DASH 18X®

Operations Manual

22834-521
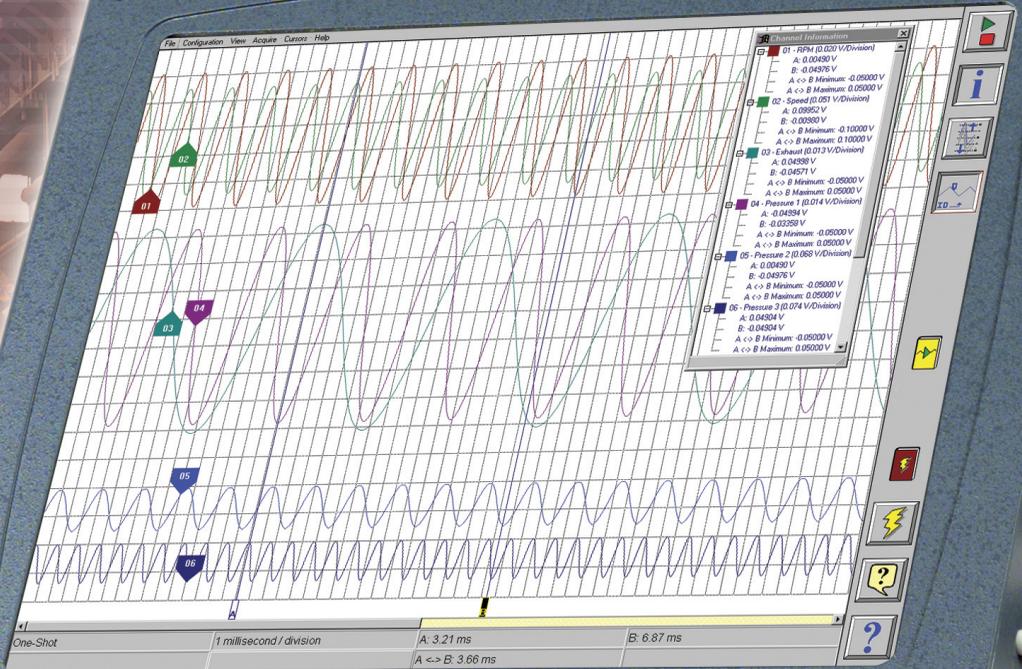
Astro-Med, Inc.

DASH-18X®

AI® **Astro-Med, Inc**
TEST & MEASUREMENT PRODUCT GROUP

Dash 18X®
Operations Manual

Part Number: 22834521

Manual Version 1.7
Software Version 3.7

Trademarks

Dash 18X and AstroVIEW are trademarks of Astro-Med, Inc.
Windows and Microsoft are trademarks of Microsoft Corporation.
Ethernet is a trademark of Xerox Corporation.

General Emissions Compliance Statement

Ferrite sleeves must be installed around any cables connected to the event or utility port with a straight through configuration to ensure compliance. Ferrite sleeves are available from Astro-Med, Inc. (part number 24394-003).

Shielded cables must be used with this unit to ensure compliance with emissions limits.

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.

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.

⏻ Standby (system shutdown - power down sequence).

◯ Off (system disconnection from mains).

❙ 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.
- Note that the power system may 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.

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 us at 401.828.4000. Astro-Med, Inc. is not responsible for your product if it is lost or damaged in transit. Astro-Med, Inc. makes no warranty either express or implied with respect to this product's fitness for a particular purpose.

Disclaimer of Warranty

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

It is good practice to thoroughly test the software with non-critical data before relying on it. It is the customer's responsibility to follow proper file backup procedures in the event of data loss. Astro-Med, Inc. is not responsible for any data loss experienced in operation of this software.

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

Contact Astro-Med, Inc.

To contact the Astro-Med, Inc. Test & Measurement Group, send us an e-mail at mtgroup@astromed.com or call 401.828.4000.

Product Identification

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

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

Model Number:
Serial Number:

Original Software Version:
Upgraded Software Version:
Date Installed:

Upgraded Software Version:
Date Installed:

Upgraded Software Version:
Date Installed:

Declaration of Conformity
Declaration de Conformité
Ubereinstimmungserklärung
Dichiarazione di Conformità

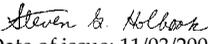|	
ID	DoC-22834521
Manufacturer's name and address Nom et adresse du fabricant Hersteller Nome del costruttore	Astro-Med, Inc. 600 East Greenwich Avenue West Warwick, RI 02893 USA
Model No. Modele No. Model Nr. Modello No.	Dash 18X
Standards to which conformity is declared Standards auquel la conformité appartient Normen für welche Übereinstimmung erklärt wird Norme per le quali si dichiara la conformità	Safety: EN 61010-1, 2nd Edition (2001) EMC: EN 61326:1998 Class A
Application of Council Directives Application des Decisions du Conseil Anwendbar fur die Richtlinien Applicazione delle Direttive del Comitato	89/336/EEC 73/23/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: <u>11/03/2008</u> Place of issue: <u>West Warwick, RI</u>

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: Introduction

Thank you for choosing the Astro-Med Dash 18X data acquisition recorder. This manual will help you learn how to use the Dash 18X features.

About the Dash 18X

The Dash 18X is a powerful data acquisition recording system for displaying, recording, and reviewing data. It combines waveform data, video, audio, and analysis in a portable system.

This system will handle all of your recording requirements in one instrument. You will not need to use a scope, meter, data recorder, data logger, or audio recorder. The large real-time waveform display, internal hard drive for capturing data, integral PC connectivity, and optional chart recorder give you data in a variety of different formats.

Getting help

In addition to this manual, several help resources are available to you. Use the following information to learn more about the Dash 18X help options.

Using context help

The context help system provides on-screen help for each of the three modes (Realtime, Scope, and Review). Context help provides an overview of the capabilities of the selected mode of operation.

To use context help:
1. If necessary, select a mode of operation by choosing Configuration >> Realtime *or* Scope *or* Review from the menu bar.

2. Open the context help screen by choosing Help >> Context Help from the menu bar. A message will display with an overview of the selected mode.

3. When you are finished reading the context help, choose anywhere on the screen to close the help message.

Using icon help

The icon help system provides on-screen help for each button. Icon help provides a brief explanation about the function of any button.

To use icon help:

1. If necessary, select a mode of operation by choosing Configuration >> Realtime *or* Scope *or* Review from the menu bar.

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

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

   To see help for other buttons, choose them and read the description.

4. When you are finished using icon help to learn about buttons, choose the Icon Help button again. The icon help function will be turned off.

Note: Remember to turn off the icon help when you are finished using it, as the buttons will not perform their functions when icon help is on.

Online manual

The Adobe Portable Document Format (PDF) version of this manual is available on the Dash 18X 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

If you require additional assistance, you can contact our Technical Support Department 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 18X.

Hardware descriptions

Use the following diagrams to familiarize yourself with the Dash 18X hardware components.

Front view diagram

The following diagram displays the front of the Dash 18X.
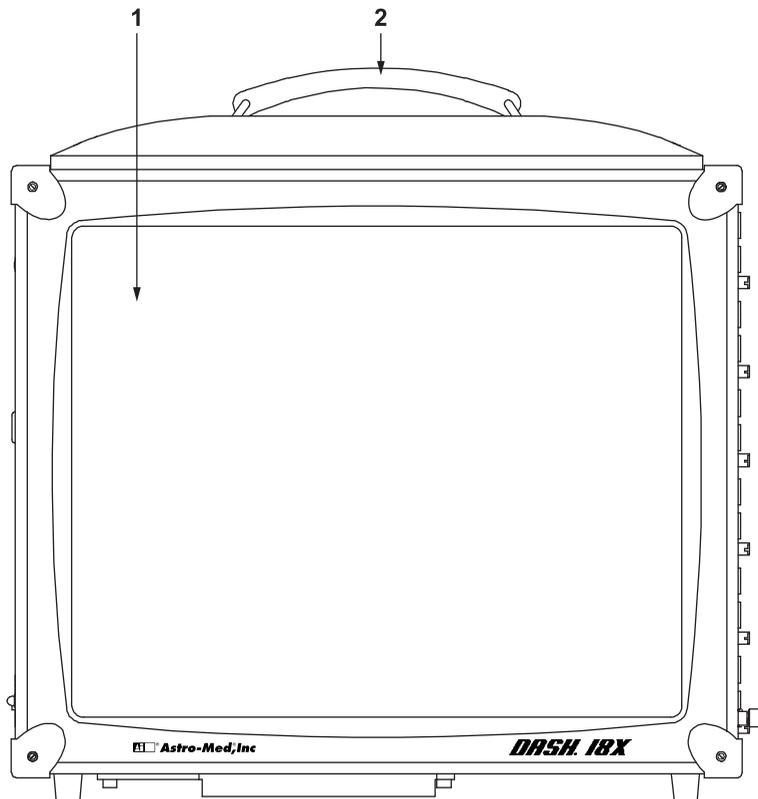
1. Touch screen   2. Carry handle

Touch-screen

The touch-screen serves as the user interface for the Dash 18X. 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 18X.

Bottom view diagram

The following diagram displays the bottom of the Dash 18X.
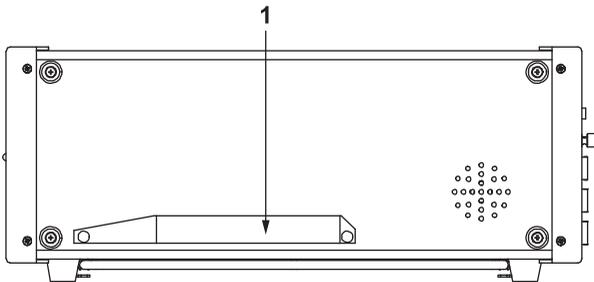
1. Module docking connector

Module docking connector

The module docking connector is used to attach peripheral devices to the Dash 18X.

Right side view diagram

The following diagram displays the right side of the Dash 18X.
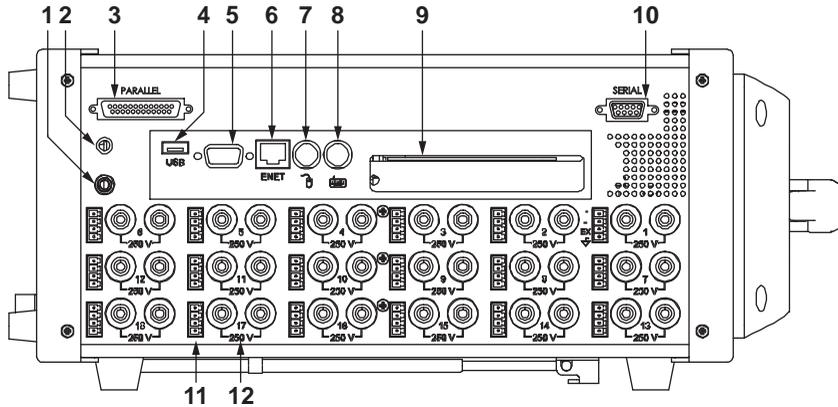
1. Ground connection
2. Thermistor
3. Parallel port
4. USB port
5. VGA port
6. Ethernet port
7. Mouse port
8. Keyboard port
9. Modem slot
10. Serial port
11. Differential input
12. Banana jack inputs

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

Thermistor
The thermistor measures the ambient temperature when thermocouple inputs are used.

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.

USB Port

The Dash 18X contains a USB 1.1 port. The USB port is used to connect accessories to the unit. This port is typically used to connect portable memory storage devices.

VGA port

The VGA port is used to connect the Dash 18X to an external monitor. The monitor must be capable of displaying a 1024 x 768 screen resolution.

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.

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.

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.

Modem slot

This slot is used to install the optional modem.

Serial port

The serial port is not currently used.

Differential input

The differential input is used for differential signal input.

Banana jack inputs

The banana jack inputs are used for isolated single-ended inputs.

Left side view diagram

The following diagram displays the left side of the Dash 18X.
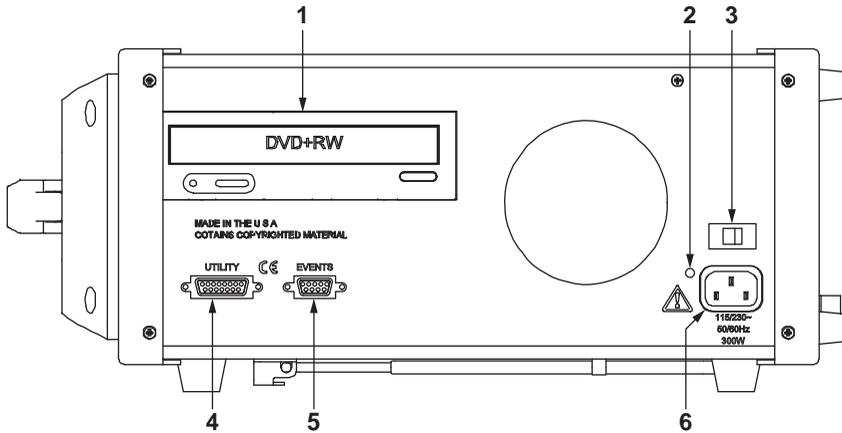
1. DVD+RW drive
2. Power indicator light
3. Power switch
4. Utility port
5. Event port
6. Power inlet

DVD+RW drive

The DVD+RW drive provides removable data storage capabilities. CDs and DVDs can be used to store, transfer, and archive data and setup files. It can also be used to upgrade the system software. You can use CD-R, CD-RW, DVD+R, DVD-R, and DVD+RW media with this drive.

Power indicator light

The power indicator light is used to visually indicate the power status of the Dash 18X. 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 switch

The power switch is used to power-up and power-down the Dash 18X. The power-up sequence requires approximately two minutes.

Utility port

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

Event port

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

Power inlet

The power inlet is used to attach the power cable to the Dash 18X.

Back view diagram

The following diagram displays the back of the Dash 18X.
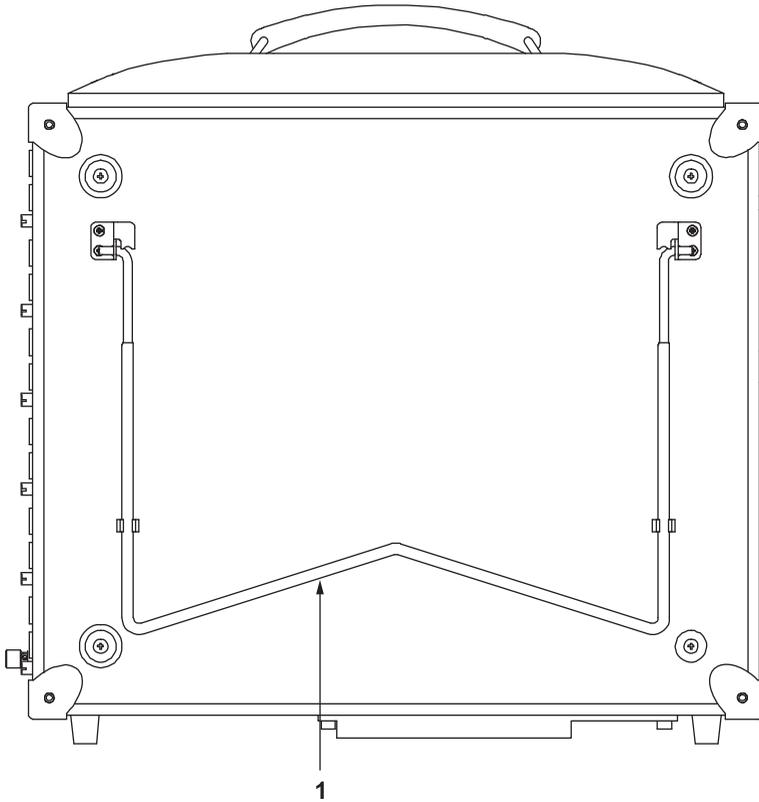
1. Stand

Stand

The built-in stand is used to position the Dash 18X on an angle.

Isolated universal inputs

Use the following information to familiarize yourself with the Dash 18X isolated universal inputs.

Isolated universal input diagram

The following diagram displays the Dash 18X isolated universal inputs.
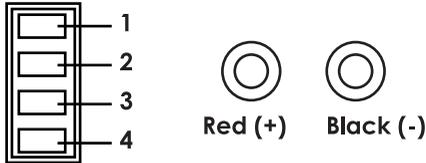
1. Differential input positive (top)
2. Differential input negative
3. Excitation input positive
4. Excitation input negative

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

Isolated single-ended voltage input

Please refer to appendix for detailed specifications.

Connection: Connect shrouded banana jacks (black and red connectors).

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

Differential voltage input

Please refer to appendix for detailed specifications.

Connection: Connect positive input to pin 1 and negative input to pin 2 of multi-pin connector.

Thermocouple input

Please refer to appendix for detailed information on temperature ranges.

Type J
Type K
Type E
Type T
Type N

Connection: Connect thermocouple between 1 and 2 of multi-pin connector.

DC bridge wiring diagrams

Please refer to appendix for detailed specifications.

Four-wire full bridge
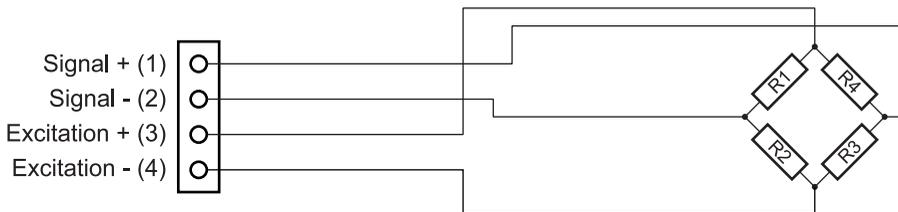
Signal + (1)
Signal - (2)
Excitation + (3)
Excitation - (4)

R1 R4
R2 R3

Three-wire half bridge

Signal + (1)
Signal - (2)
Excitation + (3)
Excitation - (4)

R1
R2

R4
R3

External Bridge
Completion
Resistors

Three-wire quarter bridge

Signal + (1)
Signal - (2)
Excitation + (3)
Excitation - (4)

R1
R2
R3

R4

External Bridge
Completion
Resistors

Two-wire quarter bridge

Signal + (1)
Signal - (2)
Excitation + (3)
Excitation - (4)

R1
R2
R3

R4

External Bridge
Completion
Resistors
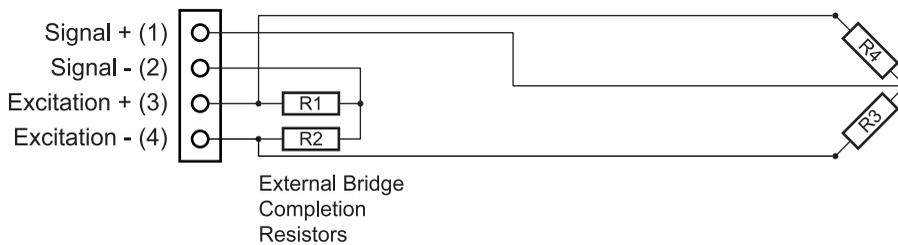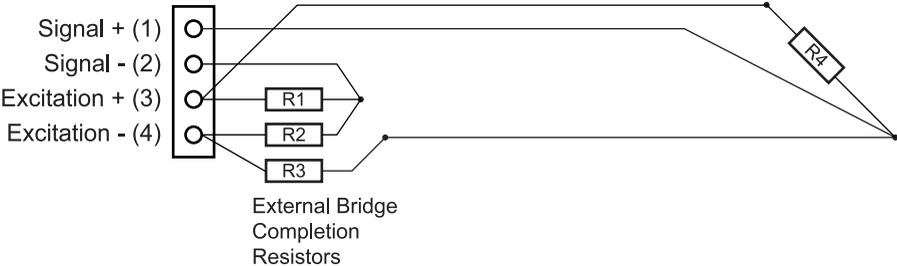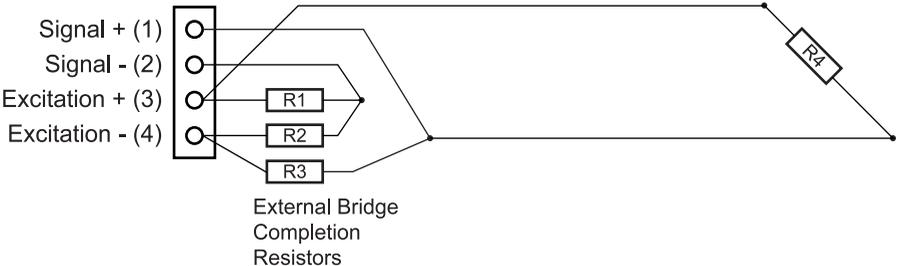
Utility port - 15 pin D-shell connector

Use the following information to familiarize yourself with the Dash 18X utility port.

Utility port diagram

The following diagram displays the Dash 18X utility port.

Pin 1

Pin 15

1. Trigger in (Input)*
2. Alarm (Output)**
3. Reserved (Do not connect)
4. Reserved (Do not connect)
5. Remote arm (Input)*
6. Security (Input)*
7. Remote abort (Input)*
8. Input (Chart Mark/Page Mark)*
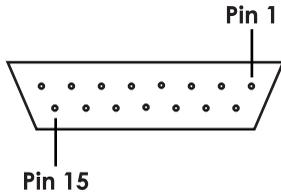
9. Trigger out (Output)**
10. External motor clock (Input)*
11. External run/halt (Input)*
12. Common
13. Input (Print Demand/Print ID/ Print Channel Labels)*
14. 5V (Through 100 Ω resistor)
15. Reserved (Do not connect)

* Active low TTL or switch closure.

** Outputs can sink up to 15 mA, and are active low TTL.

Event port - 9 pin D-shell connector

Use the following information to familiarize yourself with the Dash 18X event input port.

Event input port diagram

The following diagram displays the Dash 18X event input port. Events are TTL or switch closure to ground.

Pin 1

Pin 9
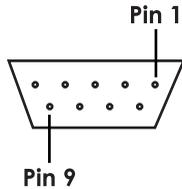
1. Event 1
2. Event 2
3. Event 3
4. Event 4
5. Event 5

6. Event 6
7. Event 7
8. Event 8
9. GND

Ethernet connection specifications

10/100 Base T

RJ-45 connector

Dash 18X cover

A cover is included with the Dash 18X. You can use the cover to protect the Dash 18X touch screen.

Protecting the screen with the cover

If harsh conditions are part of your testing environment, you can protect the Dash 18X touch screen with the cover.

To protect the screen with the cover:

1. Place the cover on the front of the Dash 18X and align the corner screw holes.
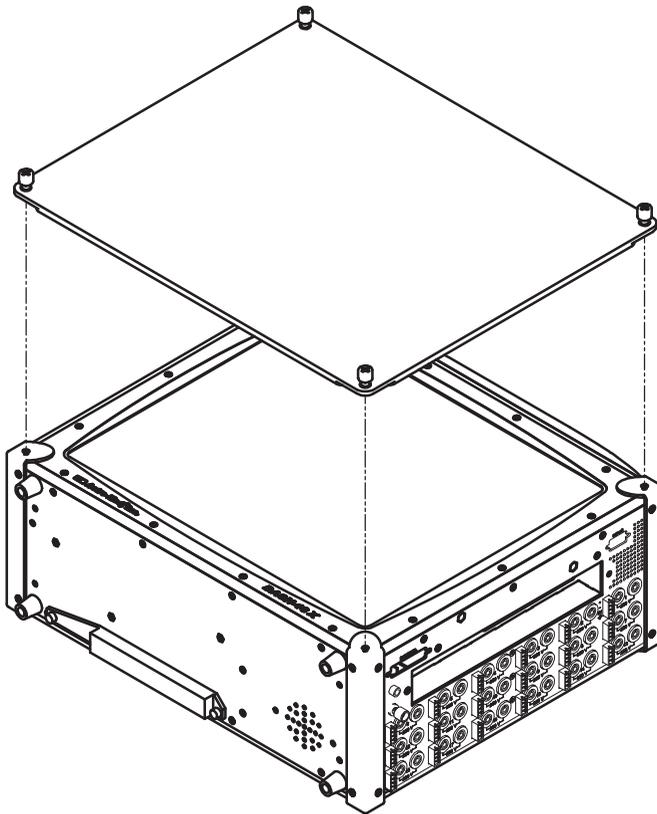
2. Attach the cover to the Dash 18X using the thumb screws.

SCR-18 printer

The SCR-18 is an optional thermal printer that connects to the Dash 18X. You will connect this printer using the module docking connector.

The illustration below displays the SCR-18 printer attached to the bottom of the Dash 18X.
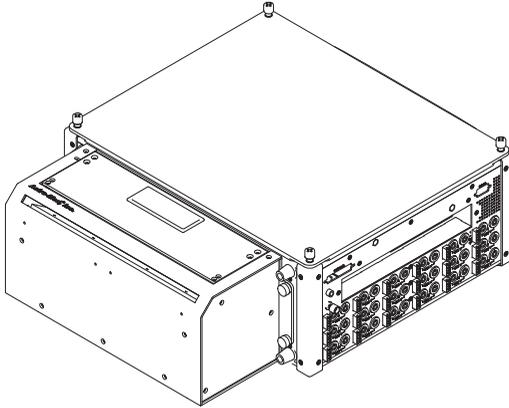
Specifications

Chart width: 11 inches

Resolution: 300 dpi

Chart Speeds: 1 mm/min to 50 mm/sec

Format: Chart format follows display format

Text Buffers: For each channel with up to 128 ASCII characters per buffer

Print Configuration: Real-time and Data Capture Playback

Weight: 9 pounds

Installing the SCR-18 printer

You will install the SCR-18 printer by attaching it to the bottom of the Dash 18X. When the printer is attached, it will automatically begin operation. You can control the printer by using the Chart menu bar options in Realtime mode.

Note: The SCR-18 cannot be used simultaneously with the PIM-18/PIM-18A power input module.

To install the SCR-18 printer:

1. Remove the SCR-18 from the packaging. Remove all protective material and visually inspect the unit.

2. Ensure that the power switch for the Dash 18X is off.

3. Remove the cover over the module docking connector. The connector is located on the bottom of the Dash 18X.
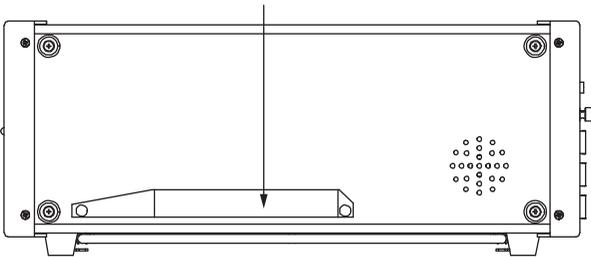
4. Attach the mating connector of the SCR-18 to the Dash 18X. The SCR-18 has only one mating connector and can only be attached to the Dash 18X in one orientation.

   Complete the physical connection by fastening the four thumbscrews of the SCR-18 to the Dash 18X chassis.

5. Power up the Dash 18X.

   After the Dash 18X boot-up process is complete, the Dash 18X will be ready for operation and the SCR-18 will begin printing.

   Note: You can use the Chart menu bar options in Realtime mode to control the chart.

Power input module (PIM-18/PIM-18A)

The PIM-18 and PIM-18A are optional components used to supply DC voltage to power the Dash 18X instead of AC power.

- The PIM-18 receives DC voltages from 9-18 volts and powers the Dash 18X.

- The PIM-18A receives DC voltages from 20-32 volts and powers the Dash 18X.
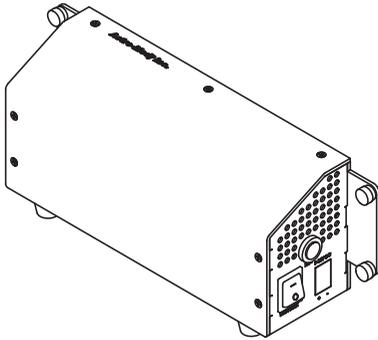
Please read the following information completely before attempting to operate the PIM-18 or PIM-18A.

Note: The PIM-18/PIM-18A cannot be used simultaneously with the SCR-18 printer.

Specifications

	PIM-18	**PIM-18A**
Input range	9-18 Volts DC	20-32 Volts DC
Input power	200 Watts maximum 150 Watts typical	200 Watts maximum 150 Watts typical
Output voltage	25 Volts DC	25 Volts DC
Fuse	User replaceable 25 Amp	User replaceable 10 Amp

Installing and using the PIM-18/PIM-18A

Power from your DC voltage source to the PIM-18/PIM-18A will need to be provided with bare "tinned" leads. You will need a small flat-head screwdriver to fasten the "tinned" leads into the input power screw terminal connection.

Your leads should be of sufficient AWG to safely handle the maximum current. The PIM-18 maximum current is 25 Amps. The PIM-18A maximum current is 10 Amps.

Note: For your safety, the PIM-18/PIM-18A will not function without being connected to the Dash 18X.

To install and use the PIM-18/PIM-18A:

1. Remove the PIM-18/PIM-18A from the packaging. Remove all protective material and visually inspect the unit.

2. Verify the following:

   - Ensure that power switches for both the Dash 18X and the PIM-18/PIM-18A are off.

   - If the Dash 18X was previously powered by an AC voltage source, ensure that the AC power cord is disconnected from the Dash 18X.

     This will ensure that the Dash 18X will not be inadvertently powered by both voltage sources.

   - Ensure that the input "tinned" leads for the PIM-18/PIM-18A power are not connected to the DC voltage source.

3. Remove the cover over the module docking connector. The connector is located on the bottom of the Dash 18X.
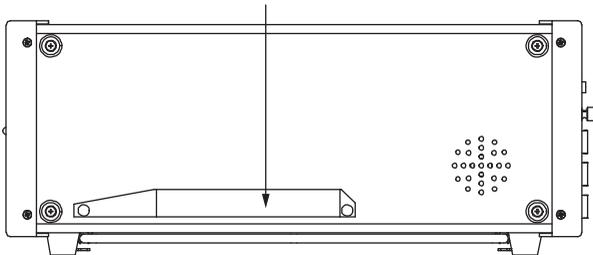
4. Attach the mating connector of the PIM-18/PIM-18A to the Dash 18X. The PIM-18/PIM-18A has only one mating connector and can only be attached to the Dash 18X in one orientation.

   Complete the physical connection by fastening the four thumbscrews of the PIM-18/PIM-18A to the Dash 18X chassis.

5. With the PIM-18/PIM-18A power switch still in the off position, install the "tinned" input power leads in the screw-terminal on the side of the PIM-18/PIM-18A. Secure them using a small, flat-headed screwdriver.

   Then connect the input leads to your active DC voltage source.

6. Turn on the power of the PIM-18/PIM-18A first, and then turn on the power for the Dash 18X.

   After the Dash 18X boot-up process is complete, the Dash 18X will be ready for operation.

7. Power down the Dash 18X.

   When you are finished using the Dash 18X, turn the power switch off on the Dash 18X. Then allow Dash 18X to shut down. When the green LED near the power switch stops flashing, the Dash 18X shut down process will be complete.

   Next, turn the power switch off on the PIM-18/PIM-18A.

   Finally, disconnect the DC voltage source before disconnecting the input power leads from the screw-terminal connection of the PIM-18/PIM-18A.

Troubleshooting the PIM-18/PIM-18A

There are a few things that can be checked if the PIM-18/PIM-18A does not operate correctly. Use the following troubleshooting tips if you encounter a problem:

- Check the connections and ensure that the PIM-18/PIM-18A is fastened securely to the chassis of the Dash 18X. Ensure that the power leads are securely tightened in the screw terminals.

- Use a voltage meter to ensure that an adequate voltage level is being supplied by the DC voltage source.

- Use a voltage meter to ensure that the polarity of the input voltage from the DC voltage source to the PIM-18/PIM-18A is correct.

Windows-based printers

The Dash 18X supports standard Windows-based printers via the parallel port.

Installing a Windows-based printer

You can connect standard Windows-compatible printers to the Dash 18X parallel port. You can use this printer for printing screen captures.

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

The installation process utilizes the Dash 18X 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 18X. A CD or USB-based memory storage device can facilitate this process.

To install a Windows-based printer:
1. Connect the printer to the Dash 18X using a parallel cable.

2. Choose Configuration >> Utilities from the menu bar to open the utilities screen.
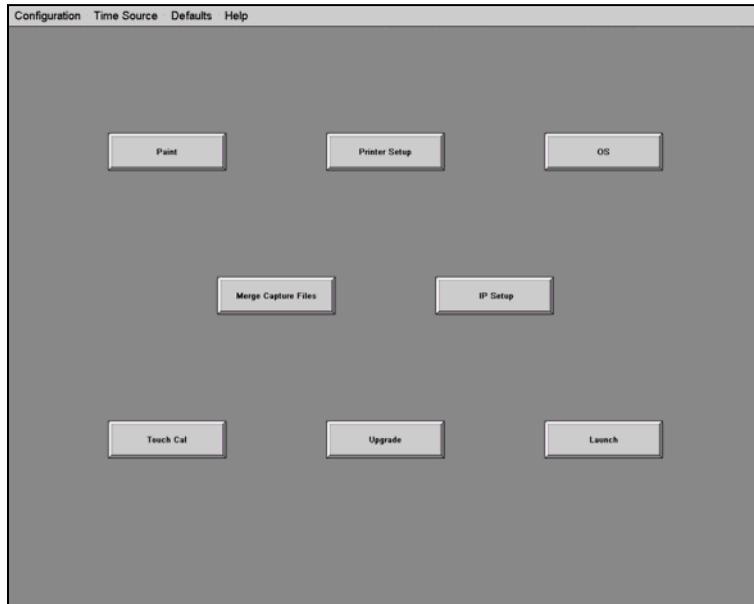
3. Choose the Printer Setup button. The Printers and Faxes window will open. This window is a part of the Microsoft ® Windows operating system.
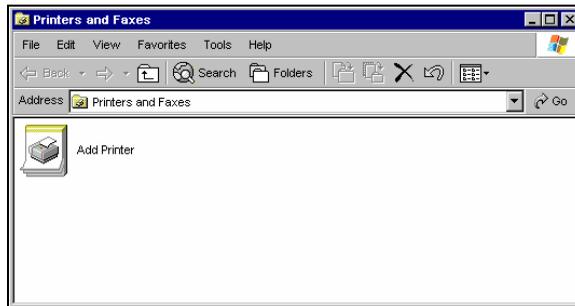
4. Double-click the Add Printer icon. The printer installation wizard will start.

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 CD or USB-based memory storage device. Insert the CD, or connect the USB-based storage device into the Dash 18X. When prompted by Windows to search for drivers, browse to the driver located on the CD or USB storage device.

6. Power down the Dash 18X and power back on before using the Windows-based printer.

Rack mount version

Use the following information to familiarize yourself with the rack mount version of the Dash 18X.

Front view diagram

The following diagram displays the front of the rack mount Dash 18X.
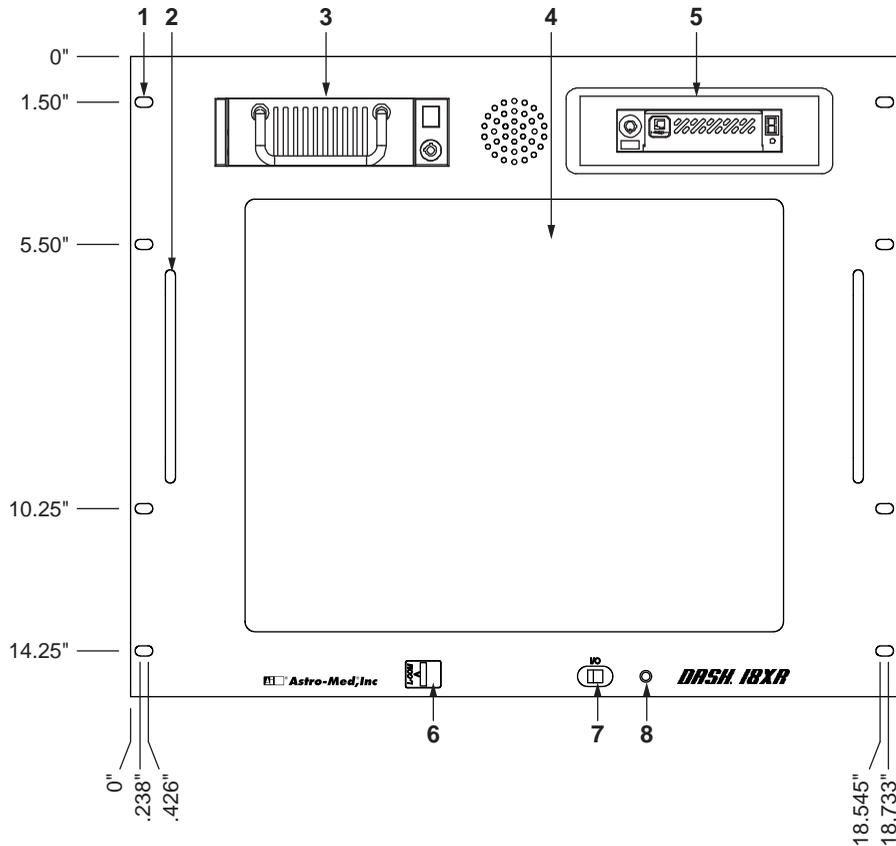
1. Rack mounting holes
2. Handle
3. Data capture drive
4. Touch screen
5. System drive
6. USB port
7. Power switch
8. Power indicator light

Back view diagram

The following diagram displays the back of the rack mount Dash 18X.
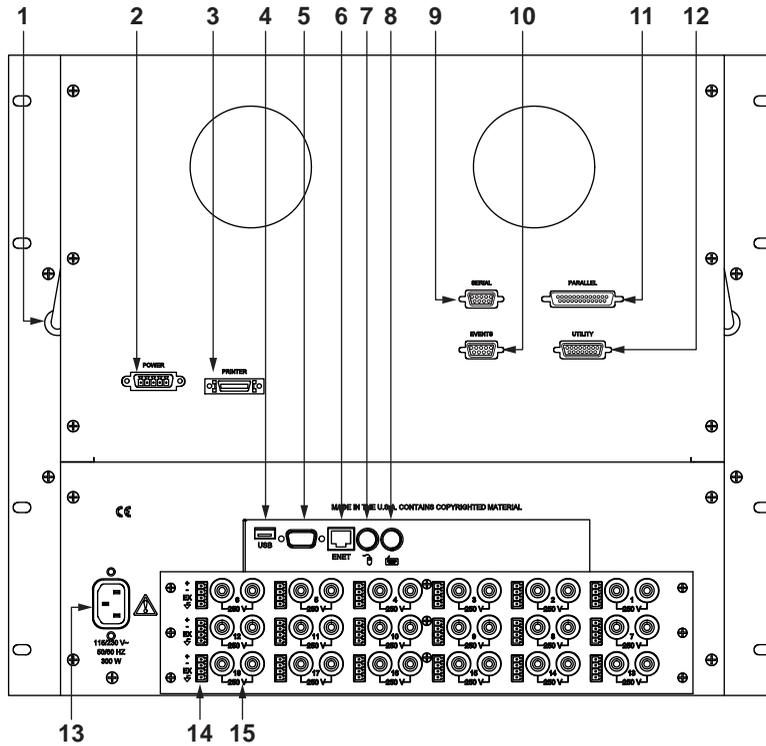
1. Carry handle
2. Printer power port
3. Printer port
4. USB port
5. Unused
6. Ethernet port
7. Mouse port
8. Keyboard port
9. Serial port
10. Event port
11. Parallel port
12. Utility port
13. Power inlet
14. Differential input
15. Banana jack inputs

Removing and loading drive modules

Use the following instructions to remove and load drive modules in the rack mount model.

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

To remove a drive module:

1. Ensure the rack mount Dash 18X power is off.

2. Insert the included drive key into the drive lock, and turn clockwise to unlock the drive.

3. Using the handle on the front of the drive, pull the drive from the drive bay.

4. Re-lock the drive by turning the drive key counter-clockwise. The rack mount Dash 18X will not start up if any of the bays are unlocked.

To load a drive module:

1. Ensure the rack mount Dash 18X power is off.

2. Insert the included drive key into the drive lock, and turn clockwise to unlock the drive. The rack mount Dash 18X will not start up if any of the bays are unlocked.

3. Slide the drive all the way into the drive bay.

4. Re-lock the drive by turning the drive key counter-clockwise.

Rack mount printer

The SCR-18R is an optional rack mount thermal printer that connects to the rack mount Dash 18X.

Front view diagram

The following diagram displays the front of the SCR-18R. Mounting hole measurements are indicated in inches.
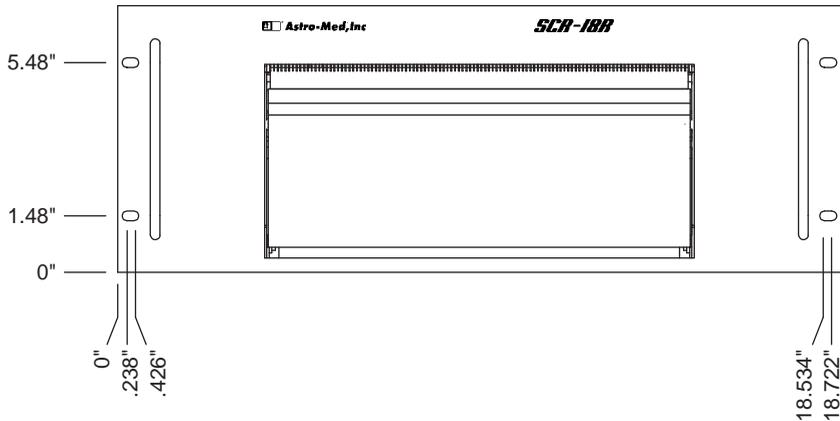
Back view diagram

The following diagram displays the back of the SCR-18R.
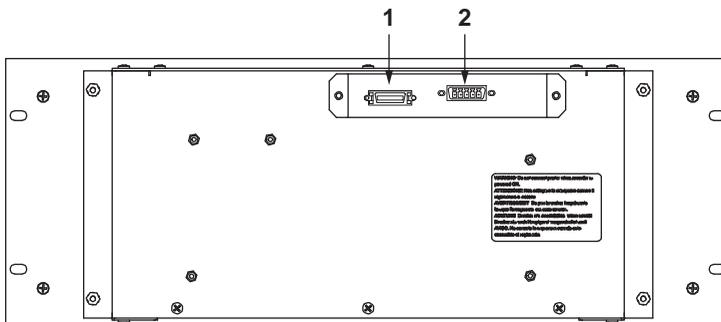
1. Printer port
2. Printer power port

Printer port

The printer port provides data to the to the SCR-18R. Use the supplied cable to connect this port to the rack mount Dash 18X printer port.

Printer power port

The printer power port provides power to the SCR-18R. Use the supplied cable to connect this port to the rack mount Dash 18X printer power port.

Specifications

Chart width: 11 inches

Resolution: 300 dpi

Chart Speeds: 1 mm/min to 50 mm/sec

Format: Chart format follows display format

Text Buffers: For each channel with up to 128 ASCII characters per buffer

Print Configuration: Real-time and Data Capture Playback

Weight: 25 pounds

IRIG version

For applications requiring external time correlation, the Dash 18X offers an optional IRIG decoding module. This module decodes IRIG A, B, H and NASA 36 time codes, providing high-speed time synchronization of your data. During real-time recording, IRIG time is shown on the display. In acquisition mode, each data point is time-stamped with IRIG time.

The Dash 18X IRIG configuration provides 12 universal input channels and 3 differential input channels.

IRIG version input diagram

The following diagram displays the input side of the Dash 18X with the optional IRIG feature installed.
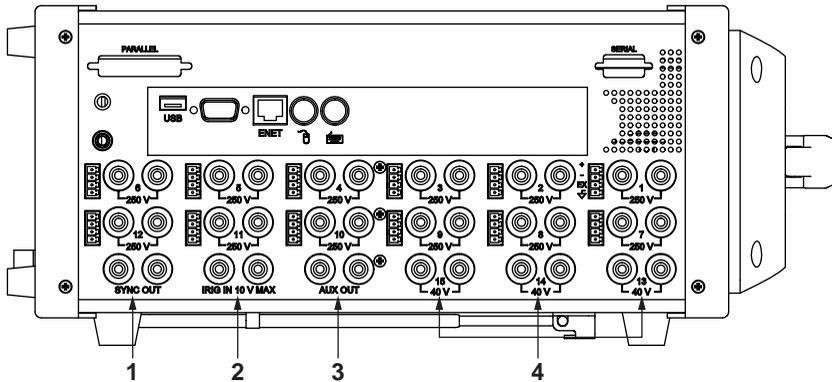
1. Sync out
2. IRIG in
3. Aux out
4. Differential inputs

Time base carrier variation

The Dash 18X can read modulated codes with carrier base variations between x1/2 and x1.5 of nominal.

Waveform input channel specifications

Cold Start Drift 40V Attenuator: 0.1% of the Attenuator

Cold Start Drift 5V Attenuator: 0.2% of the Attenuator

Baseline Accuracy: ± 0.4 % of the Attenuator

DC Accuracy: ± 0.4 % of the Attenuator

Cross Talk: 0.1% of the Attenuator

Frequency Response: 12 Khz (-3dB)

Intrinsic Noise 40V Attenuator: ± 0.1 % of the Attenuator

Intrinsic Noise 5V Attenuator: ± 0.2 % of the Attenuator

Input Impedance: 200KΩ Minimum

Overshoot: Less than 1 %

Ch1 Frequency Counter Accuracy: 10 Hz – 20 KHz   ± 0.05% of Measurement + 0.1Hz

Ch1 Frequency Counter Minimum Input Signal Level: 20% of the Attenuator

IRIG time input specifications

Max Input Voltage: 10 V

Min Input Voltage: .5 V

Aux Out: Reserved for future use.

Time Codes supported: IRIG A AM/TTL, IRIG B AM/TTL, IRIG H AM/TTL, NASA 36 AM/TTL

Accuracy - modulated: 1 ms

Accuracy – IRIG B & NASA 36 TTL: 1 second

Accuracy – IRIG A TTL: 100 ms

Output channel specifications

Sync Out: LVTTL output indicating IRIG start pulse

Aux Out: Reserved for future use.

Chapter 3: Quick start

This chapter provides information to help you immediately get started using the Dash 18X. This chapter does not provide details found in other chapters of the manual, and assumes that you are familiar with test and measurement concepts.

Start using the Dash 18X

By following these instructions, you will become familiar with basic Dash 18X operations.

To start using the Dash 18X:
1. Examine the Dash 18X before use to verify that it has not been damaged during shipment.

2. Verify input line voltage, then connect the AC power.

   Before connecting your recorder to any outlet, verify that the input line voltage is within the power specifications for the recorder.

   The Dash 18X has an auto-sensing power supply that operates from 102 – 264 VAC at 50 or 60 Hz. You do not have to set up any fuse block arrangement. The appropriate input voltage will be selected automatically and requires no adjustment once a power line is connected.

   Connect the AC power cord to the Dash 18X rear panel, then to an outlet.

3. Turn on the Dash 18X.

   When you turn on the Dash 18X, the color touch-screen will display initialization. Then the recorder will load using the last setup configurations it had prior to powering down last. The Dash 18X will take approximately two minutes to power up.

   Note: When you power down the Dash 18X, a green LED light next to the power switch will flicker for a few moments as the Dash 18X shuts down. You must allow the Dash 18X to complete the shut down sequence before turning the Dash 18X on again.

4. Use the color touch-screen display.

   The Dash 18X provides a color touch-screen display for viewing data and controlling the recorder.

   Most of the display is used to show the waveform signals. The top left of the display is used for the menu bar. If the date and time are visible at the top of the screen, press the M icon in the upper-right corner to show the menu bar. The group of keys on the right is your customizable control panel.

5. Connect your signals to the Dash 18X.

   Signal leads are not supplied with the Dash 18X. There are two physical input sections for each channel. You must only use one physical connection at a time. For the differential, DC bridge, and temperature amplifiers, use the 4-pin universal input terminal block connector. Mating connections are provided. For the single-ended amplifier, use the standard banana jacks connection.

   These instructions will use the single-ended input connection for simplicity. Insert your standard banana jack leads into the side panel of the recorder.

6. Start recording.

   Choose Configuration >> Realtime from the menu bar. The Dash 18X will begin monitoring.

   The channel labels to the right of the waveform area display full-scale values for your channels. You can modify how these channels are displayed using a menu that appears when you press a channel label.

7. Control the chart speed.

   Use the Run/Halt button to start/stop the chart display.

   With the chart running, change the speed by choosing a speed button from the upper-right section of the control panel. These buttons provide some common chart speeds. You can create custom speed buttons and add them to the control panel, if necessary.

8. Modify the display of channels.

   View basic channels by choosing View >> Basic 6, 12, or 18 Channels. You can customize the view by choosing View >> Chart/Display Wizard. The Chart/Display Wizard allows you to specify the exact number of channels you need.

   You can also move and resize channels using the channel label button on the right of a waveform.

9. Display more information using view options.

   Choose View >> Channel Information to open the Channel Information window. Then show the cursors using the Cursors menu. Observe the Channel Information window values as you move the cursors.

   Choose View >> Meter to view instantaneous numeric signal values.

   Choose View >> XY Plot to view a real-time XY plot for any two channels.

   Choose View >> AstroVision to display realtime video with signal data. You can capture video along with signal data. This feature is active only when using the optional webcam.

About the modes of operation

The Dash 18X provides the following three operating modes:

   Realtime mode provides real-time scrolling and monitoring of waveforms. Almost all signal setup options are accessed from Realtime mode.

   Scope mode provides better time-base resolution for monitoring higher frequency signals. Scope mode acts like a digital storage oscilloscope.

   Review mode allows you to review and analyze captured data.

Regardless of which configuration you choose, the Dash 18X can acquire data to the hard drive. You can also switch between configurations without interrupting your data acquisition.

Accessing Realtime mode

You will access Realtime mode by choosing Configuration >> Realtime from the menu bar.

The following illustration displays a typical Realtime mode screen. Your screen may look different depending on the options that you are using, and the custom Realtime control panel that you have set up.
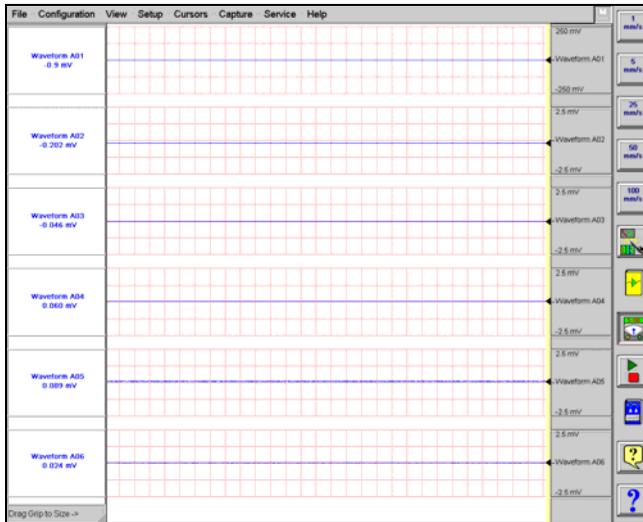
Accessing Scope mode

You will access Scope mode by choosing Configuration >> Scope from the menu bar.

The following illustration displays a typical Scope mode screen. Your screen may look different depending on the options that you are using, and the custom Scope control panel that you have set up.
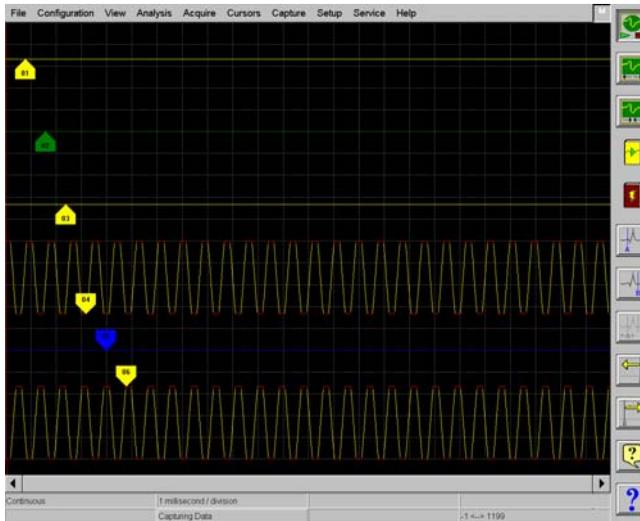
Accessing Review mode

You will access Review mode by choosing Configuration >> Review from the menu bar.

The following illustration displays a typical Review mode screen. Your screen may look different depending on the options that you are using, and the custom Review control panel that you have set up.
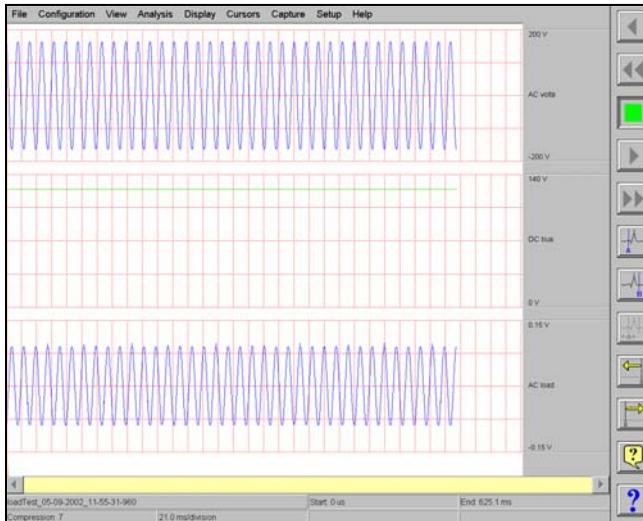
Chapter 4: Menus and buttons

Understanding the user interface of the Dash 18X will help you streamline your test and measurement process. This chapter will help you learn how to use the Dash 18X menus and buttons.

Menu bar

The menu bar is a group of drop-down menus located on the top left of the display. From this menu, you can access all of the Dash 18X modes and features.
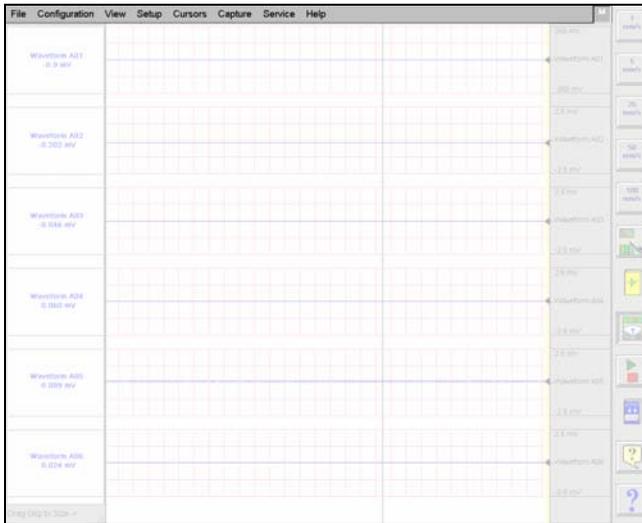
Note: If the date and time are visible on the top of the display, choose the M button in the upper right corner to display the menu bar. Choosing the M button again will hide the menu bar and display the date and time.

Control panel

The control panel is a customizable group of buttons located on the right side of the display. It allows you to launch each function with one touch and can be customized to meet your needs.

Each mode of operation (Realtime, Scope, and Review) provides a customizable control panel. These types of control panels can be individually modified and saved.
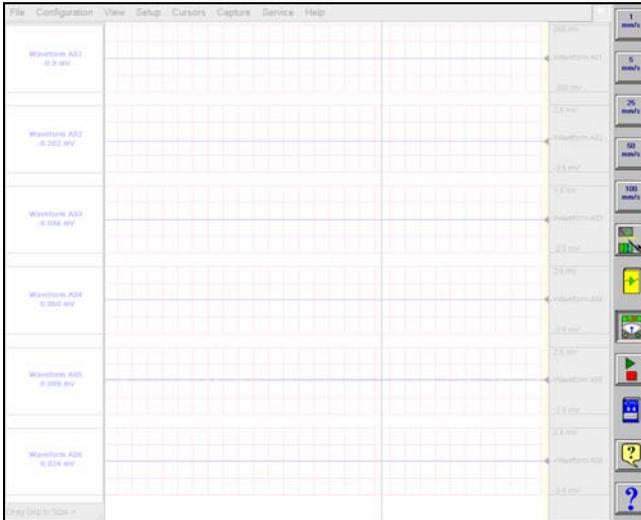
Adding frequently used functions and removing unused functions will save you time and effort during product use. For example, if you frequently use the "full page mark" function, you can save time by adding a corresponding button to the control panel. Instead of using the menu bar to choose View >> Full Page Mark, you will launch the function by simply pressing one control panel button.

You can save customized control panel configurations to a file for later use. This flexible feature allows each user to set up a personalized control panel configuration to streamline their setup and operation.

Customizing the control panel

You will remove, add, and arrange buttons to set up a customized control panel.

To customize the control panel:

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

The buttons on the left provide functions that you can add to the control panel. The columns on the right display the layout of the control panel.

Note: The size of the control panel depends on the mode that you are using. Realtime mode provides 48 button slots. Scope and Review modes provide 40 button slots.

2. Remove buttons, if necessary.

   To remove a button, select it. Then choose the Clear button.

   The selected button will be removed from the control panel.

3. Add buttons, if necessary. You can add a button to an empty slot or replace an existing button with a new one.

To add a button, select an empty slot or an existing control panel button. Then select a function from the menu on the left. A button for the function will appear in the selected control panel position. If the location previously contained a button, it will be replaced.

4. Arrange the buttons, if necessary.

   To move a button, select it. While pressing the button, move it to a different location in the control panel. Then release the button to move it to the new location. If the new location previously contained a button, it will be replaced.

5. Add or remove separator bars, if necessary.

   To add a separator bar, select an empty slot or existing control panel button. Then choose the Add/Remove Separator button.

   A separator bar will be added to the bottom of the button or slot. You can remove the separator by choosing the Add/Remove Separator button again.

6. When you are finished setting up the control panel, choose the OK button.

Saving custom control panels to files

When the control panel is set up to your desired layout, you can save the settings to a configuration file on disk.

Note: In Realtime mode, you can save your current control panel by choosing File >> Save >> Control Panel from the menu bar. This shortcut will save you time when working in Realtime mode.

To save custom control panels to files:

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

2. Choose the Save button.

The Save Control Panel Setup File window will open.

---
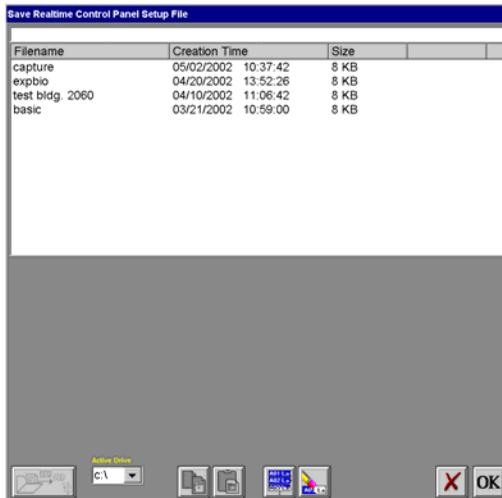
3. If necessary, select the drive to save the file on. By default, drive C is selected.

4. Press the Filename field above the list column headings. A keypad will appear.

   Enter a name for the control panel file and choose OK.

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

   Then choose OK in the Control Panel Setup window. This window will close and you will return to the main display.

Loading custom control panels from files

Before working with the Dash 18X, you can load your custom control panel from a file. You will load control panel setup files by retrieving the files from disk.

Note: In Realtime mode, you can load your custom control panel by choosing File >> Load >> Control Panel from the menu bar. This shortcut will save you time when working in Realtime mode.

To load custom control panels from files:

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

2. Choose the Load button.

The Load Control Panel Setup File window will open.
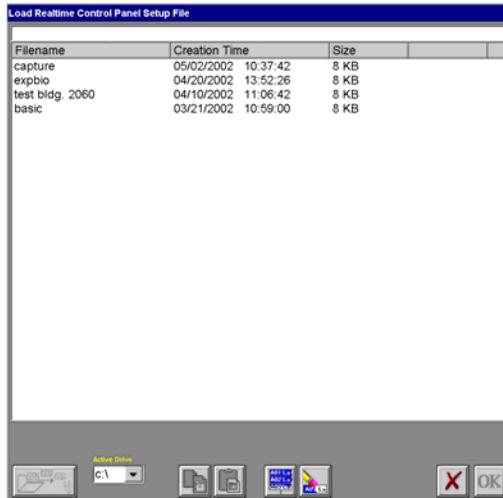
3. If necessary, select the drive where the file is located. By default, drive C is selected.

4. Select a control panel setup file and choose OK. The control panel will be loaded.

   Then choose OK in the Control Panel Setup window. This window will close and you will return to the main display.

Commonly used buttons

The following buttons are commonly used in Dash 18X windows:

OK saves the information you entered in a window and then closes the window.

Exit cancels the action you were performing in a window and closes the window without saving any changes.

Apply saves the information you 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 (Amplifier) copies the amplifier settings for the selected items to the clipboard.

Copy (File) copies the selected file to the clipboard.

Copy (User Units) copies the user unit settings for the selected items to the clipboard.

Paste (Amplifier) pastes the amplifier settings from the clipboard to the selected items.

Paste (File) pastes the file from the clipboard to the selected location.

Paste (User Units) pastes the user unit settings from the clipboard to the selected items.

Save saves current configurations to a file on your disk.

Load loads configurations from a file on your disk.

Chapter 5: Setting up channels

Before you begin measuring signals, you will need to set up your inputs and channels. This chapter will help you set up signal inputs and channel configurations.

General information and concepts

Channels

A channel is a single path for transmitting a signal. The waveform in a channel can be generated directly from a signal input, or can result from a calculation performed on multiple signal inputs. The Dash 18X provides the capability to monitor 18 individual channels.

Filters

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

There are four aspects to the Dash 18X filtering system.

Anti-aliasing filter – Limits input frequencies and prevents aliasing.

Implementation: Hardware
Type: Bessel lowpass
Cutoff (-3dB): 20 KHz
Order: 4
Roll-off: -80 dB/decade

Fixed lowpass filter – Limits high-frequency noise.

Implementation: Digital signal processing
Topology: IIR Biquad
Type: Bessel lowpass
Cutoff (-3dB): 22.3 KHz
Order: 2
Roll-off: -40 dB/decade

Built-in user-selectable filter – User-selectable filtering.

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

File-based filter – Filters frequencies based on a customized set of coefficients stored in a file.

Note: For more information on file-based filters, please contact the factory.

Grounding

A ground is generally thought of as an equipotential point or structure designed so that 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 – 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 – Analog reference point for the measuring device.

Source Common – Reference point at the voltage source.

User engineering units

The Dash 18X provides a mechanism for converting the module unit to an engineering unit specified by the user. When assigned, the user engineering unit replaces the module unit in all displays and recordings.

The relationship between the module unit and the user engineering unit is assumed to be linear, characterized by a slope and offset
(for example, $y = mx + b$).

About signal inputs

The term input refers to signal connected to a physical input on the side of the Dash 18X. The following diagram illustrates the layout of signal inputs.
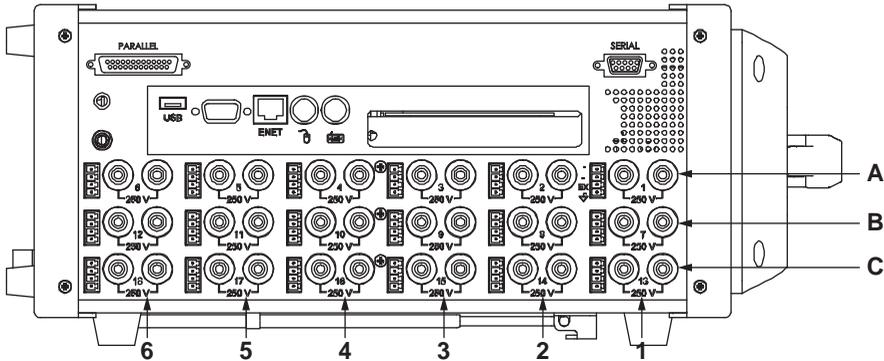
In the Dash 18X software, each signal input has an input label for identification purposes. By default, the input label corresponds to the physical location of the input. For example, A01 – Waveform A01 is the default input label of the signal connected to the A1 position.

Connecting signal inputs

Before you start taking measurements, you will connect your physical signal inputs to the Dash 18X.

To connect signal inputs:

1. Determine the appropriate connection input for your signal.

Red (+)   Black (-)

Two physical input sections are available for each channel. You will choose either the 4-pin connector or the banana jack (red & black) inputs. The input you select will vary depending on the type of signal you are measuring.

- If you need to use differential, DC bridge, or temperature amplifiers, use the 4-pin universal input connector. Mating connections are provided with the Dash 18X.

- If you need to use a single-ended amplifier, use the standard banana jack inputs.

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

If you plan to use math functions on a group of channels, you will need to connect the signals in blocks of three. For example, A01, A02, A03, or A04, A05, A06, etc.

2. Connect your signals to the Dash 18X.

Basic channel amplifier setup

The process of setting up channel amplifiers allows you to configure how your signals are generated and measured.

To set up basic channel amplifier settings:

1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.
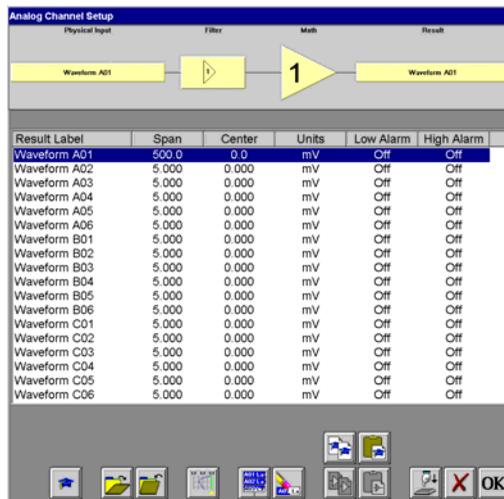
2. If necessary, modify the result label of channels by selecting a channel and choosing the Result Label column hading. A keypad will appear. Enter a new name for the channel and choose OK.

Note: The result label is the name of channel *after* filter and math functions have been applied. Waveforms in these channels can be generated directly from a signal input, or can result from a calculation performed on multiple signal inputs.

---

3. Select a span for channels by selecting a channel and choosing the Span column heading. A number pad will appear. Enter a new value for the span and choose OK. A list of allowable span ranges is provided below.

Single-ended, 4V	1 to 4V
Single-ended, 40V	1 to 40V
Single-ended, 400V	10 to 400V
Single-ended, 800V	10 to 800V
Differential, 50mV	5 to 50mV
Differential, 500mV	50 to 500mV
Differential, 1600mV	200 to 1600mV
DC Bridge, 50mV	5 to 50mV
DC Bridge, 500mV	5 to 500mV
Thermocouple, Type J	0 to 760°C
Thermocouple, Type K	0 to 1370°C
Thermocouple, Type E	-100 to 870°C
Thermocouple, Type T	-160 to 400°C
Thermocouple, Type N	0 to 1300°C

4. Specify the center or bottom values of channels.

Select whether to specify channel center or bottom values using the Bottom/Center button. The icon on the Bottom/Center button changes based on your current selection.

 Press this button to change the Center/Bottom column to Center.

 Press this button to change the Center/Bottom column to Bottom.

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* in this chapter.

Note: When user units are set up, you can copy and paste user units information among channels by using the Copy and Paste User Units buttons.
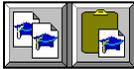
You can also copy and paste user units information among channels between the Input Setup window and Analog Channel Setup window.

---

6. Select whether to use low and high alarms by selecting a channel and choosing the High or Low Alarm column heading. A menu will appear.

   - If you do not want to use the alarm, choose the Off option.

   - If you want to use an alarm, choose the New Value option. A number pad will appear. Enter the alarm value and choose OK.

     When waveforms fall below the Low Alarm level, the waveforms will change color. When waveforms rise above the High Alarm level, the waveforms will change color. Alarms can also be used to output a signal via the utility port.

7. When you are finished setting up channel amplifiers, choose the OK button.

Note: To increase the number of decimal places displayed, please contact technical support.

Engineering unit setup

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

All signal information enters the Dash 18X 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.
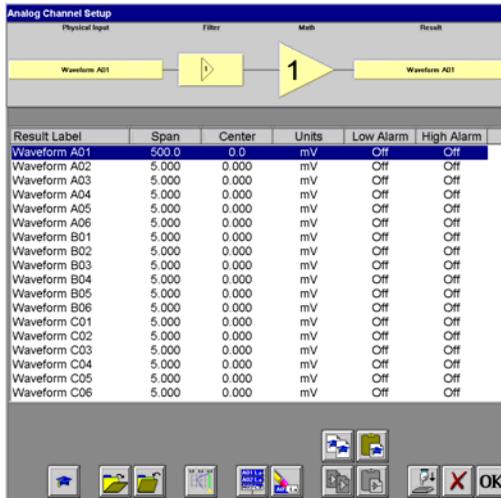
2. Set up user engineering units by selecting a waveform and choosing the Units column heading, then choose Enter User Units.
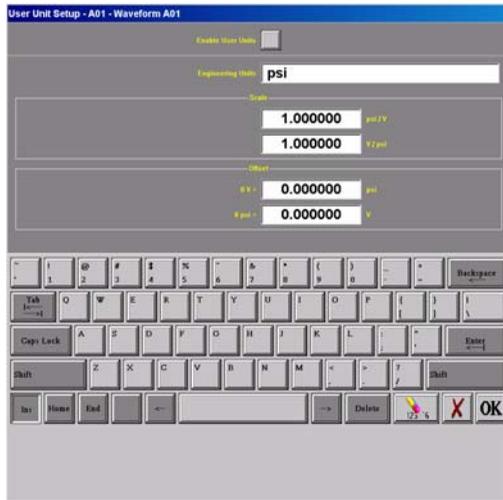
Check the Enable User Units option. This option activates engineering units.

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.

Choose the Scale field. A number pad will appear. Enter the waveform change in user engineering units that is equal to one voltage 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 voltage units

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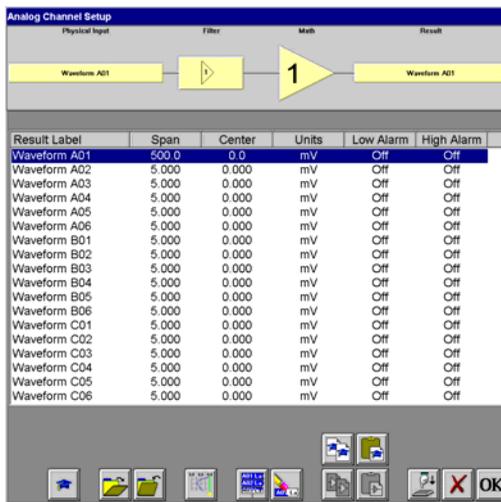
2. Set up amplifier options of the waveform that will use engineering units. Then choose the Apply button.

3. Set up user engineering units by selecting the waveform and choosing the Units column heading, then choose Scale to Levels (DC).
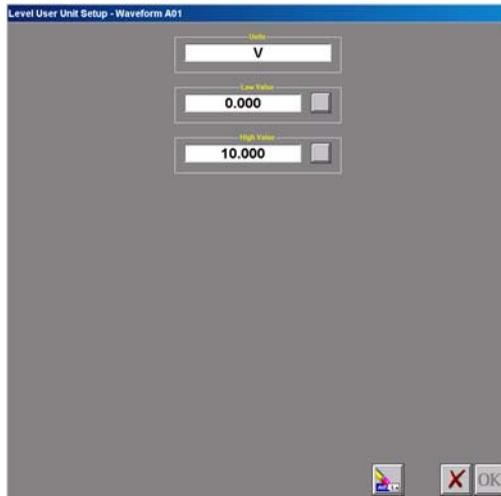
Choose the Units field and enter a name (display label) for the units. For example, PSI would be an appropriate label denoting pounds per square inch.

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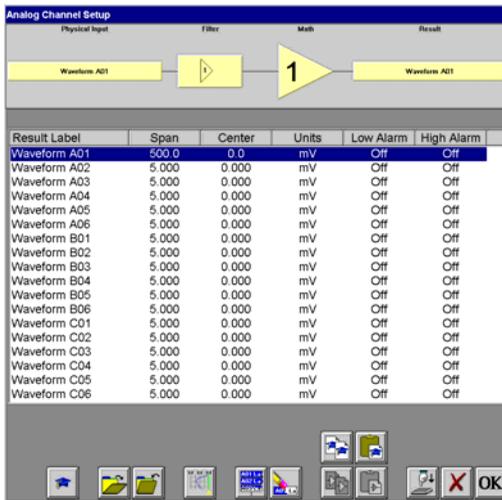
2. Set up user engineering units by selecting the waveform and choosing the Units column heading, then choose Scale to Signal (AC).

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.

Setting up signal inputs

In addition to physically connecting the input signals, you will also need to set up the Dash 18X software to accept the inputs you are using.

To set up signal inputs:

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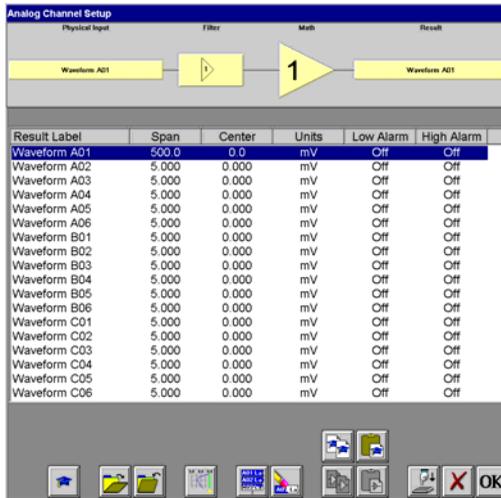
2. Choose the Advanced Setup button.

The Input Setup window will appear.
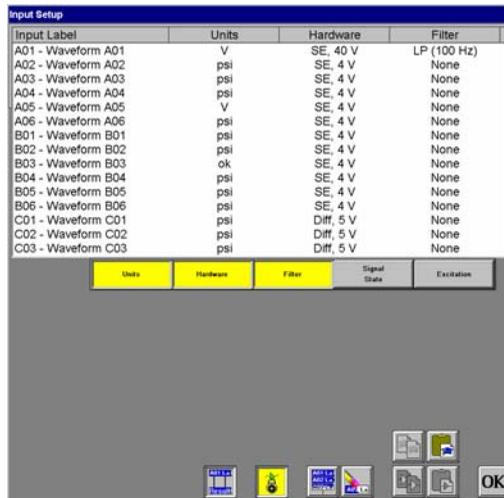
All of the columns cannot be viewed at the same time. To display columns that are not visible, choose the corresponding navigation button. The screen will scroll to display the column.

3. Enter label names for inputs by selecting an input and choosing the Input Label column heading. A keypad will appear. Enter an input name and choose OK.

4. Specify user engineering units, if necessary. This setup option should be used only for channels that will be used as math channels. To use engineering units on non-math channels, refer to the Basic channel amplifier setup process in this chapter.

Enter engineering units by selecting an input and choosing the Units column heading. The User Unit Setup window will open.
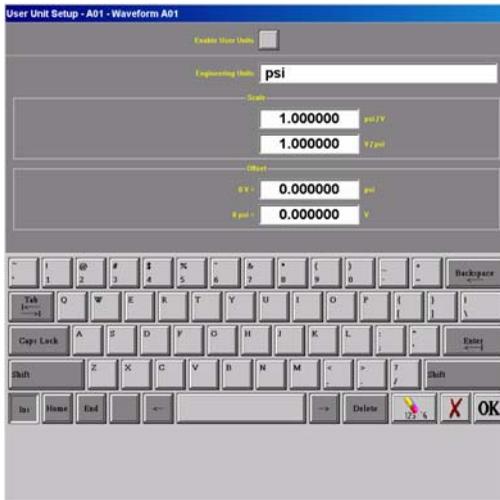
Check the Enable User Units option. Then enter the name of the units you wish to use in the Engineering Units field.

Enter a scale for the user units. Scale can be specified as either the waveform change in user units that is equivalent to one module unit or the waveform change in module units that is equivalent to one user unit. Only one entry is required; the other is derived automatically.

Enter an offset for the user units. Offset can be specified as either the number of user units that are equivalent to zero module units *or* the number of module units that are equivalent to zero user units. Only one entry is required; the other is derived automatically.

When you are finished entering user units, choose the OK button.
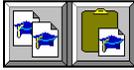
> **Note:** When user units are set up, you can copy and paste user units information among channels by using the Copy and Paste User Units buttons.
>
>
>
> You can also copy and paste user units information among channels between the Input Setup window and Analog Channel Setup window.

5. Specify the hardware type for inputs by selecting an input and choosing the Hardware column heading. The hardware type refers to how the signal is physically connected to the Dash 18X.

    A list of hardware options will appear. Choose the option that best meets your needs. The following hardware types and attenuator ranges are available:

    - Single-ended (4, 40, 400, or 800 V full scale)

    - Differential (50, 500, or 1600 mV full scale)

    - DC bridge (50 or 500 mV full scale)

    - Thermocouple (Types J, K, E, T, or N)

6. Specify a signal state for inputs by selecting an input and choosing the Signal State column heading. A list of signal state options will appear. Choose the option that best meets your needs.

    The following signal states are available:

    - Signal (passes signal)

    - Inverted signal (inverts signal polarity)

    - Ground (does not pass signal)

7. Select an excitation voltage for inputs by selecting an input and choosing the Excitation column heading. A number pad will appear. Enter an excitation voltage and choose OK. This process is necessary only for DC bridge inputs.

8. Specify whether to link the input label to the result label using the Link Input/Result Labels button. When you press this button, the button icon will change to indicate its current state.

  This icon indicates that the labels *are not* linked.

  This icon indicates that the labels *are* linked.

Note: Linking input and result labels is highly recommended for most non-math channels.

9. Specify whether to lock the attenuator ranges using the Hardware Lock button. When you press this button, the button icon will change to indicate its current state.

Locking the attenuator ranges prevents a user from changing attenuators during the Basic channel amplifier setup process detailed in this chapter.

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

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

10. When you are finished setting up the inputs, choose OK in the Input Setup window. Then choose OK in the Analog Channel Setup window.

Channel amplifier setup – filtering and math

If necessary, you will use the Analog Channel Setup window to set up advanced settings for your channels.

Advanced setup allows you to define flows of data that are not represented by a physical input, but instead result from operations performed on data from physical inputs. Data in these "virtual channels" can be assigned to a channel location for display and measurement.

Note: This section is intended for users with advanced channel setup needs. If you have basic channel setup needs, refer to the previous sections for instructions on setting up channels.

To set up filtering and math:

1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.
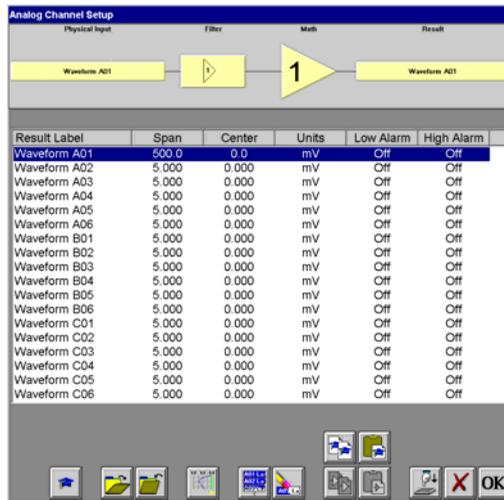
The upper portion of the window display the formula used to calculate the value of the channel selected in the lower list.

2. Select a channel location from the list. This channel will contain the result of the operations performed on input signal(s).

3. Select the physical input(s) by choosing the Physical Input button. For each channel selected, you can choose from groups of three physical blocks (1, 2, 3 or 4, 5, 6, etc.).

   The Physical Input button allows you to select the signals from the physical inputs to include in the operations you define.

4. If necessary, apply filters by choosing the Filter button. A list of filter options will appear. Choose one of the filter options.

Note: For single-ended and differential, all of the following filters are available. For DC bridge inputs, only low pass and band stop are available. For thermocouple inputs, only low pass filtering is available.

   None - All filters off

Low-pass (LP) – Filters out frequencies above a specified cutoff point. (Choice of 1 – 10,000 Hz)

High-pass (HP) – Filters out frequencies below a specified cutoff point. (Choice of 0.1, 0.5, 1, 2, 5, 10, 20, 25, 50, or 100 Hz).

Band-pass (BP) – Filters out frequencies below a specified cutoff point and above a specified cutoff point, allowing only a selected band of frequencies to pass. (Choice of lower limit from high-pass filter and upper limit from low-pass filter)

Band-stop (BS) or Notch – Filters out a specific frequency (50 or 60 Hz).

RMS - Filters the signal to its root-mean-square value. Because the RMS value is a measurement taken over some time period, a variety of response times are available to optimize the calculation for a particular input.

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

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

Note: All RMS filters use the 100 kHz sampling Rate.

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

Frequency (FREQ) – Frequency counter supported by input channels 1, 7, and 13.

Automatic - This type is a low-pass filter with a value of the sample rate divided by four.

Note: You will need to repeat this step for each channel that will use a filter.

To apply the same filter to multiple channels simultaneously, choose the Advanced Setup button to open the Input Setup window.

Select the group of channels, and then choose the Filter column heading to select a filter. This method will save you time when applying the same filter to multiple channels.

5. If necessary, apply math functions by choosing the Math button and selecting a math function. A second Physical Input button will display. Select the desired input using this button.

Note: Math functions are only available between channels in three-channel blocks. For example, channels A01, A02, A03 or A04, A05, A06, etc.

Available math functions include:

None - Math functions off

Addition - Add two selected channels

Subtraction - Subtract second channel from first

Multiplication - Multiply two channels

Division - Divide the first channel by the second

True Power, Apparent Power, or Power Factor - Refer to the following section for information on these math functions.

6. The Result box in the upper-right of the window indicates the channel location that will display the result of the operations in the formula to the left of it.

7. When you are finished setting up advanced channel settings, choose the OK button.

Power functions

True power, apparent power, and power factor calculations are available from the Math button in the Analog Channel Setup window.

True Power - Determines the power value delivered to a load. True Power is obtained by averaging the product of voltage and current waveforms over one cycle of the source. The default units value for True Power is VA.

Apparent Power - Compares voltage and current. Apparent Power is defied as Vrms x Irms, where V is the voltage across a load and I is the current flowing to the load. The default units value for Apparent Power is VA.

Power Factor - Calculates the ratio between True Power and Apparent Power. Because the Power Factor is a ratio, no units value is assigned.

Power functions are only available between channels in three-channel blocks. For example, channels A01, A02, A03 or A04, A05, A06, etc. It is assumed that the first channel in the block is a voltage and the second is a current. Each channel in each block can perform one specific power function:

Position	Power function
First channel (A01, A04, etc.)	True Power
Second channel (A02, A05, etc.)	Apparent Power
Third channel (A03, A06, etc.)	Power Factor

Use the following instructions to set up power functions for channels.

To use power functions:

1. Choose Setup >> Amplifier Channel Settings to open the Analog Channel Setup window.
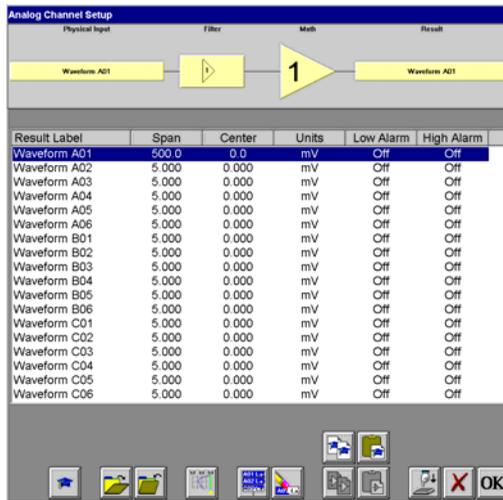
2. Select a power function type.

Note: Remember that first channel in the three-channel block should be a voltage and the second should be a current.

- To select True Power, select the first channel (A01, A04, B01, B04, C01, or C04) in a three-channel block. Then choose the Math button and select the True Power option.

- To select Apparent Power, select the second channel (A02, A05, B02, B05, C02, or C05) in a three-channel block. Then choose the Math button and select the Apparent Power option.

  A slow RMS filter will be applied to the waveforms in the first and second channels in the block.

- To select Power Factor, select the third channel (A03, A06, B03, B06, C03, or C06) in a three-channel block.

  A slow RMS filter will be applied to the waveforms in the first and second channels in the block.

When a power math function is enabled, the physical input or filter values cannot be modified.

3. When you are finished setting up the power function, choose the OK button.

Saving channel amplifier settings to a file

Once your channel amplifier settings are set up, you can save the settings to a configuration file on disk. This process allows you to load the configurations in the future by simply loading the configuration file.

Note: In Realtime mode, you can save your current amplifier settings by choosing File >> Save >> Signals from the menu bar. This shortcut will save you time when working in Realtime mode.

To save channel amplifier settings to a file:

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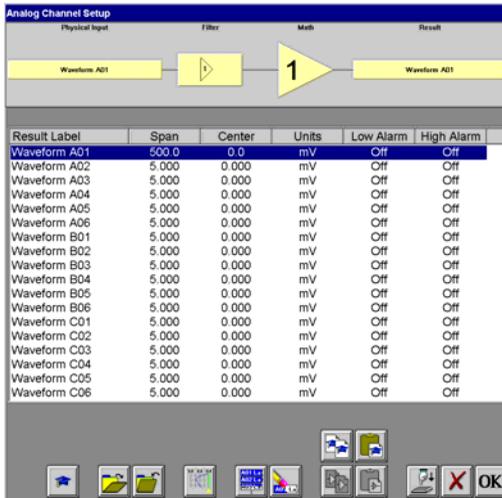
2. Choose the Save button.

The Save Signal Setup File window will open.

3. If necessary, select the drive to save the file on. By default, drive C is selected.

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

   Enter a name for the signals file and choose OK.

5. Choose OK in the Save Signal Setup File window. This window will close and you will return to the Analog Channel Setup window.

   Choose OK to close the Analog Channel Setup window.

Loading channel amplifier settings from a file

You can load your custom channel amplifier settings from a file. You will load the channel amplifier settings file by retrieving the file from disk.

Note: In Realtime mode, you can load your amplifier settings by choosing File >> Load >> Signals from the menu bar. This shortcut will save you time when working in Realtime mode.

To load channel amplifier settings from files:

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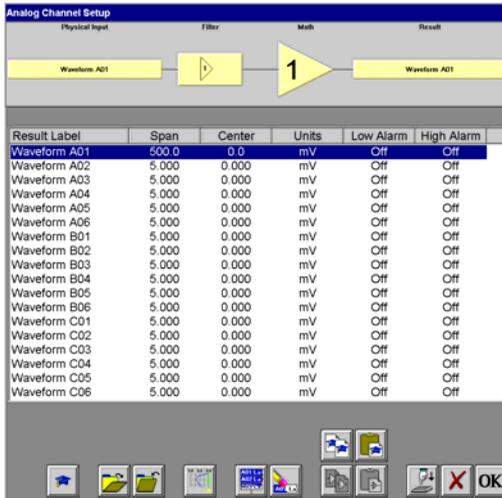
2. Choose the Load button.

The Load Signal Setup File window will open.
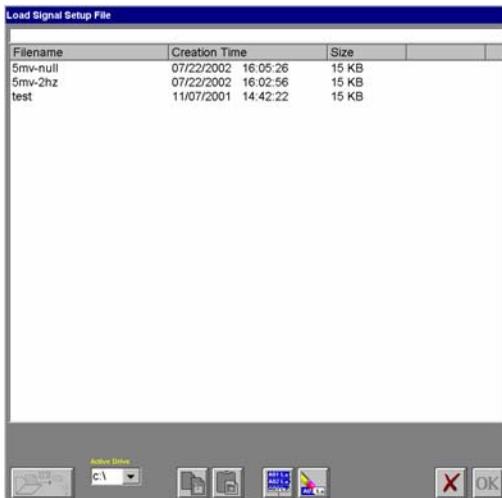
3. If necessary, select the drive where the file is located. By default, drive C is selected.

4. Select a signal setup file and choose OK. The signal configurations will be loaded.

   Choose OK in the Load Signal Setup File window. This window will close and you will return to the Analog Channel Setup window.

   Choose OK to close the Analog Channel Setup window.

Chapter 6: Viewing channels & channel information

The Dash 18X provides flexible viewing options for channels. This chapter will help you learn about the channel viewing options.

Setting up basic channel views

The Dash 18X provides basic channel views that you can access from the menu bar. These views allow you to choose a fixed number of channels to display. You can choose from these pre-set views or create your own custom view.

To view basic channels:

1. Choose View from the menu bar.

2. Do one of the following:

   - In Realtime mode, choose from Basic 6, 12, or 18 Channels views.

   - In Scope or Review mode, choose from 1, 2, 6, 12, or 18 Channel views.

Setting up custom channel views

You will use the Chart/Display Wizard to set up custom channel views. This wizard provides you with control over channel display. You will view only the channels and events that you choose. Additionally, you can set up a chart format that meets your needs.

To set up custom channel views:

1. Choose View >> Chart/Display Wizard to open the Chart/Display Wizard window.
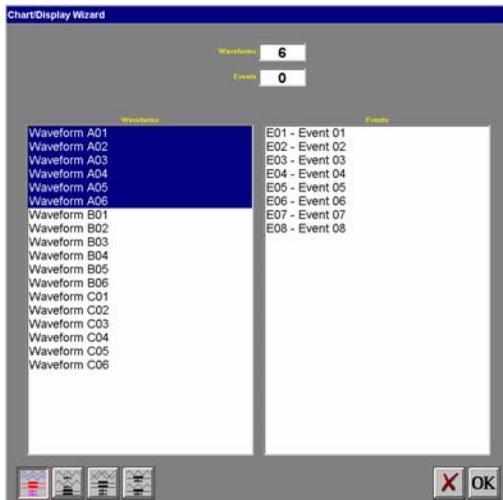
> **Note:** If the optional printer is connected, a Text field will also be available in this window. Use this option to indicate the number of text channels (1-18) to record on the printed chart.

2. Select the waveforms and events you want to display.

3. 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.

4. Choose the OK button. Your custom channel view will display.

Saving custom views to files

When the Realtime mode display is set up to your desired view, you can save the settings to a configuration file on disk.

To save custom views to files:

1. Choose File >> Save >> View from the menu bar. The Save View Setup File window will open.

2. If necessary, select the drive to save the file on. By default, drive C is selected.

3. Press the filename field above the list column headings. A keypad will appear.

    Enter a name for the view setup file and choose OK.

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

Loading custom views from files

Before working with the Dash 18X, you can load your custom view from a file. You will load view setup files by retrieving the files from disk.

To load custom views from files:

1. Choose File >> Load >> View from the menu bar. The Load View Setup File window will open.
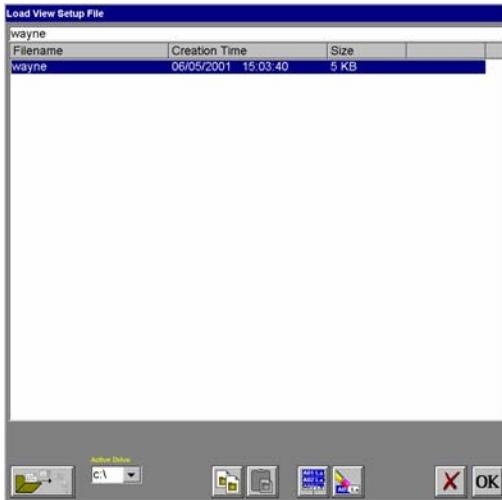
2. If necessary, select the drive where the file is located. By default, drive C is selected.

3. Select a view setup file and choose OK. The view will be loaded.

Chapter 7: Managing channels

During your test and measurement process, you might need to overlap, replace, or remove channels. This chapter will help you learn how to use the Dash 18X channel management features.

Overlapping channels

You will overlap channels by adding a new channel to an existing channel. You will accomplish this task using a menu that appears when you choose a channel label button.

To overlap channels:
1. Press the channel label button on the right side of a waveform. An options menu will appear.

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

3. Choose a channel. The channel you select will be overlapped with the existing channel.

Replacing channels

You will replace channels using a menu that appears when you choose a channel label button.

To replace channels:
1. Press the channel label button on the right side of a waveform. An options menu will appear.

2. Choose Replace. A list of channels will appear.

3. Choose a channel. The channel you select will replace the existing channel.

Note: If more than one channel is present, you will be prompted to select the channel to replace before choosing the replacement channel.

Removing channels

You will remove channels using a menu that appears when you choose a channel label button.

To remove channels:
1. Press the channel label button on the right side of a waveform. An options menu will appear.

2. Choose Remove. A list of channels will appear.

3. Choose a channel. The channel you select will be removed.

    Note: If more than one channel is present, you will be prompted to select the channel to remove.

Choosing an active channel

When there are two or more overlapping, or overlapped channels, you may need to designate one of the channels as the active channel. You will accomplish this task using a menu that appears when you choose a channel label button.

Note: This procedure applies only to overlapped channels. You cannot perform this task on a single channel. Note, however, that a single channel is always considered an active channel.

To choose an active channel:
1. Press the channel label button on the right side of a waveform. An options menu will appear.

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

3. Choose a channel. The channel you select will become active.

Chapter 8: Setting up grids

You can set up the Dash 18X to draw waveforms on top of customizable grids. These grids provide a visual aid that you can use during your data acquisition and measurement process. This chapter will help you learn about the Dash 18X grid features.

Applying grids to channels

You will apply grids to channels using a menu that appears when you choose a channel label button. Grid choices are available based on the number of divisions in the grid. You will select a grid for one channel at a time. If necessary, you can use different grids for each channel. A "No Grid" option is also available in case you need to view the channel without a grid.

To apply a grid to a channel:

1. Press the channel label button on the right side of a waveform. An options menu will appear.

2. Choose Edit Grid. A list of grids will appear.

3. Choose a grid. The grid you select will be applied to the channel.

   Note: If you need a grid that does not appear on this list, you can create custom grids with specific numbers of divisions. Refer to the following section for details on this process.

Creating custom grids

If the default grids do not meet your needs, you can create custom grids with specific numbers of divisions. You will create these grids using the Define Grids window in Realtime mode.

To create a custom grid:

1. Choose Setup >> Define Grids to open the Define Grids window.
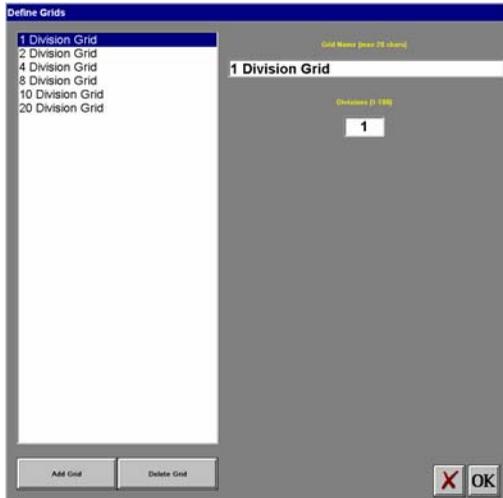
2. Choose the Add Grid button. A grid named "Untitled Grid" will be added.

3. Choose the Grid Name field. A keypad will appear.

   Replace the name "Untitled Grid" by entering a name for the new grid. Then choose the OK button.

4. Choose the Divisions field. A number pad will appear.

   Enter the number of divisions to include in the new grid. Then choose the OK button.

5. Choose the OK button in the Define Grids window. The new grid will be saved and made available for use.

Moving grids

If the default location of a grid does not meet your needs, you can move the grid vertically to a new position. You will move grids using a menu that appears when you choose a channel label button

To move a grid:

1. Press the channel label button on the right side of a waveform. An options menu will appear.

2. Choose Move Grid. A grid position slider bar will appear.
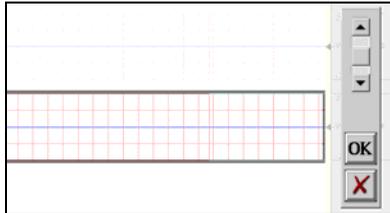
3. Move the grid vertically using the slider bar. When you are satisfied with the grid position, choose the OK button. The grid will be moved to the new position.

Note: If you move a grid on top of an existing grid, the channels will be combined into one grid.

Resizing grids

If the default size of a grid does not fit your needs, you can resize the grid to cover a larger or smaller vertical area. You will resize grids using a menu that appears when you choose a channel label button.

To resize a grid:

1. Press the channel label button on the right side of a waveform. An options menu will appear.

2. Choose Size Grid. A grid size slider bar will appear.
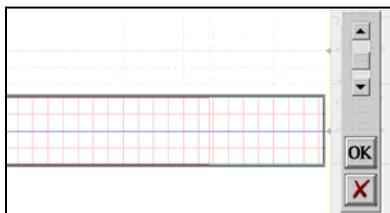
3. Resize the grid vertically using the slider bar. When you are satisfied with the grid size, choose the OK button. The grid will be resized to the new dimensions.

Selecting a grid color

If the default color of the grid does not meet your needs, you can apply a different color. You will select a grid color using the menu bar.

To select a grid color:
1. Choose View >> Grid Color. A group of color options will appear.

2. Choose a color. The color you select will be applied to all grids on the screen.

Selecting a background color

If the default color of the background does not meet your needs, you can apply a different color. You will select a background color using the menu bar.

To select a background color:
1. Choose View >> Background Color. A group of color options will appear.

2. Choose a color. The color you select will be applied to the background of the waveform display area.

Chapter 9: Setting up waveforms

As signals are measured, the Dash 18X will draw waveforms to illustrate the signals. This chapter will help you learn about setting up these waveforms.

Selecting waveform colors

You will select a waveform color using a menu that appears when you choose a channel label button.

To select a waveform color:
1. Press the channel label button on the right side of a waveform. An options menu will appear.

2. Choose Waveform Color. A list of waveform colors will appear.

3. Choose a color. The waveform color you select will be applied to the waveform in the channel.

   Note: If more than one channel is present, you will be prompted to select the channel with the waveform you wish to color before choosing the waveform color.

Selecting waveform alarm colors

You will select a waveform alarm color using a menu that appears when you choose a channel label button.

To select a waveform alarm color:
1. Press the channel label button on the right side of a waveform. An options menu will appear.

2. Choose Alarm Color. A list of waveform alarm colors will appear.

3. Choose a color. The waveform alarm color you select will be applied to the waveform in the channel.

   Note: If more than one channel is present, you will be prompted to select the channel with the waveform you wish to color before choosing the waveform alarm color.

Selecting waveform overrange colors

You will select a waveform overrange color using a menu that appears when you choose a channel label button.

To select a waveform overrange color:
1. Press the channel label button on the right side of a waveform. An options menu will appear.

2. Choose Overrange Color. A list of waveform overrange colors will appear.

3. Choose a color. The waveform overrange color you select will be applied to the waveform in the channel.

Note: If more than one channel is present, you will be prompted to select the channel with the waveform you wish to color before choosing the waveform overrange color.

Pen options

Waveforms are drawn by on-screen pens. You can control whether these pens are raised or lowered, and can also control the appearance of the pens.

Raising and lowering pens

You will use the Pen Up/Pen Down Control window to raise and lower pens. When pens are in the up position, waveforms will not be drawn. When pens are in the down position, waveforms will be drawn.

To raise and lower pens:

1. Choose Setup >> Pen Lift from the menu bar. The Pen Up/Pen Down Control window will open.
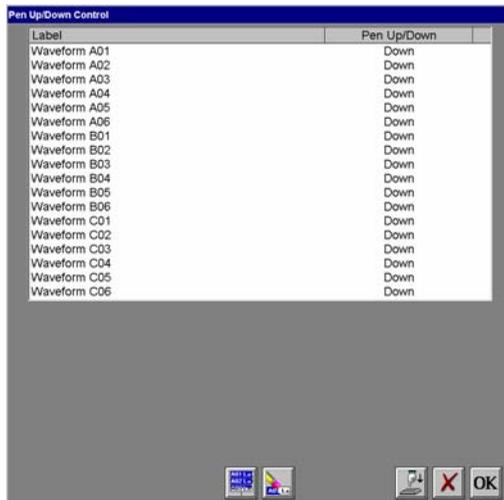
This window displays a list of waveforms and the current pen position for each waveform.

2. Raise or lower the pens as needed.

    *   To raise pens, select the rows with the pens you want to raise. Then select the Pen Up/Down column heading and choose Up. The pens for the selected waveforms will be raised.

    *   To lower pens, select the rows with the pens you want to lower. Then select the Pen Up/Down column heading and choose Down. The pens for the selected waveforms will be lowered.

3. Choose the OK button. The up and down pen settings will be applied.

Selecting pen styles

You will use the menu bar to choose from the following three pen styles:
None, Moving Origin, or Fixed Origin.
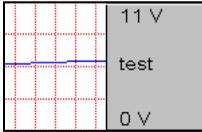
None – Does not display a pen.
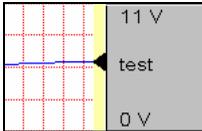
Moving Origin – Displays a pen that moves vertically as it draws the waveform.
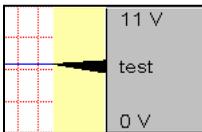
Fixed Origin – Displays a pen with a tip that tilts vertically on a fixed axis as it draws the waveform.

To select pen styles:

1. Choose your desired pen style.

   - To select the None pen style, choose View >> Pen Style >> None from the menu bar.

   - To select the Moving Origin pen style, choose View >> Pen Style >> Moving Origin from the menu bar.

   - To select the Fixed Origin pen style, choose View >> Pen Style >> Fixed Origin from the menu bar.

Chapter 10: Setting up events

Monitoring the state of events may be an important part of your measurement process. This chapter will help you learn about the Dash 18X event features.

About events

An event marker or "event" is a binary or two-state signal that, when enabled, is printed on the chart or displayed on a monitor. You can connect your event inputs using the event input port.

The state of an event is referred to as either high or low. Changes in the event input signal cause the state of an event to change. When the state of an event changes, the event's appearance on the screen changes. For example, the event may shift up and down to visually indicate changes in the event input signal.

Changes in any of three different types of input can cause the state of the event to change.

Operation of a physical switch – The event is high when the switch is open, and low when the switch is closed

Changes in an input TTL logic level – The event is high when the input level is a TTL high, and low when the input level is a TTL low.

Changes in an input DC voltage – The event is high when the DC voltage is above 0.5 volts, and low when the DC voltage is below 0.5 volts.

Caution: Maximum DC voltage is 5 volts. Higher voltages will damage the event input circuit.

Selecting event styles

When enabled, events can be printed on the screen or chart in any of four styles. The following diagram illustrates the available event styles:

Standard ⎍⎍⎍⎍⎍   Bar/line ▮ ▮ ▮ ▮ ▮

Tick | | | | | | | | Bar/off ▮ ▮ ▮ ▮
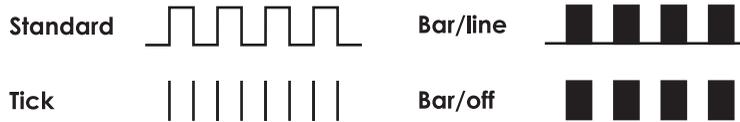
To select an event style:
1. Choose View >> Event Style from the menu bar. A list of event style will appear.

2. Choose an Event Style.

Adding/removing events

You will add and remove events using the Chart/Display Wizard window.

Note: In Realtime and Review modes, you can add, move, remove, and modify event colors by choosing Setup >> Events from the menu bar.

To add/remove events:
1. Choose View >> Chart/Display Wizard to open the Chart/Display Wizard window.
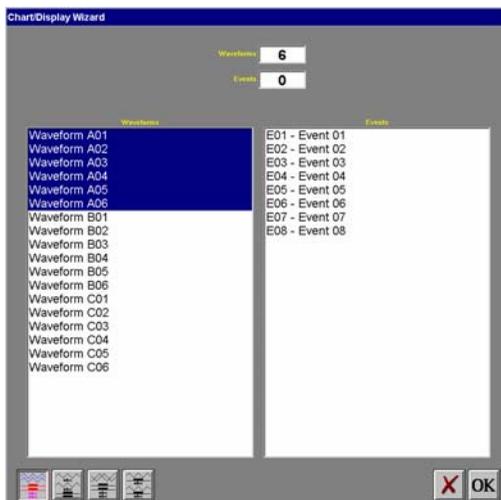
This window displays a list of events on the right side. Currently selected events will be highlighted.

2. Add and remove events.

- Select the events that you want to add.

- Deselect the events that you want to remove.

3. When you are finished adding and removing events, choose the OK button. The selected events will appear on the chart.

Chapter 11: Using the SCR-18 printer

The optional SCR-18 printer creates hard-copy output of your data. This chapter will help you learn about the features related to the SCR-18 thermal printer.

Halting and running the chart

During your measurement process, you might need to stop the chart from printing.

To freeze and run the chart:

1. Halt and run the chart by choosing Chart >> Halt from the menu bar.

2. Repeat this process to toggle between the stopped and running chart states.

Printing a page mark on the chart

During your measurement process, you might need to print a page mark line on your output.

To print a page mark on the chart:

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 in a time interval set (based on seconds) that you select.

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 the trilevel rate you want to use. The rate will be applied.

Using the demand buffer

The demand buffer is a string of text stored in memory that you can print on the chart when needed. This text prints parallel to the time axis. Use the following information to learn how to use the demand buffer.

Printing the demand buffer

Use the following instructions to print the demand buffer text on the chart.

To print the demand buffer:
1. Choose Chart >> Print Demand Buffer.

The demand buffer will print on the chart.

Editing the demand buffer

Use the following instructions to edit the text of the demand buffer.

To edit the demand buffer:
1. Choose Chart >> Edit Demand Buffer. A keypad will appear with the current demand buffer entered in the text field.

2. Enter the demand buffer text. When you are finished entering the demand buffer, choose OK.

Moving the demand buffer

Use the following instructions to change the location of the demand buffer on the printed 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.
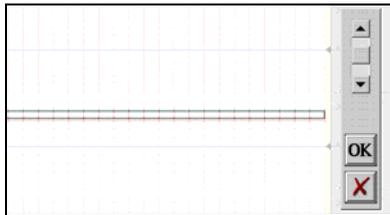
2. Move the demand buffer vertically using the slider bar. When you are satisfied with the position, choose the OK button. The demand buffer will be moved to the new position.

Printing channel limits

Use the following instructions to print the upper and lower values for each channel.

To print channel limits:
1. Choose Chart >> Print Limits from the menu bar.

   The upper and lower limits for each channel 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 measurements. Use the following instructions to edit the system log text.

To edit the system log:

1. Choose Chart >> Edit System Log from the menu bar. A keypad will appear with the current system log entered in the text field.

2. Enter the system log text. When you are finished entering the system log, choose OK.

Printing channel reports

When the channel reports option is enabled, channel scaling information will print 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. Repeat this process to turn off channel reports.

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. Repeat this process to turn off the Auto ID option.

Adjusting the waveform trace thickness

The thickness of waveforms can be changed for better contrast with the grid.

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 size you select will be used to print waveforms.

Using text buffers

The Dash 18X provides a total of 18 text buffers, which you can add and remove as needed. Buffer text will be periodically printed parallel to the time axis. You can also edit the text stored in the buffer and move the buffer to a different location on the chart.

Adding text buffers

Use the following instructions to add text buffers to your chart. The process of adding text buffers to your chart involves entering the text you wish to store in the buffer and positioning the buffer on 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 already exist will be "grayed out," and you will not be able to select them.

    A keypad will appear.

2. Enter the buffer text. When you are finished entering the buffer, choose OK. A buffer position slider bar will appear.

---
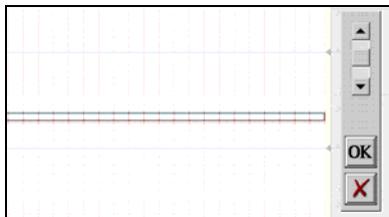
3. Move the text buffer vertically using the slider bar. When you are satisfied with the position, choose the OK button. The text buffer will be added in the position you selected.

Removing text buffers

Use the following instructions to remove text buffers from your 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 you will not be able to select them.

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 you will not be able to select them.

A keypad will appear.

2. Edit the buffer text. When you are finished editing the buffer, choose OK.

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 you will not be able to select them.

    A buffer position slider bar will appear.
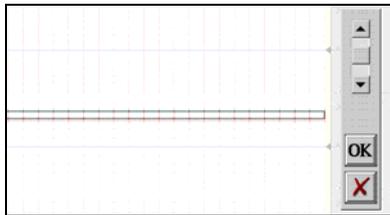
2. Move the text buffer vertically using the slider bar. When you are satisfied with the position, choose the OK button. The text buffer will be moved to the position you selected.

Chapter 12: Realtime mode general information

This chapter will help you learn about miscellaneous features in Realtime mode.

Default Realtime mode control panel

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

Chart Speed controls the rates at which the chart moves. Five default options are provided (1, 5, 25, 50, 100 mm/s).

By customizing the control panel, you can add specific speed buttons in units of millimeters per second or minute.

Analog Channel Setup opens the Analog Channel Setup window, which you will use to set up channels.

Meter opens the channel meter, which displays numeric values for signals.

Run/Freeze Display starts and stops the chart from running.

Context Help provides online help based on the mode or the menu you are using.

Icon Help provides online help for identifying the purpose of buttons.

Setting up Realtime mode

Use the following information 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.
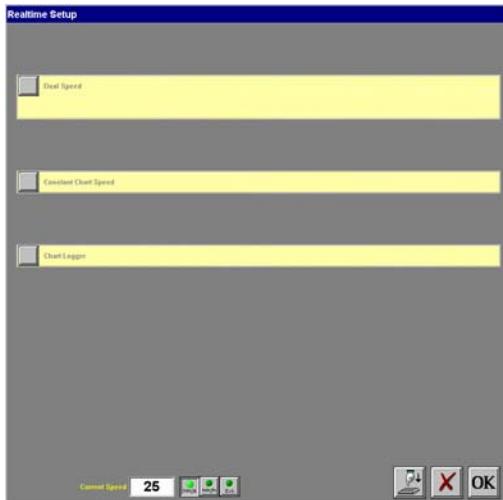
Three setup options are available in this window: Dual Speed, Constant Chart Speed, and Chart Logger. You will select one of these options to set up Realtime mode.

2. To use a dual chart speed, check the Dual Speed option. Dual speed is useful if you want to increase the chart/display speed based on a trigger. A group of dual chart speed options will appear.

Select the following options for chart speed 1:

- Enter a chart speed in millimeters per second or millimeters per minute.

- Select a duration for the speed using a specific amount of time, or until a trigger occurs.

Select the following options for chart speed 2:

- Enter a chart speed in millimeters per second or millimeters per minute.

- Select a duration for the speed using a specific amount of time, or until an abort occurs.

3. To use 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 use the chart logger, check the Chart Logger option. This option uses the SCR-18 printer.
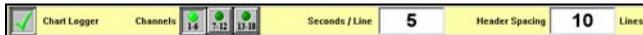
Select the channels that will be logged and specify a value for seconds/line.

Also enter a header spacing value, which indicates how frequently the header is printed. The header lines contain information such as the date, time, logger rate, and channel labels.

5. When you are finished setting up Realtime mode, choose the OK button.

View options

Use this section to learn about the various view options available in Realtime mode.

Freezing and running the display

During your measurement process, you might need to stop the chart display from running.

To freeze and run the display:

1. Freeze and run the display by doing any of the following:

    *   Choose View >> Freeze Display from the menu bar.

    *   Choose the Run/Freeze Display button from the control panel.

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

Printing channel IDs

Printing channel IDs allows you to identify channels using small visual indicators to label waveforms.

To print channel IDs:

1. Choose View >> Print IDs from the menu bar.
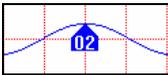
    A channel ID mark will print below each waveform.

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 chart.

To print a full page mark:

1. Choose View >> Full Page Mark from the menu bar.
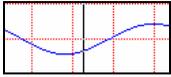
A mark will be printed vertically across all waveforms.

Viewing the time log

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

If necessary, you can customize the time log to meet your requirements.

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.

Measuring channel information with cursors

Cursors are a valuable tool that you will use for measuring channel signals. You will use cursors in conjunction with the Channel Information window to view a variety of measurements.

Note: You can save time and gain more control over cursors by adding cursor buttons to your control panel.

To measure channel information with cursors:

1. Show one or both of the cursors by doing one or both of the following:

   - Choose Cursors >> Show Cursor A

   - Choose Cursors >> Show Cursor B

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 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 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 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 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 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.

You can change the measurement type by choosing the Measurement Type option on the menu bar of this window.

3. If necessary, move one or both of the cursors. Before you move a cursor, you will need to activate it. Do one of the following:

- **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: You can also activate a cursor by pressing it once. To move it, press it again but don't release it. While pressing, drag it to a new location, and then release it.

As you move the cursors, the values in the Channel Information window will update based on the values of the new cursor locations.

4. If necessary, modify the color and type of your cursors by doing the following:

- Modify the cursor color by choosing Cursors >> Color. Then select a color from the options that appear.

- Modify the cursor type by choosing Cursors >> Type. Then select a type from the options that appear. If you select the Color option, color cursors will be used. If you select the Invert option, inverted cursors will be used, providing a pseudo-transparent cursor.

Display features

Realtime mode provides display tools to help you better understand your data. Use this section to learn more about Realtime mode display features.

Using the channel meter

Displaying the channel meter allows you to view current values for the signals you are measuring. The channel meter can be configured to display information about viewed channels or all channels.

To use the channel meter:

1. Display the channel meter by doing any of the following:

    *   Choose View >> Meter from the menu bar.

    *   Choose the Meter button from the control panel.

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

2. If necessary, modify the meter display by choosing Setup >> Meter Channels, and then choose either View Channels or All Channels.

    *   If you choose View Channels, the meter will display information only for the channels you are currently viewing.

    *   If you choose All Channels, the meter will display information for all channels, regardless of the channels you are currently viewing.

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.

Using the XY plot

Use the following instructions to learn how to use the XY plot. This plot also provides a second Y trace.

To use the XY plot:

1. Choose View >> XY Plot from the menu bar to open the X-Y-Y plot window.
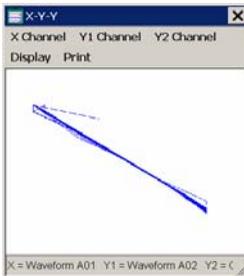
2. Select the X, Y1, and Y2 channels using the menu options.

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

   Refresh clears the window and restarts the plot drawing.

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

   Background Color allows you to select a background color for the plot window.

   Foreground Color allows you to select a color for the points and segments being drawn on the plot.

   Grid allows you to select whether to display a grid. You can also specify the number of divisions in the grid as well as the grid color.

   Point Buffer allows you to select whether turn on the point buffer. You can also specify the point buffer size.

4. If the printer is installed, you can print the screen by choosing Print >> Print Screen from the menu.

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 5: Setting up channels*.

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

Note: For information on creating control panel setup files individually, refer to *Chapter 4: 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 4: 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 4: Menus and buttons*.

View - Saves the view and display setup.

Note: For information on creating view setup files individually, refer to *Chapter 6: Viewing channels & channel information*.

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 18X 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.
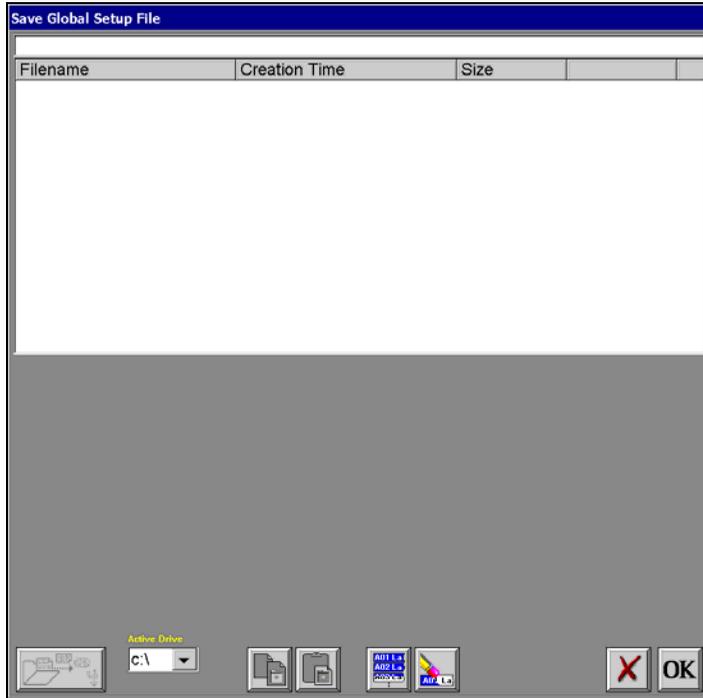
3. If necessary, select the drive to save the file on. By default, drive C is selected.

4. 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.

5. 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.

6. 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.
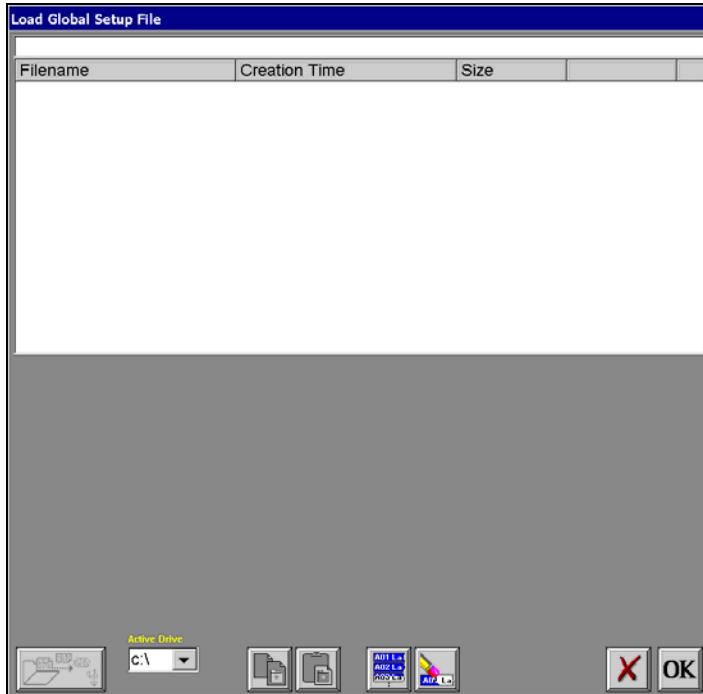
2. If necessary, select the drive where the file is located. By default, drive C is selected. Then select a file.

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

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 13: Data capture

Setting up and initiating data captures will allow you to record the information you need. This chapter will help you learn about the data capture process.

The data capture process

Data capture is the process of sampling signals and saving the sampled data in a useful format.

This process generally begins when the system is armed. Arming the system starts the pre-trigger phase, during which samples are acquired and stored up to a pre-set amount. When the pre-set amount of data has been stored, the oldest sample is replaced by the most recent, creating a circular buffer.

The sampling process continues until a valid trigger is detected and the post-trigger phase begins. In the post-trigger phase, data is acquired and stored until the user-specified number of samples are collected or the capture is aborted.

Data captures are stored on the Dash 18X internal hard drive.

Setting up data captures

Before capturing data, you will need to set up the data capture settings. This task involves specifying the data file name, channel rates, pre-trigger and post-trigger storage allocation, and other configurations.

To set up a data capture:

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

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

White – Indicates the space available for data captures.

Blue – Indicates the space already containing data captures.

Green – Indicates the amount of space the next valid data capture will use.

Red – Indicates that the next capture is too large for the hard drive

A format button is available to format the data capture hard drive. The data capture drive is formatted at the factory and only needs to be used for drive cleanup and deleting all records. Individual records can be deleted in Review mode.

2. Choose the Base Filename field. A keypad will appear. Enter a filename for the data capture and choose OK. The date and time of the capture will be added to this file name.

3. Specify your channel capture rates.

   Select a channel, and then choose the Rate column heading. A group of rate options will appear.

   - If you choose Off, data will not be captured for the selected channel.

   - If you choose Standard, data will be captured for the selected channel using the standard sample rate.

   - If you choose Slow, data will be captured for the selected channel using the slow sample rate. This option is not available when capturing pre-trigger data.

   Note: You will define values for standard and slow sample rates later in this setup process.

   Note: For IRIG models, the IRIG time code can be saved with the data. Be sure that the IRIG channel's capture rate is set to Standard.

4. Specify your pre-trigger storage allocation options.

   - If you want to capture pre-trigger data, check the Percent Pre-Trigger option. Then select the Percent Pre-Trigger field. A number pad will open.

     Enter the percentage of the capture you want used for showing the signal before the trigger occurred. If the trigger 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.

     You can select 1 to 99 percent of the capture to be pre-trigger data. Then choose OK.

   - If you do not want to capture pre-trigger data, check the No Trigger option. This option allows you to capture for a specific amount of time, regardless of triggers.

5. Select the amount of time (or samples) you wish to capture by selecting either kSamples/channel, Hours, Minutes, or Seconds, and then entering the amount.

6. Specify the sample rates you will use to collect data.

- To use a standard rate, select a Standard Rate value. Standard rates range from one sample per minute through 100,000 samples per second.

- If necessary, select a Slow Rate value. Slow rates range from one sample per minute through 50,000 samples per second. If you are capturing pre-trigger data, this option will not be available.

- To use an external rate, check the External option. This option allows you to use an external TTL sample clock up to 40,000 Hz, though rates approaching 100 KHz will work with sampling jitter of up to 10 μ.

Notes: To avoid aliasing, the sample rate must be at least two times the highest frequency in the signal to be captured. The D18 realtime software filters can be used to band limit the signal when you are unsure of possible frequency content.

You can check for aliasing by performing a test capture at 100 KHz sample rate. At this rate, the D18 hardware filters will protect against aliasing.

---

7. Specify your Automation preferences.

- If you want to re-arm the data capture automatically, check the Auto-Rearm option. This option immediately re-arms the Dash 18X for a data capture after the previous one.

  Note: Automatically rearming is helpful if you are analyzing repetitive events, but it can produce a large number of captures depending on the trigger conditions.

- If you want to archive the data capture to the system drive upon completion, check the Auto Archive button. Then choose an archive drive from the Archive Drive drop-down list.

- If you want to automatically open the DCR file in Review mode immediately after the data capture is complete, check the Auto Review option.

- If the SCR-18 thermal printer is installed, and you want to print the chart associated with the entire DCR file, check the Auto Playback option. Selecting this feature automatically enables the Auto Review option.

  Then select a playback compression value, which stretches or compresses the printed chart. Higher compression values will result in the printed chart being compressed to a smaller area. Lower compression values will result in the printed chart being stretched to a larger area.

8. Specify your Save Settings options. This feature allows you to save specific information as the first note in the data capture file.

   - If you want to save a text log as part of the data capture, check the Capture Log Text option. Then enter the text for the log.

   - If you want to save a listing of channel amplifier settings as part of the data capture, check the Amplifier Settings option.

9. When you are finished setting up the data capture, choose OK.

Triggers

Triggers are user-defined events that initiate the post-trigger data capture process. If you plan to use triggers, you will need to define them before starting a data capture. Triggers essentially define the conditions under which you will capture data. For example, when a waveform exceeds a certain level, you might want cause a trigger to capture data.

Trigger types

As part of the trigger setup process, you will need to select trigger types. Available trigger types are detailed below:

Amplitude Level OR is a commonly used method that produces a trigger when signals exceed a certain level. This type of trigger cannot be used simultaneously with the 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.

Slew OR produces a trigger when there is a change in signal slew rate, or frequency.

Slew AND produces a trigger when more than one channel must meet its respective slew rate conditions.

Events produces 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 is a commonly used method that allows the operator to produce a trigger manually via the front panel.

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.

Note: If you set up multiple triggers, a data capture will occur when any of the trigger settings are met.

Setting up triggers

Use the following instructions to set up various types of triggers.

Using the Trigger Setup Menu window

You will use the Trigger Setup Menu window to access all setup options for triggers. You will need to access this window prior to setting up each individual trigger.

To use the Trigger Setup Menu window:

1. Choose Setup >> Trigger Settings to open the Trigger Setup Menu window.
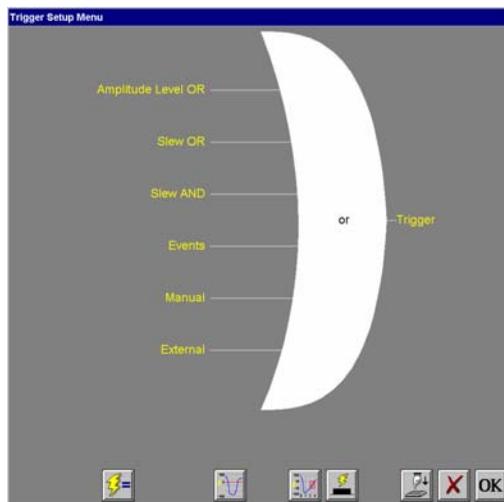
Triggers are activated and deactivated by choosing the trigger name in the graphic. Active triggers display in yellow and inactive triggers display in gray.

2. If necessary, you can view the current trigger conditions at any time by choosing the Trigger Equals button.

The current trigger configuration will be displayed.

3. Set up triggers using the following instructions. When you are finished setting up triggers, choose the OK button.

Setting up an Amplitude Level OR trigger

Use the following instructions to set up an Amplitude Level OR trigger, which starts a data capture when a signal exceeds a certain level. With the Trigger Setup Menu window open, perform the following steps to set up the trigger:

To set up an Amplitude Level OR trigger:

1. Open the Trigger Level window by choosing the Trigger Level button.

The Trigger Level window will open.

Label	Level	Slope	OR	
Waveform A01	Off	Rise	No	
Waveform A02	Off	Rise	No	
Waveform A03	Off	Rise	No	
Waveform A04	1.750	Rise	Yes	
Waveform A05	Off	Rise	No	
Waveform A06	Off	Rise	No	
Waveform B01	Off	Rise	No	
Waveform B02	Off	Rise	No	
Waveform B03	Off	Rise	No	
Waveform B04	Off	Rise	No	
Waveform B05	Off	Rise	No	
Waveform B06	Off	Rise	No	
Waveform C01	Off	Rise	No	
Waveform C02	Off	Rise	No	
Waveform C03	Off	Rise	No	
Waveform C04	Off	Rise	No	
Waveform C05	Off	Rise	No	
Waveform C06	Off	Rise	No	

2. Modify trigger levels to meet your needs.

Select a channel and choose the Level column heading. From the menu that appears, choose the New Value option. A number pad will appear. Enter the new trigger level and choose OK.

3. Select slope types.

Select a channel and choose the Slope column heading. From the menu that appears, select Rise, Fall, or Rise or Fall.

4. Select whether to enable trigger conditions for channels.

    Select a channel and choose the OR column heading. From the menu that appears, select Yes or No. If you select Yes, the trigger condition for the selected channel will be enabled. If you select No, the trigger condition for the selected channel will be disabled.

5. Choose OK in the Trigger Level window. You will return to the Trigger Setup Menu window.

6. Choose the Amplitude Level OR button. The Amplitude Level OR text will become yellow.

7. Set up other triggers, if necessary. If you are finished setting up triggers, choose OK in the Trigger Setup Menu window.

Setting up an Amplitude Window OR trigger

Use the following instructions to set up an Amplitude Window OR trigger, which starts a data capture when a signal moves outside of a pre-determined window. With the Trigger Setup Menu window open, perform the following steps to set up the trigger:
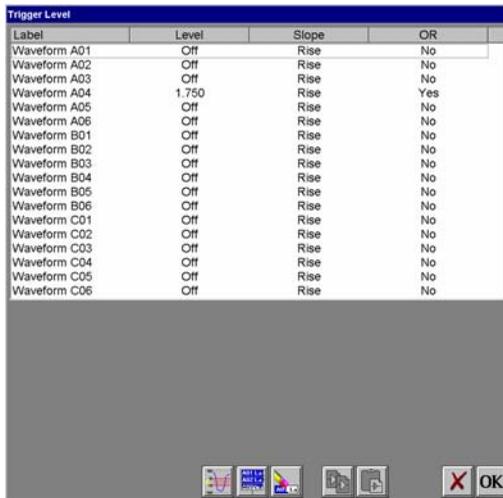
To set up an Amplitude Window OR trigger:

1. Switch from Amplitude Level OR to Amplitude Window OR.

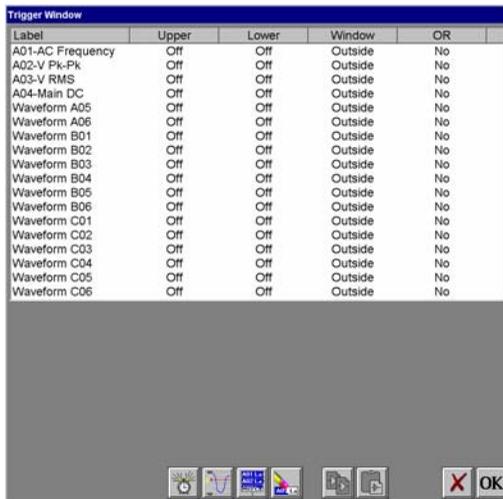   Open the Trigger Level window by choosing the Trigger Level button.

The Trigger Level window will open.

Trigger Level

Label	Level	Slope	OR
Waveform A01	Off	Rise	No
Waveform A02	Off	Rise	No
Waveform A03	Off	Rise	No
Waveform A04	1.750	Rise	Yes
Waveform A05	Off	Rise	No
Waveform A06	Off	Rise	No
Waveform B01	Off	Rise	No
Waveform B02	Off	Rise	No
Waveform B03	Off	Rise	No
Waveform B04	Off	Rise	No
Waveform B05	Off	Rise	No
Waveform B06	Off	Rise	No
Waveform C01	Off	Rise	No
Waveform C02	Off	Rise	No
Waveform C03	Off	Rise	No
Waveform C04	Off	Rise	No
Waveform C05	Off	Rise	No
Waveform C06	Off	Rise	No

Choose the Amplitude Window button.

The Trigger Window screen will appear.

Trigger Window

Label	Upper	Lower	Window	OR
A01-AC Frequency	Off	Off	Outside	No
A02-V Pk-Pk	Off	Off	Outside	No
A03-V RMS	Off	Off	Outside	No
A04-Main DC	Off	Off	Outside	No
Waveform A05	Off	Off	Outside	No
Waveform A06	Off	Off	Outside	No
Waveform B01	Off	Off	Outside	No
Waveform B02	Off	Off	Outside	No
Waveform B03	Off	Off	Outside	No
Waveform B04	Off	Off	Outside	No
Waveform B05	Off	Off	Outside	No
Waveform B06	Off	Off	Outside	No
Waveform C01	Off	Off	Outside	No
Waveform C02	Off	Off	Outside	No
Waveform C03	Off	Off	Outside	No
Waveform C04	Off	Off	Outside	No
Waveform C05	Off	Off	Outside	No
Waveform C06	Off	Off	Outside	No

2. Enter upper and lower window limits.

Select a channel and choose the Upper or Lower column heading. A number pad will appear. Enter a value and choose OK.

3. Specify whether the trigger should occur when the signal moves outside or inside of the window limits.

   Select a channel and choose the Window column heading. From the menu that appears, choose either Inside or Outside.

4. Select whether to enable trigger conditions for channels.

   Select a channel and choose the OR column heading. From the menu that appears, select Yes or No. If you select Yes, the trigger condition for the selected channel will be enabled. If you select No, the trigger condition for the selected channel will be disabled.

5. Choose OK in the Trigger Level window. You will return to the Trigger Setup Menu window.

6. Choose the Amplitude Window OR button. The Amplitude Window OR text will become yellow.

7. Set up other triggers, if necessary. If you are finished setting up triggers, choose OK in the Trigger Setup Menu window.

Setting up Slew OR & Slew AND triggers

Use the following instructions to set up Slew OR & Slew AND triggers. Slew OR produces a trigger when there is a change in signal frequency, Slew AND produces a trigger when more than one channel must meet its respective 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 trigger.

Portion of Input Signal

Voltage (Delta) 5V

2 sec
Time (dT)
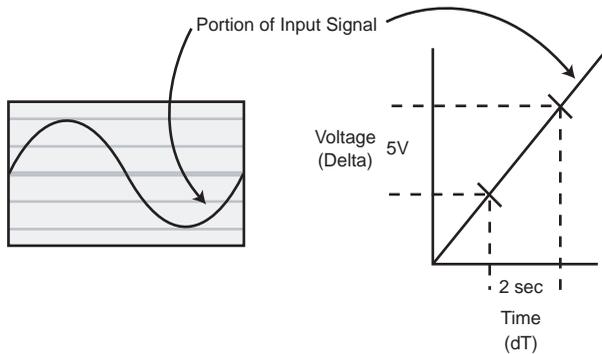
Delta - The Delta indicates the slew voltage span. The slope of this voltage span with respect to "dT" and "Window" will set the trigger point.

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

5V

7V

2 sec

Time
(dT)
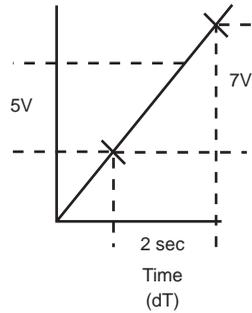
- If using a "below window," the following scenario would result in a trigger: Signal Slew Rate = 3V / 2 sec = 1.5 V/s
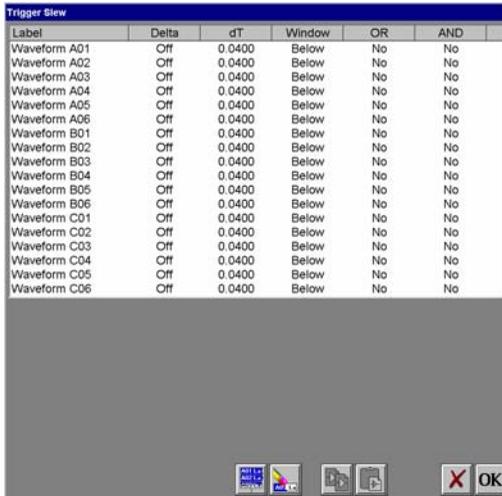

5V

3V

2 sec

Time
(dT)
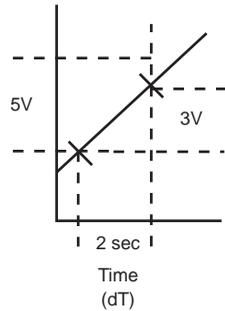
To set up Slew OR & Slew AND triggers:

1. Open the Trigger Slew window by choosing the Trigger Slew button.

The Trigger Slew window will open.

2. Select delta values.

    Select a channel and choose the Delta column heading. From the menu
    that appears, choose the New Value option. A number pad will appear.
    Enter a delta value and choose OK.

3. Select delta time values.

    Select a channel and choose the dT column heading. A number pad will
    appear. Enter a delta time value and choose OK.

4. Select whether the trigger will occur when the slew rate of the waveform
    is above or below the chosen slew rate.

    Select a channel and choose the Window column heading. From the
    menu that appears, choose the Above or Below option.

5. Select whether to enable trigger conditions for channels.

    Select a channel and choose the OR column heading. From the menu that
    appears, select Yes or No. If you select Yes, the trigger condition for the
    selected channel will be enabled. If you select No, the trigger condition
    for the selected channel will be disabled.

6. Select whether to require multiple conditions to initiate a trigger.

For each channel trigger condition that you want to require, select the channel and choose the AND column heading. Then choose Yes from the menu that appears. Triggers will occur only when all conditions are met.

7. Choose OK in the Trigger Slew window. You will return to the Trigger Setup Menu window.

8. Choose the Slew OR or Slew AND button. The text will become yellow.

9. Set up other triggers, if necessary. If you are finished setting up triggers, choose OK in the Trigger Setup Menu window.

Setting up an Event trigger

Use the following instructions to set up an Event trigger, which produces a trigger when event inputs meet their change in logic state. With the Trigger Setup Menu window open, perform the following steps to set up the trigger:

To set up an Event trigger:

1. Open the Event Trigger window by choosing the Event Trigger button.

The Event Trigger window will open.
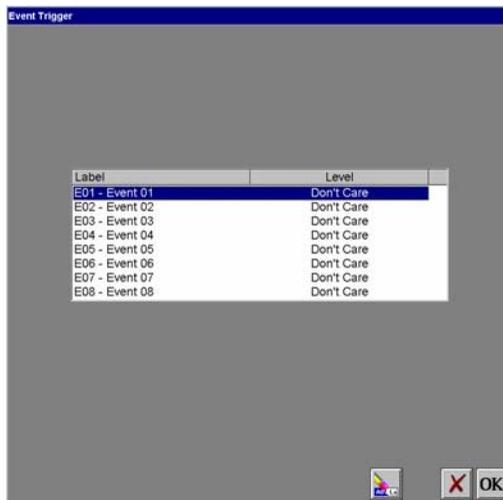
2. Select event level values that will cause a trigger.

Select a channel and choose the Level column heading. From the menu that appears, choose either Don't Care, High, or Low.

- If you choose Don't Care, changes in the selected event will not cause a trigger.

- If you choose High, a trigger will occur when the selected event changes to its high state.

- If you choose Low, a trigger will occur when the selected event changes to its low state.

3. Choose OK in the Event Trigger window. You will return to the Trigger Setup Menu window.

4. Choose the Event button. The text will become yellow.

5. Set up other triggers, if necessary. If you are finished setting up triggers, choose OK in the Trigger Setup Menu window.

Setting up a Manual trigger
Use the following instructions to set up a Manual trigger, which allows the operator to produce a trigger manually via the front panel. With the Trigger Setup Menu window open, perform the following steps to set up the trigger:

To set up a Manual trigger:
1. In the Trigger Setup Menu window, choose the Manual button. The text will become yellow.

2. Set up other triggers, if necessary. If you are finished setting up triggers, choose OK in the Trigger Setup Menu window.

3. To initiate a manual trigger, choose Capture >> Manual Trigger from the menu bar. To streamline this process, you can add a manual trigger button to the control panel, if necessary.

Setting up an External trigger
Use the following instructions to set up an External trigger, which 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. With the Trigger Setup Menu window open, perform the following steps to set up the trigger:

To set up an External trigger:

1. In the Trigger Setup Menu window, choose the External button. The text will become yellow.

2. Set up other triggers, if necessary. If you are finished setting up triggers, choose OK in the Trigger Setup Menu window.

Aborts

Aborts are user-defined events that stop the data capture process. Aborts define the conditions under which you will halt the data capture process. For example, when a waveform meets a specific slew condition, you might want cause an abort to stop the data capture.

Abort types

As part of the abort setup process, you will need to select abort types. Available abort types are detailed below:

Slew OR produces an abort when there is a change in signal frequency.

Slew AND produces an abort when more than one channel must meet its respective slew rate conditions.

Events produces an abort when event inputs meet their change in logic state.

Manual is a commonly used method that allows the operator to produce an abort manually via the front panel.

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.

Note: If you set up multiple aborts, the data capture will be stopped when any of the abort settings are met.
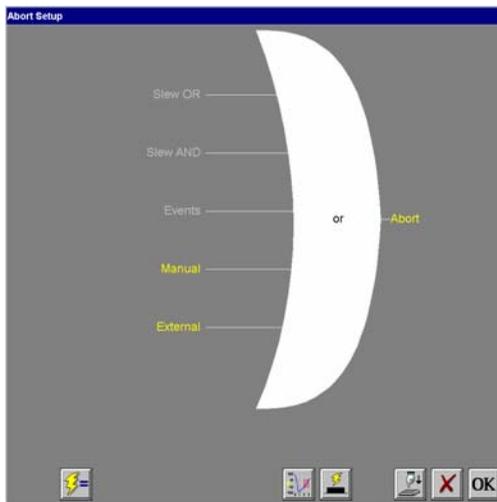
Setting up aborts

Use the following instructions to set up various types of aborts.

Using the Abort Setup window

You will use the Abort Setup window to access all setup options for aborts. You will need to access this window prior to setting up each individual abort.

To use the Abort Setup Menu window:

1. Choose Setup >> Abort Settings to open the Abort Setup window.

    Aborts are activated and deactivated by choosing the abort name in the graphic. Active aborts display in yellow and inactive aborts display in gray.

2. If necessary, you can view the current abort conditions at any time by choosing the Abort Equals button.

    The current abort configuration will be displayed.

3. Set up aborts using the following instructions. When you are finished setting up aborts, choose the OK button.

Setting up Slew OR & Slew AND aborts

Use the following instructions to set up Slew OR & Slew AND aborts. Slew OR produces an abort when there is a change in signal frequency, Slew AND produces an abort when more than one channel must meet its respective slew rate conditions.

Note: The slew concepts for this abort are similar to the concepts described in the *Setting up Slew OR & Slew AND* triggers section. However, instead of producing a trigger, an abort will occur under the specified conditions.

To set up Slew OR & Slew AND aborts:

1. Open the Abort Slew window by choosing the Abort Slew button.

   The Abort Slew window will open.

2. Select delta values.

   Select a channel and choose the Delta column heading. From the menu that appears, choose the New Value option. A number pad will appear. Enter a delta value and choose OK.

3. Select delta time values.

Select a channel and choose the dT column heading. A number pad will appear. Enter a delta time value and choose OK.

4. Select whether the abort will occur when the slew rate of the waveform is above or below the chosen slew rate.

    Select a channel and choose the Window column heading. From the menu that appears, choose the Above or Below option.

5. Select whether to enable abort conditions for channels.

    Select a channel and choose the OR column heading. From the menu that appears, select Yes or No. If you select Yes, the abort condition for the selected channel will be enabled. If you select No, the abort condition for the selected channel will be disabled.

6. Select whether to require multiple conditions to initiate an abort.

    For each channel abort condition that you want to require, select the channel and choose the AND column heading. Then choose Yes from the menu that appears. Aborts will occur only when all conditions are met.

7. Choose OK in the Abort Slew window. You will return to the Abort Setup window.

8. Choose the Slew OR or Slew AND button. The text will become yellow.

9. Set up other aborts, if necessary. If you are finished setting up aborts, choose OK in the Abort Setup window.
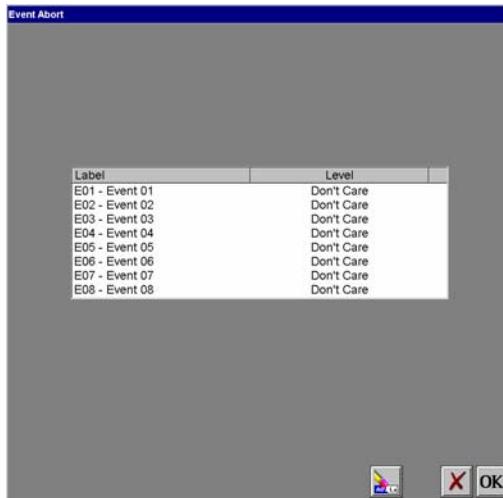
Setting up an Event abort

Use the following instructions to set up an Event abort, which produces an abort when event inputs meet their change in logic state. With the Abort Setup window open, perform the following steps to set up the abort:

To set up an Event abort:

1. Open the Event Abort window by choosing the Event Abort button.

The Event Abort window will open.

2. Select event level values that will cause an abort.

    Select a channel and choose the Level column heading. From the menu that appears, choose either Don't Care, High, or Low.

    •   If you choose Don't Care, changes in the selected event will not cause an abort.

    •   If you choose High, an abort will occur when the selected event changes to its high state.

    •   If you choose Low, an abort will occur when the selected event changes to its low state.

3. Choose OK in the Event Abort window. You will return to the Abort Setup window.

4. Choose the Event button. The text will become yellow.

5. Set up other Aborts, if necessary. If you are finished setting up aborts, choose OK in the Abort Setup window.

Setting up a Manual abort

Use the following instructions to set up a Manual abort, which allows the operator to produce an abort manually via the front panel. With the Abort Setup window open, perform the following steps to set up the abort:

To set up a Manual abort:

1. In the Abort Setup window, choose the Manual button. The text will become yellow.

2. Set up other aborts, if necessary. If you are finished setting up aborts, choose OK in the Abort Setup window.

3. To initiate a manual abort, choose Capture >> Manual Abort from the menu bar. To streamline this process, you can add a manual abort button to the control panel, if necessary.

Setting up an External abort

Use the following instructions to set up an External abort, which 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. With the Abort Setup window open, perform the following steps to set up the abort:

To set up an External abort:

1. In the Abort Setup window, choose the External button. The text will become yellow.

2. Set up other aborts, if necessary. If you are finished setting up aborts, choose OK in the Abort Setup window.

Initiating a data capture

Once data capture settings have been set up, the Dash 18X is ready to capture data.

To initiate a data capture:

1. Arm the data capture by choosing Capture >> Arm from the menu bar.

    - If you set up the system to capture pre-trigger data, the Dash 18X will begin recording pre-trigger data.

    - If you set up the system without a trigger, the Dash 18X will immediately begin the post-trigger capture.

2. If necessary, wait for a trigger to occur based on your trigger settings, or manually trigger the data capture (Capture >> Manual Trigger).

    If appropriate, the data capture will abort based on your abort settings, or you can cause a manual abort to stop the data capture (Capture >> Manual Abort).

3. Complete the data capture process.

    The data capture will complete automatically and save the results to a file. If you enabled automatic re-arming, however, the system will continually restart the data capture process after each data capture.

Additional data capture features

The Dash 18X provides the capability to record notes, video, and audio related to your data captures.

Capturing notes

Use the following instructions to captures notes during data capture.

To capture notes:

1. When a data capture is in progress, the Capture Progress window will open:
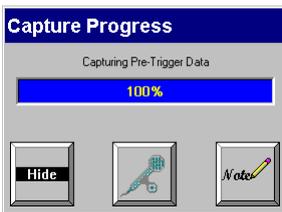
2. You can add notes by choosing the Note button.

A keypad will appear.

3. Enter the note. When you are finished entering the note, choose the OK button.

    If necessary, you can enter more notes by repeating this process.

Frequently used notes

The bottom of the keypad consists of six buttons used to store frequently used notes. These notes can be added by simply pressing the button, which is faster and easier than typing them in multiple times.

The content of these buttons is determined by the "notelist.ini" file, located in the C:/d18 directory. The file looks like this :

```
[Note1]
NoteValid=1
ButtonText=Text 1
NoteText=First note.

[Note2]
NoteValid=1
ButtonText=Note 2
NoteText=Note number 2.

[Note3]
NoteValid=1
ButtonText=Start Test
NoteText=Test started.


[Note4]...
```

The "NoteValid" parameter is used to activate or deactivate a note button. To activate a button, set its "NoteValid" value to 1.

The "ButtonText" parameter determines the label of the button, *not the text of the note*.

The "NoteText" parameter determines the actual text of the note itself.

Capturing video with the CAM-18 option

Use the following instructions to capture real-time video snapshots as part of your data capture.

To capture video:
1. Ensure that the CAM-18 USB webcam is connected to the Dash 18X USB port.

2. Choose View >> AstroVision from the menu bar. The Snapshot Camera window will open.

This window displays the current view being monitored by the webcam.

- To automatically capture a video snapshot in response to a previously defined trigger condition, choose the Trigger Snapshot button.

- When a data capture is in progress, you can choose the Snapshot button to capture a video snapshot at any time during the capture.

3. When you are finished capturing video snapshots, choose X to close the Snapshot Camera window.

Capturing audio with the AUD-18 option

Use the following instructions to capture real-time audio as part of your data capture.

To capture audio:

1. Ensure that the USB headset/microphone is connected to the Dash 18X USB port.

2. When a data capture is in progress, the Capture Progress window will open:
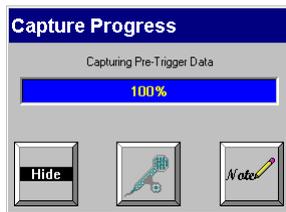
3. Start recording audio by choosing the Microphone button.

    Speak into the microphone to record your message, or position your microphone at the desired sound source. When you are finished recording the audio, choose the Microphone button again to stop recording.

    Repeat this step to record additional audio clips.

4. When the data capture process ends, you will no longer be able to record audio as part of the data capture.

Chapter 14: Scope captures

This chapter will help you learn about Scope mode and the scope capture process.

About scope captures

The scope capture process is very similar to the Realtime data capture process. The concepts of triggers, aborts, and arming the system apply to the scope capture process.

In addition to the concepts of Realtime data captures, it is helpful to understand the following information about Scope mode data captures:

- Scope mode operates much like a traditional oscilloscope, allowing the capture and display of high-speed snapshots.

- Designed for timing and synchronization analysis, transient capture, and high-speed testing, Scope mode can be used while you are continuously capturing data to the hard drive as well as monitoring your signals on the display.

Scope captures can also be embedded into realtime data captures so you can review high-speed and trend data at the same time.

Default Scope mode control panel

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

Arm/Abort arms and aborts data captures.

Increase Time Base increases the time base used for scope captures.

Decrease Time Base decreases the time base used for scope captures.

Analog Channel Setup opens the Analog Channel Setup window, which you will use to set up channels.

Trigger Setup opens the Trigger Setup Menu window, which you will use to set up triggers.

Show Cursor A displays cursor A.

Show Cursor B displays cursor B.

Change Active Cursor changes the active cursor configuration between A, B, and A + B.

Move Cursor Left moves the active cursor(s) to the left each time you choose it.

Move Cursor Right moves the active cursor(s) to the right each time you choose it.

Context Help provides online help based on the mode or the menu you are using.

Icon Help provides online help for identifying the purpose of buttons.

Setting up scope captures

The Scope mode uses the same trigger, abort, and setup parameters as Realtime mode. You can access these options using the capture and setup options from the menu bar.

Note: For more detailed information on triggers, aborts, and setup parameters, refer to *Chapter 13, Data capture.*

Additionally, there are specific scope mode features that you will need to set up before performing a scope capture.

To set up scope captures:
1. If necessary, set up trigger, abort, and capture settings to meet your needs. This process is identical to Realtime mode.

2. Choose Acquire from the menu bar, then choose one of the following acquisition options:

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.   Choose Acquire from the menu bar, then choose a time base option. Time base options include the following:

100, 200, or 500 microseconds/division

1, 2, 5, 10, 20, 50, 100, 200, 500 milliseconds/division

1 second/division

4.   Choose Acquire from the menu bar, then choose the 1 screen or 2 screens option.

One screen display allows all of the data acquired to be viewed on one screen. Two screen captures twice as much data and displays it on two screens with a scrolling bar on the bottom.

5.   Choose Acquire from the menu bar, then choose Pre-Trigger Percent. A number pad will appear. Enter the percentage of the data capture to record before a trigger.

6.   If you choose Capture >> Auto Save Scope with Capture, the scope capture will be embedded in the previously set up Realtime data capture.

Note: Auto Save Scope with Capture will allow you to save high-speed scope captures while collecting data to the hard drive. Care should be taken 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, a trigger occurs, and data is recorded until the capture is completed or aborted.

For triggered scope modes, triggers will be ignored until all of the requested pre-trigger memory is filled.

Note: For information on initiating data captures, refer to *Chapter 13, Data capture*.
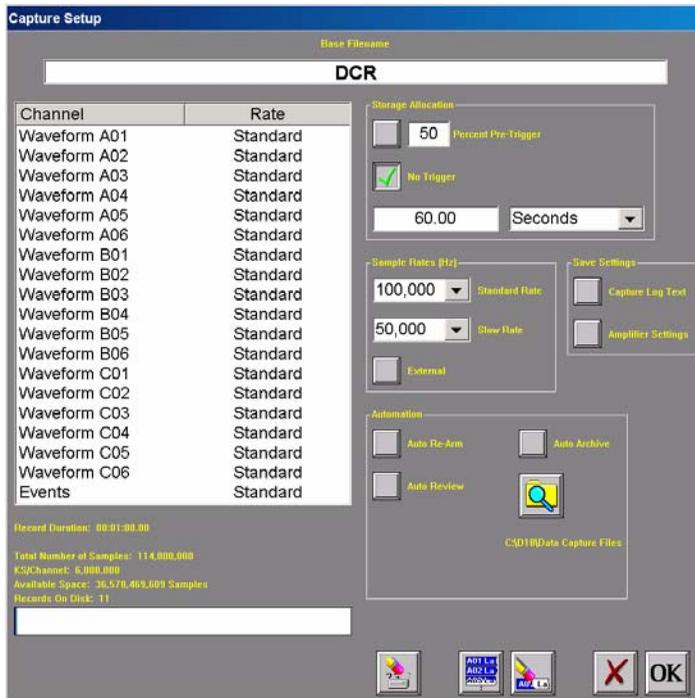
Embedded scope captures

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.
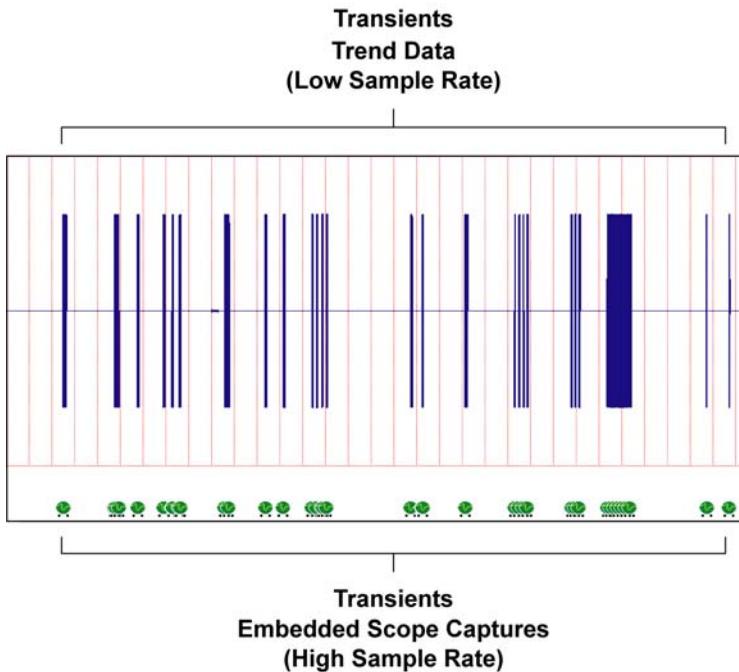
To set up embedded scope captures:

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.

Capture Setup

Base Filename

DCR

Channel	Rate
Waveform A01	Standard
Waveform A02	Standard
Waveform A03	Standard
Waveform A04	Standard
Waveform A05	Standard
Waveform A06	Standard
Waveform B01	Standard
Waveform B02	Standard
Waveform B03	Standard
Waveform B04	Standard
Waveform B05	Standard
Waveform B06	Standard
Waveform C01	Standard
Waveform C02	Standard
Waveform C03	Standard
Waveform C04	Standard
Waveform C05	Standard
Waveform C06	Standard
Events	Standard

Storage Allocation

50 Percent Pre-Trigger

✓ No Trigger

60.00 Seconds

Sample Rates (Hz)

100,000 Standard Rate

50,000 Slow Rate

External

Save Settings

Capture Log Text

Amplifier Settings

Automation

Auto Re-Arm   Auto Archive

Auto Review

C:\D1R\Data Capture Files

Record Duration: 00:01:00.00

Total Number of Samples: 114,000,000
KS/Channel: 6,000,000
Available Space: 36,570,469,609 Samples
Records On Disk: 11

Note: More information about using the Capture Setup window is
located in *Chapter 13: 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 13: 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 either 1 Screen or 2 Screens.

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.

Measuring channel information with cursors

Cursors are a valuable tool that you will use for measuring channel signals. You will use cursors in conjunction with the Channel Information window to view a variety of measurements.

Note: You can save time and gain more control over cursors by adding cursor buttons to your control panel.

To measure channel information with cursors:

1. Show one or both of the cursors by doing one or both of the following:

    • Choose Cursors >> Show Cursor A

    • Choose Cursors >> Show Cursor B

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

Max-Min displays the value 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 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 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 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.

You can change the measurement type by choosing the Measurement Type option on the menu bar of this window.

3. If necessary, move one or both of the cursors. Before you move a cursor, you will need to activate it. Do one of the following:

    - **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: You can also activate a cursor by pressing it once. To move it, press it again but don't release it. While pressing, drag it to a new location, and then release it.

As you move the cursors, the values in the Channel Information window will update based on the values of the new cursor locations.

4. If necessary, modify the color and type of your cursors by doing the following:

  - Modify the cursor color by choosing Cursors >> Color. Then select a color from the options that appear.

  - Modify the cursor type by choosing Cursors >> Type. Then select a type from the options that appear. If you select the Color option, color cursors will be used. If you select the Invert option, inverted cursors will be used, providing a pseudo-transparent cursor.

View options

Use this section to learn about the various view options available in Scope mode.

Setting up scope channel appearances

You will use the Scope Channel Setup window to modify the appearance of channels in Scope mode.

To set up scope channel appearances:

1. Choose View >> Channel Setup from the menu bar. The Scope Channel Setup window will open.
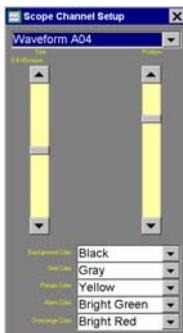
2. Select a channel using the drop-down list.

3. Adjust the channel size and 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.

> **Primary Color** – The color of the channel 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.

5. Set up other channels, if necessary.

6. When you are finished, close the Scope Channel Setup window by choosing the X in the upper-right corner of the window.

Displaying the trigger line

Use the following information to display the trigger line, which indicates the point when the trigger occurred during the scope capture.

To display the trigger line:

1. Choose View >> Show Trigger Line from the menu bar. The trigger line will appear.

2. To hide the trigger line, choose View >> Show Trigger Line again.

Displaying channel IDs

Viewing channel IDs allows you to identify channels using small visual indicators to label waveforms.

To display channel IDs:

1. Choose View >> Show Channel IDs from the menu bar.

A channel ID mark will appear next to each waveform.

Using the FFT window

Use the following instructions to measure channels with the FFT window.

To use the FFT window:

1. Choose Analysis >> FFT Window from the menu bar. The FFT window will open.

2. Choose the Channel option. A list of channels will appear. Select the channel you want to analyze.

3. If necessary, select the Settings menu option to modify the following options:

> **Background Color** allows you to select a color for the background of the FFT window.
>
> **Channel Color** allows you to select a color for the channel in the FFT window.
>
> **Grid** allows you to select whether to display a grid. You can also specify the grid color as well as the amplitude axis and frequency axis.
>
> **Plot** allows you to choose between linear magnitude, logrithmic magnitude, or magnitude^2.
>
> **Horizontal Size** allows you to adjust the horizontal size of the FFT window by choosing from small, medium, or large configurations.
>
> **Frequency Expansion** allows you 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 18X functions.

6. If necessary, you can use the Help menu option to learn more about the FFT window.

7. When you are finished, close the FFT window by choosing the X in the upper-right corner of the window.

Archiving files

You can archive scope files in data capture record or ASCII format.

Archiving as a data capture record

Use the instructions below to archive scope captures as data capture records.

To archive as a data capture record:

1. Choose File >> Archive as Data Capture Record. The Save Scope Capture File window will open.
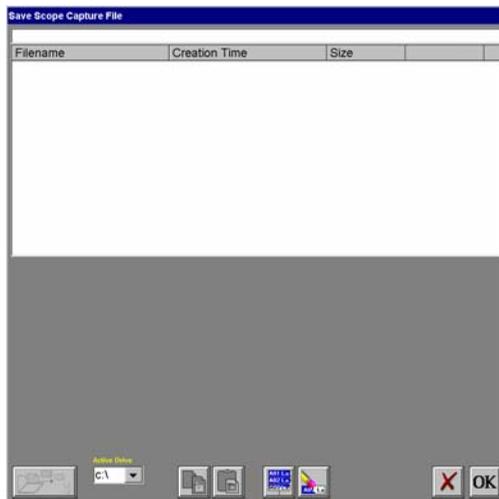
2. If necessary, select the drive to save the file on. By default, drive C is selected.

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

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

Archiving as ASCII

You can save scope capture files in ASCII format for archiving and analysis in other programs.

To archive as ASCII:

1. Choose File >> Archive as ASCII from the menu bar. The Save ASCII Capture File window will open.

2. If necessary, select the drive to save the file on. By default, drive C is selected.

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

Enter a name for the archive file and choose OK. The ASCII Archive
Setup window will appear.
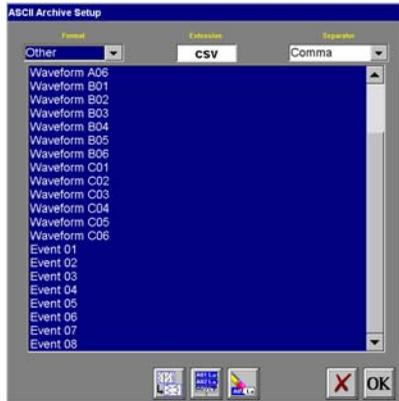
4.   Specify the file setup option.

Select your file format options.

- If you choose Excel format, the file will be saved with a csv extension
  and the delimiters will be commas.

- If you choose Other format, you will be able to enter an extension
  and select delimiters.

Select the channels and events to include in the archive. By default, all
channels and events are selected.

5.   Choose OK. The archive file will be saved.

Chapter 15: Data capture review & analysis

When your data capture is complete, you will use Review mode to analyze the data. This chapter will help you learn about the data review and analysis features of Review mode.

Data capture files

Data capture files are stored on the hard drive of the Dash 18X. You can open these files and analyze them in Review mode. If necessary, you can also delete these files by using the delete button on the file open screen.

Opening data capture files

When you launch Review mode, you will be prompted to open a data capture file. Use the following instructions to learn how to open a data capture file.

Note: When already in Review mode, you can open files by choosing File >> Load New Capture File from the menu bar.

To open data capture files:

1. Launch Review mode by choosing Configuration >> Review. A file management screen will appear.

2. Open the file that you want to analyze.

    •   To open a file, select it and choose the OK button.

    •   To open a scope capture file, choose the scope capture button.

    Select a scope capture file and choose the OK button. The scope capture will be opened.

    •   To open an archived data capture, choose the archive data capture button.

    Select an archive file and choose the OK button. The archive 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 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: You can save time by adding Show Next File and Show Previous File buttons to your control panel.

Deleting data capture files

Use the following instructions to learn how to delete data capture files from the Dash 18X hard drive.

Note: In addition to deleting files from the Review mode file selection window, you can also delete individual files during review.

To delete the data capture file currently being reviewed, choose File >> Delete Current Capture File. A confirmation message will appear. Choose Yes to delete the file and return to the Review mode file selection window.

To delete data capture files:

1. Launch Review mode by choosing Configuration >> Review. A file management screen will appear.

2. Select the file(s) you want to delete, and then choose the Send File to Trash button.

The file will be transferred to the Dash 18X trash folder.

If you accidentally selected the wrong file, you can retrieve it from the trash by choosing the Retrieve from Trash button.
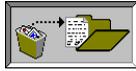
You can specify whether to display deleted files. If you want to display deleted files, check the Show Deleted Files option.

3. If necessary, you can empty the trash to free hard drive space. Emptying the trash permanently deletes files currently in the trash. You should use caution with this feature to ensure that you don't lose files that you may later need.

    To empty the trash, choose the Empty Trash button.
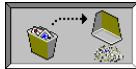
Default Review mode control panel

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

Scroll Back 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 Cursor A displays cursor A.

Show Cursor B displays cursor B.

Change Active Cursor changes the active cursor configuration between A, B, and A + B.

Move Cursor Left moves the active cursor(s) to the left each time you choose it.

Move Cursor Right moves the active cursor(s) to the right each time you choose it.

Context Help provides online help based on the mode or the menu you are using.

Icon Help provides online help for identifying the purpose of buttons.

File navigation

This section describes the variety of ways you can use to navigate through your data capture files.

Scroll bar

The following illustration displays a portion of the scroll bar.

You can move forward and backward in a file by using the scroll bar on the bottom of the screen.

Control panel

The default control panel provides a variety of buttons to help you move through the file.

Menu bar

The menu bar provides the following methods of navigating through your data capture file:

Goto allows you to navigate directly to the start, end, trigger point, or cursor location in a file. You can access an advanced search that allows you to navigate directly to locations that meet your specified conditions. If the file contains notes, a note search option allows you to navigate the file based on note locations.

Page Back scrolls the chart back one page. Pressing on an open portion of the scroll bar accomplishes the same function.

Page Forward scrolls the chart forward one page. Pressing on an open portion of the scroll bar accomplishes the same function.

Rewind scrolls the chart backwards quickly.

Scroll Back scrolls the chart backwards.

Scroll Stop stops the chart from scrolling.

Scroll Forward scrolls the chart forward.

Fast Forward scrolls the chart forward quickly.

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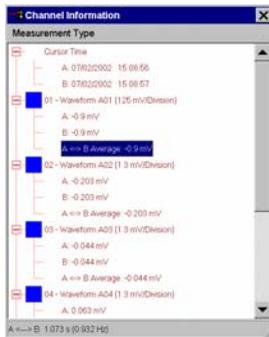
Measuring channel information with cursors

Cursors are a valuable tool that you will use for measuring channel signals. You will use cursors in conjunction with the Channel Information window to view a variety of measurements.

Note: You can save time and gain more control over cursors by adding cursor buttons to your control panel.

To measure channel information with cursors:
1. Show one or both of the cursors by doing one or both of the following:

    -   Choose Cursors >> Show Cursor A

    -   Choose Cursors >> Show Cursor B

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. Additionally, when both cursors are shown, relative data elements are displayed.

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 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 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 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 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.

You can change the measurement type by choosing the Measurement Type option on the menu bar of this window.

3. If necessary, move one or both of the cursors. Before you move a cursor, you will need to activate it. Do one of the following:

- **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: You can also activate a cursor by pressing it once. To move it, press it again but don't release it. While pressing, drag it to a new location, and then release it.

As you move the cursors, the values in the Channel Information window will update based on the values of the new cursor locations.

4. If necessary, modify the color and type of your cursors by doing the following:

   - Modify the cursor color by choosing Cursors >> Color. Then select a color from the options that appear.

   - Modify the cursor type by choosing Cursors >> Type. Then select a type from the options that appear. If you select the Color option, color cursors will be used. If you select the Invert option, inverted cursors will be used, providing a pseudo-transparent cursor.

View options

Review mode provides many of the view options of Realtime mode. In addition to these options, Review mode provides other functions to help you analyze your data.

Changing the status text format

You can change the type of information displayed in the status text area, which is in the lower-right corner of the screen.

To change the status text format:

1. Choose View >> Status Text Format from the menu bar. A list of status text formats will appear.

2. Choose samples, relative time, absolute time, or percent. The status text in the lower-right corner of the display will be updated based on your selection.
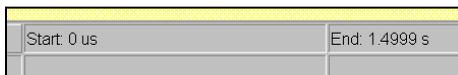

```
Start: 0 us                        End: 1.4999 s
```

Displaying channel IDs

Viewing channel IDs allows you to identify channels using small visual indicators to label waveforms.

To display channel IDs:

1. Choose View >> Show Channel IDs from the menu bar.
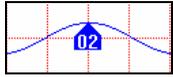
A channel ID mark will appear next to each waveform.

Viewing pictures with the CAM-18 option

If the file you are reviewing contains video images, you can view them by using the following instructions.

To view pictures:

1. Choose View >> Picture Viewer from the menu bar. The Snapshot Viewer window will open.

> **Note:** You can also open the Snapshot Viewer window by scrolling to and choosing a camera icon in the file.

The Find button allows you to scroll to the picture location in the file, if necessary.

2. You can use the scroll bar on the bottom of the picture to move to other images stored in the file.

3. When you are finished viewing the pictures, choose the X button to close the Snapshot Viewer window.

Viewing, editing, and adding notes

You can view, add, and edit text notes by using the following instructions.

To view/edit notes:

1. Launch Review mode by choosing Configuration >> Review. A file management screen will appear.

2. Select a file. If the file contains notes, the Note button will become active.

3. Choose the Note button. The Notetag Viewer window will open.
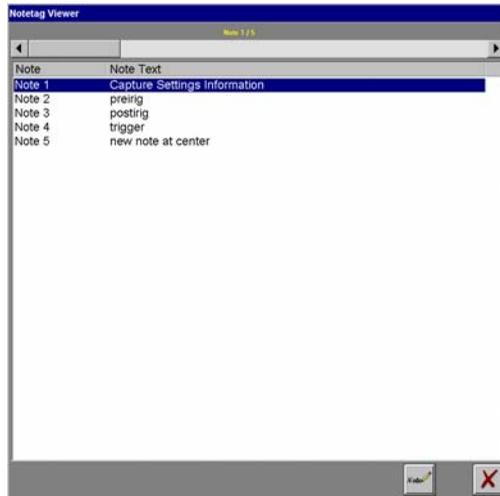
This window displays a list of notes contained in the data capture file. The text of each note is also visible.
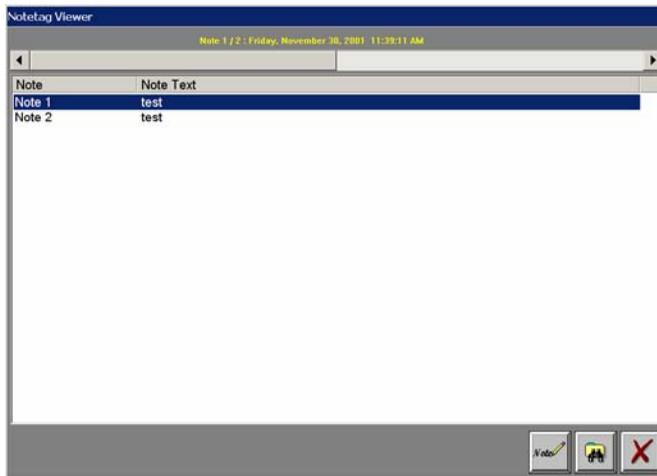
4. Select a note and choose the Note button. A keypad will appear.

5. If necessary, edit the contents of the note and choose the OK button.

To view/edit notes during data capture review:

1. With a data capture file open in Review mode, chose View >> Notes >> Note Viewer from the menu bar. The Notetag Viewer window will open.

> **Note:** You can also open the Notetag Viewer window by scrolling to and choosing a pencil icon in the file.

2. To scroll to a specific note location in the data capture file, select the note and choose the Find button.

3. To edit a note, select the note and choose the Note button. A keypad will appear.

Edit the note and choose the OK button.

4. When you are finished viewing and editing notes, choose the X button to close the Notetag Viewer window.

To add notes during data capture review:

1. With a data capture file open in Review mode, chose View >> Notes >> Add Note from the menu bar. The Add Notetag window will open.


2. Enter the note text.

3. Choose the location of the note. Notes can be placed at the center of the screen, or at either cursor point if cursors are displayed.

4. Choose the OK button. The note will be added to the data capture file.

Viewing scope captures

If the file you are reviewing contains embedded scope captures, you can view them by using the following instructions.

To view scope captures:

1. Choose View >> Scope Viewer from the menu bar. The Scope window will open.
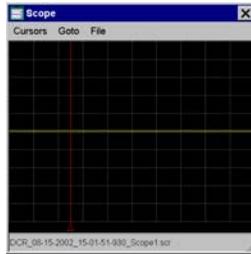
> **Note:** You can also open the Scope window by scrolling to and choosing a scope icon in the file.

2. If necessary, choose the Cursors menu option to use cursors. The cursors in this window function similar to cursors in other Dash 18X functions.

3. If necessary, you can choose Goto >> Scope Location from the menu to scroll to the scope location in the file.

4. If necessary, you can move to the next or previous scope capture in the file by choosing File >> Next *or* Previous.

5. When you are finished viewing the scope capture, choose the X in the upper-right corner to close the Scope window.

Using the Sound Player window to play sound with the AUD-18 option

If the file you are reviewing contains embedded audio, you can play the audio by using the following instructions.

To use the sound player window to play sound:

1. Ensure that the USB headset/microphone is connected to the Dash 18X USB port.

2. Choose View >> Sound Player from the menu bar. The Sound Player window will open.

> **Note:** You can also open the Sound Player window by scrolling to and choosing a sound icon in the file.

3. Choose the Play Sound button to play the audio clip.


4. If necessary, you can choose Goto >> Sound Location from the menu to scroll to the sound location in the file.

5. If necessary, you can move to the next or previous sound in the file by choosing File >> Next *or* Previous.

6. When you are finished listening to the sound captures, choose the X in the upper-right corner to close the Sound Player window.

Displaying the trigger line

Use the following information to display the trigger line, which indicates the trigger point in the data capture.

To display the trigger line:

1. Choose Display >> Show Trigger Line from the menu bar. The trigger line will appear.
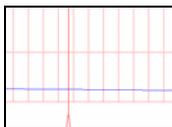
2. To hide the trigger line, choose Display >> Show Trigger Line again.

Viewing the entire file in one window

Instead of scrolling to view the contents of a file, you can choose to display the entire file by compressing it into one window.

To view the entire file in one window:

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

Setting up file compression

You can set a specific compression ratio to stretch or compress the file. The maximum amount of compression allowed for a file is that which will cause the file to be displayed on one page.

To set up file compression:

1. Choose File >> Compression. A number pad will appear.

2. Enter the compression factor you wish to use. Then choose OK. The view of the file will be modified based on the compression you selected.

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.
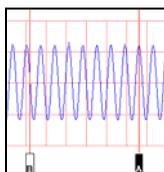
3. Choose Display >> Autoscale Between Cursors. Amplitude scaling will be applied to all channels based on the area between the cursors.
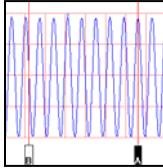
4. To remove the auto scaling, choose Display >> Clear Autoscale.

Analysis features

Review mode provides data analysis tools to help you better understand your data. Use this section to learn more about Review mode analysis features.
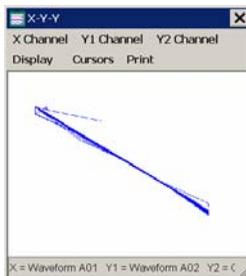
Using the XY plot

Use the following instructions to learn how to use the XY plot. This plot also provides a second Y trace.

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 use the XY plot:

1. Choose View >> XY Plot from the menu bar to open the X-Y-Y plot window.

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

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

>   **Refresh** clears the window and restarts the plot drawing.

>   **Connect Points** draws lines to connect the sample points on the plot. When this option is active, segments will be drawn.

>   **Background Color** allows you to select a background color for the plot window.

>   **Foreground Color** allows you to select a color for the points and segments being drawn on the plot.

>   **Grid** allows you to select whether to display a grid. You can also specify the number of divisions in the grid as well as the grid color.

4. If necessary, choose the Cursors menu option to use cursors. The cursors in this window function like cursors in other Dash 18X functions, with the exception of the following features.

    *   Cursor A is a vertical cursor and cursor B is a horizontal cursor.
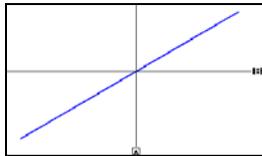
    *   To restrict cursor movement to the data points, choose Cursors >> Lock On Data. The starting and ending data points will be highlighted automatically. The Lock On Data option is unavailable when the Y2 plot is used.

        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.
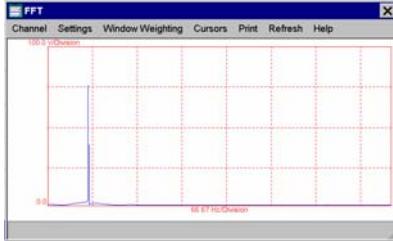
5. If the printer is installed, you can print the screen by choosing Print >> Print Screen from the menu.

Using the FFT window

Use the following instructions to measure channels with the FFT window.

To use the FFT window:

1. Choose Analysis >> FFT Window from the menu bar. The FFT window will open.

2. Choose the Channel option. A list of channels will appear. Select the channel you want to analyze.

3. If necessary, select the Settings menu option to modify the following options:

    Background Color allows you to select a color for the background of the FFT window.

    Channel Color allows you to select a color for the channel in the FFT window.

    Grid allows you to select whether to display a grid. You can also specify the grid color as well as the amplitude axis and frequency axis.

    Plot allows you to choose between linear magnitude, logrithmic magnitude, or magnitude^2.

    Horizontal Size allows you to adjust the horizontal size of the FFT window by choosing from small, medium, or large configurations.

    Frequency Expansion allows you 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 18X functions. Refer to the *Measuring channel information with cursors* section in this chapter for more information on cursor use.

6. If necessary, you can use the Help menu option to learn more about the FFT window.

7. When you are finished, close the FFT window by choosing the X in the upper-right corner of the window.

Using the Filter window

Use the following instructions to analyze channels with the Filter window.

To use the Filter window:

1. Choose Analysis >> Filter Window from the menu bar. The Filter window will open.
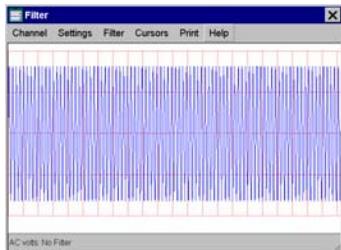
2. Choose the Channel option. A list of channels will appear. Select the channel you want to analyze.

3. If necessary, select the Settings menu option to modify the following options:

> **Background Color** allows you to select a color for the background of the Filter window.

> **Channel Color** allows you to select a color for the channel in the Filter window.

Grid allows you to select whether to display a grid. You can also specify the grid color as well as the number of divisions.

4. Choose the Filter menu option to apply one of the following filters:

   None – No filter

   Low-pass (LP) – Filters out frequencies above a specified cutoff point. (Choice of 1 – 10,000 Hz)

   High-pass (HP) – Filters out frequencies below a specified cutoff point. (Choice of 0.1 - 100 Hz).

   Band-pass (BP) – Filters out frequencies below a specified cutoff point and above a specified cutoff point, allowing only a selected band of frequencies to pass. (Choice of lower limit from high-pass filter and upper limit from low-pass filter)

5. If necessary, choose the Cursors menu option to use cursors. The cursors in this window function similar to cursors in other Dash 18X functions. Refer to the *Measuring channel information with cursors* section in this chapter for more information on cursor use.

6. If the printer is installed, you can print the screen by choosing Print >> Print Screen from the menu.

Using the zoom window

The zoom window allows you to magnify a portion of a data capture record 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.
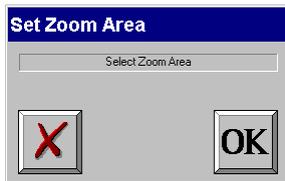
2. Create the zoom area by pressing anywhere on the waveform display area. While pressing, drag diagonally away from the first point to create a rectangular area.

3. When you are satisfied with the zoom area, choose OK in the Set Zoom Area window. The Zoom window will appear.
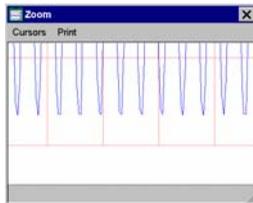
   The Zoom window displays the rectangular area you selected. You can use cursors to measure the zoom area. Refer to the *Measuring channel information with cursors* section in this chapter for more information on cursor use.
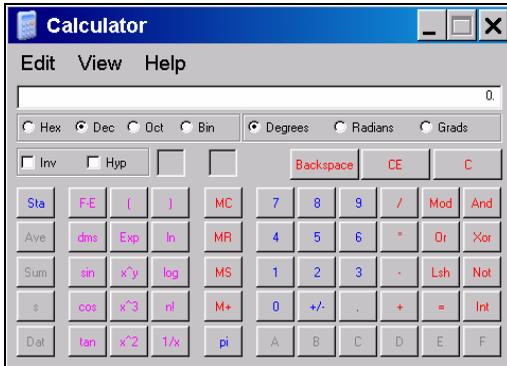
   You can also print the zoom area if the printer is installed.

Using the calculator

Use the following instructions to access the Calculator program.

To use the calculator:

1. Choose Analysis >> Calculator from the menu bar. The Calculator program will open. This program is part of the Windows® operating system.

2. When you are finished using the Calculator program, choose the X button in the upper-right corner to close it.

Archiving files

Archiving files is an important step if you plan to transfer these files to a PC. Use the information in this section to learn about archiving files. You can archive files in data capture record or ASCII format.

Note: Files larger than 2 GB are not compatible with FAT32 file systems. When archiving files larger than 2 GB, you will be prompted to split the file.

Choose Yes to continue and split the file into partitions no greater than 2 GB each. Choose No to cancel the archiving process.

Archiving as data capture records

Use the following information to archive as data capture records. This binary format is preferable if you plan to use Dash 18X Offline or AstroVIEW on your PC.

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.
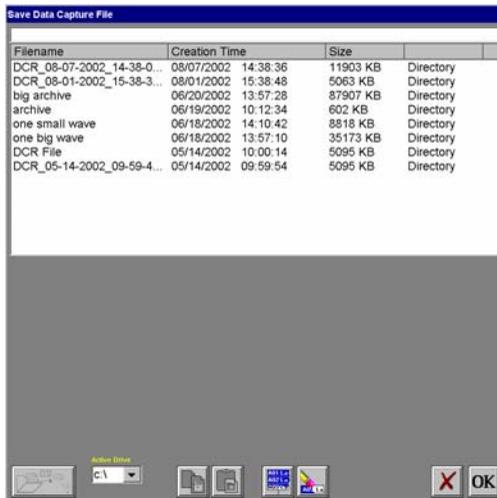
2. If necessary, select the drive to save the file on. By default, drive C is selected.

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

   Enter a name for the archive file and choose OK.

4. Choose OK in the Save Data Capture File window. The archive file will be saved.

Archiving as ASCII

You can save capture files in ASCII format for archiving and analysis in other programs. Because the ASCII format results in large file sizes, this format is not recommended for larger data capture files. It is ideal for small parts of the capture, however.

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.
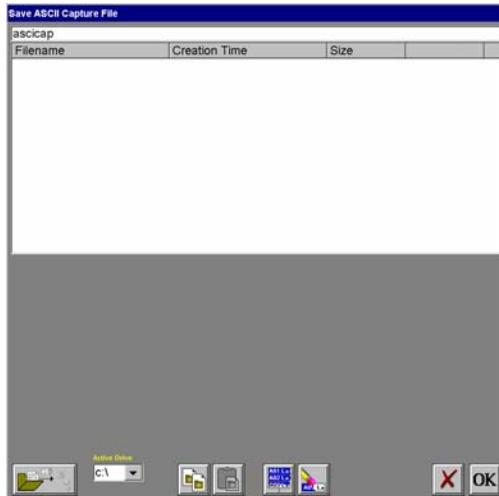
2. If necessary, select the drive to save the file on. By default, drive C is selected.

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

   Enter a name for the archive file and choose OK. The ASCII Archive Setup window will appear.
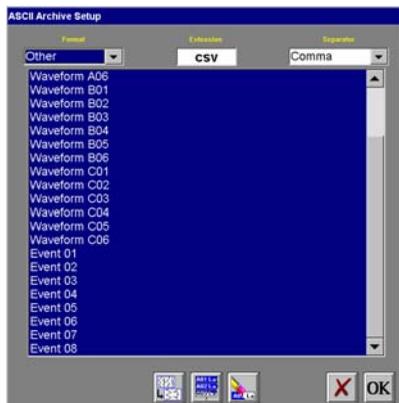
4. Specify the file setup options.

   Select your file format options.

- If you choose Excel format, the file will be saved with a csv extension and the delimiters will be commas.

- If you choose Other format, you will be able to enter an extension and select delimiters.

Select the channels and events to include in the archive. By default, all channels and events are selected.

5. 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 16: Service options

Several service options are available to help you maintain your Dash 18X. This chapter will help you learn about these Dash 18X service options.

Calibration

This section provides information on the calibration features of the Dash 18X.

Calibrating channels

The calibrate service allows you to calibrate each Dash 18X input channel.

To calibrate channels:

1. Choose Service >> Calibration. A password entry window will open. Enter the system password and choose OK. When the password has been correctly entered, the Calibration window will open.

   Note: The factory default password on the Dash 18X is "dash18" (without the quotation marks). Refer to the *Setting up a password* section of this chapter for information on changing the password.
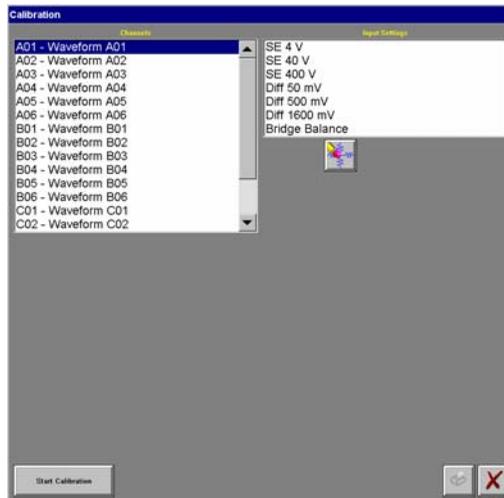
2. Select a channel and the input settings for that channel.

3. Choose the Start Calibration button. You will be prompted to place known voltages at the inputs. The selected channel will be automatically calibrated.

4. Calibrate other channels, if necessary.

5. If you are using bridge inputs, you can use the bridge balance function to apply an offset to the signal to make it equal to zero.

   When you select a channel that uses bridge inputs, the Bridge Balance option will become available in the Input Settings list. Select the Bridge Balance option and choose the Start Calibration button. The offset will be applied to make the signal equal to zero.

   If necessary, you can choose the Bridge Reset button to clear the Bridge Balance offset.

6. When you are finished calibrating channels, choose the X button.

Thermocouple calibration

The thermocouple calibration feature allows you to calibrate thermocouple channels.

When a thermocouple channel is successfully calibrated, calibration factors are saved to a file (C:\D18\calfactors.acf). This file allows the Dash 18X to recall the calibration settings if an input is changed from a thermocouple to another input type, and then changed back to a thermocouple.

Use the following instructions to calibrate thermocouple channels. This process involves setting up a thermocouple input and then calibrating the channel.

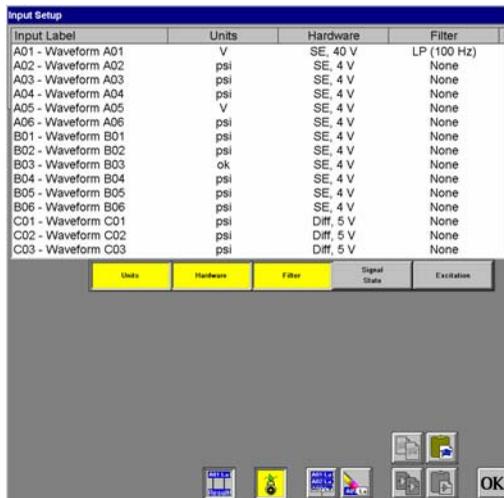**To calibrate thermocouple channels:**

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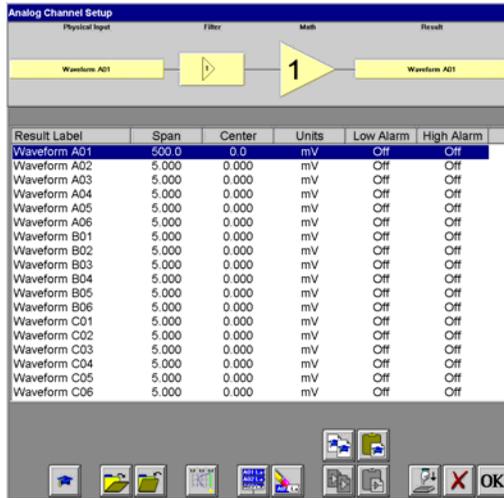
2. Choose the Advanced Setup button.

The Input Setup window will appear.

3. Specify a thermocouple type for inputs by selecting an input and choosing the Hardware column heading. The hardware type refers to how the signal is physically connected to the Dash 18X.

   A list of hardware options will appear. Choose the thermocouple option that best meets your needs. The following thermocouple types are available: J, K, E, T, or N.

   Note: When a thermocouple is selected, user units will automatically be enabled. The scale and offset values will be provided by the "calfactors.acf" file, which is located in the D18 folder.

4. Choose the OK in the Input Setup window. In the Analog Channel Setup window, adjust the span and center/bottom to the temperature range that will be calibrated.

   Then choose OK in the Analog Channel Setup window.

5. Choose Service >> Calibration. A password entry window will open. Enter the system password and choose OK. When the password has been correctly entered, the Calibration window will open.
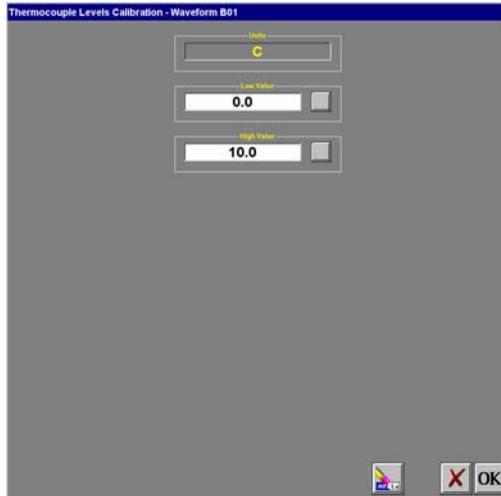
   Note: The factory default password on the Dash 18X is "dash18" (without the quotation marks). Refer to the *Setting up a password* section of this chapter for information on changing the password.
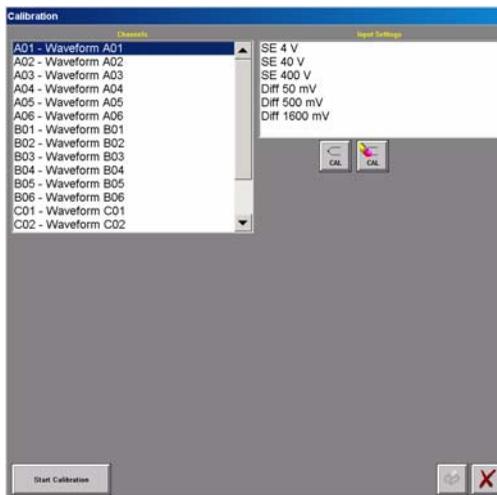
6. Select a thermocouple channel and choose the CAL button.

The Thermocouple Levels Calibration window for the selected channel will open.

7. Calibrate the thermocouple using the desired custom low and high values.

- Select the Low Value field. A number pad will open. Enter the desired low temperature calibration value and choose the OK button.

  Input the temperature you selected to the channel, and then choose the button to the right of the Low Value field.

- Select the High Value field. A number pad will open. Enter the desired high temperature calibration value and choose the OK button.

  Input the temperature you selected to the channel, and then choose the button to the right of the High Value field.

When both the low and high values have been set successfully, choose the OK button in the Thermocouple Levels Calibration window. The

calibration factors will be saved to the "calfactors.acf" file, which is located in the D18 folder. These values will automatically be applied as the user scale and offset values for the channel.

8. If necessary, you can clear the thermocouple calibration settings for the selected channel by choosing the Clear Thermocouple Calibration button.


9. When you are finished calibrating thermocouple channels, choose the X button in the Calibration window.

Viewing and modifying utility port settings

You will use the Utility Port Setup window to view and modify your utility port settings. This window provides a graphical representation of the utility port with setup options, where appropriate.

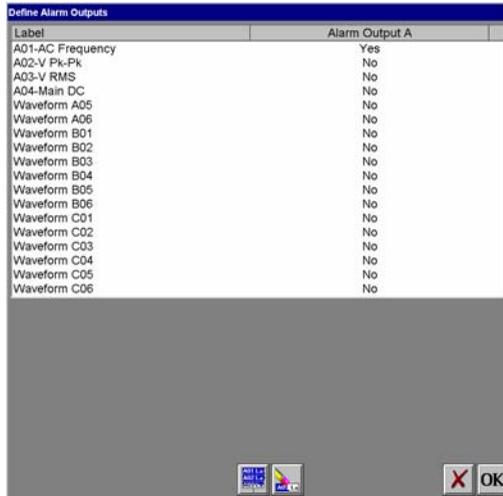**To view and modify utility port settings:**

1. Choose Service >> Utility Port to open the Utility Port Setup window.
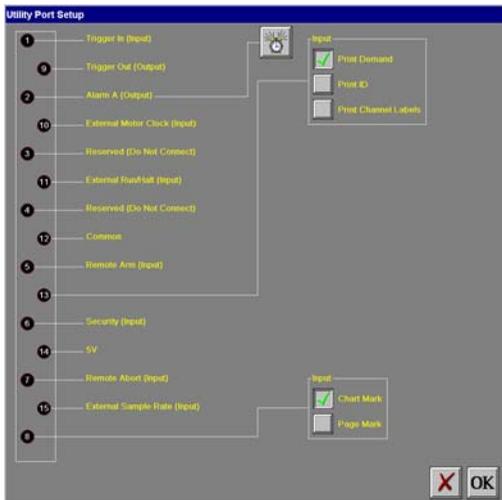
2. Modify the alarm output, if necessary, by choosing the Alarm button.

The Define Alarm Outputs window will open.

Select a channel (or multiple channels with "or" logic) and choose the Alarm Output A column heading. Then select Yes or No to indicate whether the selected alarm output will be used.

When you are finished setting up alarm outputs, choose the OK button.

3. Select a print option, if necessary, by selecting one of the options in the port 13 input box. The following print options are available:

    - Print Demand

    - Print ID

    - Print Channel Labels

4. Select a chart/page option in the port 8 input box. The following chart/page options are available:

    - Chart Mark

    - Page Mark

5. When you are finished viewing and modifying the utility port settings, choose OK. Your modifications will be saved.

Modifying IP information

The Dash 18X can be configured to obtain an IP address automatically using Dynamic Host Configuration Protocol (DHCP). Alternately, a specific IP address can be assigned.

Note: The factory default IP address for the Dash 18X is 192.168.255.1. The default subnet mask is 255.255.255.000.
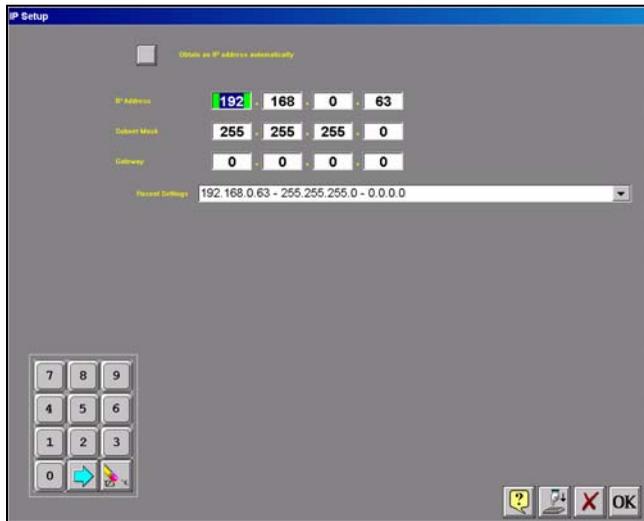
Use the following instructions to modify the Dash 18X IP information.

To modify IP information:
1. Open the IP Setup window using one of the following methods.

    • In Realtime mode, choose Service >> Network >> IP Configuration.

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

Choose the IP Setup button.

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 an 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.

Setting up a password

Use the following information to set up a Dash 18X password. This password is used for locking the menu bar to prevent unauthorized access.

To set up a password:

1. Choose Service >> Security >> Set Password from the menu bar. A keypad will appear with a field for your old password.

2. Enter your old password, and then choose OK.

   Note: The factory default password on the Dash 18X is "dash18" (without the quotation marks).

   When you have correctly entered the password, the New Password field will appear.

3. Enter your new password, and then choose OK. The Verify New Password field will appear.

4. Re-enter your new password, and then choose OK. The password will be changed.

Locking the menu bar

Use the following information to lock the Dash 18X menu bar. You can lock the menu bar to prevent unauthorized users from accessing menu bar functions.

To lock the menu bar:

1. Choose Service >> Security >> Menu Lock from the menu bar. A keypad will appear.

2. Enter your password, and then choose OK.

   Note: The factory default password on the Dash 18X is "dash18" (without the quotation marks). Refer to the previous section for information on changing the password.

   When you have correctly entered the password, the menu bar will be locked. When the menu bar is locked, it displays the date, time, mode,

and other information about the status of the system. It does not, however, allow access to menu bar options.

The following icon will appear on the right side of the menu bar.


3. To unlock the menu, choose the menu lock icon. A keypad will appear. Enter your password, and then choose OK. When you have correctly entered the password, the menu bar will be unlocked.

Chapter 17: Utilities

This chapter will help you learn about Dash 18X utility options.

Accessing utilities

You will access utilities by choosing Configuration >> Utilities from the menu bar.
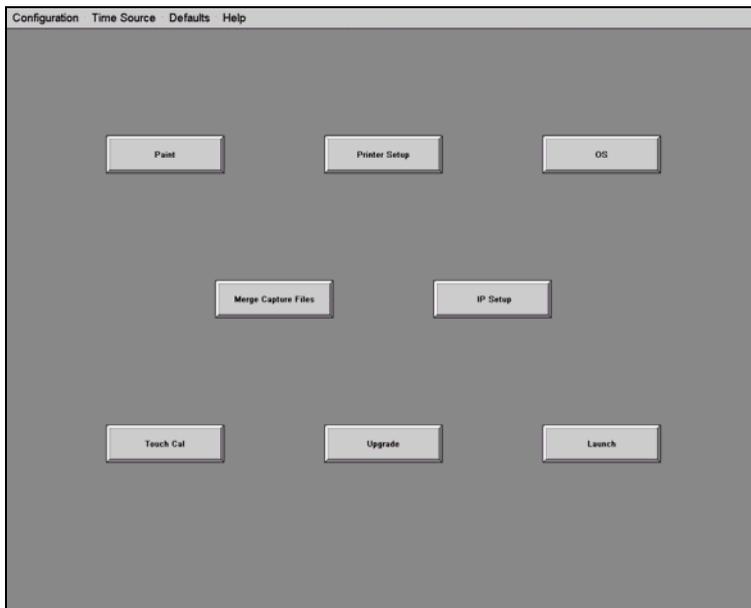
You will need to be in this utilities screen to perform the tasks detailed in the following sections.

Selecting a time source

The Dash 18X provides an internal time reference. It is also possible to use an external time reference.

An optional configuration of the Dash 18X is equipped with the SM-IRIG. With the SM-IRG installed, the Dash 18X supports a number of standard codes using modulated or TTL signals.

To select a time source:

1. Choose the Time Source option from the menu bar. A list of available IRIG time options will appear.

2. Select the IRIG time option that best meets your needs.

Restoring default settings

There may be times when you want to restore settings to their original state. Use the following instructions to return the Dash 18X settings to the factory-default configuration.

To restore default settings:

1. Choose Defaults from the menu bar. The following default options will appear:

   Inputs – Restores the default settings for the channel inputs.

   Grids – Restores the default settings for the channel grids.

   Realtime View – Restores the default view settings for Realtime mode.

   Realtime Panel – Restores the default Realtime mode control panel.

   Scope View – Restores the default view settings for Scope mode.

   Scope Panel – Restores the default Scope mode control panel.

   Review Panel – Restores the default Review mode control panel.

   System – Restores all settings to their default system state.

   Select the default option you wish to use.

2. A default verification prompt will appear.

   - To confirm the default settings, choose Yes. The default settings you chose will be applied.

   - To cancel the default settings, choose No. The default settings will not be applied.

Selecting a language

If necessary, you can change the language settings of the Dash 18X to meet the needs of users.

To select a language:
1. Choose Help >> Language from the menu bar. A list of language options will appear. The following language options are available:

   - American
   - British
   - French
   - Italian
   - German

2. Select a language. The language will be applied to Dash 18X text.

Other utilities

Use the following information to learn about the utilities that you can launch with the buttons in the utility screen.

Paint – Use this option to launch Microsoft ® Paint.

Printer Setup – Use this option to set up your printer.

OS – Use this option to exit the Dash 18X software and access the Microsoft ® Windows operating system. You will need to enter your system password before accessing the operating system.

Note: You can also access the operating system by choosing File >> Exit from the Realtime mode menu bar.

Merge Capture Files – Use this function to combine multiple data capture records into one file. Refer to the *Data capture merge utility* section in this chapter for detailed information.

IP Setup – Use this function to modify IP information. Refer to *Chapter 16: Service options* for detailed information.

Touch Cal – Use this function to set up your touch-screen calibration and other display options.

Upgrade – Use this function to upgrade your Dash 18X software. Insert a CD that contains the software upgrade. Then choose the Upgrade button. The software upgrade will be installed.

Launch – Use this function to launch other software applications. Refer to the *Launch utility* section in this chapter for detailed information.

Launch utility

In some instances, other software applications can be used while the Dash 18X is running. In these cases, you can use the Launch utility to launch programs.

Note: Astro-Med, Inc. does not guarantee that you can run any application while using the Dash 18X. Contact technical support before using this feature to ensure that performance will not be affected by your selected application.

To use the Launch utility:

1. From the Utilities screen, choose the Launch button. A window will appear with a text entry box, list box, and keypad.

2. Use the keypad to enter the full path of a program's ".exe" file into the text entry box and choose the OK button. The program will be launched.

   When a program is successfully launched, it will be added to the list in this window. To launch the program in the future, you can select it from the list and choose the OK button instead of entering the path again.

Up to 100 paths will be stored and displayed with the most recently launched paths on the top of the list.

Data capture merge utility

The data capture merge utility allows you to combine multiple data capture records into one file.

All files that will be merged must be compatible. They must have identical setup options such as display and channel configurations. Additionally, captures must contain only post-trigger data; they cannot contain a pre-trigger data percentage. Files must also contain channels captured at the standard sample rate; a dual rate (standard *and* slow) capture cannot be used.

Use the following instructions to merge multiple data capture files.

To use the data capture merge utility:

1. From the Utilities screen, choose the Merge Capture Files button. The Merge Data Capture Files window will open.
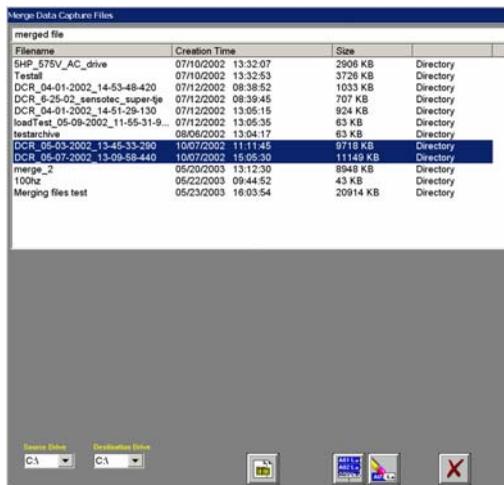
Files are sorted by creation time, listing the oldest file first. Note that files will be merged in the order in which they appear on the list.

2. Use the Source Drive list to select the drive that contains the data capture files that will be merged.

Use the Destination Drive list to select the drive to save the merged file on. By default, drive C is selected.

3. Press the Filename field above the list column headings. A keypad will appear.

    Enter a name for the merged data capture file and choose OK.

4. Select two or more files to merge, and then choose the Merge button.

    If the selected files are compatible, they will be copied and merged into one new file. The selected files will remain intact as distinct files.

    In the merged file, merge points will be marked with a note icon.

5. When you are finished merging files, choose the X button. The Merge Data Capture Files window will close and you will return to the Utilities screen.

Chapter 18: Transferring Data & Setup Files

This chapter will help you learn about the optional Dash 18X offline software.

About the offline software

The offline software is the Dash 18X program designed to run on a personal computer running Windows 98/ME/2000/XP. It provides most of the functionality of the standard Dash 18X program and can be used for a variety of purposes.

Some common uses of Dash 18X offline software include:

Creating setup files – You can use the offline software to create view, control panel, signals, and global setup files. You can later transfer these files to the Dash 18X for use via CD, DVD, USB memory device, or network connection.

Viewing data records – You can use the offline software to view data files captured on the Dash 18X. Once the data capture files are transferred to the Dash 18X, you can analyze the files using Review mode.

Note: Dash 18X offline software is not to be confused with AstroVIEW software. These are two distinct products.

Installation

The Dash 18X offline software is installed using a wizard-based installation common to most software products. The installation program should run automatically when you insert the CD into your CD-ROM drive.

Note: If the program does not automatically run, launch the "setup.exe" file from the CD. You can do this using Windows ® Explorer or the Run option from the Start menu (e.g. d:/setup.exe – where d: is your CD-ROM drive).

Transferring data and setup files using FTP

You can use File Transfer Protocol (FTP) to transfer files between the Dash 18X and a computer connected to the same network. This process is initiated using the Dash 18X offline software.

Transferring setup files to the Dash 18X

It is common for users to configure setup files using a computer running the Dash 18X offline software. These files can later be transferred to a Dash 18X via an ethernet network connection.

To transfer setup files to the Dash 18X:

1. Ensure the Dash 18X is on and properly connected to your network.

   Ensure that the Dash 18X offline software is installed on a computer connected to your network.

2. In the Dash 18X offline software, choose Service >> Network >> FTP from the menu bar. The FTP window will open.
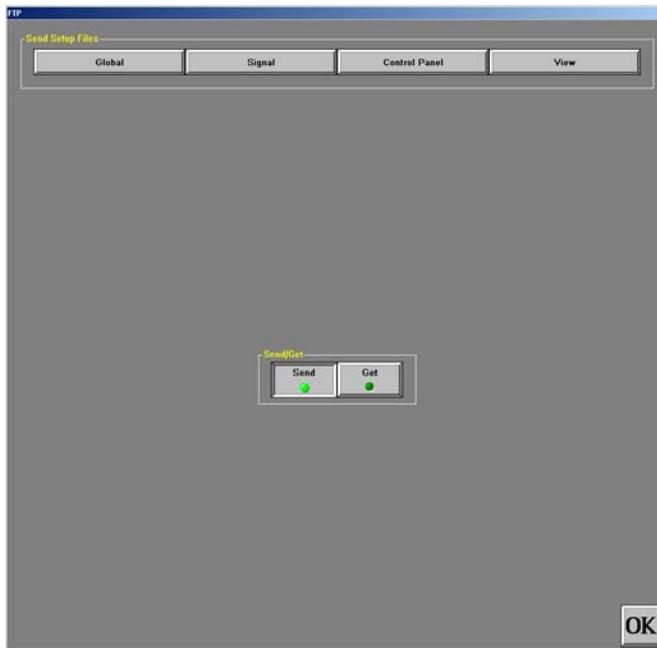
3. Choose the type of file you wish to send to the Dash 18X and press the corresponding button. You can send Global, Signal, Control Panel, and View files. The Send File window will open.
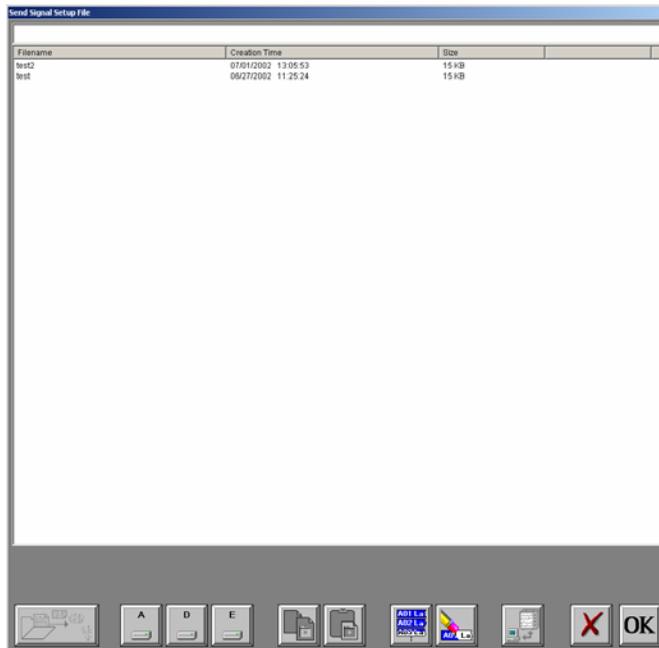
4. Select the file you wish to transfer. Then choose the FTP button.

The Select Remote Machine window will open.
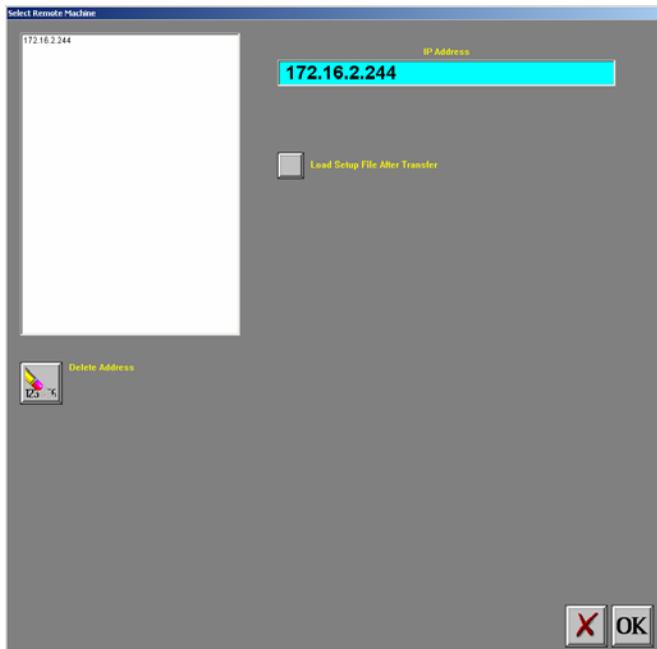
5. Choose the IP Address field to specify an IP address. A keypad will open. Use the keypad to enter the IP Address of the Dash 18X that will receive the transferred file, then choose the OK button to close the keypad.
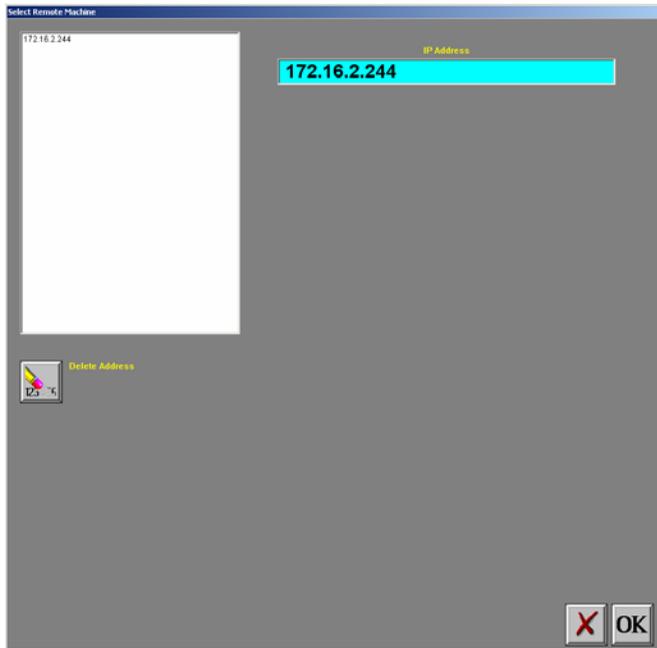
    When you enter an IP address, it will be added to the list in this window. To choose the IP address in the future, you can select it from the list instead of entering it again. To remove IP addresses from this list, select the address and choose the Delete Address button.

6. Choose whether to load the setup file automatically after transfer. To load the file after transfer, check the "Load Setup File After Transfer" button.

7. Choose the OK button. The file will be transferred and you will return to the Send File window. A "Transfer Complete" message will display on the bottom of this window.
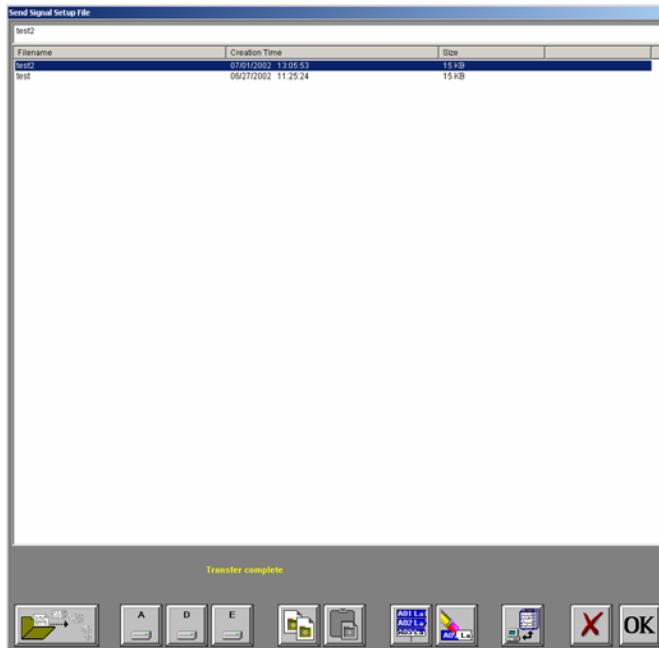
8. Choose the OK button in the Send File window.

9. Choose the OK button in the FTP window.

Transferring data files from the Dash 18X

It is common for users to analyze data capture files using the Dash 18X offline software. These files can later be transferred from a Dash 18X to a computer via an ethernet network connection.

To transfer data files from the Dash 18X:

1. Ensure the Dash 18X is on and properly connected to your network.

   Ensure that the Dash 18X offline software is installed on a computer connected to your network.

2. In the Dash 18X offline software, choose Service >> Network >> FTP from the menu bar. The FTP window will open.
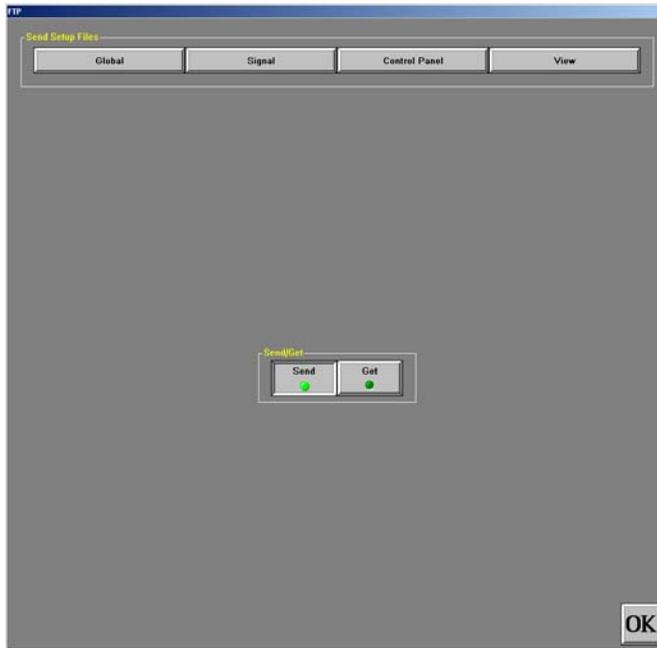
3. Choose the Get button. The Get Data Files options will appear in the window.
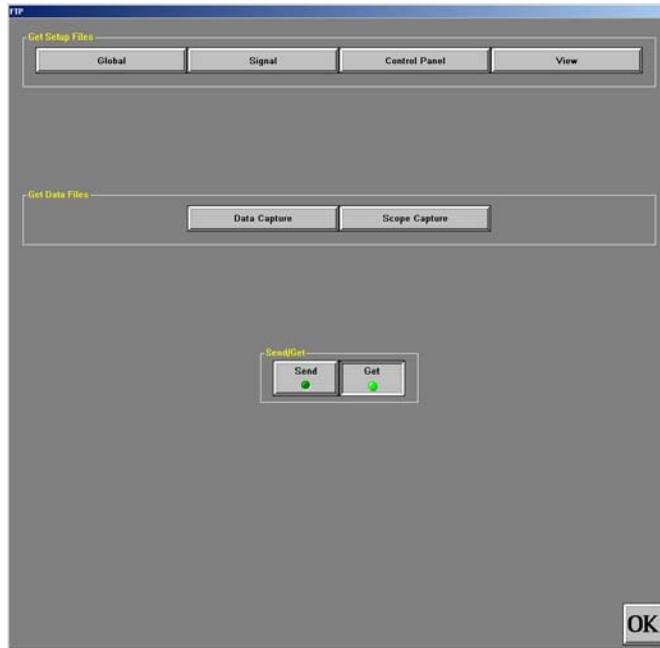

4. Choose the type of file you wish to get from the Dash 18X and press the corresponding button. You can transfer Global, Signal, Control Panel, View, Data Capture and Scope Capture files. The Select Remote Machine window will open.

5. Choose the IP Address field to specify an IP address. A keypad will
    open. Use the keypad to enter the IP Address of the Dash 18X that you
    will transfer the file from, then choose the OK button to close the
    keypad.

    When you enter an IP address, it will be added to the list in this window.
    To choose the IP address in the future, you can select it from the list
    instead of entering it again. To remove IP addresses from this list, select
    the address and choose the Delete Address button.

6. Choose the OK button. The Get File window will open displaying a list of data files on the Dash 18X. If a large number of files are present, this process might take a few moments to complete.
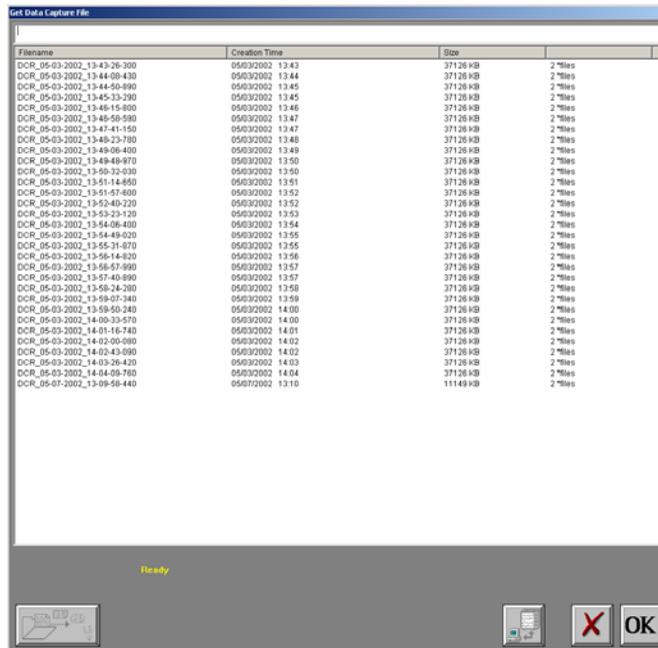
7. Select the file you wish to transfer. Then choose the FTP button.

The file will be transferred from the Dash 18X to your computer. 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, you can cancel the transfer by choosing the Cancel FTP button.

8. Choose the OK button.

Transferring data capture files to your PC using CDs or DVDs

You can transfer data captures files from the Dash 18X to your computer using CDs or DVDs. If you do not use the offline software, you can still use these instructions to learn how to transfer files to a CD or DVD, which you can use for backup purposes.
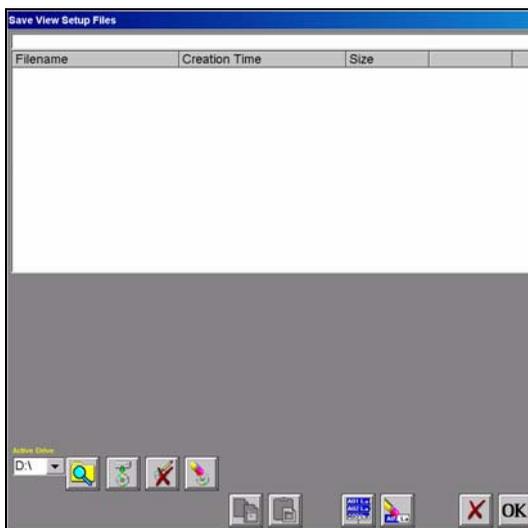
Formatting DVD+RW media

If you are using DVD+RW media, you must format the disc the first time you use it in the Dash 18X. Once formatted, the disc will be ready to use. If necessary, you can later reformat the disc to erase it.

Note: Only DVD+RW media requires formatting prior to use. If you are using another media type, please skip the following procedure.

To format DVD+RW media:

1. Insert a DVD+RW disc into the DVD+RW drive.

2. Open any file save or load window. In this example, the Save View Setup File window is used.

3. Select the D drive from the Active Drive list.

4. Choose the Format Disc button.

A confirmation message will appear.

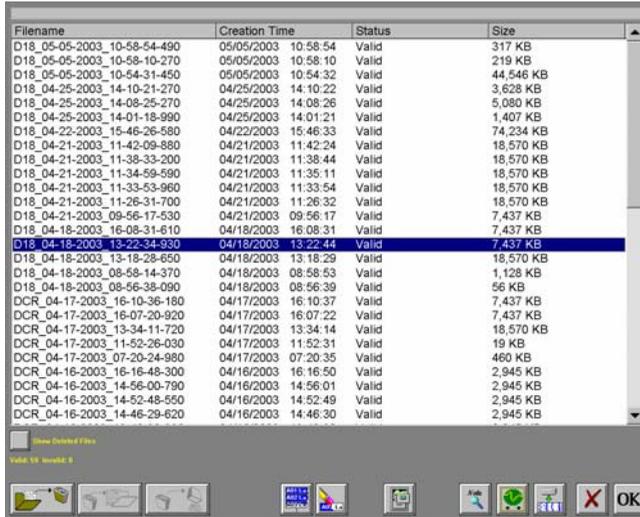To continue and format the disc, choose Yes. To cancel, choose No.

Note: Use caution when formatting DVD+RW media to ensure that no needed files are deleted. Formatting a DVD+RW disc may require up to 30 minutes to complete.

Transferring data capture files to your PC using CDs or DVDs

Use the following instructions to transfer data capture files to your PC using CDs or DVDs.

To transfer data capture files to your PC using CDs or DVDs:

1. Ensure that you have a CD or DVD in the Dash 18X DVD+RW drive.
    Then launch Review mode on the Dash 18X by choosing Configuration
    >> Review from the menu bar.

2. Choose whether to transfer scope capture files or archived data capture
    files.

    - To transfer a scope capture file, choose the Scope Capture button.

    - To transfer a data capture file, choose the Data Capture file button.

    The Load Scope Capture File or Load Data Capture File window will
    appear.

3. Select the file(s) you want to transfer. Then choose the Copy button.

4. Choose the D Drive from the drop-down menu. Then choose the Paste button.

The files you selected will be copied to the CD or DVD.

5. When the process is complete, remove the CD or DVD from your Dash 18X and insert it in your PC CD or DVD drive. Ensure that your drive can read the type of media used.

You will be able to access the files(s) using the Dash 18X Offline software by choosing the CD or DVD when you open a file.

Closing DVDs and CDs

After being written to, a recordable CD or DVD must be "closed" before its contents can be read by another machine. Once a disc is closed, it can no longer be written to. Do not close a disc if you need to be able to write additional files to it.

To close a disc:

1. Open any file save or load window. In this example, the Save View Setup File window is used.
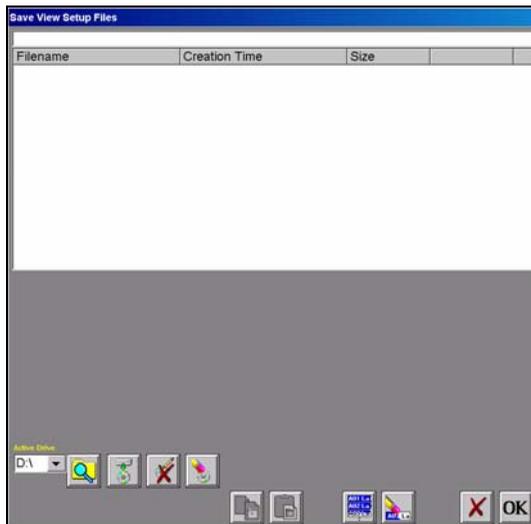
2. Select the D drive from the Active Drive list.

3. Choose the Close Disc button.

A confirmation message will appear.

Choose Yes to continue and close the disc. Choose No to cancel this process and leave the disc open.
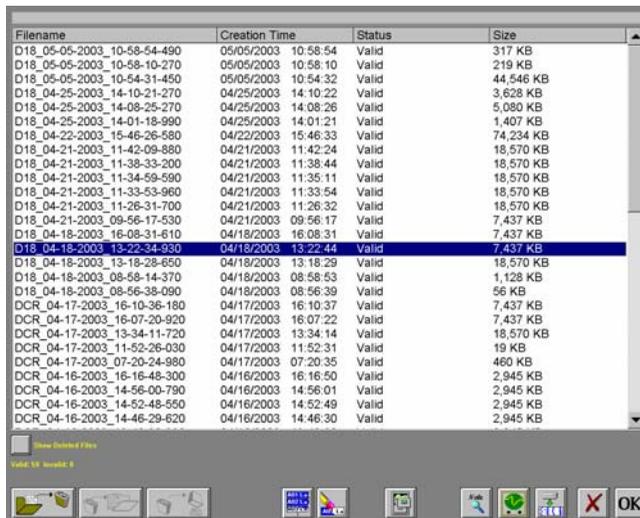
When the closing process is complete, you will be able to use the disc in other devices.

Transferring data capture files to your PC using a USB storage device

The typical transfer process involves copying files to the Dash 18X clipboard, pasting the files to a USB storage device, and retrieving files from the USB storage device with the Dash 18X offline software. These instructions can also be used to create backup copies of files for archive purposes..

To transfer data capture files to your PC using USB storage devices:

1. Ensure that a USB storage device is properly connected to the Dash 18X. Then launch Review mode on the Dash 18X by choosing Configuration >> Review from the menu bar.

2. Choose whether to transfer scope capture files or archived data capture files.

    *   To transfer a scope capture file, choose the Scope Capture button.

    *   To transfer a data capture file, choose the Data Capture file button.

The Load Scope Capture File or Load Data Capture File window will appear.

3. Select the file(s) you want to transfer. Then choose the Copy button.

4. Choose the E Drive from the drop-down menu. Then choose the Paste button.

    The files in the Dash 18X clipboard will be pasted to the selected drive location.

5. Remove the USB storage device from the Dash 18X and connect it to the computer running Dash 18X offline software.

    The files on the USB storage device can be accessed using the Dash 18X offline software. To open a file from the device, choose the corresponding drive letter when using a load file window.

Chapter 19: FTP and web browser features

This chapter will help you learn about the FTP and web browser features available when the Dash 18X is connected to your network..

Using FTP applications

In addition to transferring files with the Dash 18X offline software, It is possible to transfer files via other FTP programs.

Note: This feature should only be used by individuals familiar with FTP applications and concepts.

Some web browsers provide FTP functionality. When using a browser, you will need to enter the FTP address in the address bar, and then enter your username and password. For a more direct method of using a web browser to download data capture files, refer to the following section.

Use the following information to log into the Dash 18X using FTP programs.

Username: dash18
Password: dash18
FTP Address: ftp:// 192.168.255.1/ (or other IP address if changed)

All Dash 18X files are stored in a folder located in the root directory. The path to this folder is: C:\D18. Within this folder, the following sub folders are present:

ASCII Capture Files – This folder contains all data capture files 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, you will need to copy the entire folder. The data capture folders contains a DCR file, which contains the data capture record, and an IDX file, which contains a listing of external items used in a data capture (images, notes, sounds, and scope captures). Additionally, these external items (BMP, TXT, WAV, and SCR) are also stored in this folder.

Note: You can open and analyze DCR files using AstroVIEW 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, you will need to 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 18X software and automatically load control panel, signal, and view setup files. When you transfer a setup file 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. When you transfer global setup files to this folder, you will need to follow a specific sequence:

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.

View Setup Files – This folder contains view setup files. These files have an extension of DVS.

Web browser interface

The Dash 18X provides a browser-based interface that can be accessed from any computer on your network. This feature provides basic functionality to support remote use.

Opening the browser interface

The features described in the rest of this section assume that you have the browser interface to the Dash 18X up and running. Use the following instructions to access the Dash 18X browser interface.

To open the browser interface:

1. Ensure the Dash 18X is on and properly connected to your network.

2. Launch a web browser using a computer connected to your network.

3. Enter the following address in the address bar:

    http:// 192.168.255.1/ dash18

    If you changed the IP address from the default, you will need to enter the new IP address instead.

    The web browser interface will open and display the Status page.
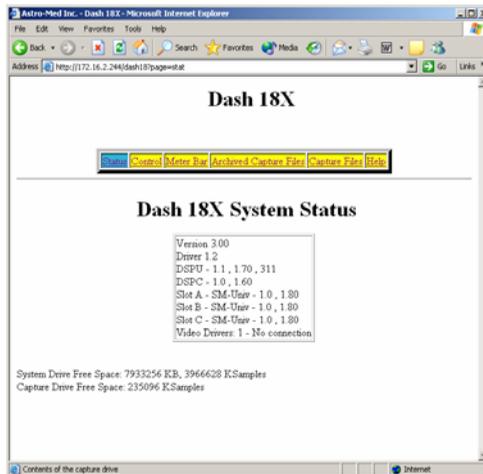
The Status page displays the current system settings of the Dash 18X. Listed are the system software version, the DSPU, the DSPC, the

modules installed, and printer status. This is the same information shown by the Dash 18X during startup. The available space on the capture and archive drives are also displayed.

Note: A help page that provides descriptions of other page functions is also available.

Using the control functions

Use the following instructions to control data captures from a computer connected to your network. A data capture can be armed, triggered, and 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 option. The Control page will open.

2. You can choose to arm, trigger, or abort a data capture from this page by choosing the appropriate button.

    The data capture process in this case uses capture settings that are currently configured on the Dash 18X.

3. If necessary, you can format the data capture drive by pressing the Quick-Format Capture Drive button.

> **Warning:** This function will erase the capture drive. This function cannot be aborted or undone.

Viewing meter bar values

Use the following instructions to view the meter bar signal measurement values from a computer connected to your network. The Meter Bar screen functions exactly as the meter bar on the Dash 18X; it shows the realtime numerical values of the channels in the current view.

To view meter bar values:

1. With the browser interface open, choose the Meter Bar option. The 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 rate selected.

3. To stop refreshing the meter bar values, choose the Stop Refreshing option.

Accessing archived data capture files

Use the following instructions to download data capture files to a computer connected to your network. You can also delete files using this feature.

To access archived data capture files:

1. With the browser interface open, choose the Archived Capture Files option. The Archive page will open.
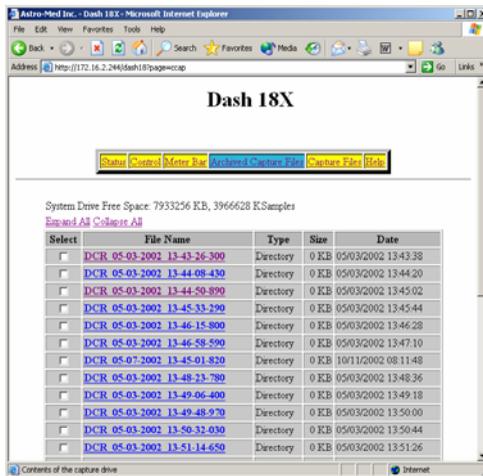
2. Choose your viewing preference by selecting the Expand All or Collapse All options. The expanded view provides a listing of all files within a data capture record folder. The collapsed view provides a listing of data capture record folders.

   You can also expand and collapse individual data capture record folders by choosing them individually.

3. To download a file to your computer, choose the link for the file. A download prompt will open. Choose to save the file and save it to a location on your computer.

4. To delete a data capture record, select the checkbox next to the record name. Then type the word "Yes" into the "Are you sure?" box and choose the Delete Selected Files button. The selected files will be deleted.

Archiving capture files

Use the following instructions to archive capture files to the Dash 18X system hard drive from a computer connected to your network.

When capture files are archived, you will be able to download them using the Archived Capture Files page.

To archive capture files:

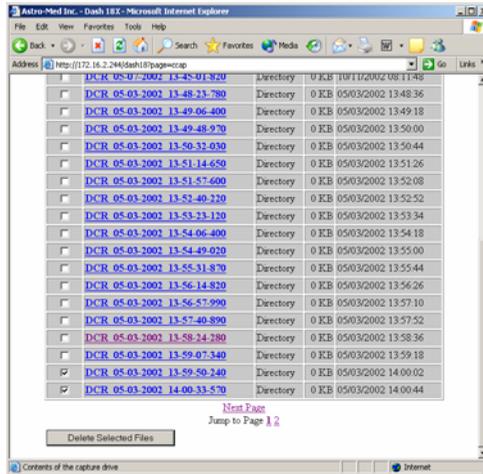1. With the browser interface open, choose the Capture Files option. The Capture Files page will open.
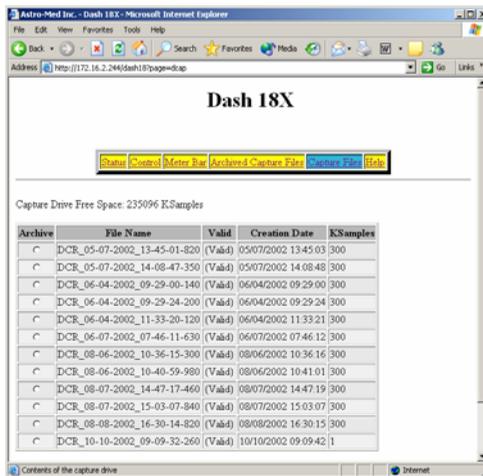
This page displays the contents of the data capture drive.

2. To archive a file, mark the Archive radio button next to the file you wish to archive. You can only select one file at a time.

    Select whether to overwrite an existing file, if one exists, using the check box. Checking this option will allow you to overwrite a file if it has the same name as the file you are archiving.

    Then choose the Archive Data Capture button. The file will be archived.
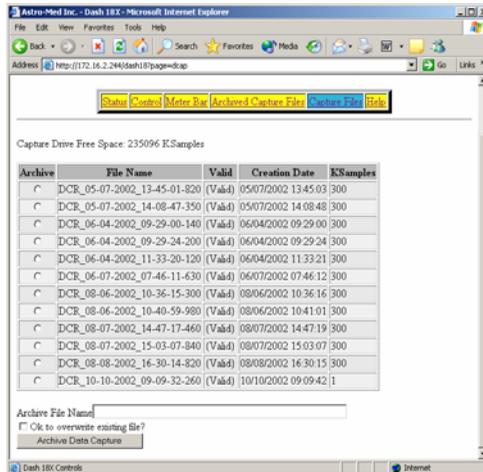
Note: Archiving files does not remove them from the data capture drive. If necessary, you can remove all files from the data capture drive remotely using the Control page.

Appendix A: Specifications

Color Display

Type..Active matrix color LCD (TFT) with 17" display
Resolution...............................1280 x 1024
Touch ....Full screen, resistive
Functions ....User interface with touch-based icons and menus; Real-time waveform monitoring; Review previous waveform records while recording; Overlay numeric values in Engineering Units

Signal Modules

Number of Modules..............Up to 3
Number of Channels.............Maximum of 18
A/D ....14 bit

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 > 0.01 msec

Real-time Signal Processing

Filters......................................Low Pass Stops: 10 Hz to 10,000 Hz; High Pass Starts: 0.1 Hz to 100 Hz; Notch: 50 or 60 Hz center
RMS..Time constant selectable from 0.002 to 2 seconds
Frequency to Voltage ....5 Hz to 20 kHz, ± 0.05 % of measurement + 0.1 Hz
Cross Channel........................Sum, Difference, Ratio, Product (Power)

Real-time Signal Testing

Basic..All active signals monitored simultaneously
Test Types..............................Tests include window, slope/level, slew and event pattern
Outputs ....Result available for alarms, trigger, abort and external TTL

Data Capture

Recording Method................ Internal 73 GByte hard drive
Max. Sample Rate .... 100,000 samples/second each channel (all channels)
Min. Sample Rate.................. 1 sample/minute
Dual Sample Rate .... Second slower rate available
Total Capacity .... Over 16 billion samples
Maximum Record .... 2 billion sample periods
Header................................... Information on units, range, sample rates, etc. saved
with data
Events................................... All standard event inputs captured with waveforms
Trigger Point Lock................ Amount of pre- and post-trigger is user adjustable
Auto Re-Arm......................... Allows automatic stacking of captures
Ext. Sample rate .... External TTL sample clock to 40,000 Hz
Rates approaching 100 KHz can be used if up to a
10-microsecond jitter is acceptable.
Timebase Accuracy .... 50 PPM

Signal Conditioner Specifications (SM-U)

General Specifications (all input types unless specified otherwise)
Channels per module........... 6
Isolation .... 250 Vrms (iso-common to chassis and other iso-
commons)
Bandwidth............................. 12 kHz (-3 dB)
Input Coupling .... DC
User Engineering Units .... Yes
Calibration............................ Semi-automated to external reference
Input Type .... Overvoltage Category II, Pollution Degree 2

Single-ended Voltage Input

Connector .... Guarded banana jack
Max Rated Input.................... ± 250 Vrms
Max Transient Input............. ± 400 V
Specified Ranges................... 40 to 400 VFS; 4 to 40 VFS; 0.4 to 4 VFS
Accuracy (25 °C) .... ± 0.5% of attenuator
Intrinsic Noise...................... Less than .1% of attenuator.
Min Input Impedance .... 1 Megohm

Differential Voltage Measurements (use for bridge measurements)

Input Type .... Isolated, differential
Connector .... Screw Terminal Header
Absolute Max Input .... ± 40 V differential
Measuring Ranges .... 200 to 1600 mVFS; 50 to 500 mVFS; 5 to 50 mVFS
Accuracy (25°C) .... ± 0.5% of attenuator
Min Input Impedance .... 200 KOhm
CMR at 60 Hz .... > 80 dB
Intrinsic Noise .... +/- 0.1% of attenuator (10 KHz LPF on 50 mV attenuator)
Auto Balance .... Yes (limited by maximum span)

Thermocouple Measurements

Input Type .... Isolated, differential with Screw Terminal Header
                                     Connector
Absolute Max Input .... ± 40 V
Specified Range .... Type J: 0 to 760 °C;
                                     Type K: 0 to 1370 °C;
                                     Type T: -160 to 400 °C;
                                     Type E: -100 to 870°C;
                                     Type N: 0 to 1300°C
Accuracy (25°C) .... 0.5% of measurement +2 °C
Bandwidth .... 10 Hz (-3 dB)
Intrinsic Noise .... 0.5 °C (1 Hz filter)
Linearization .... NIST polynomial
Units .... °C or °F

Excitation

Connector .... Screw Terminal Header
Excitation .... Isolated 10.0 V @ 20 mA
Adjustable .... 0.1 to 10.1 V

Frequency Measurements

Method .... Zero-crossing, channels 1, 7 and 13 only
Signal Amplitude .... Peak to peak minimum is 20% of attenuator
Hysteresis .... Yes

Removable Drive

Type....................................... DVD +/- RW
Format.................................... CDFS (CD-R or CD-RW), UDFS (DVD-RW)
Function................................. Setup files, software upgrades, data transfer/archive

Power

Input Voltage Range .... 100 to 250 VAC
Frequency Range .... 47 Hz to 63 Hz
Power Consumption .... 300 W maximum, 130 W typical w/o printer

Interface

Ethernet 10/100BaseT
DVD....................................... Supports archive of data to CD-R, CD-RW, DVD-R,
                                                      DVD+R & DVD+RW
VGA....................................... For displaying data on an external monitor
USB 1.1 .... For external peripherals and file export

Physical

Enclosure .... Aluminum
Dimensions............................ 15" L x 16" W x 6.4" H
Weight.................................... 26 lbs

Environmental

Operating Temp.................... 5 to 40 °C (40 to 105 °F)
Non-operating Temp .... -20 to 60 °C (-4 to 140 °F)
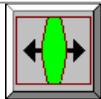Operating Humidity .... 10% to 90% non-condensing

Optional Chart Recorder

Chart Width........................... 11"
Max. Chart Speed .... 50 mm/sec

Optional DC Input Power

Input Range............................9 - 18 V
Input Power ....200 W max
Fuse ....User replaceable 25 A

Compliance

Safety.....................................EN 61010-1, 2nd Edition (2001)
　　　　　　　　　　　　　　UL 61010-1:2004, 2nd Edition
　　　　　　　　　　　　　　CAN/CSA C22.2 No. 61010-1:2004 2nd Edition
EMC.......................................FCC Part 15, Subpart B, Class A,
　　　　　　　　　　　　　　EN 61326:1998 Class A

Appendix B: Button definitions

	Abort Setup opens the Abort Setup window, which you will use to set up aborts.
	Add/Remove Channels opens the Chart/Display Wizard window in Scope mode. You will use this window to set up the display characteristics of channels.
	Advanced Search opens the Advanced Search window in Review mode. You will use this window to navigate through the file using specific search criteria.
	Amp Down decreases the span of the indicated waveform. Press and hold this button to quickly decrease the value.
	Amp Up increases the span of the indicated waveform. Press and hold this button to quickly increase the value.
	Analog Channel Setup opens the Analog Channel Setup window, which you will use to set up channels.
	Apply saves the information you modified in a window without closing the window.
	Arm/Abort arms and aborts data and scope captures.
	Archive File archives the entire file, the currently displayed page of the file, or the portion of the file between cursors.
	AstroVision opens the Snapshot Camera window, which you will use to record realtime video snapshots. This function requires the optional CAM-18 webcam accessory.

	Calibration opens the Calibration window, which you will use to calibrate channel inputs.
	Calculator opens the Calculator program that is part of the Windows® operating system.
	Capture Indicator indicates when a data capture is in progress by illuminating.
	Capture Setup opens the Capture Setup window, which you will use to configure the data capture settings.
	Change Active Cursor changes the active cursor configuration between A, B, and A + B.
	Channel Setup opens the Scope Channel Setup window, which you will use to set up display characteristics of channels in Scope Mode.
	Chart Mark prints a full page mark across the chart. This function requires the optional SCR-18 printer.
	Chart Run/Halt starts and stops the chart from printing.
25 mm/s	**Chart Speed** controls the rates at which the chart moves. Five default options are provided (1, 5, 25, 50, 100 mm/s). By customizing the control panel, you can add specific speed buttons in units of millimeters per second or minute.
	Clear Selection removes the selections/highlights from all items in a list box.
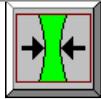
		Compress compresses the display of files in Review mode to fit more of the file on the screen.
		Context Help provides online help based on the mode or the menu you are using.
		Copy (Amplifier) copies the amplifier settings for the selected items to the clipboard.
		Copy (File) copies the selected file to the clipboard.
		Copy (User Units) copies the user unit settings for the selected items to the clipboard.
		Decrease Time Base decreases the time base used for scope captures.
		Exit cancels the action you were performing in a window and closes the window without saving any changes.
		Expand expands the display of files in review mode and fits less of the file on the screen.
		Fast Forward scrolls the chart forward quickly.
		Feed advances the paper in the SCR-18 printer.
		Full Page Mark displays a full page mark across the scrolling waveform display area.
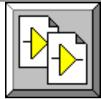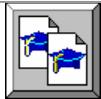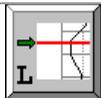
		Goto Cursor A scrolls the Review mode display to the location of cursor A.
		Goto Cursor B scrolls the Review mode display to the location of cursor B.
		Goto Cursors A + B compresses or expands the review mode display to the portion of the file between cursors A and B.
		Goto End scrolls the display in Review mode to the end of the file.
		Goto Realtime Mode launches Realtime mode.
		Goto Review Mode launches Review mode.
		Goto Scope Mode launches Scope mode.
		Goto Start scrolls the display in Review mode to the start of the file.
		Goto Trigger scrolls the display in Review mode to the trigger point in a file.
		Goto Utilities launches the Utilities screen.
		Icon Help provides online help for identifying the purpose of buttons.

	Increase Time Base increases the time base used for scope captures.
	Load loads a file (or configurations from a file) on your disk.
	Load Control Panel File loads the specified control panel file.
	Load Global File loads the specified global file.
	Load New Capture File launches the file open screen in Review mode.
	Load Signal File loads the specified signal file.
	Load View File loads the specified view file.
	Manual Abort stops the data or scope capture currently in progress.
	Manual Trigger triggers a data or scope capture manually.
	Meter opens the channel meter, which displays numeric values for signals.
	Monitor/Chart Wizard opens the Chart/Display Wizard window, which you will use to set up the display characteristics of channels.
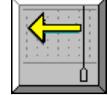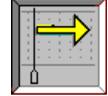
	Move Cursor Left moves the active cursor(s) to the left each time you choose it.
	Move Cursor Right moves the active cursor(s) to the right each time you choose it.
OK	**OK** saves the information you entered in a window and then closes the window.
FFT	**Open FFT Window** opens the FFT window in Scope and Realtime modes.
Hz	**Open Filter Window** opens the Filter window in Review mode.
	Open Zoom Window opens the Zoom window in Review mode.
	Paste (Amplifier) pastes the amplifier settings from the clipboard to the selected items.
	Paste (File) pastes the file from the clipboard to the selected location.
	Paste (User Units) pastes the user unit settings from the clipboard to the selected items.
	Pen Up/Down Control opens the Pen Up/Down Control window, which you will use to raise and lower pens.
ID	**Print IDs** prints the IDs of each channel next to the waveforms.
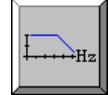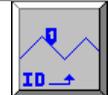
		Print Labels prints channel label information on the chart. This function requires the optional SCR-18 printer.
		Print On-Demand Buffer prints the demand buffer, a string of text stored in memory, on the chart. This function requires the optional SCR-18 printer.
		Print prints the display.
		Realtime Setup opens the Realtime Setup window, which you will use to define Realtime setup options.
		Realtime XY opens the XY window.
		Rewind scrolls the chart backward quickly.
		Run/Freeze Display starts and stops the chart from running.
		Save saves current configurations to a file on your disk.
		Scope 1 or 2 Screens selects whether to capture an amount of data that will fit on one or two screens.
		Scope Pre-Trigger Percent opens the Pre-Trigger Percent window, which you will use to enter the percent of the scope capture used for pre-trigger data.
		Scroll Back scrolls the chart backward.

	Scroll Forward scrolls the chart forward.
	Scroll Stop stops the chart from scrolling.
	Select All selects/highlights all items in a list box.
	Separator adds a separator bar between control panel buttons.
	Show All compresses the file in Review mode to fit the entire file on the display.
	Show/Hide Channel IDs displays and hides identification labels for channels.
	Show/Hide Cursor A displays and hides cursor A.
	Show/Hide Cursor B displays and hides cursor B.
	Show/Hide Trigger Line displays and hides the trigger line in a scope capture file.
	Solitaire opens the Solitaire program that is part of the Windows® operating system.
	Start Capture initiates the data or scope capture.

	Timebase Down decreases the timebase used in scope captures.
	Timebase Up increases the timebase used in scope captures.
	Trigger Indicator indicates when triggers occur by displaying a yellow circle.
	Trigger Setup opens the Trigger Setup Menu window, which you will use to set up triggers.

Index