OK button. The file will be transferred and the Send File window will re-appear. A “Transfer Complete” message will display on the bottom of the window.



9. Choose the OK button in the Send File window.

10. Choose the OK button in the FTP window.

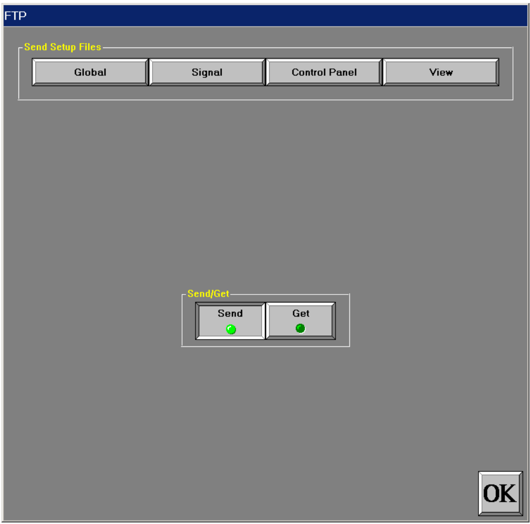
To get setup and data files from a Dash 8Xe:

1. Ensure that a Dash 8Xe and a computer running Dash 8Xe offline software are on and connected to the same network.

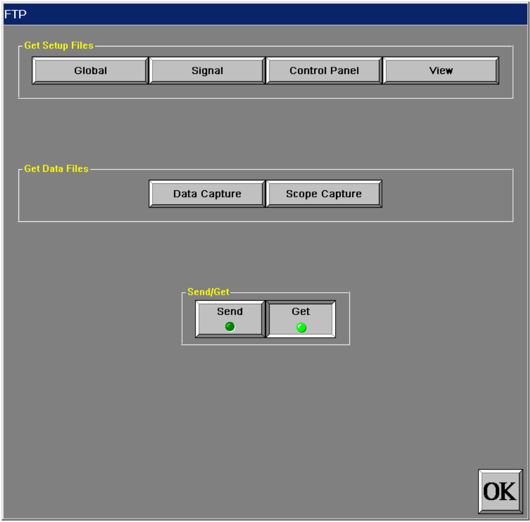
2. On the Dash 8Xe unit, choose Configuration >> Utilities to open the Utilities screen. Then choose FTP/Web >> Enabled to activate the Dash 8Xe FTP/Web function.

   A check mark will be displayed next to the Enabled option on the FTP/Web menu to indicate the FTP/Web functions are active.

3. In the Dash 8Xe offline software, choose Service >> Network >> FTP to open the FTP window.

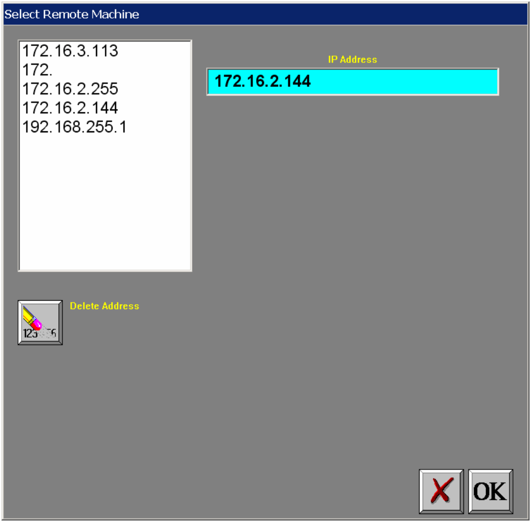


4. Choose the Get button. The Get Files options will appear in the window.



5. Choose the type of file to get from the Dash 8Xe and press the corresponding button. Global, signal, control panel, and view setup files can be transferred. Data capture and scope capture files can also be transferred.

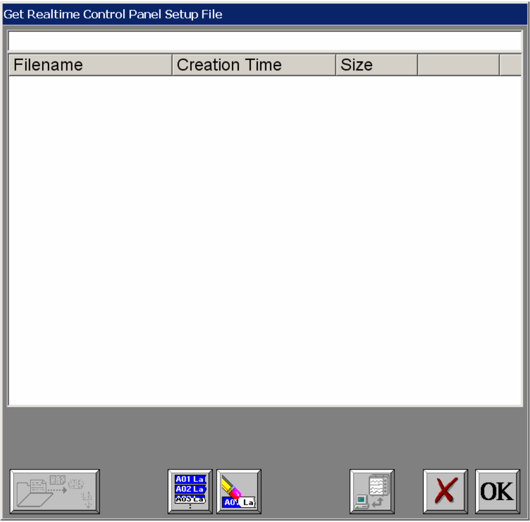
The Select Remote Machine window will open.



6. Choose the IP Address field to specify an IP address. A keypad will appear. Use the keypad to enter the IP address of the Dash 8Xe that contains the files that will be transferred, then choose the OK button.

   When an IP address is entered, it will be added to the list in this window. To choose the IP address in the future, select it from this list instead of entering it again. To remove an IP address from this list, select it and choose the Delete Address button.

7. Choose the OK button. The Get File window will open.



When this window opens, it will display a list of files on the Dash 8Xe. If a large number of files are present, this process may take a few moments to complete.

8. Select the file to transfer, then choose the FTP button.

The file will be transferred from the Dash 8Xe to the PC running Dash 8Xe offline. A progress bar will appear to indicate the status of the transfer. When the transfer is complete, the message "Transfer Complete" will appear on the bottom of the Get File window.

If necessary, the Cancel FTP button can be used to stop the file transfer.

9. Choose the OK button.

Selecting modules

The Dash 8Xe offline software provides the capability to select signal input modules that are installed in each slot.

This feature is helpful when creating setup files using the offline software. Setup files can be created with the appropriate modules installed in each slot, thus matching the configuration of the actual Dash 8Xe that will use the files.

Use the following instructions to select modules.

To select modules:

1. Choose Configuration >> Utilities to open the Utilities screen.
2. Choose Slot Configuration from the menu bar. A list of slots will appear.

   Select the slot to configure. A list of available modules will appear. Then select a module from the list.

   The selected module will be assigned to the selected slot.

Appendix A: Specifications

This appendix provides detailed specifications for the Dash 8Xe.

Color Display

Type..Active matrix color LCD (TFT)
Viewing Area ....15.0 inch (diagonal)
Resolution...1024 x 768
Touch ....Full screen, resistive

Signal Modules

Maximum Modules............................8
Maximum Waveforms.......................8
User Engineering Units ....Yes
Calibration..Semi-automated to external reference

Standard Event Inputs

Number of Inputs...............................8
Connector ....D-shell (9 pin)
Input Type..TTL with pull up (0 to 5 V)
Response...Detects if duration > 0.005 msec

Recording

Recording Method.............................Internal disk drive
Maximum Sample Rate......................200,000 samples/second each channel (all channels)
Minimum Sample Rate ....1 sample/minute
Dual Sample Rate ....Yes
Total Capacity.....................................Over 8 billion samples
Maximum Record...............................2 billion sample periods
Time Stamp ....Time and Date automatically saved with data
Timebase accuracy ....50 ppm
Header ....Information on units, range, sample rates, etc. saved with data
Events..All standard event inputs captured with waveforms
Trigger Point Loc................................Amount of pre and post trigger is user adjustable
Auto Re-Arm.......................................Allows automatic stacking of captures
Auto Playback ....No
Auto Review ....Yes
External sample rate...........................External TTL sample clock to 100+ KHz.

External sample clock duty cycle ..... 10 - 90%

Power

Input Voltage .... 115 / 230 VAC
Input Voltage Range .... 100 to 250 VAC
Frequency .... 50 / 60 Hz
Frequency Range .... 47 Hz - 63 Hz
Power Factor Correction.................... Yes
Power Consumption .... 150 W maximum (130 W typical)
External DC Input .... Regulated 24.0 V from AM DC-DC option only

Compliance

Safety... EN 61010-1:2001, UL 61010A-1, CSA C22.2 No. 1010.1-92
EMC .... FCC Part 15, Subpart B, Class A, EN 61326
Power Harmonics.............................. EN 61000-3-2:1995

Physical

Enclosure .... Aluminum
Dimensions (inches).......................... 12.6″ H x 16.7″ W x 5.8″ T (without feet)
Weight... 19 lbs (no modules)
22 lbs with 8 NDV1 modules
Module Size.. 1.25″ W x 3″ T x 4.5″ D
(approximate size not including front panel)

Environmental

Operating Temp .... 5 to 40 °C (40 to 105 °F)
Operating Humidity .... 10% to 90% non condensing

Interface

Ethernet 10/100/1000BaseT
VGA .... For displaying data on an external monitor
USB 2.0.. For external peripherals and file export
Link Ports .... For synchronizing data captures on up to eight systems

Optional Scope Card

Number of Channels.......................... 4
Input connector.................................. BNC
Input type .... Single ended, DC coupled
A/D resolution .... 12 bit
Input impedance .... 1.5 MΩ

Abs Max Input ....+/- 15V
Frequency Timebase ....50 ppm
Input voltage range ....+/- 1V and +/- 5V
Maximum Sample Rate......................10 MSamp/second per channel
Minimum Sample Rate ....5 KSamp/second per channel
Record Sizes ....Up to 9 choices.
1 KSamp / channel to 256 KSamp / channel
Timebase selections ....16 choices: 100 ns / div to 10 ms / div
Pretrigger Capable ....Yes, 1-7999 sample periods
Sync w/ data capture..........................Yes
Sync accuracy......................................1 second, 110 ms typical.

Note 1 : When pretrigger in use or records bigger than 16K in use, some captures may be missed.
Note 2 : Fastest 5 timebase selections use expansion (zoom)
Note 3 : When triggered captures are in use, some of the smaller record sizes are not available depending on sample rate.

Optional Chart Recorder

Print Width..8.5"
Chart speed ....1 mm/min to 50 mm/sec
Accuracy ....+ or - 2%
External chart drive............................Yes, via main unit utility port
Max ext. chart drive clock ....780 Hz (65 mm/sec)
Resolution...300 dpi x 300 dpi
Enclosure ....Aluminum
Dimensions (inches)...........................12.6" H x 16.7" W x 2.125" T (without feet)
Weight..9 lbs without paper, 11 lbs with paper
Input Voltage ....115 / 230 VAC
Input Voltage Range ....100 to 250 VAC
Frequency ....50 / 60 Hz
Frequency Range ....47 Hz - 63 Hz
Power Factor Correction....................Yes
Power Consumption ....130 W maximum

NDV1 - Non-isolated differential input module

Connector ....Guarded banana jacks (red/black)
Isolation ....No
Bandwidth (5V Attenuator) ....28 KHz (-3db)
Bandwidth (40V Attenuator) ....34 KHz (-3db)
Input...Differential, DC coupled
Off Ground Measurements ....Yes
Zero Suppression................................Yes, digital.
Measurement Ranges.........................+/- 5 V (5 VFS or 10 VFS w/ zero offset)

	+/- 40 V (40 VFS or 80 VFS w/ zero offset)
Max Rated Input	+/- 40 V
Max Transient Input	+/- 40 V
A/D	14 bit SAR
Anti-Aliasing Filter	4 pole Bessel
Input Resolution	5.621 mV/ADStep (40 V Attenuator)
	700.8 uV/ADStep (5 V Attenuator)
Accuracy (25°C)	0.25 % of attenuator (12.5 mV and 100 mV)
Cold Start Drift	Less than 0.5% of attenuator
Overshoot	Less than 1% of attenuator
Intrinsic Noise	0.1 % of attenuator
Min Input Impedance	425 K
CMR at 60 Hz	Better than -60 dB
Frequency Counter Capability	No

IHV1 - Isolated single ended high voltage input module

Connector	Guarded banana jacks (red/black)
Isolation	250 VRMS or DC, Cat II (iso-common to chassis and other iso-commons)
Bandwidth (100V attenuator)	39 KHz (-3db)
Bandwidth	40 KHz (-3db)
Input	Single-ended, DC coupled
Zero Suppression	Yes, digital.
Measurement Ranges	+/- 400 V (400 VFS or 800 VFS w/ zero offset)
	+/- 200 V (200 VFS or 400 VFS w/ zero offset)
	+/- 100 V (100 VFS or 200 VFS w/ zero offset)
	+/- 50 V (50 VFS or 100 VFS w/ zero offset)
	+/- 40 V (40 VFS or 80 VFS w/ zero offset)
	+/- 20 V (20 VFS or 40 VFS w/ zero offset)
	+/- 10 V (10 VFS or 20 VFS w/ zero offset)
	+/- 5 V (5 VFS or 10 VFS w/ zero offset)
Max Rated Input	+/- 250 Vrms or DC
Max Transient Input	+/- 800 V (not to exceed 250 Vrms)
A/D	16 bit SAR
Anti-Aliasing Filter	4 pole Bessel
Input Resolution	14.185 mV/ADStep (400 V Attenuator)
	7.093 mV/ADStep (200 V Attenuator)
	3.546 mV/ADStep (100 V Attenuator)
	1.773 mV/ADStep (50 V Attenuator)
	1.420 mV/ADStep (40 V Attenuator)
	698.309 uV/ADStep (20 V Attenuator)

	353.250 uV/ADStep (10 V Attenuator)
	175.538 uV/ADStep (5 V Attenuator)
Accuracy (25°C)	+/- 0.15% of Attenuator
Cold Start Drift	Less than 0.2% of attenuator
Overshoot	Less than 2% of attenuator
Intrinsic Noise	0.1 % of Attenuator
Min Input Impedance	1 M ohm
IMR at 60 Hz	> 75 dB
Frequency Counter Capability	Yes. Software selectable.
Frequency ctr range (menu)	2 – 80 KHz
Frequency ctr range (spec'd)	2 – 60 KHz
Frequency ctr accuracy (spec'd)	2 – 60 KHz + 0.05% of Measurement
Min input amplitude	20% of attenuator
Uni-Polar Frequency Detection	No

IBR1 - Isolated bridge input module

Connector	5 wire screw terminal
Isolation	250 VRMS or DC, Cat II (iso-common to chassis and other iso-commons)
Bandwidth	40 KHz (-3dB) 2V, 200 mV and 50 mV Attenuators
	38 KHz (-3dB) 20 mV Attenuator
Input	Differential, DC coupled
Zero Suppression	Yes, digital.
Absolute Max Input	+/- 12V (either input referenced to iso-common)
Measurement Ranges	+/- 2 V
	+/- 200 mV
	+/- 50 mV
	+/- 20 mV
Max Transient Input	60 V
A/D	16 bit SAR
Anti-Aliasing Filter	4 pole Bessel
Input Resolution	71.8 uV (2 V Attenuator)
	7.13 uV (200 mV Attenuator)
	1.78 uV (50 mV Attenuator)
	725.4 nV (20 mV Attenuator)
Accuracy (25°C)	+/- 0.15 % of attenuator
Cold Start Drift	Less than 0.2% of attenuator
Overshoot	Less than 2% of attenuator
Intrinsic Noise (pk-pk)	0.1 % of attenuator (2V, 200 mV Attenuators)
	0.2 % of attenuator (50 mV Attenuator)

	0.2 % of attenuator (20 mV Attenuator)
Min Input Impedance	250 K (balanced to signal common)
CMR at 60 Hz	> 90 dB
Excitation	DC adjustable to 10 V @ 30 mA
Excitation Range	0.1 to 10.0 V
Excitation Accuracy	0.05 V
Auto Balance	Yes (limited by maximum span)
Frequency Counter Capability	Yes. Software selectable.
Frequency ctr range (menu)	2 – 80 KHz
Frequency ctr range (spec'd)	2 – 60 KHz (200 mV and 2V attenuators)
Frequency ctr range (spec'd)	2 – 40 KHz (20 mV and 50 mV attenuators)
Frequency ctr accuracy	± 0.05% of Measurement + .002 Hz
Min input amplitude	25% of attenuator
Uni-Polar Frequency Detection	No. (Subject to Change)
Bridge Completion Resistors:	Yes, internal. Includes location for 3-wire quarter bridge.

IHV2 - Isolated differential very high voltage module

Connector	Guarded Banana Jacks
Isolation	600 VRMS or DC Cat II (either input to chassis and across inputs) 300 VRMS or DC Cat III
Bandwidth	34 KHz (-3db)
Input Type	Isolated differential, balanced to internal common.
Coupling	DC
Zero Suppression	Yes, digital.
Measurement Ranges	+/- 1000 V +/- 800 V +/- 400 V +/- 200 V +/- 100 V
Max Rated Input	600 Vrms or DC
Max Transient Input	+/- 1000 V (not to exceed 600 Vrms)
A/D	16 bit SAR
Anti-Aliasing Filter	4 pole Bessel
Input Resolution	34.269 mV/ADStep (1000 V Attenuator) 27.630 mV/ADStep (800 V Attenuator) 14.209 mV/ADStep (400 V Attenuator) 7.026 mV/ADStep (200 V Attenuator) 3.484 mV/ADStep (100 V Attenuator)
Accuracy (25°C)	+/- 0.15% Attenuator

Cold Start Drift ....Less than 0.2% of attenuator
Overshoot ....Less than 1% of attenuator
Intrinsic Noise (pk-pk).......................0.15 % Attenuator
Min Input Impedance ....4M (differential)
IMR...> 70 dB @ 60 Hz
Frequency Counter Capability..........Yes. Software selectable.
Frequency ctr range (menu).............2 – 80 KHz
Frequency ctr range (spec'd)............2 – 40 KHz
Frequency ctr accuracy ....± 0.05% of Measurement
Min input amplitude..........................20% of attenuator
Uni-Polar Frequency Detection ....No.

ITCU - Universal thermocouple input module

Connector ....Type U miniature thermocouple
Isolation ....250 VRMS or DC, Cat II.
Bandwidth..5 Hz update rate (TC sampled at 2.5 Hz)
Absolute Max Input ....+/- 10V
Specified Range Type J: ....-210 to 1200 °C
Specified Range Type K:....................-200 to 1372 °C
Specified Range Type E: ....-200 to 1000 °C
Specified Range Type T: ....-200 to 400 °C
Specified Range Type N: ....-200 to 1300 °C
Specified Range Type B: ....600 to 1820 °C (250 to 1820 on menu)
Specified Range Type R:....................-20 to 1768 °C
Specified Range Type S:.....................-20 to 1768 °C
A/D ....24 bit Sigma Delta
Resolution...0.01 °C
Thermocouple types...........................J,K,E,T,N,B,R,S
Accuracy (25°C) J...............................+/- 0.8 °C
Accuracy (25°C) K ....+/- 1.0 °C
Accuracy (25°C) E..............................+/- 0.8 °C
Accuracy (25°C) T (- 100 to 400)........+/- 1.0 °C
Accuracy (25°C) T (- 200 to - 100) .....+/- 2.0 °C
Accuracy (25°C) N.............................+/- 0.8 °C
Accuracy (25°C) B..............................+/- 2.0 °C
Accuracy (25°C) R ....+/- 2.0 °C
Accuracy (25°C) S ....+/- 2.0 °C
Accuracy (25°C) 100 mV....................+/- 0.01% of attenuator
Cold Junction Compensation............Yes
Compensation Error...........................Included in above accuracy specification
Intrinsic Noise ....(p-pk, J,K,E,T,N) < 0.07 °C

Intrinsic Noise ....(pk-pk, B,R,S) < 0.5 °C
IMR...> 110 dB @ DC
Linearization ....NIST ITS-90
Frequency Counter Capability ....No

Note: Specified accuracy does not include probe errors.

IRTD - Isolated resistance temperature detector input module

Connector ....4 wire screw terminal
Isolation ....Yes
Bandwidth...2.5 Hz update rate
Absolute Max Input ....+/- 10V
Measurement Ranges:........................Pt100(385) -200 to 800 °C
Pt100(3916) -200 to 630 °C (-200 to 800 on menu)
Pt100(3926) -200 to 630 °C (-200 to 800 on menu)
resistance 0 to 400 Ω (0 to 450 Ω on menu)
Minimum full scale ....10 °C
A/D...24 bit Sigma Delta
Anti-Aliasing Filter ....Inherent
Resolution..0.01 °C
Accuracy (25°C).................................0.05% of measurement + 0.2 °C
Accuracy (resistance 25°C)...............0.02% of measurement + 0.05 Ω
Cold Start Drift ....< 0.3 °C
Intrinsic Noise (pk-pk)......................< 0.03 °C
Intrinsic Noise (resistance pk-pk).....< 0.01 Ω
Linearization ....Yes
Supported RTD Probe types ....Pt 100 - 385 (DIN 43760, IEC751 and ASTM 1137)
Pt 100 - 3916 (JIS C1604)
Pt 100 - 3926 (reference grade)
Excitation..1 mA constant current
Frequency Counter Capability ....No

Note 1: Specified accuracy does not include probe errors or 2-wire connection wire errors.
Note 2: 4-wire connection RTDs supported when R45 and R10 removed from PCB.

LIVM - Low Impedance Voltage Mode amplifier input module

A/D...16 bit SAR
Anti-Aliasing Filter ....4 pole Bessel
Isolation ....Yes, 250 Vrms or DC to chassis common. *
Connector ....Isolated BNC
Bandwidth..6 Hz to 30 KHz (-3db)
Input...AC coupled transducers with integral electronics

Coupling Time Constant ....300 ms (+/- 20%)
Zero Suppression...............................N/A
Absolute Max Input ....+/- 40V
Measurement Ranges........................+/- 5000 mV
+/- 500 mV
+/- 200 mV
+/- 50 mV
Max Transient Input..........................60 V
Input Resolution ....178.035 μV/ADStep (5000 mV Attenuator)
18.098 μV/ADStep (500 mV Attenuator)
7.203 μV/ADStep (200 mV Attenuator)
1.78 μV/ADStep (50 mV Attenuator)
Accuracy LIVM mode (25°C)...........2 % of attenuator
Accuracy Diff mode (25°C) ....0.15 % of attenuator
Cold Start Drift ....Less than 2% of attenuator
Intrinsic Noise - 50 mV att ....2% of attenuator (pk-pk)
Intrinsic Noise - other atten's ....1% of attenuator (pk-pk)
Min Input Impedance ....25 KΩ (differential)
IMR at 60 Hz - 5000 mV att ....80 dB
IMR at 60 Hz - other atten's..............90 dB
Excitation ....4 mA DC current
Excitation Accuracy...........................20% (25 ºC)
Excitation Compliance Voltage ....18V (23V open circuit voltage)
Excitation Protection ....Short Circuit Protected
Frequency Counter Capability..........No

Note : Isolation limited to 30 Vrms or 60V DC when non-insulated mating BNC connector used since hazardous voltage would otherwise be accessible.

IDCV - Isolated high accuracy, wide dynamic range DC Voltmeter input module

Connector ....Guarded banana jacks (red/black)
Isolation ....250 VRMS or DC, Cat II. (iso-common to chassis and other iso-commons)
Input Type ....Single-ended, Isolated
Input Coupling ....DC
Minimum Input Impedance.............> 1 MΩ
Zero Suppression...............................Yes, digital.
Bandwidth ....6 Hz update rate (.65 Hz 3dB point typical)
Max Rated Input ....- 5 V to +130V
Max Transient Input..........................+/- 250 V
A/D ....24 bit Sigma Delta
Anti-Aliasing Filter ....Inherent
Resolution...7.6 μV/ADstep

Accuracy (25°C) .... +/- 55 ppm of reading + .7 mV
Intrinsic Noise (pk-pk) .... < 1.0 mV (.7 mV typ)
IMR (DC) .... > 110 dB
Frequency Counter Capability .... No
DSP Filtering Capability .... None

Note: Minimum full scale span is .1 VFS.

IDCV1 - Isolated high accuracy, wide dynamic range DC Voltmeter with 160V attenuator.

Connector .... Guarded banana jacks (red/black)
Isolation 250 VRMS or DC, Cat II. (iso-common to chassis and other iso-commons)
Input Type .... Single-ended, Isolated
Input Coupling .... DC
Minimum Input Impedance .... > 1 MΩ
Zero Suppression .... Yes, digital.
Bandwidth .... 6 Hz update rate (.65 Hz 3dB point typical)
Max Rated Input .... - 5 V to +160V
Max Transient Input .... +/- 250 V
A/D .... 24 bit Sigma Delta
Anti-Aliasing Filter .... Inherent
Resolution .... 9.6 μV/ADstep
Accuracy (25°C) .... +/- 60 ppm of reading + .8 mV
Intrinsic Noise .... (pk-pk) < 1.0 mV (.7 mV typ)
IMR (DC) .... > 110 dB
Frequency Counter Capability .... No
DSP Filtering Capability .... None

Note: .... Minimum full scale span is .1 VFS.

IHV3 - Isolated single ended high voltage module

Connector .... Guarded banana jacks (red/black)
Isolation .... 250 VRMS or DC, Cat II. (iso-common to chassis and other iso-commons)
Bandwidth .... 22 KHz (-3db) (350 through 40 V Atts)
20 KHz (-3db) (20 V Atts)
17 KHz (-3db) (10000 through 500 mV Atts)
12 KHz (-3db) (100 through 50 mV Atts)
Input .... Single-ended, DC coupled
Zero Suppression .... Yes. digital
Measurement Ranges .... +/- 350 V (350 VFS or 700 VFS w/ 0 offset)
+/- 200 V (200 VFS or 400 VFS w/ 0 offset)

+/- 100 V (100 VFS or 200 VFS w/ 0 offset)
+/- 50 V (50 VFS or 100 VFS w/ 0 offset)
+/- 40 V (40 VFS or 80 VFS w/ 0 offset)
+/- 20 V (20 VFS or 40 VFS w/ 0 offset)
+/- 10000 mV (10000 mVFS or 20000 mVFS w/ 0 offset)
+/- 5000 mV (5000 mVFS or 10000 mVFS w/ 0 offset)
+/- 1000 mV (1000 mVFS or 5000 mVFS w/ 0 offset)
+/- 500 mV (500 mVFS or 1000 mVFS w/ 0 offset)
+/- 100 mV (100 mVFS or 200 mVFS w/ 0 offset)
+/- 50 mV (50 mVFS or 100 mVFS w/ 0 offset)

Max Rated Input.................................+/- 250 Vrms or DC
Max Transient Input...........................+/- 800 V (not to exceed 250 Vrms)
A/D...16 bit SAR
Anti-Aliasing Filter ....4 pole Bessel
Input Resolution.................................12.151 mV/ADStep (350 V Attenuator)
6.944 mV/ADStep (200 V Attenuator)
3.472 mV/ADStep (100 V Attenuator)
1.736 mV/ADStep 50 V Attenuator)
1.389 mV/ADStep (40 V Attenuator)
694.37 uV/ADStep (20 V Attenuator)
359.671 uV/ADStep (10000 mV Attenuator)
173.593 uV/ADStep (5000 mV Attenuator)
173.593 uV/ADStep (1000 mV Attenuator)
34.719 uV/ADStep (500 mV Attenuator)
3.597 uV/ADStep (100 mV Attenuator)
1.798 uV/ADStep (50 mV Attenuator)
Accuracy (25°C).................................+/- 0.15% of Attenuator
Cold Start Drift ....Less than 0.5% of attenuator
Overshoot..Less than 1% of attenuator (1000mV thru 350V atts)
Less than 2% of attenuator (500 mV attenuator)
Less than 10% of attenuator (100 mV attenuator)
Less than 15% of attenuator (50 mV attenuator)
Intrinsic Noise (pk-pk).......................< 0.2 % of Attenuator (Volt Attenuators)
< 0.2 % of Attenuator + 2 mV (mV Attenuators)
Min Input Impedance ....1 M ohm *
IMR at 60 Hz (Volt Attenuators) ......> 50 dB
IMR at 60 Hz (mVolt Atts) ....> 60 dB
Frequency Counter Capability..........Yes. Software selectable.
Frequency ctr range (menu)..............2 – 50 KHz
Frequency ctr range (spec'd).............2 – Attenuator bandwidth + 5 KHz
Frequency ctr min amplitude ....50% of att (mV atts) 25% of att (V atts)

Frequency ctr accuracy ....+ 0.1% of Measurement + 1 Hz (50 mV / 100 mV atts)
Frequency ctr accuracy ....+ 0.05% of Measurement (other attenuators)
Uni-Polar Frequency Detection ....No.

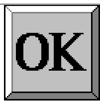
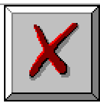
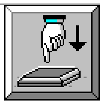
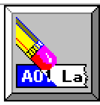
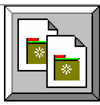
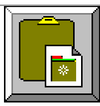
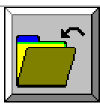
* Note: While inputs are protected as stated above, voltages applied to the input that significantly exceed the rating of the mV attenuators will result in up to a 12 % lower input impedance during the fault condition.

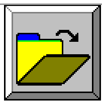
Appendix B: Button definitions

This appendix provides detailed information about the Dash 8Xe icon buttons.

System buttons

The following icon buttons are commonly used in Dash 8Xe windows, regardless of the mode of operation.

	**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.
	**Copy** copies the selected items (or characteristics of selected items) to the clipboard.
	**Paste** pastes the clipboard contents to the selected location.
	**Save** saves files.

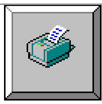
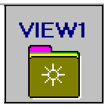
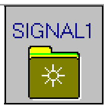
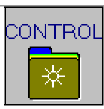
	**Load** loads files.

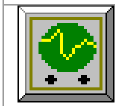
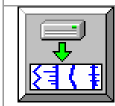
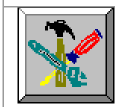
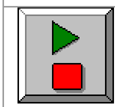
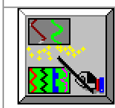
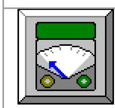
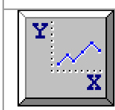
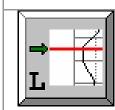
Realtime mode control panel

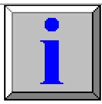
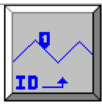
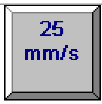
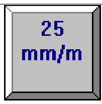
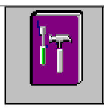
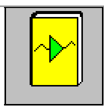
This section describes all icon buttons that can be added to the Realtime mode control panel.

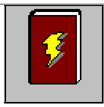
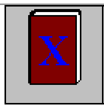
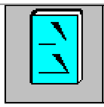
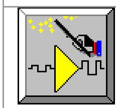
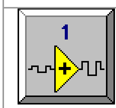
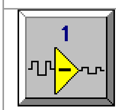
Note: At the bottom of each description, a selection path is noted. This information appears in an *italic typeface*.

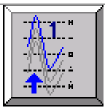
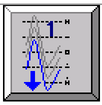
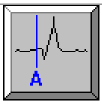
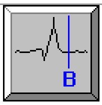
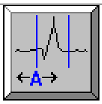
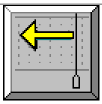
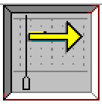
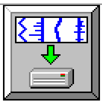
While in the Control Panel Setup window for Realtime mode, use this selection path to add the associated icon button to the control panel.

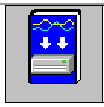
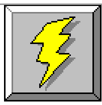
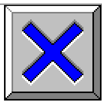
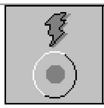
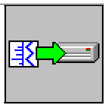
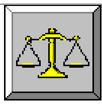
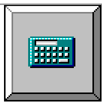
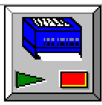
	**Print Screen** prints the contents of the display. This function is available only when the optional printer is installed. *File >> Print Screen*
	**Load View File** loads the specified view file. In this example, a view file named VIEW1 will be loaded when this button is pressed. *File >> View*
	**Load Signal File** loads the specified signal file. In this example, a signal file named SIGNAL1 will be loaded when this button is pressed. *File >> Signals*
	**Load Control Panel File** loads the specified control panel file. In this example, a control panel file named CONTROL will be loaded when this button is pressed. *File >> Control Panel*

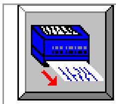
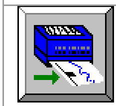
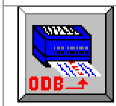
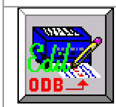
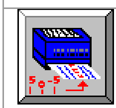
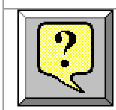
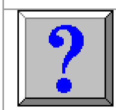
	**Load Global File** loads the specified global file. In this example, a global file named GLOBAL1 will be loaded when this button is pressed. *File >> Global*
	**Go to Scope Mode** launches Scope mode. *Configuration >> Goto Scope*
	**Go to Review Mode** launches Review mode. *Configuration >> Goto Review*
	**Go to Utilities** launches the Utilities screen. *Configuration >> Goto Utilities*
	**Run/Halt Monitor and Chart** starts and stops the on-screen chart from running. *View >> Freeze Display*
	**Monitor/Chart Wizard** opens the Chart/Display wizard, which is used to set up signal views and layouts. *View >> Monitor/Chart Wizard*
	**Show/Hide Meter** displays and hides the channel meter, which provides a numeric data logger of signal values updated once per second. *View >> Meter*
	**Show/Hide XY** displays and hides the Realtime mode XY window. *View >> Realtime XY*
	**Print Full Page Mark** prints a full page mark across the scrolling waveform display area. *View >> Full Page Mark*

	**Show/Hide Channel Information** displays and hides the Channel Information window, which is used in conjunction with cursors to measure signals. *View >> Channel Information*
	**Print Signal IDs** prints an identification label for each signal in the waveform display area. *View >> Print IDs*
	**Set Chart Speed (mm/s)** changes the speed of the on-screen and printed charts. In this example, the chart speed will be changed to 25 mm/s when the button is pressed. Custom speed icons can be configured during the control panel setup process. *Setup >> mm/s*
	**Set Chart Speed (mm/m)** changes the speed of the on-screen and printed charts. In this example, the chart speed will be changed to 25 mm/m when the button is pressed. Custom speed icons can be configured during the control panel setup process. *Setup >> mm/m*
	**Realtime Setup** opens the Realtime Setup window, which is used to define Realtime mode setup options. *Setup >> Realtime Setup*
	**Analog Channel Setup** opens the Analog Channel Setup window, which is used to set up channel amplifiers and filters. *Setup >> Analog Channel Setup*

	**Trigger Setup** opens the Trigger Setup Menu window, which is used to set up data capture triggers. *Setup >> Trigger Setup*
	**Abort Setup** opens the Abort Setup window, which is used to set up data capture aborts. *Setup >> Abort Setup*
	**Pen Up/Down Control** opens the Pen Up/Down Control window, which is used to raise and lower on-screen pens. *Setup >> Pen Up/Down Control*
	**Auto Range** opens the Auto Range window, which is used to specify channel spans based on a signal input. *Setup >> Auto Range*
	**Amp Up** increases the span of the indicated waveform. Pressing this button increases the value. Pressing and holding the button increases the value quickly. In this example, the span of Waveform 1 will be affected. *Setup >> Amp Up*
	**Amp Down** decreases the span of the indicated waveform. Pressing this button decreases the value. Pressing and holding the button decreases the value quickly. In this example, the span of Waveform 1 will be affected. *Setup >> Amp Down*

	**Offset Up** moves the indicated waveform up as much as 50% from the default location on the grid. Pressing this button decreases the value. Pressing and holding the button decreases the value quickly. In this example, the span of Waveform 1 will be affected.
	**Offset Down** moves the indicated waveform down as much as 50% from the default location on the grid. Pressing this button decreases the value. Pressing and holding the button decreases the value quickly. In this example, the span of Waveform 1 will be affected.
	**Show/Hide Cursor A** displays and hides cursor A. *Cursors >> Show/Hide Cursor A*
	**Show/Hide Cursor B** displays and hides cursor B. *Cursors >> Show/Hide Cursor B*
	**Set Active Cursor** changes the active cursor between A, B, or A & B. *Cursors >> Set Active Cursor*
	**Move Cursor Left** moves the active cursor(s) to the left each time the button is pressed. *Cursors >> Move Cursor Left*
	**Move Cursor Right** moves the active cursor(s) to the right each time the button is pressed. *Cursors >> Move Cursor Right*
	**Arm Data Capture** arms (initiates) a data capture using the configured data capture setup options. *Capture >> Arm*

	**Capture Setup** opens the Capture Setup window, which is used to configure data capture setup options. *Capture >> Capture Setup*
	**Manual Trigger** triggers a data capture manually. *Capture >> Manual Trigger*
	**Manual Abort** cancels the data capture in progress. *Capture >> Manual Abort*
	**Trigger Indicator** indicates when a trigger occurs by displaying a yellow-colored circle. *Capture >> Trigger Indicator*
	**Capture Indicator** indicates when a data capture is in progress by illuminating. *Capture >> Capture Indicator*
	**Calibration** opens the Calibration window, which is used to calibrate signal inputs. *Service >> Calibration*
	**Calculator** launches the Calculator application, which is part of the Windows operating system. *Service >> Calculator*
	**Chart Run/Halt** starts and stops the printed chart from running. This function is available only when the optional printer is installed. *Chart >> Chart Run/Halt*

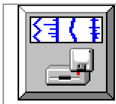
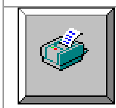
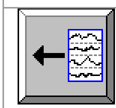
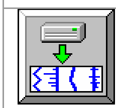
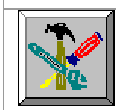
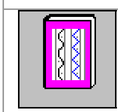
	**Chart Feed** feeds paper in the thermal printer. This function is available only when the optional printer is installed. *Chart >> Feed*
	**Chart Mark** prints a full page mark across the chart. This function is available only when the optional printer is installed. *Chart >> Chart Mark*
	**Print On-Demand Buffer** prints the demand buffer, a string of text stored in memory, on the chart. This function is available only when the optional printer is installed. *Chart >> Print Demand Buffer*
	**Edit On-Demand Buffer** brings up the on-screen keyboard used to change the contents of the On-Demand Buffer. This function is available only when the optional printer is installed. *Chart >> Edit Demand Buffer*
	**Print Labels** prints channel labels on the chart. This function is available only when the optional printer is installed. *Chart >> Print Labels*
	**Context Help** provides on-screen help based on the mode or window in use. *Help >> Context Help*
	**Icon Help** provides on-screen help for the purpose of identifying icon buttons. *Help >> Icon Help*

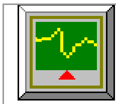
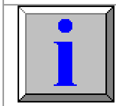
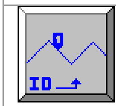
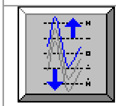
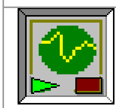
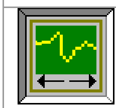
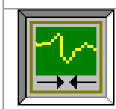
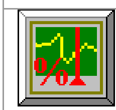
Scope mode control panel

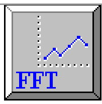
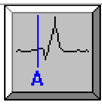
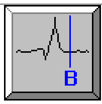
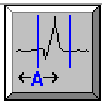
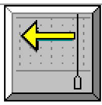
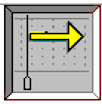
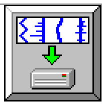
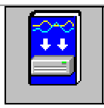
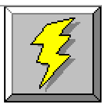
This section describes all icon buttons that can be added to the Scope mode control panel.

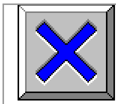
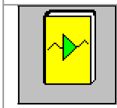
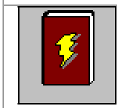
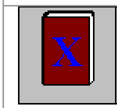
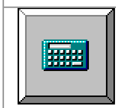
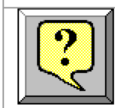
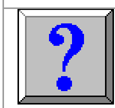
Note: At the bottom of each description, a selection path is noted. This information appears in an *italic typeface*.

While in the Control Panel Setup window for Scope mode, use this selection path to add the associated icon button to the control panel.

	**Archive** 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*
	**Print Screen** prints the contents of the display. This function is available only when the optional printer is installed. *File >> Print*
	**Go to Realtime Mode** launches Realtime mode. *Configuration >> Goto Realtime*
	**Go to Review Mode** launches Review mode. *Configuration >> Goto Review*
	**Go to Utilities** launches the Utilities screen. *Configuration >> Goto Utilities*
	**Add/Remove Channels** opens the Chart/Display wizard, which is used to set up signal views. *View >> Add/Remove Channels*

	**Show/Hide Trigger Line** shows and hides the trigger line in a scope capture. *View >> Show/Hide Trigger Line*
	**Show/Hide Channel Information** displays and hides the Channel Information window, which is used in conjunction with cursors to measure signals. *View >> Show Channel Information*
	**Show/Hide Channel IDs** displays an identification label for each signal in the scope waveform display area. *View >> Show/Hide Channel IDs*
	**Channel Setup** opens the Scope Channel Setup window, which is used to set up display characteristics of signals in Scope Mode. *View >> Channel Setup*
	**Scope Arm/Abort** arms and aborts scope captures. *Acquire >> Arm/Abort*
	**Increase Timebase** increases the timebase used for scope captures. *Acquire >> Timebase Up*
	**Decrease Timebase** decreases the timebase used for scope captures. *Acquire >> Timebase Down*
	**Scope 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. *Acquire >> Set Pre-Trigger Percent*

	**FFT** opens the Fourier Transform window. *Analysis >> Open Fourier Transform Window*
	**Show/Hide Cursor A** displays and hides cursor A. *Cursors >> Show/Hide Cursor A*
	**Show/Hide Cursor B** displays and hides cursor B. *Cursors >> Show/Hide Cursor B*
	**Set Active Cursor** changes the active cursor between A, B, or A & B. *Cursors >> Change Active Cursor*
	**Move Cursor Left** moves the active cursor(s) to the left each time the button is pressed. *Cursors >> Move Cursor Left*
	**Move Cursor Right** moves the active cursor(s) to the right each time the button is pressed. *Cursors >> Move Cursor Right*
	**Arm Data Capture** arms (initiates) a data capture using the configured data capture setup options. *Capture >> Arm*
	**Capture Setup** opens the Capture Setup window, which is used to configure data capture setup options. *Capture >> Capture Setup*
	**Manual Trigger** triggers a data capture manually. *Capture >> Trigger*

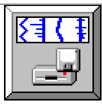
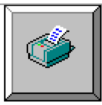
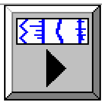
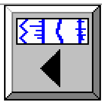
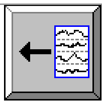
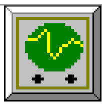
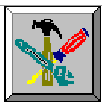
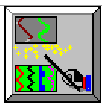
	**Manual Abort** cancels the data capture in progress. *Capture >> Abort Data Capture*
	**Amp Setup** opens the Analog Channel Setup window, which is used to set up channel amplifiers and filters. *Setup >> Amp Setup*
	**Trigger Setup** opens the Trigger Setup Menu window, which is used to set up data capture triggers. *Setup >> Trigger Setup*
	**Abort Setup** opens the Abort Setup window, which is used to set up data capture aborts. *Setup >> Abort Setup*
	**Calculator** launches the Calculator application, which is part of the Windows operating system. *Service >> Calculator*
	**Context Help** provides on-screen help based on the mode or window in use. *Help >> Context Help*
	**Icon Help** provides on-screen help for the purpose of identifying icon buttons. *Help >> Icon Help*

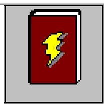
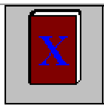
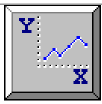
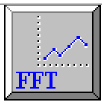
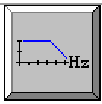
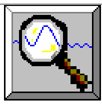
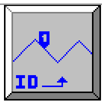
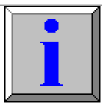
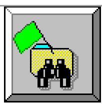
Review mode control panel

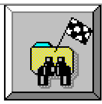
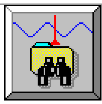
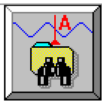
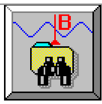
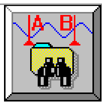
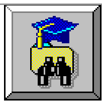
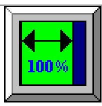
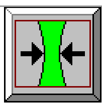
This section describes all icon buttons that can be added to the Review mode control panel.

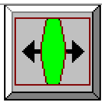
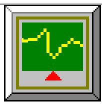
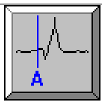
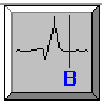
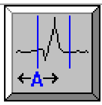
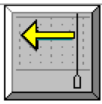
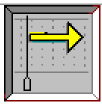
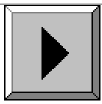
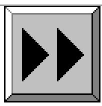
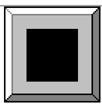
Note: At the bottom of each description, a selection path is noted. This information appears in an *italic typeface*.

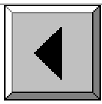
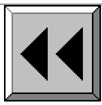
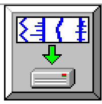
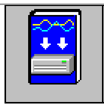
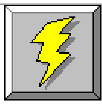
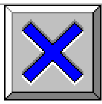
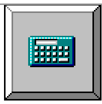
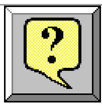
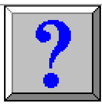
While in the Control Panel Setup window for Review mode, use this selection path to add the associated icon button to the control panel.

	**Archive** archives an entire scope 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 the optional printer is installed. *File >> Print*
	**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*
	**Go to Realtime Mode** launches Realtime mode. *Configuration >> Goto Realtime*
	**Go to Scope Mode** launches Scope mode. *Configuration >> Goto Scope*
	**Go to Utilities** launches the Utilities screen. *Configuration >> Goto Utilities*
	**Display Wizard** opens the Chart/Display wizard, which is used to set up signal views and layouts. *View >> Display Wizard*

	**Trigger Setup** opens the Trigger Setup Menu window, which is used to set up data capture triggers. *View >> Trigger Setup*
	**Abort Setup** opens the Abort Setup window, which is used to set up data capture aborts. *View >> Abort Setup*
	**Show/Hide XY** displays and hides the XY window. *Analysis >> Open XY Window*
	**FFT** opens the Fourier Transform window. *Analysis >> Open Fourier Transform Window*
	**Open Filter Window** opens the Filter window. *Analysis >> Open Filter Window*
	**Open Zoom Window** opens the Zoom window. *Analysis >> Open Zoom Window*
	**Show Signal IDs** displays an identification label for each signal in the waveform display area. *Analysis >> Show Channel IDs*
	**Show/Hide Channel Information** displays and hides the Channel Information window, which is used in conjunction with cursors to measure signals. *Analysis >> Show Channel Information*
	**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*
	**Goto 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 + B** compresses or expands the display to the portion of the file between cursors A and B. *Display >> Goto Cursors A + B*
	**Advanced Search** opens the Advanced Search window, which is used to navigate through the file using specific search criteria. *Display >> Advanced Search*
	**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*
	**Show/Hide Trigger Line** shows and hides the trigger line. *Display >> Show/Hide Trigger Line*
	**Show/Hide Cursor A** displays and hides cursor A. *Cursors >> Show/Hide Cursor A*
	**Show/Hide Cursor B** displays and hides cursor B. *Cursors >> Show/Hide Cursor B*
	**Set Active Cursor** changes the active cursor between A, B, or A & B. *Cursors >> Change Active Cursor*
	**Move Cursor Left** moves the active cursor(s) to the left each time the button is pressed. *Cursors >> Move Cursor Left*
	**Move Cursor Right** moves the active cursor(s) to the right each time the button is pressed. *Cursors >> Move Cursor Right*
	**Scroll Forward** scrolls the chart forward. *Scroll >> Scroll Forward*
	**Fast Forward** scrolls the chart forward quickly. *Scroll >> Fast Forward*
	**Scroll Stop** stops the chart from scrolling. *Scroll >> Scroll Stop*

	**Scroll Backward** scrolls the chart backward. *Scroll >> Scroll Backward*
	**Rewind** scrolls the chart backward quickly. *Scroll >> Rewind*
	**Arm Data Capture** arms (initiates) a data capture using the configured data capture setup options. *Capture >> Arm*
	**Capture Setup** opens the Capture Setup window, which is used to configure data capture setup options. *Capture >> Capture Setup*
	**Manual Trigger** triggers a data capture manually. *Capture >> Trigger*
	**Manual Abort** cancels the data capture in progress. *Capture >> Abort Data Capture*
	**Calculator** launches the Calculator application, which is part of the Windows operating system. *Service >> Calculator*
	**Context Help** provides on-screen help based on the mode or window in use. *Help >> Context Help*
	**Icon Help** provides on-screen help for the purpose of identifying icon buttons. *Help >> Icon Help*

Index