DATA TRANSLATION

UM-25242-E

QuickDAQ User's Manual

for QuickDAQ 2014

Fifth Edition
November, 2014

Data Translation, Inc.
100 Locke Drive
Marlboro, MA 01752-1192
(508) 481-3700
www.datatranslation.com
Fax: (508) 481-8620
E-mail: info@datx.com

Table of Contents

About this Manual

The manual describes the features of QuickDAQ 2104, how to get started using QuickDAQ 2014, and how to create a typical application to acquire and analyze data from a data acquisition device.

Note that QuickDAQ 2014 is referred to as QuickDAQ throughout this manual.

Intended Audience

This document is intended for engineers, scientists, technicians, or others responsible for acquiring and analyzing measurement data. It is assumed that you are familiar with the Windows operating environment, that you are familiar with measurement principles.

Organization of this Manual

This manual is organized as follows:

- Chapter 1, "Introduction to QuickDAQ," provides an overview of the features of QuickDAQ.
- Chapter 2, "Quick Start," describes how to install and start QuickDAQ.
- Chapter 3, "Understanding the Interface," describes the basic elements of the Data Logger and FFT Analyzer interfaces of QuickDAQ.
- Chapter 4, "Configuring Channels, Acquisition Settings, and Digital Filters," describes how to configure the channels and acquisition settings of a supported device using QuickDAQ.
- Chapter 5, "Configuring the Display of the Data Logger Interface," describes how to configure the settings of the Channel Plot, Channel Display, and Statistics windows when using the Data Logger interface of QuickDAQ.
- Chapter 6, "Configuring the Display of the FFT Analyzer Interface," describes how to configure the settings of the Channel Plot, Channel Display, and Statistics windows when using the FFT Analyzer interface of QuickDAQ.
- Chapter 7, "Examples Using the Data Logger Interface," provides step-by-step instructions that you can follow to create applications when using the Data Logger interface of QuickDAQ.
- Chapter 8, "Examples Using the FFT Analyzer Interface," provides step-by-step instructions that you can follow to create applications when using the FFT Analyzer interface of QuickDAQ.
- Chapter 9, "Support," provides technical support information.

A glossary and index complete this manual.

Conventions Used in this Manual

The following conventions are used in this manual:

- Notes provide useful information that requires special emphasis, cautions provide information to help you avoid losing data or damaging your equipment, and warnings provide information to help you avoid catastrophic damage to yourself or your equipment.
- Items that you select or type are shown in **bold**.

Related Information

This manual is intended to be used with the documentation for your data acquisition device.

Where to Get Help

Should you encounter problems installing or using QuickDAQ, the Data Translation Technical Support Department is available to provide technical assistance. Refer to Chapter 9 starting on page 383 for information on how to contact the Technical Support Department.

1

Introduction to QuickDAQ

Features

Three versions of QuickDAQ 2014 are available:

- **QuickDAQ 2014 Base –** This free-of-charge application allows you to perform the following functions:
  - Discover and select your devices.
  - Configure all input channel settings for the attached sensors.
  - Load/save multiple hardware configurations.
  - Generate output stimuli (fixed waveforms, swept sine waves, or noise signals).
  - On each supported data acquisition device, acquire data from all channels supported in the input channel list.
  - Choose to acquire data continuously or for a specified duration.
  - Choose software or triggered acquisition.
  - Log acquired data to disk in an .hpf file.
  - Display acquired data during acquisition in either a digital display using the Channel Display window or as a waveform in the Channel Plot window.
  - Choose linear or logarithmic scaling for the horizontal and vertical axes.
  - View statistics about the acquired data, including the minimum, maximum, delta, and mean values, and the standard deviation in the Statistics window.
  - Export time data to a .csv or .txt file; you can open the recorded data in Microsoft Excel® for further analysis.
  - Read a previously recorded .hpf data file.
  - Customize many aspects of the acquisition, display, and recording functions to suit your needs, including the acquisition duration, sampling frequency, trigger settings, filter type, and temperature units to use.
- **QuickDAQ 2014 FFT Analysis Option –** When enabled with a purchased license key, the FFT Analysis option includes all the features of the Base package plus these features:
  - The ability to switch between the Data Logger time-based interface and the FFT Analyzer block/average-based interface.
  - Supports software, freerun, or triggered acquisition with accept and reject controls for impact testing applications.
  - Allows you to perform single-channel FFT (Fast Fourier Transform) operations, including AutoSpectrum, Spectrum, and Power Spectral Density, on the acquired analog input data. You can configure a number of parameters for the FFT, including the FFT size, windowing type, averaging type, integration type, and so on.
  - Allows you to display frequency-domain data as amplitude or phase.
  - Supports dB or linear scaling with RMS (root mean squared), peak, and peak-to-peak scaling options
  - Supports linear or exponential averaging with RMS, vector, and peak hold averaging options.

- Supports windowed time channels.
- Supports the following response window types: Hanning, Hamming, Bartlett, Blackman, Blackman Harris, and Flat top.
- Supports the ability to lock the waveform output to the analysis frame time.
- Allows you to configure and view dynamic performance statistics, including the input below full-scale (IBF), total harmonic distortion (THD), spurious free dynamic range (SFDR), signal-to-noise and distortion ratio (SINAD), signal-to-noise ratio (SNR), and the effective number of bits (ENOB), for selected time-domain channels in the Statistics window.
- Supports digital IIR (infinite impulse response) filters.

• **QuickDAQ 2014 Advanced FFT Analysis Option –** When enabled with a purchased license key, this version includes all the features of the Base package with the FFT Analysis option plus these features:
  - Allows you to designate a channel as a Reference or Response channel.
  - Allows you to perform two-channel FFT analysis functions, including Frequency Response Functions (Inertance, Mobility, Compliance, Apparent Mass, Impedance, Dynamic Stiffness, or custom FRF) with H1, H2, or H3 estimator types, Cross-Spectrum, Cross Power Spectral Density, Coherence, and Coherent Output Power.
  - Supports the Exponential response window type.
  - Supports the following reference window types: Hanning, Hamming, Bartlett, Blackman, Blackman Harris, FlatTop, Exponential, Force, and Cosine Taper windows.
  - Supports real, imaginary, and Nyquist display functions.
  - Allows you to save data in the .uff file format.

Supported Devices

You can acquire and analyze data from a number of devices using QuickDAQ, including the following:

- All DT-Open Layers devices that support A/D streaming
- MEASURpoint instruments
- TEMPpoint instruments
- VOLTpoint instruments
- DT8824 and DT8824-HV instrument modules
- DT8837 instrument modules (analog input operations only)

Analog output operations are supported on devices that support D/A streaming. Single-value analog output operations are currently not supported.

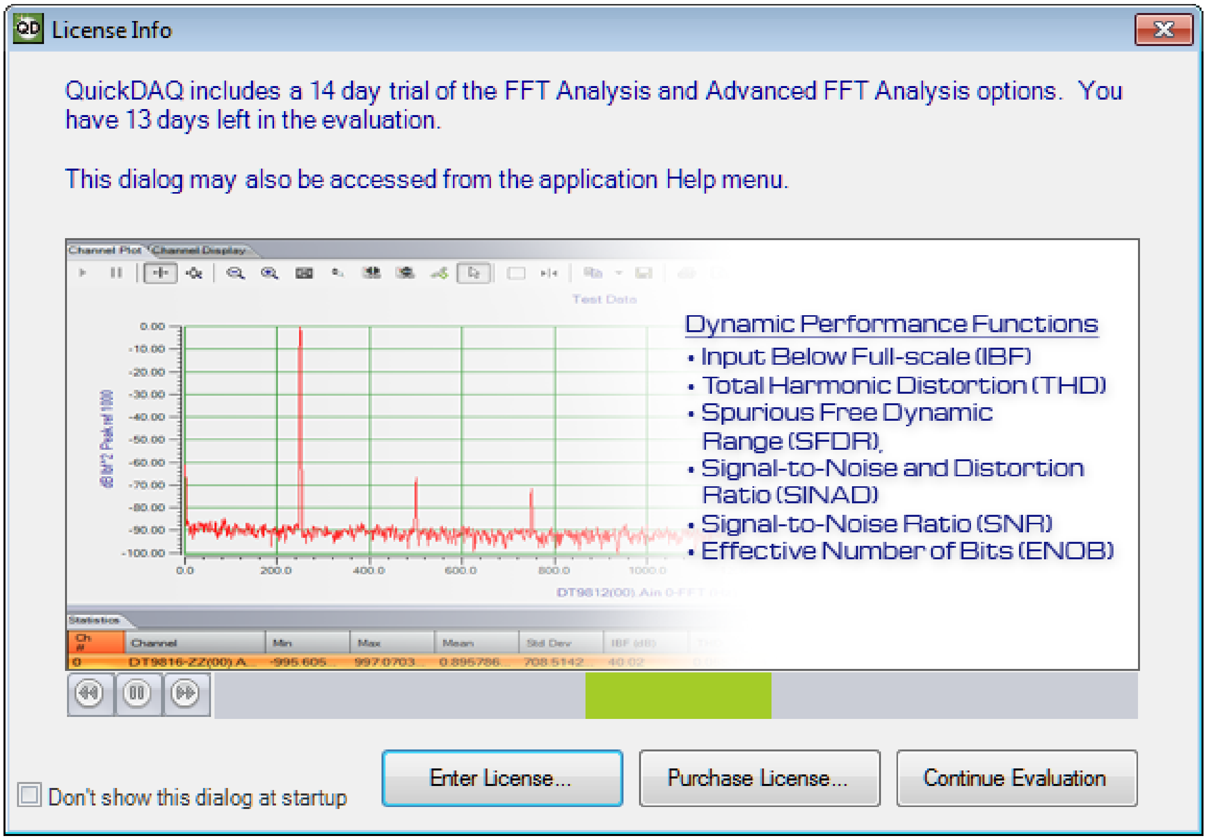
Modes of Operation

You can use QuickDAQ in one of two operation modes:

- Acquisition mode – Use this mode if you want to acquire data in the analog input stream and/or output an analog waveform. This mode requires use of at least one of the supported Data Translation devices; refer to page 18 for a list of supported devices.
- File reader mode – Use this mode if you want to view a previously recorded data file. Note that you cannot acquire new data using the File reader mode.

Licensing Requirements

QuickDAQ 2014 includes a 14-day trial version of the FFT Analysis and Advanced FFT Analysis options. Once you start the software, you have 14 consecutive calendar days in which to try the features of the application.



To use the trial version, click **Continue Evaluation**.

Note: If you do not want to show the License Info dialog, select the **Don't show this dialog at startup** checkbox. You can access the License Info dialog from the **License Information**... option of the **Help** menu.

When the 14-day trial period has elapsed, you can continue using the Base package, but the features of the FFT Analysis and Advanced FFT Analysis options are no longer available. The License Info dialog will no longer be shown.

If you want to continue using the analysis features, you must purchase either the FFT Analysis license key or the Advanced FFT Analysis license key depending on the features that you require. See page 18 for the list of features that each option provides. To purchase a license, perform the following steps:

1. Click **Purchase License**.
   The appropriate Data Translation web page is displayed.
2. Purchase the license key for the analysis package that you want.
3. Click **Enter License** from the License Info dialog.
   The Enter License Key dialog is displayed.
4. If desired, you can copy the license key to the clipboard.
5. Enter the license key or paste the license key from the clipboard using the **Paste from Clipboard** button, and click **Apply License and Continue**.
   If the application was running when you entered the license key, the application will restart. Otherwise, the application continues with either the Device Selection dialog, if an existing configuration file is not found, or the main interface.

Note: If, after the trial period has expired, you try to load a configuration file that was saved with the FFT Analysis or Advanced FFT Analysis option, the configuration will not load and an error message will be generated.

If you have a license file for the FFT Analysis option but try to load a configuration file that was saved with the Advanced FFT Analysis option, the configuration will load but any FRF channels or other non-supported parameters will be missing.

Resolution Requirements

A minimum vertical screen resolution of 768 is required for proper operation of QuickDAQ.

In addition, it is recommended that you set the default font size to 100%. To do this, select **Control Panel**, **Display**, and then choose **Smaller (100%)**.

2

Quick Start

Installing the Software

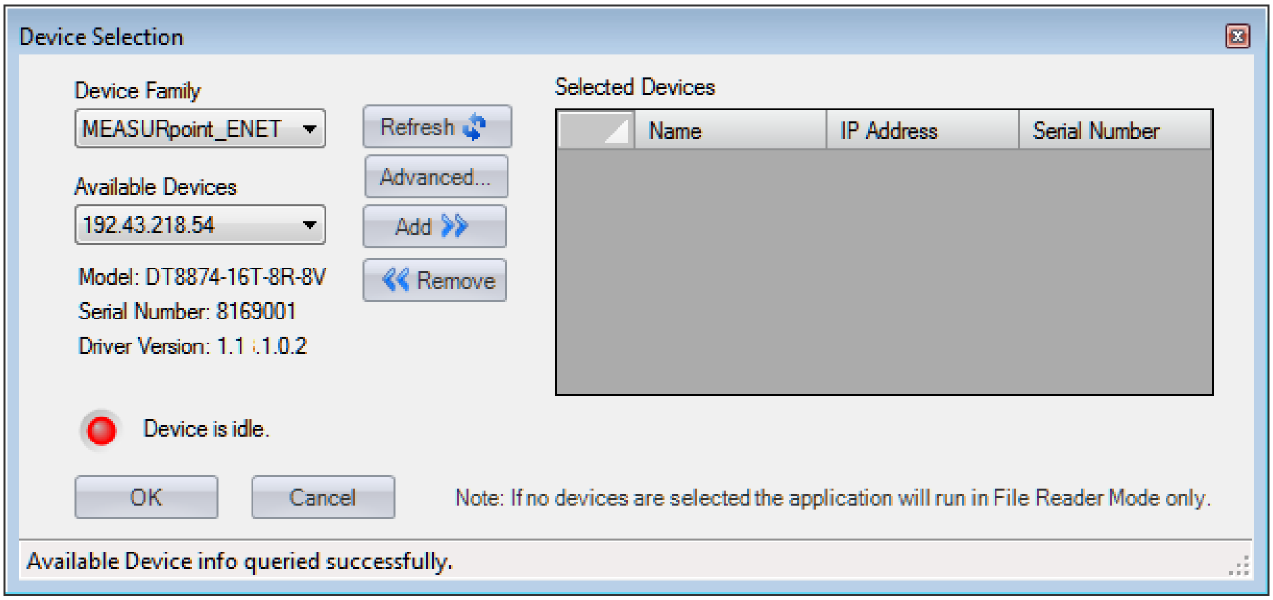
The QuickDAQ software is installed automatically when you install the device driver for your hardware.

No other software installation is required.

Getting Started

To get started with QuickDAQ, follow these steps:

1. Connect a supported device to the network (Ethernet/LXI devices) or to the USB port of your computer (for USB devices), and connect your sensors to the device.
2. Start QuickDAQ.
   The Device Selection window appears.



Note: If you ran QuickDAQ before and the devices that were used are still present and are not acquiring data, the Device Selection window is bypassed. If the devices support password protection, you must supply the correct password before the application launches. Refer to the user's manual for your device to determine the default password.

For devices that do not support password protection and that are currently acquiring data, a warning dialog box appears notifying you that another application is using the device. You cannot continue with QuickDAQ until the device is not acquiring data.

For devices that support password protection and that are currently acquiring data, the software alerts you that the device is acquiring data and gives you the option to abort acquisition and continue, or to cancel.

3. By default, the application "discovers" all devices that are available for the specified device family and displays their IP address (for Ethernet devices) or module name (for USB devices) in the drop-down list. If you want to refresh this list to determine if other devices are available, click **Refresh**.

Notes: The Device Family dropdown shows only the interfaces that are installed on your system.

OpenLayersDevices is the Device Family name for all Open Layers modules.

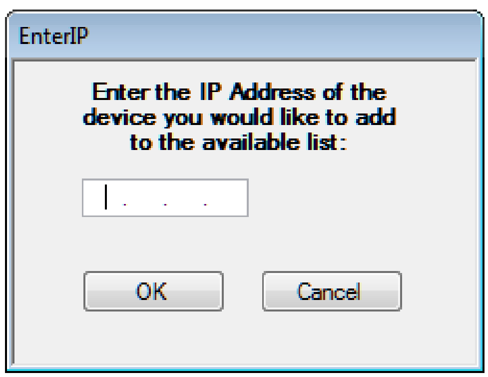
MEASURpoint-USB is the Device Family name for the USB versions of MEASURpoint, TEMPpoint, and VOLTpoint instruments.

MEASURpoint-ENET is the Device Family name for all Ethernet versions of MEASURpoint, TEMPpoint, and VOLTpoint instruments.

DT8824 is the Device Family name for all DT8824 Series instrument modules.

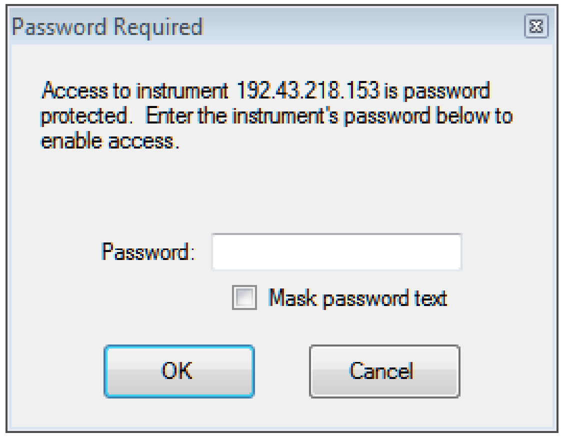
DT8837 is the Device Family name for the DT8837 instrument module.

4. Select the IP address (for Ethernet devices) or module name (for USB devices) for the device that you want to use from the list of Available Devices, and click **Add**.
   Information about the device, including the model number, serial number, firmware version, driver version, and scanning status is displayed.
5. For Ethernet devices, if your supported device is not included in the list of available devices, but you want to manually connect to it, do the following:
   a. Click **Advanced**.

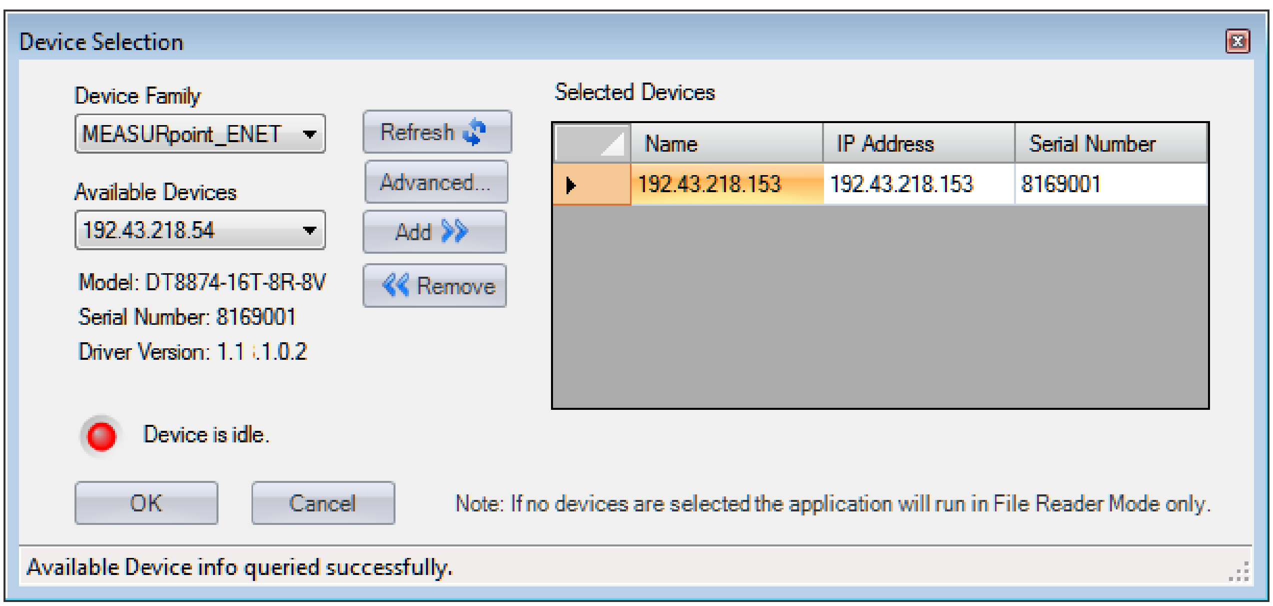


   b. Enter the IP address of the device that you want to connect to, and click **OK**.
      Information about the device, including the model number, serial number, firmware version, driver version, and scanning status is displayed.
   c. From the Device Selection window, click **Add**.

d. If the device supports password protection, the Password Required dialog box appears:



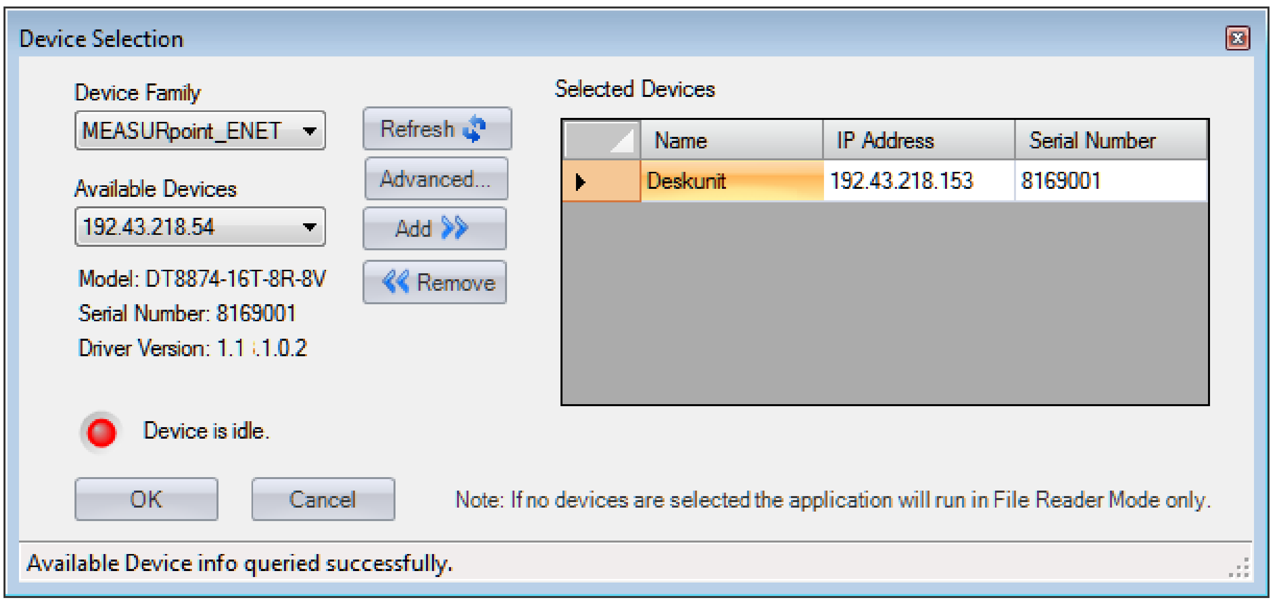
e. If prompted, enter the password for the device. Refer to the user’s manual for your device to determine the default password. By default, the **Mask password text** box is checked so that you cannot read the password as it is being typed. If desired, you can uncheck this box, if desired. (Note that the software does not allow you to continue if you do not enter the appropriate password for the device.)



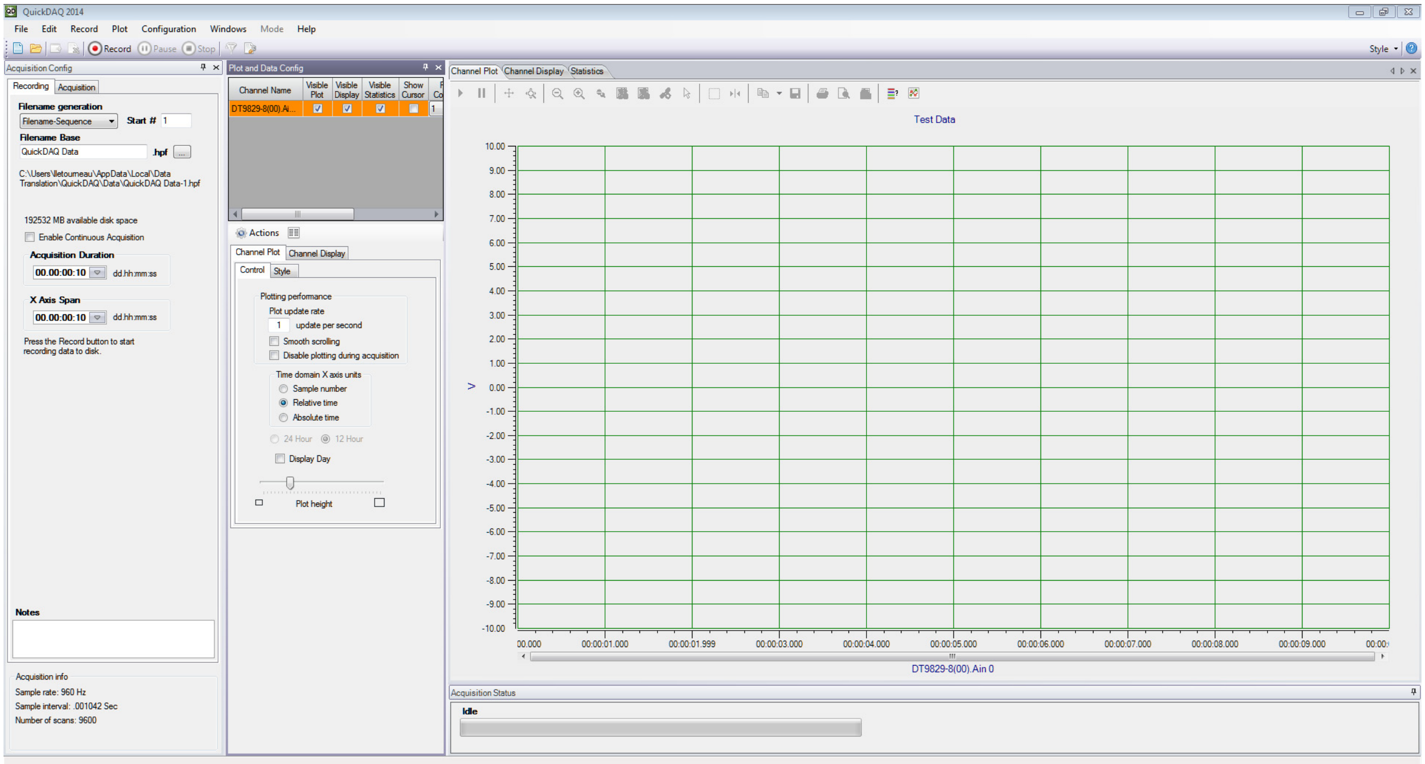
Note: If the device that you want to add does not support password protection and the device is currently acquiring data, a warning dialog box appears notifying you that another application is using the device. You cannot continue with QuickDAQ until the device is not acquiring data.

If the device that you want to add supports password protection and the device is currently acquiring data, the software alerts you that the device is acquiring data and gives you the option to abort acquisition and continue, or to cancel.

6. (Recommended) If you want to rename your device, do the following:
   a. Click the Row Selector button for the device.
   b. Click the IP address or module name in the **Name** column to highlight it and enter a meaningful name to represent each available device.



7. (Optional) If you want to remove a device from list of selected devices, click the Row Selector button for the device, and then click **Remove**.
8. Once you have added all the devices that you want to use with the application, click **OK**. *The latest state is saved and used when the application is next run, and the QuickDAQ interface is displayed.*



9. Configure the input channels of the device. (See page 84.)
10. If you are using the FFT Analysis option or Advanced FFT Analysis option, choose either the Data Logger or FFT Analyzer interface mode. (See page 47).
The Base package provides the Data Logger interface mode only.
11. If you are using the Data Logger interface mode, perform these steps:
    a. Configure the recording settings of the device using the Recording tab of the Acquisition Config window. (See page 119.)
    b. Configure the frequency settings of the device using the Freq tab of the Acquisition Config window. (See page 125.)
    c. If you are using analog output channels, configure the parameters of the Output tab in the Acquisition Config window. (See page 138.)
    d. If you are using digital filters, configure the digital filter settings. (See page 162.)
    e. Set up the parameters of the Channel Plot, Channel Display, and/or Statistics windows. (See Chapter 5 starting on page 167.)
12. If you are using the FFT Analyzer interface mode, perform these steps:
    a. Configure the recording settings of the device using the Recording tab of the Acquisition Config window. (See page 122.)
    b. Configure the frequency settings of the device using the Freq tab of the Acquisition Config window. (See page 128.)
    c. Configure the trigger parameters of the device using the Trigger tab of the Acquisition Config window. (See page 131.)

d. Configure the window functions for FFT channels using the Window tab of the Acquisition Config window. (See page 135.)

e. If you are using analog output channels, configure the parameters of the Output tab of the Acquisition Config window. (See page 138.)

f. If you are using digital filters, configure the digital filter settings. (See page 162.)

g. Set up the parameters of the Channel Plot, Channel Display, and/or Statistics windows. (See Chapter 6 starting on page 189.)

13. To start acquisition, click the **Record** menu and then click **Start**, click the **Record** toolbar button (

), or press the **F5** key.
The results are displayed in the Channel Plot window and in the Channel Display window.

3

Understanding the Interface

Overview of the Interface

The Base package of QuickDAQ provides a Data Logger interface, shown in Figure 1.

If you using the FFT Analysis option or Advanced FFT Analysis option, you can choose either the Data Logger or FFT Analyzer interface using the **Mode** menu. Figure 2 shows the FFT Analyzer interface of QuickDAQ.

Both interface modes include menus, toolbar buttons, a configuration area, which includes the Acquisition Config window and the Plot and Data Config window, a display area, which includes the Channel Display window, Channel Plot window, and Statistics window, and the Acquisition Status window.

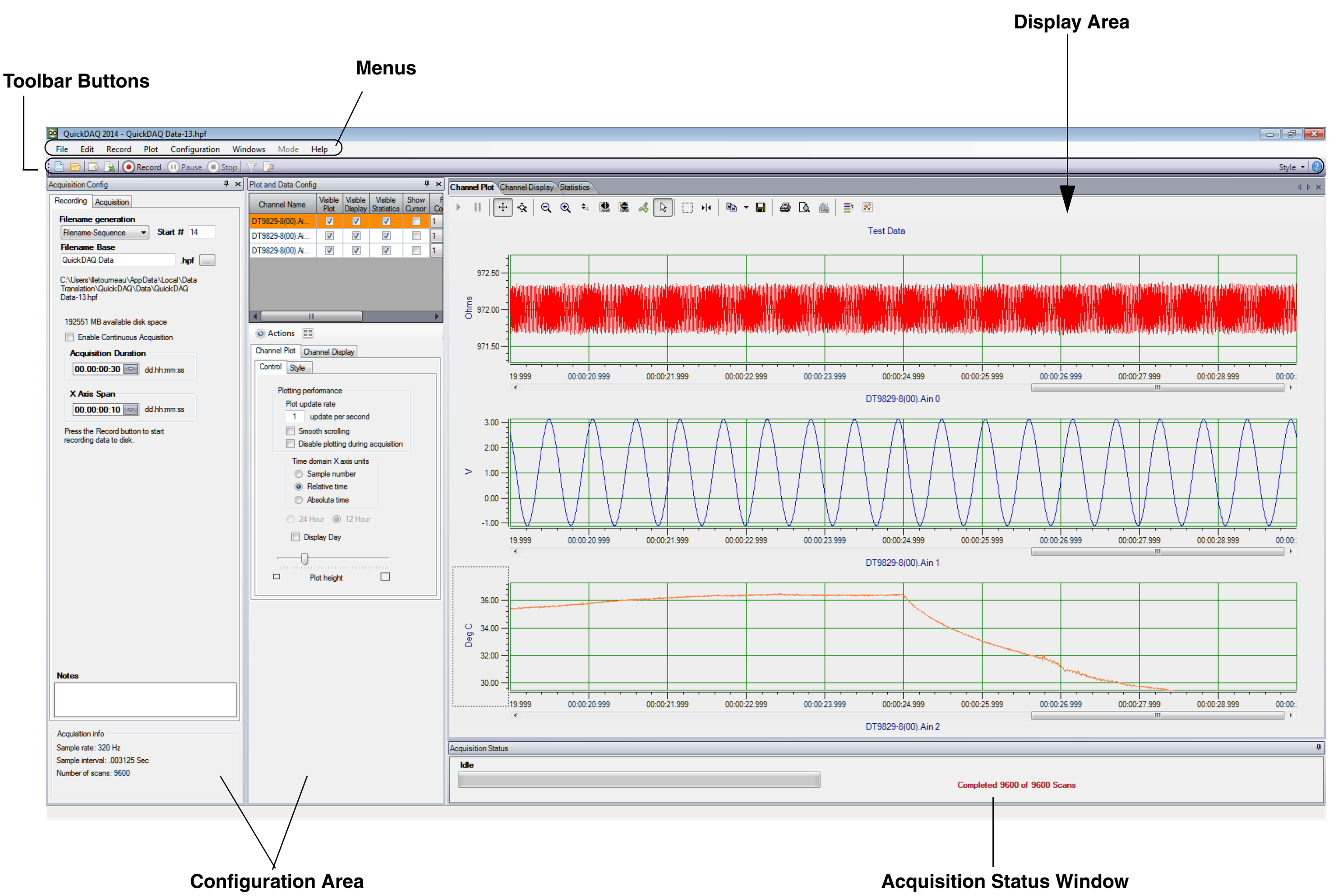


Figure 1: Data Logger Interface of QuickDAQ

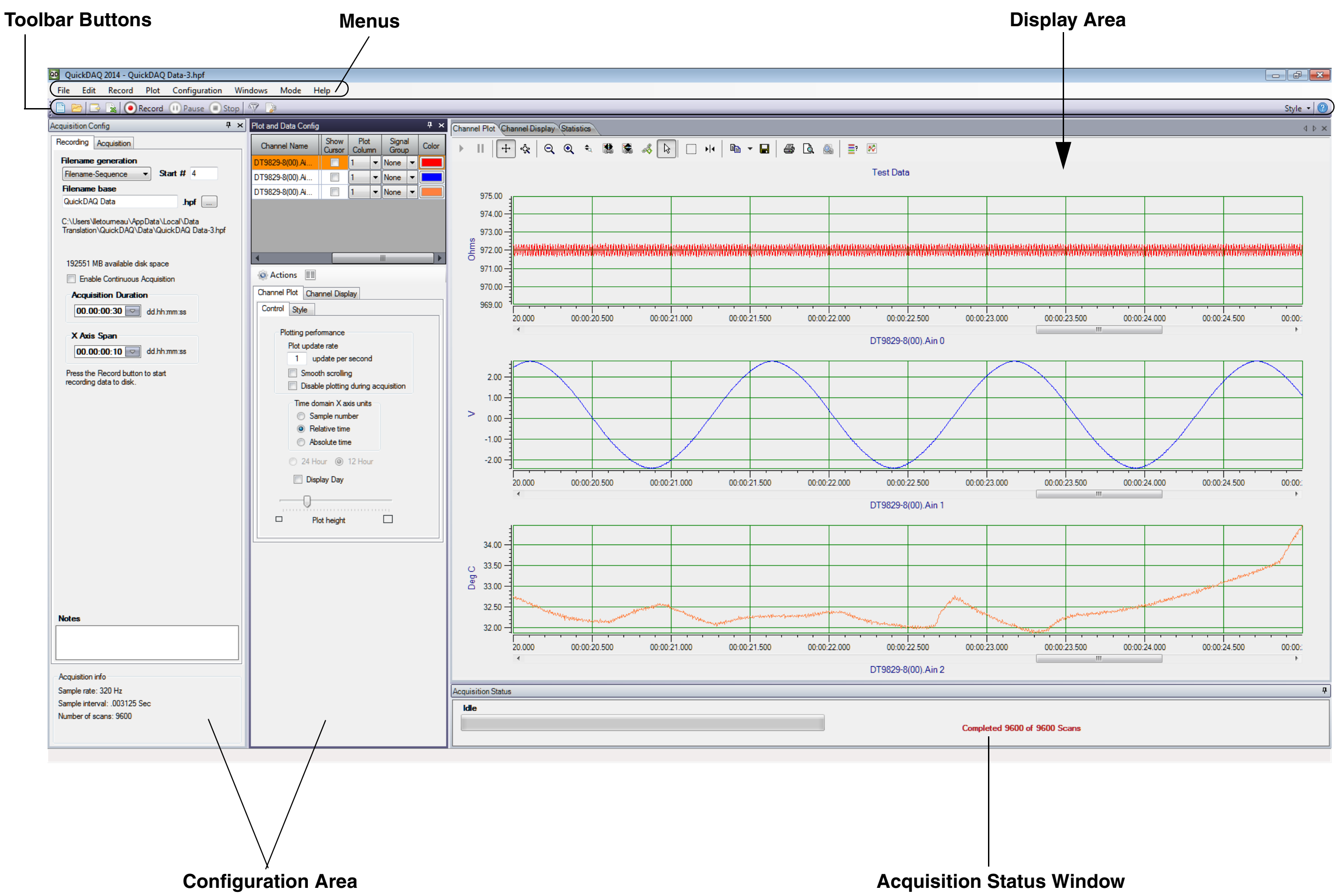


Figure 2: FFT Analyzer Interface of QuickDAQ

This chapter describes the features of both interface modes of QuickDAQ.

Note: You can customize the look of the interface, as desired. However, if you want to revert to the default layout, select **Windows** from the list of menus and select **Reset Window Layout to Default Layout**.

Menus

QuickDAQ provides the following menus: **File**, **Edit**, **Record**, **Plot**, **Configuration**, **Windows**, **Mode**, and **Help**.

File Menu

This section describes the options provided in the File menu.

New Hpf Data File

When you click the **New Hpf Data File** option of the File menu or click the **New HPF Data File** toolbar button  , you can create a new (empty) data file for use with QuickDAQ. You can name and save the data file, as desired. The format of this file is High Performance binary (.hpf).

By default, the data file is stored in the following location:

For Windows Vista, Windows 7, or Windows 8:
C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

For Windows XP:
C:\Documents and Settings\<user name>\Local Settings\Application Data\
Data Translation\QuickDAQ\Data

Note: To see the AppData folder, ensure that you set your view options to show hidden files. For example, in Windows 7, select **Control Panel** -> **Appearance and Personalization** -> **Folder Options**, select the **View** tab, and then select the **Show hidden files, folders, and drives** checkbox. Consult the documentation for your operating system for more information.

Open Hpf Data File

When you click the **Open Hpf Data File** option of the File menu or click the **Open HPF Data File** toolbar button  , you can open an .hpf data file that was created using QuickDAQ.

A sample data file, called QuickDAQ Data.hpf, is provided, by default, in the following location:

For Windows Vista, Windows 7, or Windows 8:
C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

For Windows XP:
C:\Documents and Settings\<user name>\Local Settings\Application Data\
Data Translation\QuickDAQ\Data

Close Hpf Data File

The **Close Hpf Data File** option closes an opened data file (.hpf), restoring the previous state of QuickDAQ.

Export Data

When you click the **Export Data** menu option from the File menu, or click the **Export Data** toolbar button  , the File Export dialog appears. This dialog is different for each interface mode.

File Export Dialog for the Data Logger Interface

The File Export dialog for the Data Logger interface is shown in Figure 3.

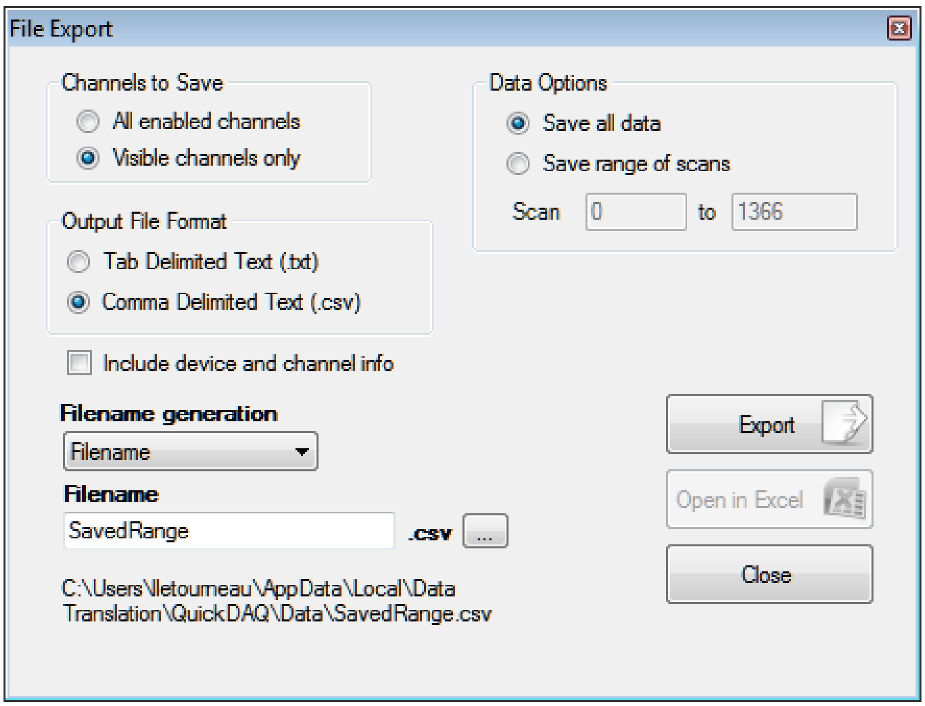


Figure 3: File Export Dialog for the Data Logger Interface

Configure this dialog as follows:

1. Under the **Channels to Save** area of the dialog, you can select whether to save the data from all channels or only those that are visible in the Channel Plot window or Channel Display window.

   To save data from all channels, select **All channels**.

   To save data from only the visible channels, select **Visible channels only**.

2. Under the **Data Options** area of the dialog, select which data to save.

   You can save all scans by selecting **Save all data**, or save a subset of scans by selecting **Save range of scans** and entering the first scan and the last scan to save.

3. Under the **Output File Format** area of the dialog, select the file format to use when exporting the data:

   – **Tab Delimited Text (.txt) –** This file format is used by a number of applications, including Microsoft Excel.

   – **Comma Delimited Text (.csv) –** This file format is used by a number of applications, including Microsoft Excel.

Note: In tab delimited text (.txt) files, the time column shows the relative time in seconds of when the sample was acquired.

In comma delimited text (.csv) files, the time column shows the Excel time, where the number represents the number of days since 1900-Jan-0, plus a fractional portion of a 24 hour day. If you chose to display data as absolute time, then the first sample represents the start time of the acquisition. If you chose to display data in relative time, then the first sample represents time 0. Within Excel, you can format this column to use a more meaningful time option if you wish by right-clicking the selected column, choosing **Format Cells**, choosing **Time**, and selecting the option you want to use.

   If you wish, you can export information about the device and channels that correspond to the saved data, by selecting the **Include device and channel info** checkbox.

4. Under **Filename generation**, specify how to create the name of the file. You can choose one of the following options:

   – Filename – The specified filename is created when you click the **Export** button. Each time you click the **Export** button, you will be prompted to overwrite the file. SavedRange.uff is an example of a filename that was created using the Filename option.

   – Filename-Sequence – The specified filename is created and a starting number that you define is appended to the filename when you click the **Export** button. Each time you click the **Export** button, the number that is appended to the filename is incremented by one. SavedRange.txt-1 is an example of a filename that was created using the Filename-Sequence option.

   – Filename-DateTime – The specified filename is created and the current date and time is appended to the filename when you click the **Export** button. Each time you click the **Export** button, the filename with the current date and time is created. SavedRange-2012-06-25_02-39-59-PM.txt is an example of a filename that was created using the Filename-DateTime option.

5. In the **Filename** text box, enter or browse to the name of the file that will contain the exported data.
   The path showing where the file is located is shown.

   The default location for saving the data files is as follows:

   <u>*For Windows Vista, Windows 7, or Windows 8:*</u>
   C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

   <u>*For Windows XP:*</u>
   C:\Documents and Settings\<user name>\Local Settings\Application Data\Data Translation\QuickDAQ\Data

6. Click the **Export** button to export the data to the file you specified.
7. Once the file is created, you can open the file in Microsoft Excel by clicking the **Open in Excel** button.
8. When you are finished exporting the data, close this dialog box by clicking the **Done** button.

File Export Dialog for the FFT Analyzer Interface

The File Export dialog for the FFT Analyzer interface is shown in Figure 4.

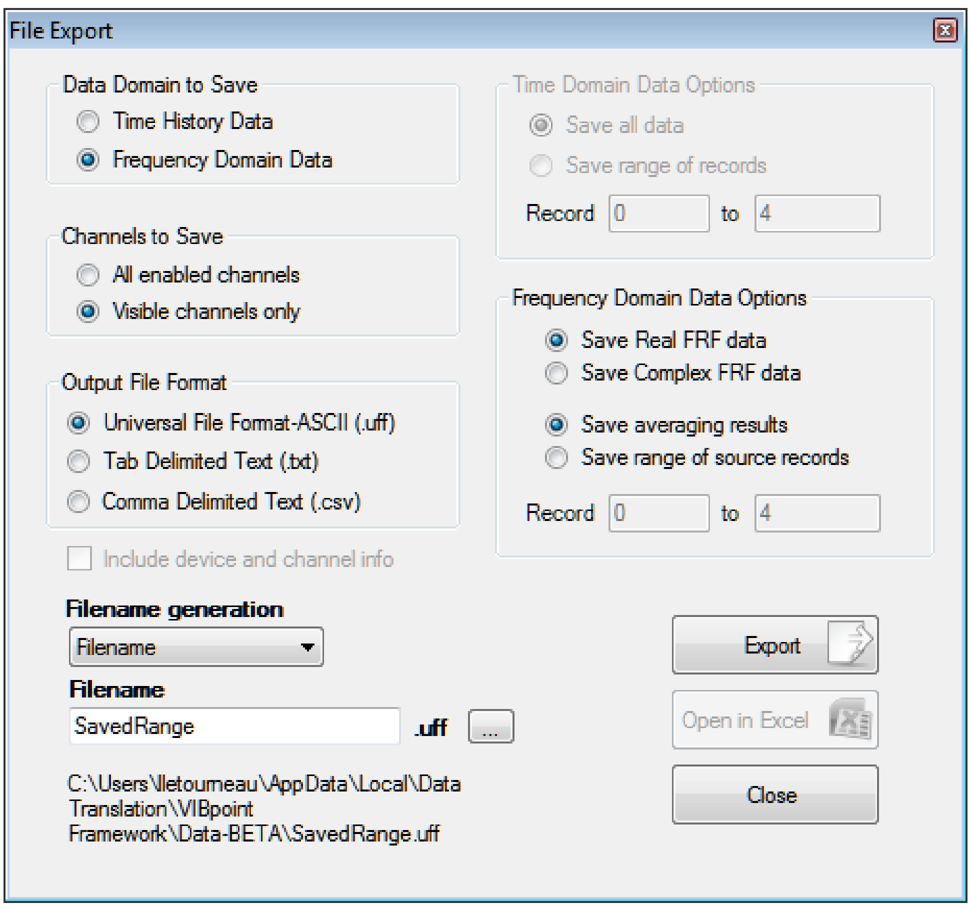


Figure 4: File Export for the FFT Analyzer Interface

Configure this dialog as follows:

1. Under the **Data Domain to Save** area of the dialog, choose the type of data to save.

   For time domain data, select **Time History Data**.

   For frequency domain data, select **Frequency Domain Data**.

2. If you selected **Time History Data**, select which records to save under the **Time Domain Data Options** area of the dialog.

   You can save all the records by selecting **Save all data**, or save a subset of records by selecting **Save range of records** and entering the first record and the last record to save.

3. If you select **Frequency Domain Data**, you can select which data to save under the **Frequency Domain Data Options** area of the dialog.

   You can save the average of all records by selecting **Save averaging results**, or save a subset of source records by selecting **Save range of source records** and entering the first record and the last record to save.

   If you are using the Advanced FFT Analysis option and are using FRF channels, you also choose whether to save real data by selecting **Save Real FRF data** or complex data by selecting **Save Complex FRF data**.

4. Under the **Channels to Save** area of the dialog, you can select whether to save the data from all channels or only those that are visible in the Channel Plot window.

   To save data from all channels, select **All channels**.

   To save data from only the visible channels in the Channel Plot window, select **Visible channels only**.

5. Under the **Output File Format** area of the dialog, select the file format to use when exporting the data:

   – **Universal File Format-ASCII (.uff) –** Available only in the Advanced FFT Analysis option of QuickDAQ, this file format is used by a number of applications, including ME'scope™ from Vibrant Technology.

   – **Tab Delimited Text (.txt) –** This file format is used by a number of applications, including Microsoft Excel.

     If this file format is selected, you can also choose to export information about the device and channels that correspond to the saved data, by selecting the **Include device and channel info** checkbox.

   – **Comma Delimited Text (.csv) –** This file format is used by a number of applications, including Microsoft Excel.

     If this file format is selected, you can also choose to export information about the device and channels that correspond to the saved data, by selecting the **Include device and channel info** checkbox.

Note: In tab delimited text (.txt) files, the time column shows the relative time in seconds of when the sample was acquired.

In comma delimited text (.csv) files, the time column shows the Excel time, where the number represents the number of days since 1900-Jan-0, plus a fractional portion of a 24 hour day. If you chose to display data as absolute time, then the first sample represents the start time of the acquisition. If you chose to display data in relative time, then the first sample represents time 0. Within Excel, you can format this column to use a more meaningful time option if you wish by right-clicking the selected column, choosing **Format Cells**, choosing **Time**, and selecting the option you want to use.

---

6. Under **Filename generation**, specify how to create the name of the file. You can choose one of the following options:
   - Filename – The specified filename is created when you click the **Export** button. Each time you click the **Export** button, you will be prompted to overwrite the file. SavedRange.uff is an example of a filename that was created using the Filename option.
   - Filename-Sequence – The specified filename is created and a starting number that you define is appended to the filename when you click the **Export** button. Each time you click the **Export** button, the number that is appended to the filename is incremented by one. SavedRange.uff-1 is an example of a filename that was created using the Filename-Sequence option.
   - Filename-DateTime – The specified filename is created and the current date and time is appended to the filename when you click the **Export** button. Each time you click the **Export** button, the filename with the current date and time is created. SavedRange-2012-06-25_02-39-59-PM.uff is an example of a filename that was created using the Filename-DateTime option.
7. In the **Filename** text box, enter or browse to the name of the file that will contain the exported data.
   The path showing where the file is located is shown.

   The default location for saving the data files is as follows:

   <u>*For Windows Vista, Windows 7, or Windows 8:*</u>
   C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

   <u>*For Windows XP:*</u>
   C:\Documents and Settings\<user name>\Local Settings\Application Data\
   Data Translation\QuickDAQ\Data
8. Click the **Export** button to export the data to the file you specified.
9. Once the file is created, you can open the file in Microsoft Excel by clicking the **Open in Excel** button.
10. When you are finished exporting the data, close this dialog box by clicking the **Done** button.

Save Current Plot Image

The **Save Current Plot Image** option saves the current plot image from the Channel Plot window to a file.

To save the image to a file, specify the name of the file and choose one of the extensions to save the file as a graphic.

The default location for saving the current plot image files is as follows:

For Windows Vista, Windows 7, or Windows 8:
C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

For Windows XP:
C:\Documents and Settings\<user name>\Local Settings\Application Data\
Data Translation\QuickDAQ\Data

Open Current Data in Excel

The function of the **Open Current Data in Excel** menu option depends on the interface mode, described below.

Note: Before using this option, ensure that Microsoft Excel is installed on your computer. In addition, you must stop acquisition before trying to use this option, or the application cannot open the file.

Open Current Data in Excel Using the Data Logger Interface

If you choose the **Open Current Data in Excel** menu option from the File menu or click the **Open Current Data in Excel** toolbar button , the application opens a tab-separated (tmp.tvs) file in Microsoft Excel that contains all the time domain data for the visible channels in the Channel Plot window.

You can then perform further analysis of your data using Excel or other utilities.

Open Current Data in Excel Using the FFT Analyzer Interface

If you are using the FFT Analyzer Interface and choose the **Open Current Data in Excel** menu option from the File menu or click the **Open Current Data in Excel** toolbar button , you can specify either **Time History Data** or **Frequency Domain Data**.

If you select **Time History Data**, the application opens a tab-separated (tmp.tvs) file in Microsoft Excel that contains all the time domain data for the visible channels in the Channel Plot window.

If you select **Frequency Domain Data**, the application opens a tab-separated (tmp.tvs) file in Microsoft Excel that contains the average frequency domain data for the visible channels in the Channel Plot window.

You can then perform further analysis of your data using Excel or other utilities.

Load Configuration

The **Load Configuration** option opens an .xml file of stored configuration settings.

Save Configuration

The **Save Configuration** option saves the current application configuration to a user-specified filename with the .xml extension.

The default location for saving the configuration file is as follows:

For Windows Vista, Windows 7, or Windows 8:
C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Config.

For Windows XP:
C:\Documents and Settings\<user name>\Local Settings\Application Data\
Data Translation\QuickDAQ\Config

Open Hpf Data Folder

The **Open Hpf Data Folder** option opens Windows Explorer in the current folder that is used to store .hpf data files. This provides a quick way to browse your data files.

The default location for saving the data files is as follows:

For Windows Vista, Windows 7, or Windows 8:
C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

For Windows XP:
C:\Documents and Settings\<user name>\Local Settings\Application Data\
Data Translation\QuickDAQ\Data

Open Export Data Folder

The **Open Export Data Folder** option opens Windows Explorer in the current folder that is used to store exported data files. This provides a quick way to browse your exported files.

The default location for saving exported files is as follows:

For Windows Vista, Windows 7, or Windows 8:
C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

For Windows XP:
C:\Documents and Settings\<user name>\Local Settings\Application Data\
Data Translation\QuickDAQ\Data

Page Setup

The **Page Setup** option allows you to configure the page before you print it. Options are available for configuring the size, orientation, and margins of the page.

Print Preview

The **Print Preview** option allows you to preview the image from the Channel Plot window before you print it.

Print

The **Print** option prints the picture from the Channel Plot window to a printer.

Exit

The **Exit** option closes QuickDAQ, and saves the current device configuration in the QuickDAQConfiguration.xml file in the default configuration folder. You can also close the application by clicking the  button in the top, right-hand side of the application.

Notes: A backup configuration file (QuickDAQConfiguration.xml.bak) is saved in the default configuration folder when QuickDAQ is exited. If, when starting QuickDAQ, an error occurs and the configuration file cannot be restored, the backup file is used to restore the configuration.

If the device is acquiring data through another application, a warning dialog box appears notifying you that you cannot save the current configuration if you close the application.

Exit without saving current configuration

The **Exit without saving current configuration** option closes QuickDAQ, but does not save the current device configuration.

Edit Menu

The Edit menu has the following menu options:

- **Copy plot data to clipboard** – Copies the data from the Channel Plot window to the Windows clipboard.
- **Copy plot image to clipboard** – Copies the image from the Channel Plot window to the Windows clipboard.

Record Menu

The Record menu has the following menu options:

- **Start** – Starts data acquisition on the device and logs the data to a High Performance binary file (.hpf). This is equivalent to clicking the **Record** button ( ) on the toolbar or pressing the **F5** key. You can specify the name of the data file by opening a new Hpf file, described on page 36, or by using the **Recording** tab, described on page 119.

Note: You can use the scrollbar on the x-axis to pause tracking and scroll back through the data to the beginning of acquisition. Refer to page 70 and page 71 for more information on pausing and resuming tracking.

For long duration acquisitions, ensure that your computer's power options are set to never hibernate; refer to page 384 for more information.

- **Pause** – Pauses recording. This is equivalent to clicking the **Pause** button ( ) on the toolbar or pressing the spacebar. To resume recording, click the **Pause** menu option, click the **Pause** button, or press the spacebar again.
- **Stop** – Stops the application from receiving data from the device. This is equivalent to clicking the Stop button ( ) on the main menu or holding down the **Shift** key and pressing the **F5** key. You must stop acquisition to make configuration changes.

Plot Menu

The Plot menu has the following menu options:

- **Plot Area Mouse Mode** – You can select one of the following controls for the Channel Plot window:
  - **Select** – Allows you to select items in the view area to zoom or to scroll. This is equivalent to using the Select control ( ) of the Channel Plot window, or holding down the **Ctrl** and **Alt** keys while pressing the **V** key.
  - **Zoom** – Allows you to select a region in the view to magnify (zoom). This is equivalent to using the Zoom Box ( ) control of the Channel Plot window, or holding down the **Ctrl** and **Alt** keys while pressing the **Z** key.

    Once in zoom mode, you can hold down the left mouse button, dragging the mouse to select the region that you want to magnify, and release the mouse button to magnify the area. The zoom action affects both axes.
  - **Cursor** – Allows you to move the position of a data-cursor on the curve. This is equivalent to using the Data-Cursor ( ) control of the Channel Plot window, or holding down the **Ctrl** and **Alt** keys while pressing the **C** key.

    Once in Data-Cursor mode, hold down the left mouse button and drag the mouse to move the data cursor. When the **Data-Cursor** is active, right-click the mouse to change the cursor style, line style, and/or color of the cursor; refer to page 74 for more information.

- **Axis Mouse Mode –** You can select one of the following controls for the axis:
  - **Scroll –** Allows you to scroll through the axes. This is equivalent to using the Axes Scroll () control of the Channel Plot window.

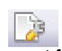
    Once in this mode, hold down the left mouse button and drag the mouse over the axis (up and down for the y-axis, or right and left for the x-axis) to scroll through the axes.

    If you want to change the magnification (zoom) of the axes while this mode is selected, hold down the Ctrl button while holding down the left mouse button, and drag the mouse over the axis (up and down for the y-axis, or right and left for the x-axis).
  - **Zoom –** Allows you to change the magnification (the zoom) of the axes. This is equivalent to using the Axes Zoom () control of the Channel Plot window.

    Once in this mode, you can change the magnification (the zoom) of the axes by holding down the left mouse button and dragging the mouse over the axis (up and down for the y-axis, or right and left for the x-axis).

Configuration Menu

The Configuration menu has the following menu options:

- **Input Channel Configuration –** When you click this menu option or click the **Input Channel Configuration** toolbar button (), the **Configure Devices** dialog box appears. Using this dialog box, you can specify which input channels are sampled, and configure the sensor type, engineering units, number of millivolts per engineering unit, and the test point number for each channel.

  Refer to page 84 for more information about configuring channels.
- **Digital Filter Configuration –** This menu option is enabled only in the QuickDAQ FFT Analysis and Advanced FFT Analysis packages.

  When you click this menu option or click the **Digital Filter Configuration** toolbar button (), the **Digital Filter Configuration** dialog box appears. Using this dialog box, you can specify the type of IIR (infinite impulse response) filter that you want to apply for each channel and the parameters for the each filter, including the filter category (low pass, high pass, band pass, band stop), filter order, cutoff frequencies, and dB ripple.

  Refer to page 162 for more information about configuring digital filters.
- **Device Selection –** Opens the Device Selection window, allowing you to select the devices for use with QuickDAQ in acquisition mode. Refer to page 27 for more information about using the Device Selection window.

Windows Menu

The Windows menu provides the following menu options:

- **Acquisition Config** window – Check this window option to show the Acquisition Config window. If this window option is unchecked, the Acquisition Config window is hidden. The Acquisition Config window allows you to configure the acquisition parameters of the data acquisition devices. Refer to Chapter 4 **starting on page 83** for more information on this window.

- **Plot and Data Config** window – Check this window option to show the Plot and Data Config window. If this window option is unchecked, the Plot and Data Config window is hidden. The Plot and Data Config window allows you to configure the appearance of the display. Refer to Chapter 5 **starting on page 167** for more information on this window.
- **Channel Plot** window – Check this window option to show the Channel Plot window. If this window option is unchecked, the Channel Plot window is hidden. This window allows you view the data from your device in a plot view. Refer to page 66 and page 169 for more information on this window.
- **Channel Display** window – Check this window option to show the Channel Display window. If this window option is unchecked, the Channel Display window is hidden. This window allows you view the data from your device in a digital display that resembles the front panel of the device. Refer to page 78 and page 169 for more information on this window.
- **Channel Statistics** window – Check this window option to show the Channel Statistics window. If this window option is unchecked, the Channel Statistics window is hidden. This window allows you view statistics about the data that you acquired. Refer to page 79 for more information on this window.
- **Reset Window Layout to Last Saved** – Allows you to change the window layout to the last saved configuration.
- **Reset Window Layout to Default Layout** – Allows you to change the window layout to the default state.

Mode Menu

The **Mode** menu is disabled (grayed out) if you are using the Base package of QuickDAQ.

If you using the FFT Analysis option or Advanced FFT Analysis option of QuickDAQ, the **Mode** menu provides the following options:

- **Data Logger** – This option selects the Data Logger interface. This is the interfaced provided in the Base package of QuickDAQ. Use this interface if you want to acquire data, display time-domain data, and log time-domain data to disk, and do not want to perform FFT analysis on the data using this application.
- **FFT Analyzer** – This option selects the FFT Analyzer interface. Use this interface if you want to acquire data, perform FFT analysis on the data, display the time-domain and/or frequency-domain data, and log the data to disk.

Note: If you are using the FFT Analysis option or Advanced FFT Analysis option of QuickDAQ, you can toggle between the two interfaces. The application remembers the configuration between toggles. However, when the application is closed, only the active interface mode is saved.

If you want the configuration of both interface modes to be saved, you must manually save the configuration for the interface that is not active before the application is closed.

Help Menu

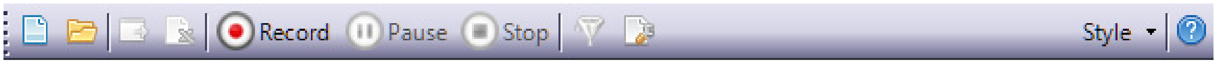
The Help menu provides the following options:

- **User Manual** – Opens this manual, which describes the functionality of QuickDAQ.
- **License Info** – Opens the License Info dialog, described in page 21, which allows you to purchase and enter a license key for the FFT Analysis or Advanced FFT Analysis option of QuickDAQ.
- **About** – Provides version information about the QuickDAQ application.

You can also open this manual using the **Help** toolbar button ( ).

Toolbar Buttons

The following buttons are provided on the toolbar:



-  – **New HPF Data File** – Creates a new (empty) data file for use with QuickDAQ. This is equivalent to using the **New Hpf Data file** menu option under the File menu, described on page 36.

  You can name and save the data file, as desired. The format of this file is High Performance binary (.hpf).

  By default, the data file is stored in the following location:

  For Windows Vista, Windows 7, or Windows 8:
  C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

  For Windows XP:
  C:\Documents and Settings\<user name>\Local Settings\Application Data\Data Translation\QuickDAQ\Data

-  – **Open HPF Data File** – Opens an .hpf data file that was created using QuickDAQ. This is equivalent to using the **Open Hpf Data file** menu option under the File menu, described on page 36.

  A sample data file, called QuickDAQ Data.hpf, is provided, by default, in the following location:

  For Windows Vista, Windows 7, or Windows 8:
  C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

  For Windows XP:
  C:\Documents and Settings\<user name>\Local Settings\Application Data\Data Translation\QuickDAQ\Data

-  – **Export Data** – Opens the Export Data dialog box. This is equivalent to selecting the **Export Data** menu option of the File menu, described on page 37.
-  – **Open Current Data in Excel** – After you stop recording, you can click this button to open a tab-separated (tmp.tvs) file in Microsoft Excel that contains all the time domain data for the visible channels in the Channel Plot window.

  You can then perform further analysis of your data using Excel or other utilities.

  This is equivalent to selecting the **Open Current Data in Excel** menu option of the File menu, described on page 42.

Note: Before using this option, ensure that Microsoft Excel is installed on your computer.

- 

 – **Record** – Starts data acquisition on the device and logs the data to a High Performance binary (.hpf) file. This is equivalent to selecting the **Start** menu option of the Record menu or pressing the **F5** key. You can specify the name of the data file using the **Recording** tab, described on page 119.

Note: You can use the scrollbar on the x-axis to pause tracking and scroll back through the data to the beginning of acquisition. Refer to page 70 and page 71 for more information on pausing and resuming tracking.

For long duration acquisitions, ensure that your computer's power options are set to never hibernate; refer to page 384 for more information.

- – **Pause** – Pauses recording. This is equivalent to selecting the **Pause** menu option of the Record menu or pressing the spacebar. To resume recording, click the **Pause** button, click the **Pause** menu option, or press the spacebar again.
- – **Stop** – Stops acquisition. This is equivalent to the **Stop** menu option under the **Record** menu or holding down the **Shift** key and pressing the **F5** key. You must stop acquisition to make configuration changes.
- – **Digital Filter Configuration** – Opens the Digital Filter Configuration dialog. This is equivalent to using the **Digital Filter Configuration** menu option of the Configuration menu, described on page 46.

Note: This toolbar button is enabled only in the QuickDAQ FFT Analysis and Advanced FFT Analysis packages.

- – **Input Channel Configuration** – Opens the Configure Devices dialog. This is equivalent to using the **Input Channel Configuration** menu option of the Configuration menu, described on page 46.
- – **Style** – Allows you to configure the theme colors used by the application. You can select one of the following theme colors:
  - Office 2010 Blue
  - Office 2010 Silver
  - Office 2010 Black
  - Visual Studio 2010
  - Windows 7
  - Office 2007 Blue
  - Office 2007 Black
  - Office 2007 Silver
  - Vista Glass
  - Custom Scheme – Allows you to select your own color scheme to use for the application.
- – **Help** – Opens the this manual to describe QuickDAQ. This is equivalent to clicking the **User's Manual** menu option of the Help menu, described on page 48.

Window Positioning

The configuration area of the interface contains the Acquisition Config and the Plot and Data Config windows. The display area of the interface contains the Channel Plot, Channel Display, and Channel Statistics windows. The Acquisition Status window is located under the Channel Plot, Channel Display, and Statistics windows.

This section describes how to move, resize, dock, and hide windows in the interface. The remaining sections of this document describe each window in more detail.

Moving, Resizing, and Docking Windows

Note: The Acquisition Status window is docked automatically. You cannot undock the Acquisition Status window.

You can move a window anywhere on the desktop by clicking the title bar or tab of the window and, while holding down the mouse button, dragging the window to the location you want. When you release the mouse button, this "floating" window remains in the location that you selected.

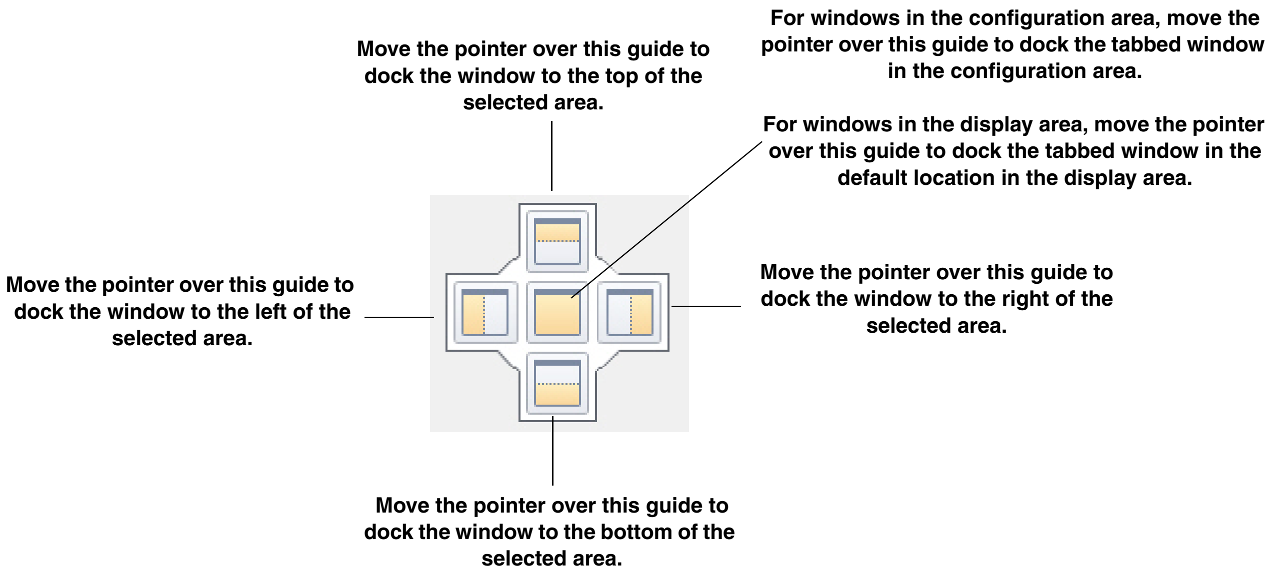
You can resize a window by clicking on any of the outside edges of the window and, while holding down the mouse button, dragging the window to the desired size. When you release the mouse, the window is scaled automatically.

You can also dock a window by doing the following:

1. Click the title bar or tab of the window.
2. Drag the window to where you want to dock it.
   A guide diamond appears.

   For windows in the configuration area, the four guides on the outside of the diamond represent the four sides of the configuration area. The center square docks the tabbed window in the configuration area. Additional guides are provided when you move the windows in the configuration area, allowing you to dock the window to the left side of the configuration area or to the right or left side of the display area.

   For windows in the display area, the four guides on the outside of the diamond represent the four sides of the display area. The center square docks the tabbed window in the display area.



3. Move the pointer over the corresponding portion of the guide (the designated area is shaded), and release the mouse.

For example, if you want to dock the Acquisition Config window to the left side of the configuration area, drag the Acquisition Config window toward the middle of the configuration area, move the pointer over the guide on the far left, and then release the mouse button.

Notes: To return a window in the display area to its most recent docked location, press **CTRL** while you double-click the title bar of the window.

To dock/undock a window in the configuration area, press **CTRL** while you double-click the title bar of the window.

Hiding a Window

For all windows, except the Acquisition Status window, you can hide the window by clicking the "X" at the top, right corner of the window. To show the window again, click the **Windows** menu, and the check the name of the window to show. Any window name that is not checked under the **Windows** menu is hidden.

You can hide the Acquisition Status window and the windows in the configuration area, if desired, by clicking the **Auto-Hide** pin ( ) in the top, right corner of the window. When the window is hidden, a tab for the window is created; the size of the display area is increased. When the window is shown, the size of the display area is decreased. To see the window again, hover over the tab for the window. Click the **Auto-Hide** pin again to display the window.

Resetting the Window Layout

If you want to reset the windows to their default view, select the **Windows** menu, and select **Reset Window Layout to Default Layout**.

You can also reset the display area to its last saved configuration by selecting the **Windows** menu, and selecting **Reset Window Layout to Last Saved**.

Acquisition Config Window

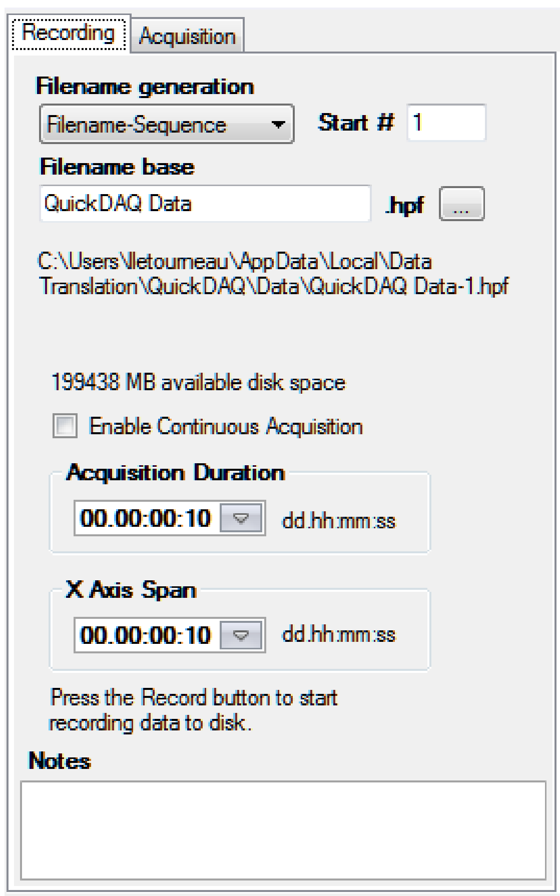
The contents of the Acquisition Config window depend on the interface mode that is selected, as described below.

Acquisition Config Window for the Data Logger Interface

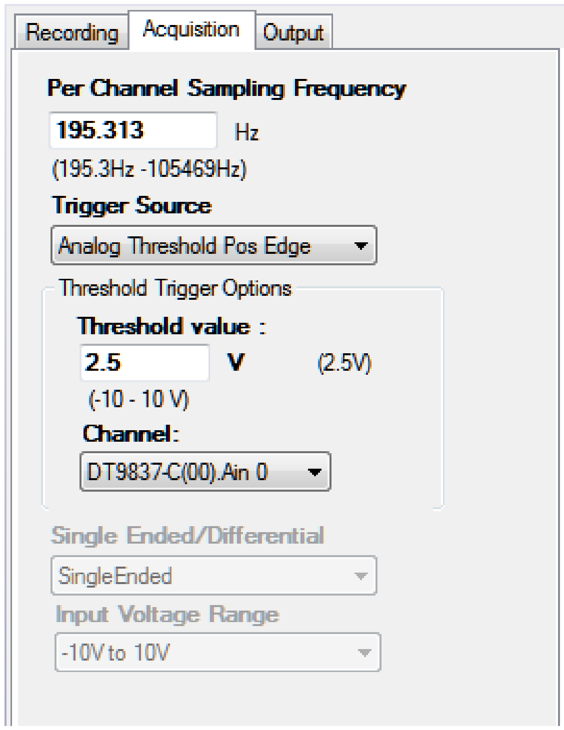
When using the Data Logger interface, the Acquisition Config window provides the following tabs for configuring the acquisition and output settings used by the device. Based on the information you set, the current sample rate, sample interval, and number of scans used by the device are displayed.

- **Recording tab** – This tab allows you to set up the file name for the data file, enable or disable continuous acquisition, and to specify the acquisition duration and x-axis span.

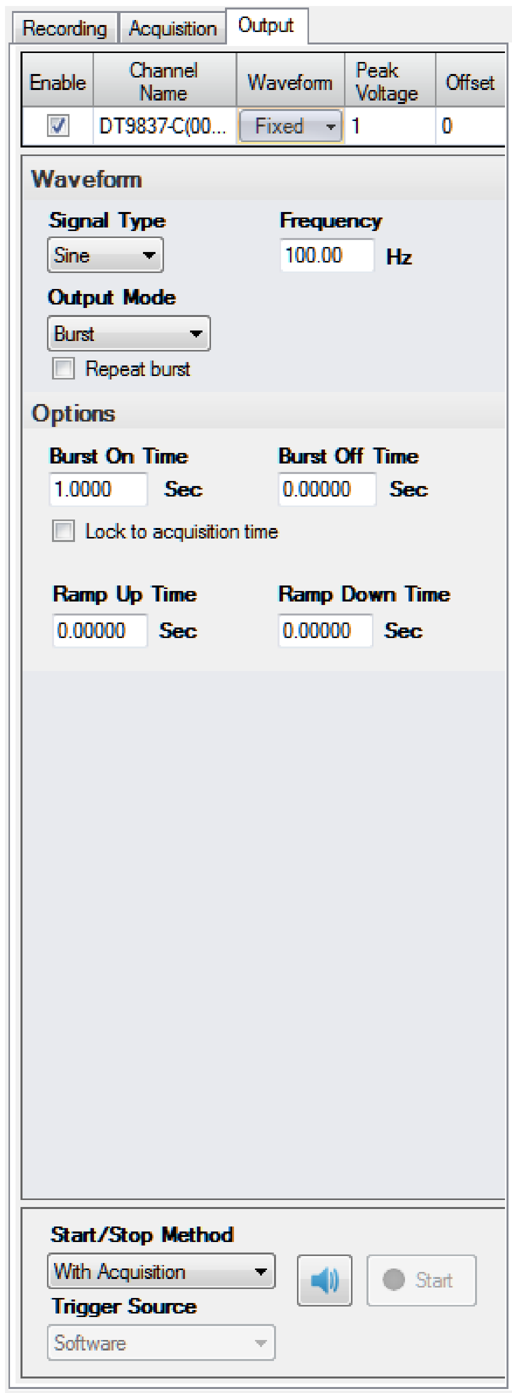
  Refer to page 119 for more information about using this tab to configure the recording settings.



- **Acquisition tab –** This tab allows you to set up the parameters that apply to the analog input subsystem, including the per channel sampling frequency of the device, the trigger source that starts the acquisition, the channel configuration (single-ended or differential), input voltage range, the temperature units to use when converting voltage data to temperature, and/or the filter type. Refer to page 121 for more information about using this tab to configure the acquisition settings.



- **Output tab** – This tab appears if your device supports continuous analog output operations. It allows you to set up the parameters for an analog output signal. You can define the type of waveform to output (fixed, sweep, or noise), and all the parameters that apply to that waveform type, including the signal or noise type, sweep mode, whether the signal is output continuously or as a burst, the frequency of the output signal, whether to ramp up or down, and how to start the operation. Refer to page 138 for more information about using this tab to configure the output settings.



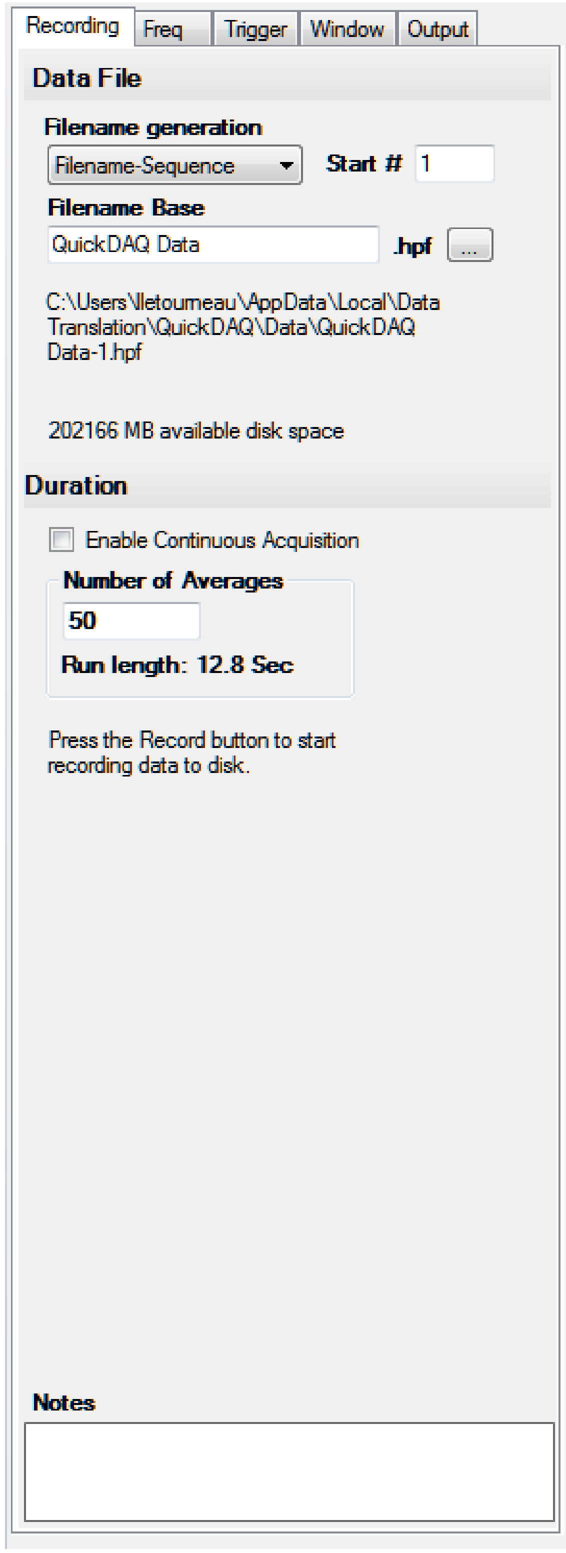
Acquisition Config Window for the FFT Analyzer Interface

If you are using the FFT Analyzer interface, the Acquisition Config window provides the following tabs for configuring the acquisition and output settings used by the device:

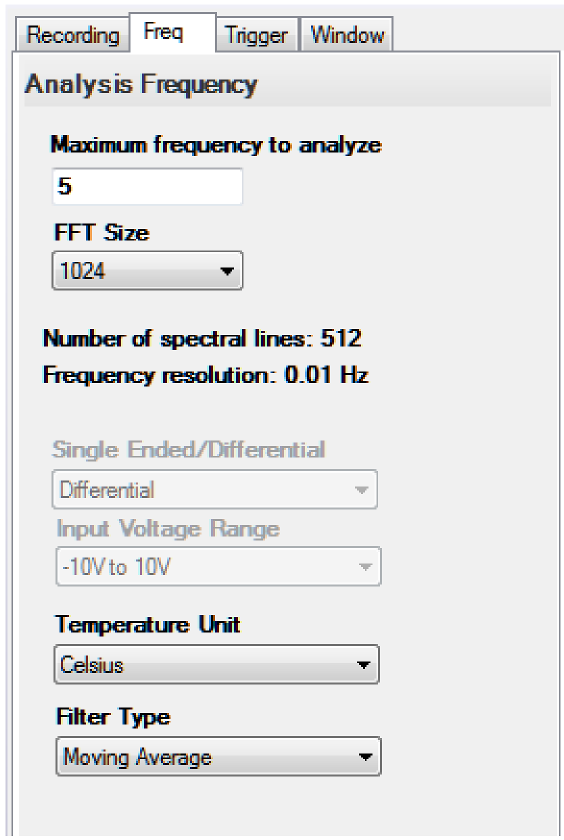
- **Recording tab –** This tab allows you to set up the data file name, enable or disable continuous acquisition, specify how many records to acquire, process, and average, and enter notes about the acquisition.

  A *record* is a block of data whose length is equal to the size of the FFT. An *average* is a series of measurements that are combined to create a result. The results from the last completed measurement are combined, point by point, with the results from the previous measurement. The combined results are retained and displayed as they are updated.

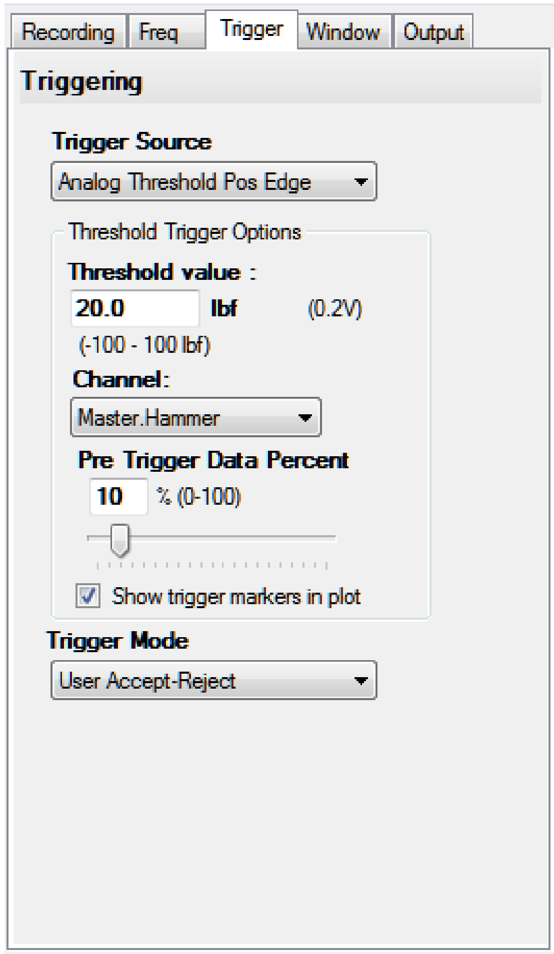
  Refer to page 122 for more information about using this tab to configure the recording settings.


Recording
Freq
Trigger
Window
Output
Data File
Filename generation
Filename-Sequence
Start #
1
Filename Base
QuickDAQ Data
.hpf
...
C:\Users\lletourneau\AppData\Local\Data Translation\QuickDAQ\Data\QuickDAQ Data-1.hpf
202166 MB available disk space
Duration
Enable Continuous Acquisition
Number of Averages
50
Run length: 12.8 Sec
Press the Record button to start recording data to disk.
Notes

- **Freq tab –** This tab allows you to set up the maximum frequency that you are interested in analyzing, the FFT size, and other parameters that apply to the analog input subsystem, including the channel configuration (single-ended or differential), input voltage range, the temperature units to use when converting voltage data to temperature, and/or the filter type. Refer to page 128 for more information about using this tab to configure the frequency settings.



- **Trigger tab –** This tab allows you to specify the trigger source that starts acquisition on the device, and the trigger mode. Refer to page 131 for more information about using this tab to configure the triggers settings.



- **Window tab** – This tab allows you to specify a windowing function to apply to the FFT channel to reduce spectral leakage errors. If you are using the FFT Analysis option, the Window tab appears as shown in Figure 5. If you are using the Advanced FFT Analysis option, the Window tab appears as shown in Figure 6.

  Refer to page 135 for more information about using this tab to configure a windowing function.

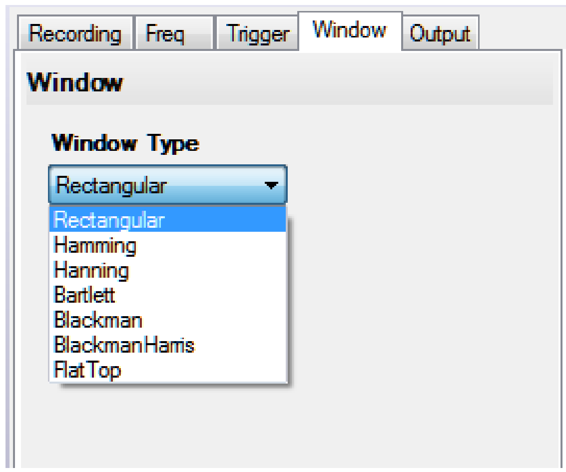


Figure 5: Window Tab for the FFT Analysis Option

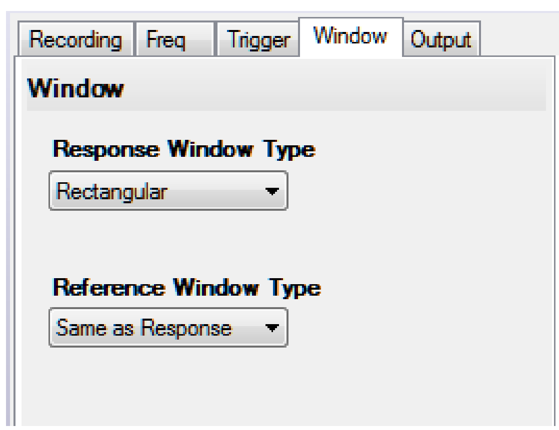
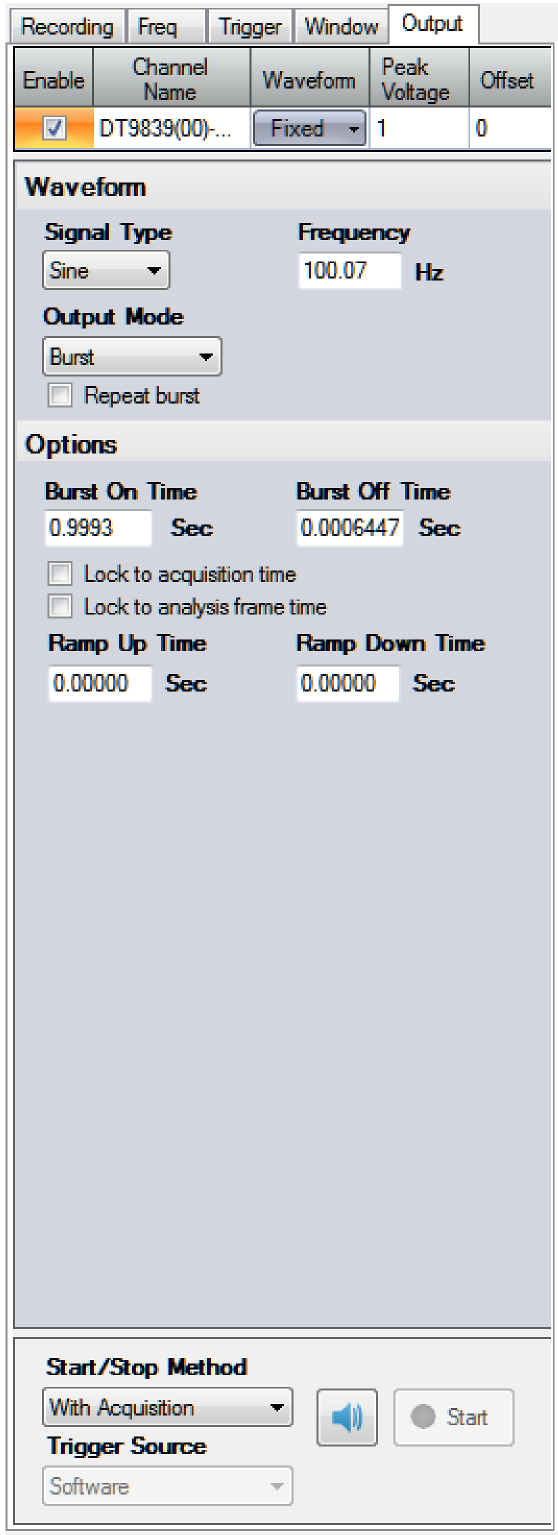


Figure 6: Window Tab for the Advanced FFT Analysis Option

- **Output tab** – This tab allows you to set up the parameters for an analog output signal, if your device supports analog output operations. You can define the type of waveform to output (fixed, sweep, or noise), and all the parameters that apply to that waveform type, including the signal or noise type, sweep mode, whether the signal is output continuously or as a burst, the frequency of the output signal, whether to ramp up or down, and how to start the operation.

Refer to page 138 for more information about using this tab to configure the output settings.



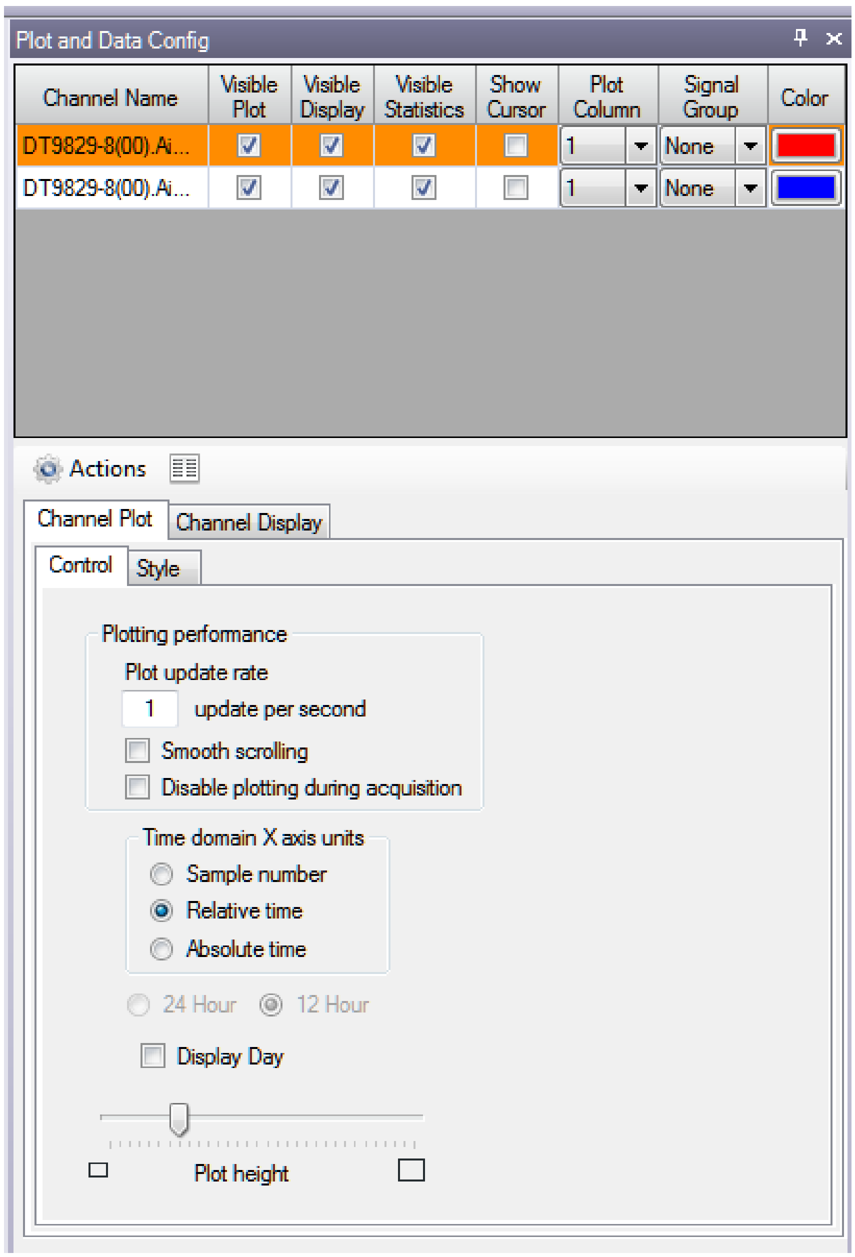
Plot and Data Config Window

The contents of the Plot and Data Config window depend on the interface mode that is selected, as described below.

Plot and Data Config Window for the Data Logger Interface

When you are using the Data Logger interface, the Plot and Data Config window allows you to select which channels you want to plot and specify the appearance of the display area.

Refer to Chapter 5 for information about how to configure the settings of the Plot and Data Config window.



Plot and Data Config Window for the FFT Analyzer Interface

When you are using the FFT Analyzer interface, the Plot and Data Config window allows you to select which channels you want to plot, add and remove single FFT functions, add and remove two-channel FFT functions (if you purchased the Advanced FFT Analysis option of QuickDAQ), specify the type of data analysis operations to perform on the selected channels, and specify the appearance of the display area. Figure 7 shows the Plot and Data Config window for the FFT Analysis option.

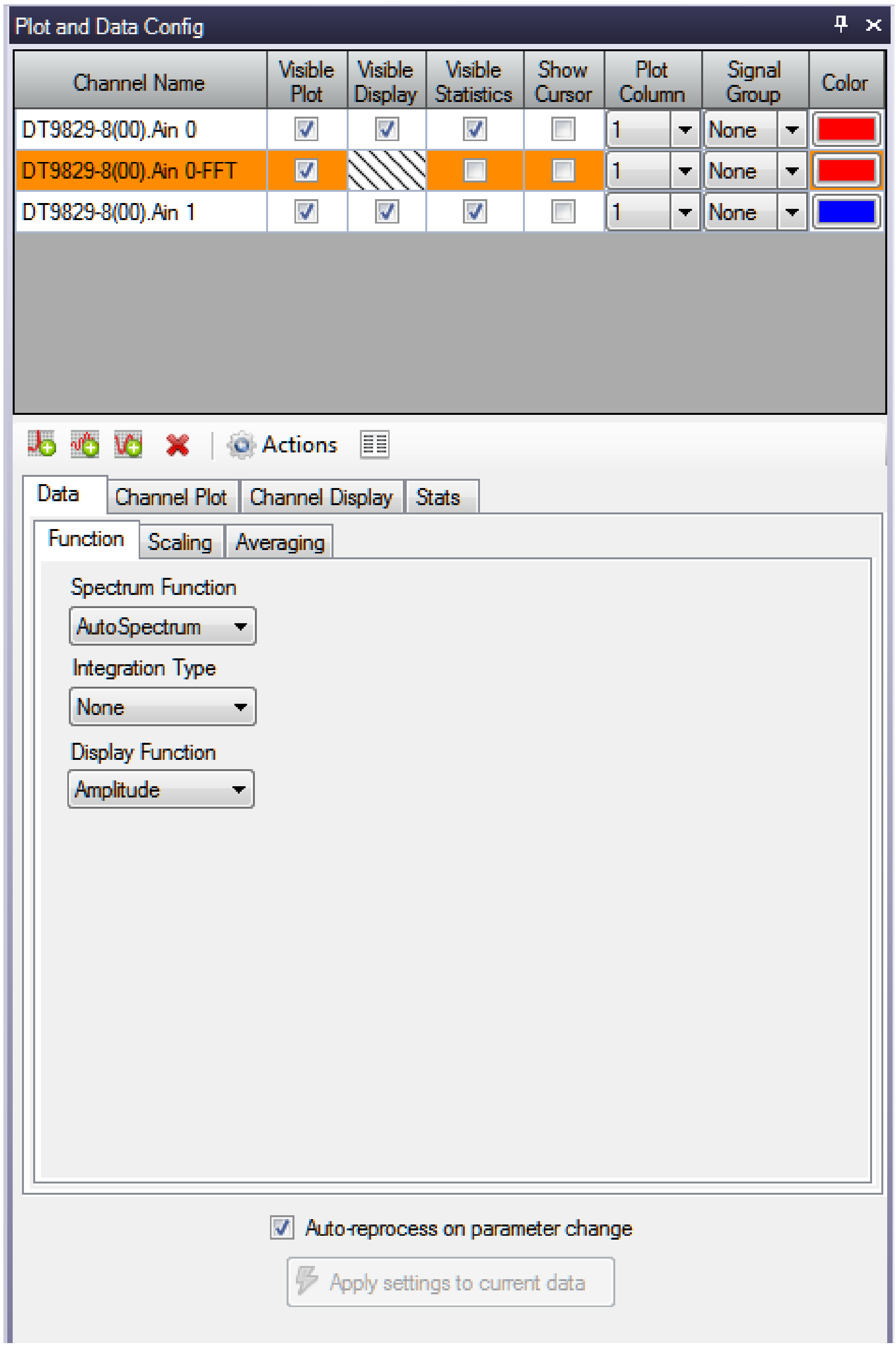


Figure 7: Plot and Data Config Window for the FFT Analysis Option

Figure 7 shows the Plot and Data Config window for the Advanced FFT Analysis option.

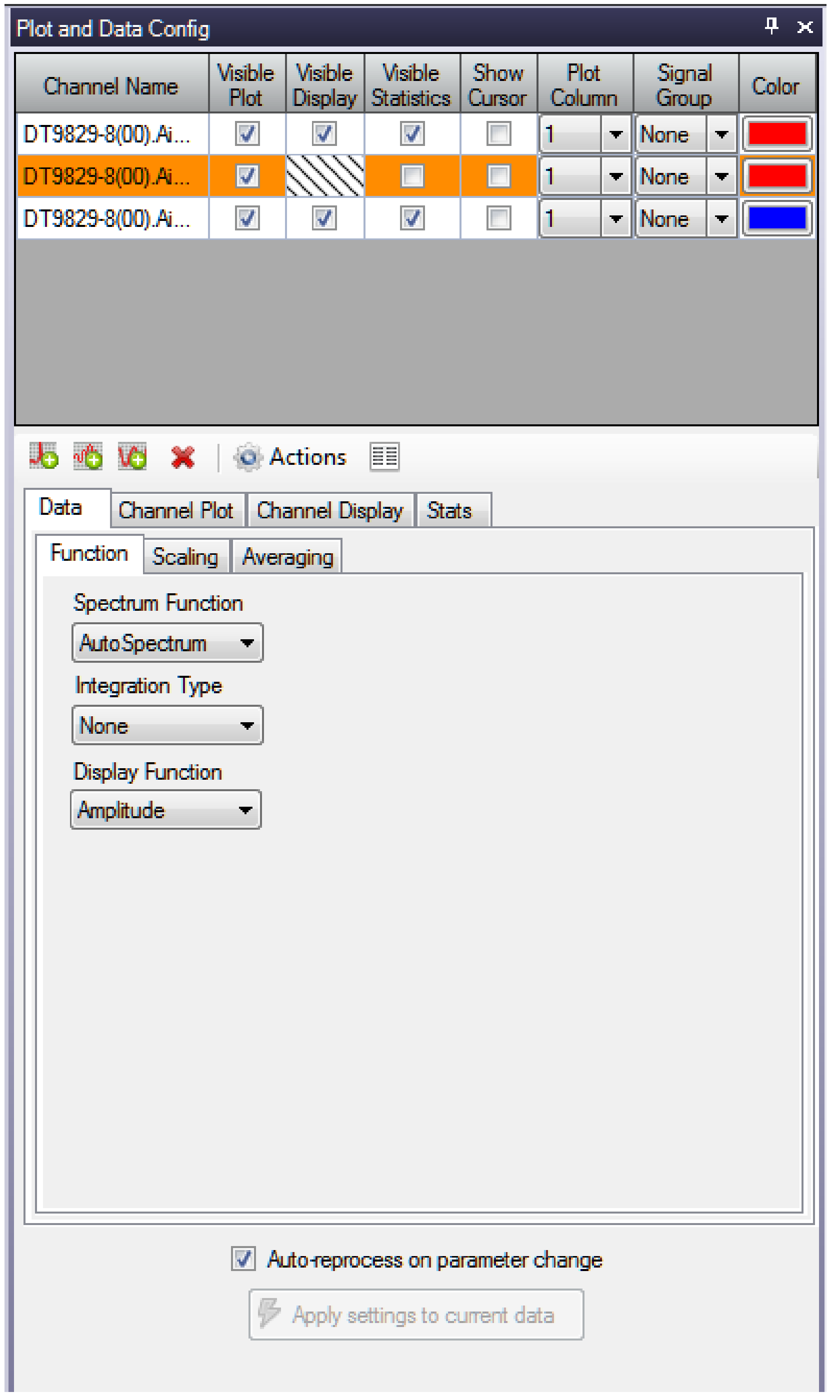


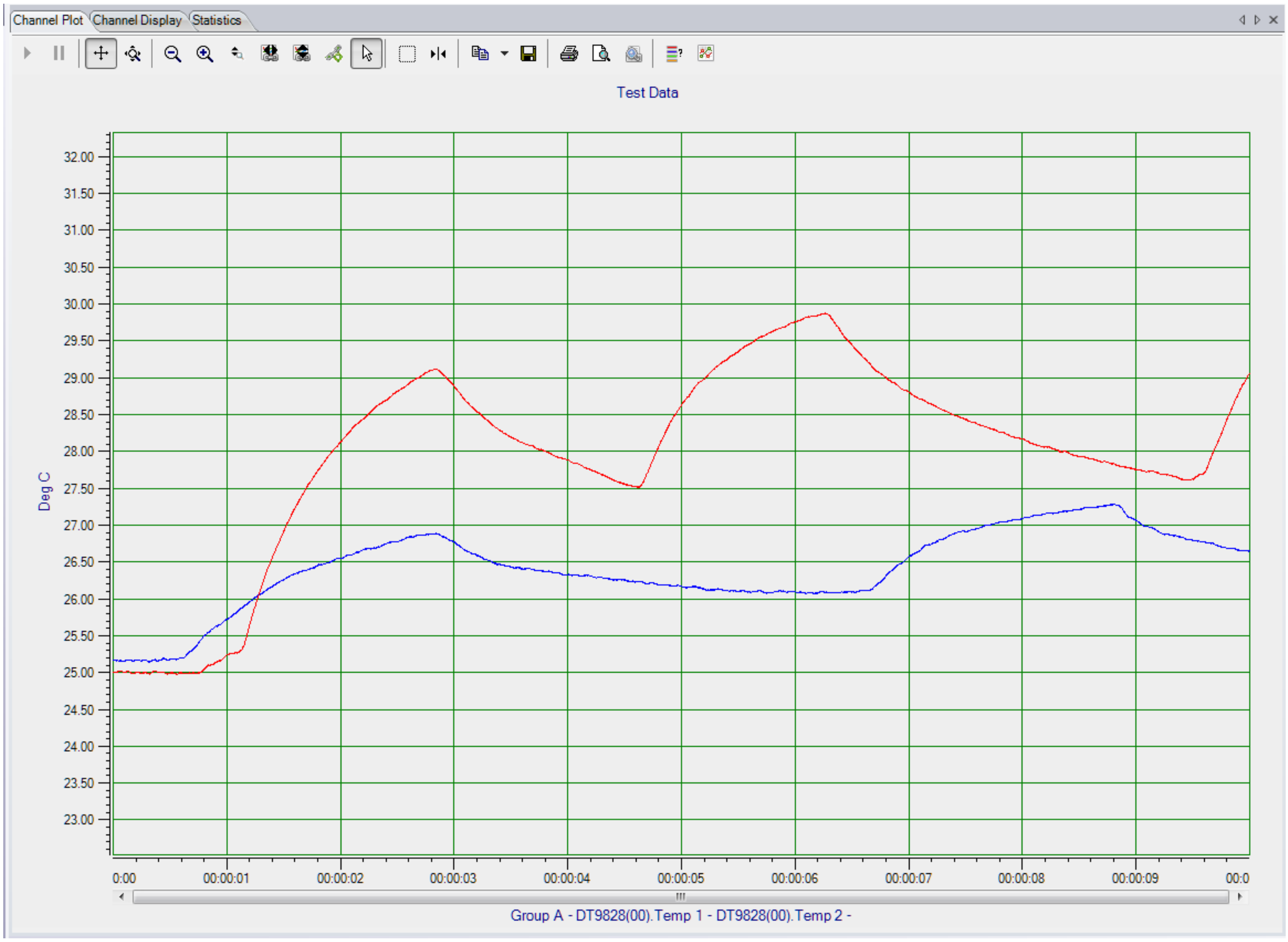
Figure 8: Plot and Data Config Window for the Advanced FFT Analysis Option

Refer to Chapter 6 for information about how to configure the settings of the Plot and Data Config window.

Channel Plot Window

You can access the Channel Plot window by clicking the **Channel Plot** tab in the display area of the interface or by clicking the **Channel Plot** menu option under the **Windows** menu.

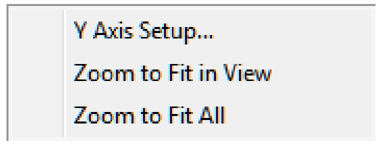
During acquisition, the results are plotted in the Channel Plot window in real time. In file-reader mode, the results are read from the file and plotted in the Channel Plot window.



The following sections describe how to set up the scaling for the y- and x-axes on the Channel Plot window, how to use the scrolling options for the Channel Plot window, and how to use the controls provided for the Channel Plot window. Refer to Chapter 5 starting on page 167 for more information about the Channel Plot window.

Setting Up the Y-Axis

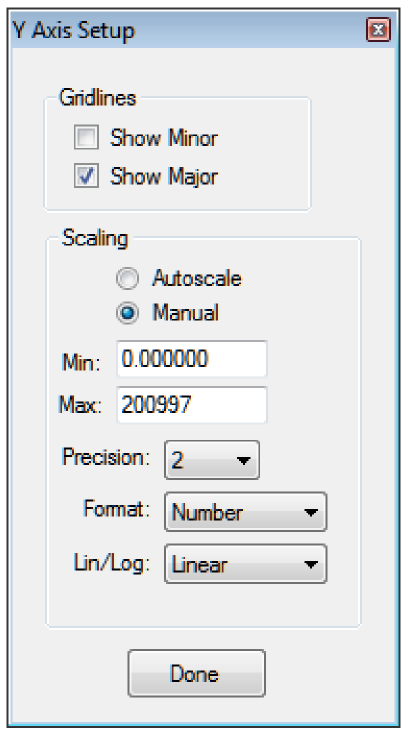
If you right-click on the y-axis, the following dialog box is displayed:



These options are described below.

Y Axis Setup

If you select **Y Axis Setup**, the following dialog appears:



You can set up the following parameters for the y-axis:

- **Gridlines** – Select **Show Minor** if you want to display the minor gridlines on the y-axis; select **Show Major** if you want to display the major gridlines on the y-axis.
- **Scaling** – Select **Autoscale** if you want the application to determine the scale of the y-axis automatically. If you want to specify the scale of the y-axis, select **Manual** and then enter the minimum and maximum values for the y-axis.
- **Precision** – You can select the number of significant digits after the decimal point to display on the y-axis and in the cursor text; values range from 0 to 8 (2 is the default).

- **Format** – You can change for the format of the axis to see either the actual **Number** (such as 200) or the number in **Exponent** form (such as 2.0e+002).
- **Lin/Log** – For time-domain channels, you can change the y-axis of the plotted data to either **Linear** or **Log10**.

  If you are using the FFT Analyzer interface and **Linear** scaling is selected for the FFT channel (using the **Data - Scaling** tab of the Plot and Data Config window), you can change the y-axis of the plotted data to either **Linear** or **Log10**. If you are using the FFT Analyzer interface and **dB** scaling is selected for the FFT channel (using the **Data - Scaling** tab of the Plot and Data Config window), this setting is not applicable.

Zoom to Fit in View

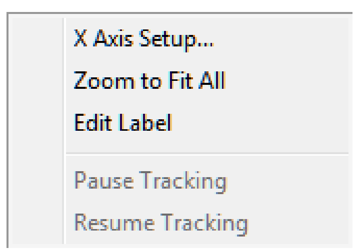
Select this menu option to scale the y-axis for the selected channel to the minimum and maximum values of the actual input signal(s) that are in the current view.

Zoom to Fit All

Select this menu option to scale the y-axis for the selected channel to the minimum and maximum values of the actual input signal(s) that were acquired, regardless of whether the data is in the current view.

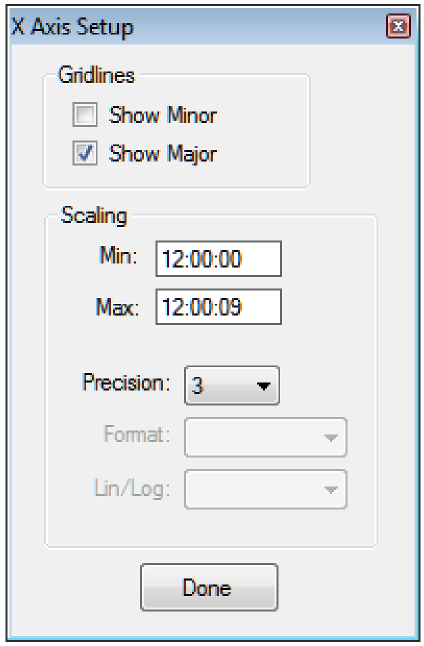
Setting Up the X-Axis

If you right-click on the x-axis, the following dialog box is displayed:



X Axis Setup

If you select **X Axis Setup**, the following dialog appears:



You can set up the following parameters for the x-axis:

- **Gridlines** – Select **Show Minor** if you want to display the minor gridlines on the y-axis; select **Show Major** if you want to display the major gridlines on the y-axis.
- **Scaling** – Specify the scale of the x-axis by entering the minimum and maximum values for the x-axis.
- **Precision** – You can select the number of significant digits after the decimal point to display on the x-axis and in the cursor text; values range from 0 to 8 (3 is the default).
- **Format** – If you are using the FFT Analyzer interface, you can change for the format of the x-axis to see either the actual **Number** (such as 200) or the number in **Exponent** form (such as 2.0e+002).
- **Lin/Log** – If you are using the FFT Analyzer interface, you can change the axis of the plotted data to either **Linear** or **Log10**.

Zoom to Fit All

Select this menu option to scale the x-axis for the selected channel to the minimum and maximum values of the actual input signal(s) that were acquired.

Edit Label

Select this menu option to change the text that is displayed for the x-axis.

Pause Tracking

If the application is recording, you can click this popup menu option to pause tracking for a specific plot. The plot stops updating, but data is still recorded to disk. The tracking for other plots is unaffected by this action.

A specific plot can also be paused by using the scrollbar for the x-axis to scroll through the data. After tracking is paused, the scrollbar continues to be updated so that you can scroll to the data as it is being acquired.

If you want to pause tracking for all channels, use the Tracking Pause button on the toolbar instead. Refer to page 71 for more information.

While tracking is paused, you can use the controls for the Channel Plot window, described on page 71, to analyze the data that was previously recorded.

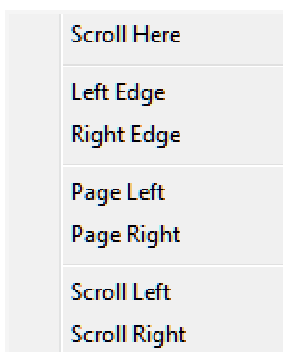
Resume Tracking

If recording is still in progress, you can resume tracking for a specific plot by clicking the Resume Tracking popup menu option. The tracking for other plots is unaffected.

To resume tracking for all plots whose tracking was paused, click the Tracking Resume button on the toolbar. Refer to page 71 for more information.

Scrolling Options

If you right-click on the scrollbar of the x-axis, the following dialog box is displayed:



The options are as follows:

- **Scroll Here** – Moves the scroll bar to the point of the mouse click that opened the scroll bar dialog.
- **Left Edge** – Moves the scroll bar to the left edge of the acquired data.
- **Right Edge** – Moves the scroll bar to the right edge of the acquired data.
- **Page Left** – Updates the content of the scroll area with the page of data to the left of the data is currently being displayed.

- **Page Right** – Updates the content of the scroll area with the page of data to the right of the data is currently being displayed.
- **Scroll Left** – Scrolls to the left.
- **Scroll Right** – Scrolls to the right.

Using the Channel Plot Window Controls

A toolbar, shown in Figure 9, provides a number of controls for the Channel Plot window.

Figure 9: Toolbar of the Channel Plot Window

Tracking Resume 

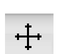
If the application is recording and tracking was paused, you can click this button to resume tracking for all plots on the display. The display is then updated with the current data in real time.

To resume tracking on a specific plot rather than on all plots, right-click the x-axis and select the Resume Tracking popup menu option; refer to page 70 for more information.

Tracking Pause 

If the application is recording, you can click this button to pause tracking on all plots of the display. The display stops updating, but data is still recorded to disk. While tracking is paused you can use the other controls to analyze the data that was previously recorded.

To pause tracking on a specific plot rather than on all plots, right-click the x-axis and select the Pause Tracking popup menu option or use x-axis scrollbar to scroll through the data; refer to page 70.

Axes Scroll 

When you click this button, you can scroll through the axes by holding down the left mouse button and dragging the mouse over the axis (up and down for the y-axis, or right and left for the x-axis).

If you want to change the magnification (zoom) of the axes while this button is selected, hold down the Ctrl button while holding down the left mouse button, and drag the mouse over the axis (up and down for the y-axis, or right and left for the x-axis).

When this button is selected, the Axes Zoom button is unselected.

Axes Zoom

When you click this button, you can change the magnification (the zoom) of the axes by holding down the left mouse button and dragging the mouse over the axis (up and down for the y-axis, or right and left for the x-axis).

When this button is selected, the Axes Scroll button is unselected.

Zoom-Out All

When you click this button, the magnification of all axes is reduced (zoomed out) by a factor of 2.

Zoom-In All

When you click this button, all axes are magnified (zoomed in) by a factor of 2.

Zoom All to Fit in View Y

When you click this button, the scale of all y-axes are adjusted to the minimum and maximum values of the actual input signal(s).

Link X Axes to Selected

Click the x-axis for the plot that you want to link to (the master plot), then click the **Link X Axes to selected** button to link the x-axes of all the other plots (the slave plots) with the same data domain (time domain or frequency domain). You can then use the scrolling controls on the master plot to affect the view of all the slave plots with the same data domain at the same time.

For example, assume that you have three plots (a time-domain plot in volts, a frequency-domain plot, and a time-domain plot in degrees C). If you choose the x-axis of the plot in volts and click the **Link X Axes to selected** button, the time-domain plot in volts is the master plot and the time-domain plot in degrees C is the slave plot. When you zoom into the data or scroll through the x-axis of the master plot, the application also zooms into the data or scrolls through the x-axis for the slave plot. The frequency-domain plot is unaffected.

If you zoom into or scroll through the x-axis of the slave plot, none of the other plots is affected. To resynchronize the slave plot with the master plot, either zoom into or scroll through the x-axis of the master plot, or unlink the x-axes in the master plot and relink them in the master plot using the **Link X Axes to selected** button.

Link Y Axes to Selected 

Click the y-axis for the plot that you want to link to (the master plot) then click the **Link Y Axes to selected** button to link the y-axes of all the other plots (the slave plots) that have the same data domain (either time domain or frequency domain). You can then use the scrolling controls on the master plot to affect the view of all the slave plots with the same data domain at the same time.

For example, assume that you have three plots (a time-domain plot in volts, a frequency-domain plot, and a time-domain plot in degrees C). If you choose the y-axis of the plot in volts and click the **Link Y Axes to selected** button, the time-domain plot in volts is the master plot and the time-domain plot in degrees C is the slave plot. When you scroll through the y-axis of the master plot, the application also scrolls through the y-axis for the slave plot. The frequency-domain plot is unaffected.

If you scroll through the y-axis in the slave plot, none of the other plots is affected. To resynchronize the slave plot with the master, either scroll through the y-axis in the master plot, or unlink the y-axes in the master plot and relink them using the **Link X Axes to selected** button.

Link Cursors to Selected 

Locks data cursors together so that by moving the locked data cursor, all the data cursors move together. To use this button, click the **Data-Cursor** button, click the data cursor that you want to lock to, and then click the **Link Cursors to Selected** button. Once it is locked, moving the locked data cursor moves all the data cursors.

Select 

When you click this button, you can select items in the view area to zoom or scroll. This is equivalent to holding down the **Ctrl** and **Alt** keys while pressing the **V** key, or selecting the **Plot** menu, selecting **Plot Area Mouse Mode**, and choosing the **Select** menu option.

Zoom Box 

When you click this button, you can select a region in the view to magnify (zoom) by holding down the left mouse button, dragging the mouse to select the region that you want to magnify, and releasing the mouse button. The zoom action affects both axes.

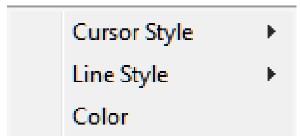
This is equivalent to holding down the **Ctrl** and **Alt** keys while pressing the **Z** key, or selecting the **Plot** menu, selecting **Plot Area Mouse Mode**, and choosing the **Zoom** menu option.

Data-Cursor 

When you click this button, you can move the position of a data-cursor on the curve by holding down the left mouse button and dragging the mouse.

This is equivalent to holding down the **Ctrl** and **Alt** keys while pressing the **C** key, or selecting the **Plot** menu, selecting **Plot Area Mouse Mode**, and choosing the **Cursor** menu option.

When the **Data-Cursor** is active, right-click the mouse to change the cursor style, line style, and/or color of the cursor; the following menu appears:

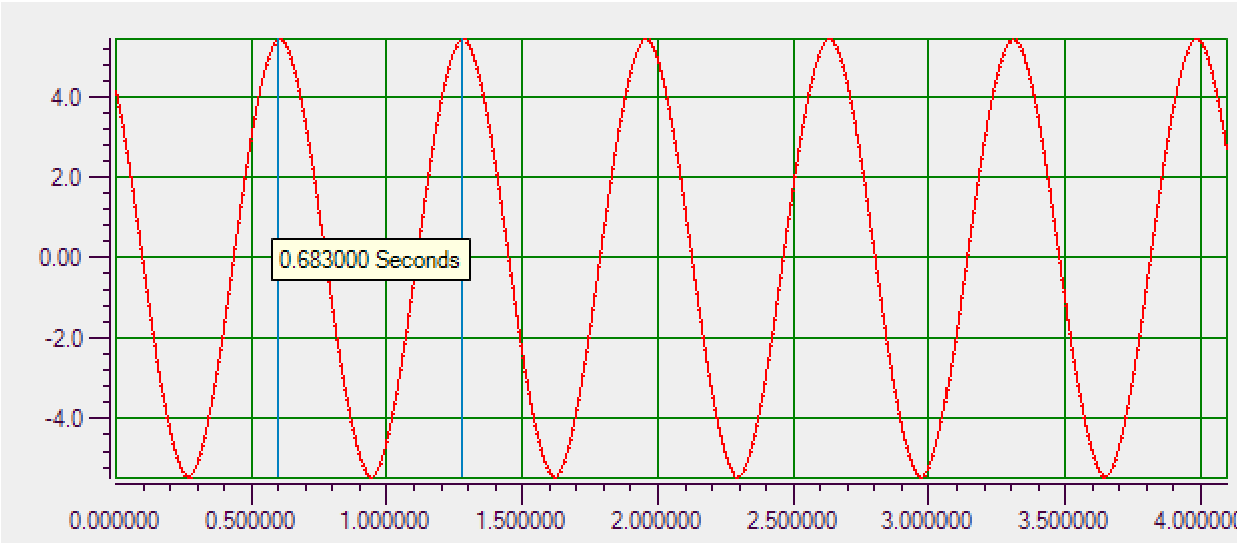


You can change the cursor style to one of the following settings:

- **Value-XY –** Choose this style to display a single value at a particular point on the X- and Y-axes.
- **Period –** Choose this style to display the period, in seconds, between two vertical cursors. You can drag each cursor independently.
- **Frequency –** Choose this style to display the frequency, in Hertz, between two vertical cursors. You can drag each cursor independently.
- **Peak-Peak –** Choose this style to display the value between two horizontal cursors. This is the number of volts or sensor units. You can drag each cursor independently.
- **Band Power –** Available with the FFT Analysis and Advanced FFT Analysis options. When the spectrum function (selected on the **Data - Function** tab of the Plot and Data Config window) is **AutoSpectrum** or **PSD**, and the scaling type is **Linear** (selected on the **Data - Scaling** tab of the Plot and Data Config window), select this style to display the square root of the sum of the values between the cursor lines.

  When the spectrum function (selected on the **Data - Function** tab of the Plot and Data Config window) is **Spectrum**, and the scaling type is **Linear** (selected on the **Data - Scaling** tab of the Plot and Data Config window), select this style to display the square root of the sum of the squared values between the cursor lines.

The following example shows a period data cursor, where two data cursors show the period, in seconds, between two data points:



You can change the line style to one of the following settings:

- **Solid** – Choose this style to display the cursors as solid lines.
- **Dash** – Choose this style to display the cursors as a dashed lines.
- **Dot** – Choose this style to display the cursors as a dotted lines.
- **DashDot** – Choose this style to display the cursors as lines of dash-dot elements.

You can change the color of the cursors to one of the provided basic colors or to any custom color you desire.

Copy to Clipboard 

When you click this button, you can copy either the graph (the picture) or the data that is displayed in the Channel Plot window to the clipboard.

Save Plot Image 

When you click this button, you can save the image from the Channel Plot window to a file. Specify the name of the file and choose one of the extensions to save the file as a graphic.

The default location for saving the image files is as follows:

For Windows Vista, Windows 7, or Windows 8:
C:\Users\<username>\AppData\Local\Data Translation\QuickDAQ\Data

For Windows XP:
C:\Documents and Settings\<user name>\Local Settings\Application Data\Data Translation\QuickDAQ\Data

Print 

When you click this button, you can print the picture from the Channel Plot window to a printer.

Preview 

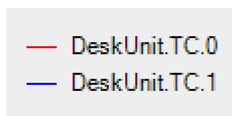
When you click this button, you can preview the image from the Channel Plot window before you print it.

Page Setup 

When you click this button, you can configure the page before you print it. Options are available for configuring the size, orientation, and margins of the page.

Show Legend 

When you click this button, you can show or hide the legend that defines the colors used for each channel on the plot. An example follows:

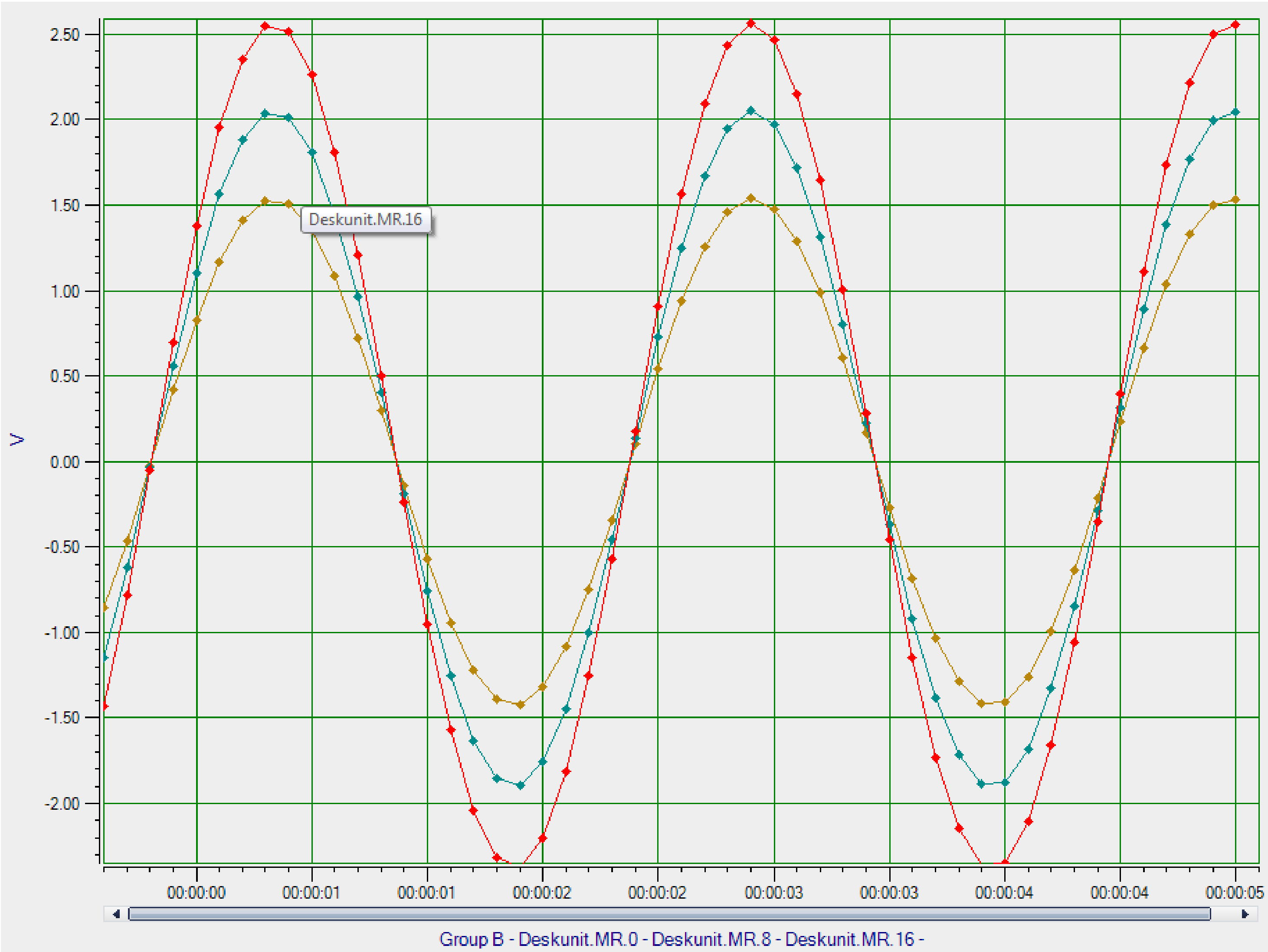


Show Data Point Markers 

When you click this button, you can show or hide markers that indicate the actual samples on the curve, as shown in the following example. The diamond markers indicate the actual samples.

If you click on a data point marker, a label for the channel is displayed.

Note: When moving the data-cursor with the data point markers shown, the data-cursor snaps to the closest sample on the curve.



2.50
2.00
1.50
1.00
0.50
0.00
-0.50
-1.00
-1.50
-2.00
V
Deskunit.MR.16
00:00:00
00:00:01
00:00:01
00:00:02
00:00:02
00:00:03
00:00:03
00:00:04
00:00:04
00:00:05
Group B - Deskunit.MR.0 - Deskunit.MR.8 - Deskunit.MR.16 -

Channel Display Window

You can access the Channel Display window by clicking the **Channel Display** tab in the display area of the interface or by clicking the **Channel Display** menu option under the **Windows** menu.

During acquisition, the newest value for each enabled channel is displayed in the Channel Display window in real time. The Channel Display window allows you to view data from your device in a digital display. By default, the layout represents the channel locations on the front panel of the device. Refer to Chapter 5 starting on page 167 for more information about the Channel Display window.

Statistics Window

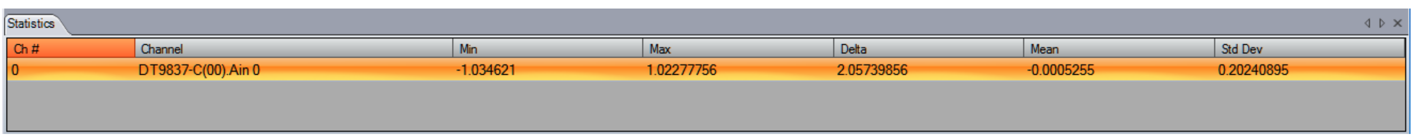
You can access the Statistics window by clicking the **Statistics** tab in the display area of the interface or by clicking the **Channel Statistics** menu option under the **Windows** menu.

Statistics are handled differently depending on the interface mode you selected, as described below.

Statistics in the Data Logger Interface

If you are using the Data Logger interface, statistics are displayed in the Statistics window in real time as data is acquired, provided that the **Visible Statistics** checkbox in the Plot and Data Config window is selected for the channel.

The Statistics window shows the minimum, maximum, and mean values of the data, along with the difference between the minimum and maximum values (called the delta) and the standard deviation for each enabled channel:



In file-reader mode, the results are read from the file and the statistics are shown in the Statistics window.

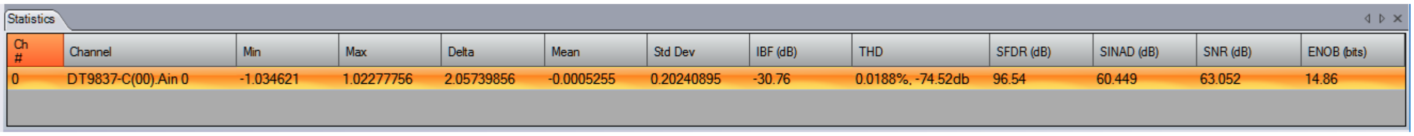
Note: To save the data that is displayed in the Statistics window, either take a screen shot of the Statistics window or click the Statistics tab, press Ctrl-A (to select all listed channels), and press Ctrl-C to past the data into another application.

Refer to page 186 for more information about statistics in the Data Logger interface.

Statistics in the FFT Analyzer Interface

If you are using the FFT Analyzer interface, the statistics that you select using the **Stats** tab, described on page 234, are displayed in the Statistics window in real time as data is acquired, provided that the **Visible Statistics** checkbox in the Plot and Data Config window is selected for the channel.

In file-reader mode, the results are read from the file and the statistics that you selected using the **Stats** tab are shown in the Statistics window:



Using the **Stats** tab, you can choose to display the following statistics for the enabled channels:

- Minimum value, maximum value, mean value, delta of minimum and maximum values, and standard deviation statistics
- Dynamic statistics – Allows you to quantify the analog characteristics of the data that is acquired through the A/D converter on the device. You can select either RMS (Root Mean Squared) or Vector averaging to compute the following statistics:
  - IBF – Input Below Full-Scale
  - THD – Total Harmonic Distortion
  - SFDR – Spurious Free Dynamic Range
  - SINAD – Signal-to-Noise And Distortion
  - SNR – Signal-to-Noise Ratio
  - ENOB – Effective Number of Bits

Note: To save the data that is displayed in the Statistics window, either take a screen shot of the Statistics window or click the Statistics tab, press Ctrl-A (to select all listed channels), and press Ctrl-C to past the data into another application.

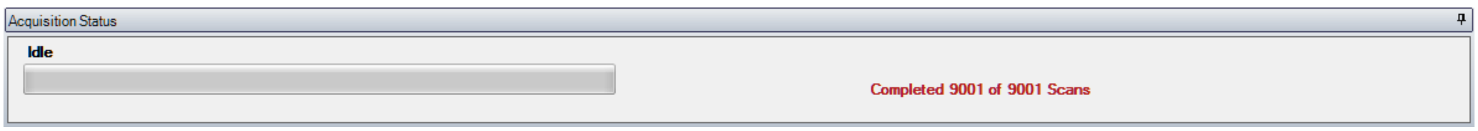
Refer to page 234 for more information about statistics in the FFT Analyzer interface.

Acquisition Status Window

The Acquisition Status window, located under the Channel Plot, Channel Display, and Statistics windows, contains different information depending on the interface mode you are using, as described below.

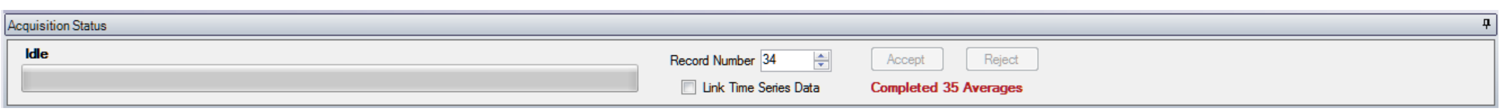
Acquisition Status Window When Using the Data Logger Interface

If you are using the Data Logger interface, the Acquisition Status window shows the status of the device, including whether the device is acquiring data or idle and how many scans have been completed.



Acquisition Status Window When Using the FFT Analyzer Interface

If you are using the FFT Analyzer interface, the Acquisition Status window shows the status of the device, as well as the **Record Number** and the **Link Time Series Data** checkbox, described on page 227, and the **Accept** and **Reject** buttons, described on page 134.



4

Configuring Channels, Acquisition Settings, and Digital Filters

Configuring Channels of a Device

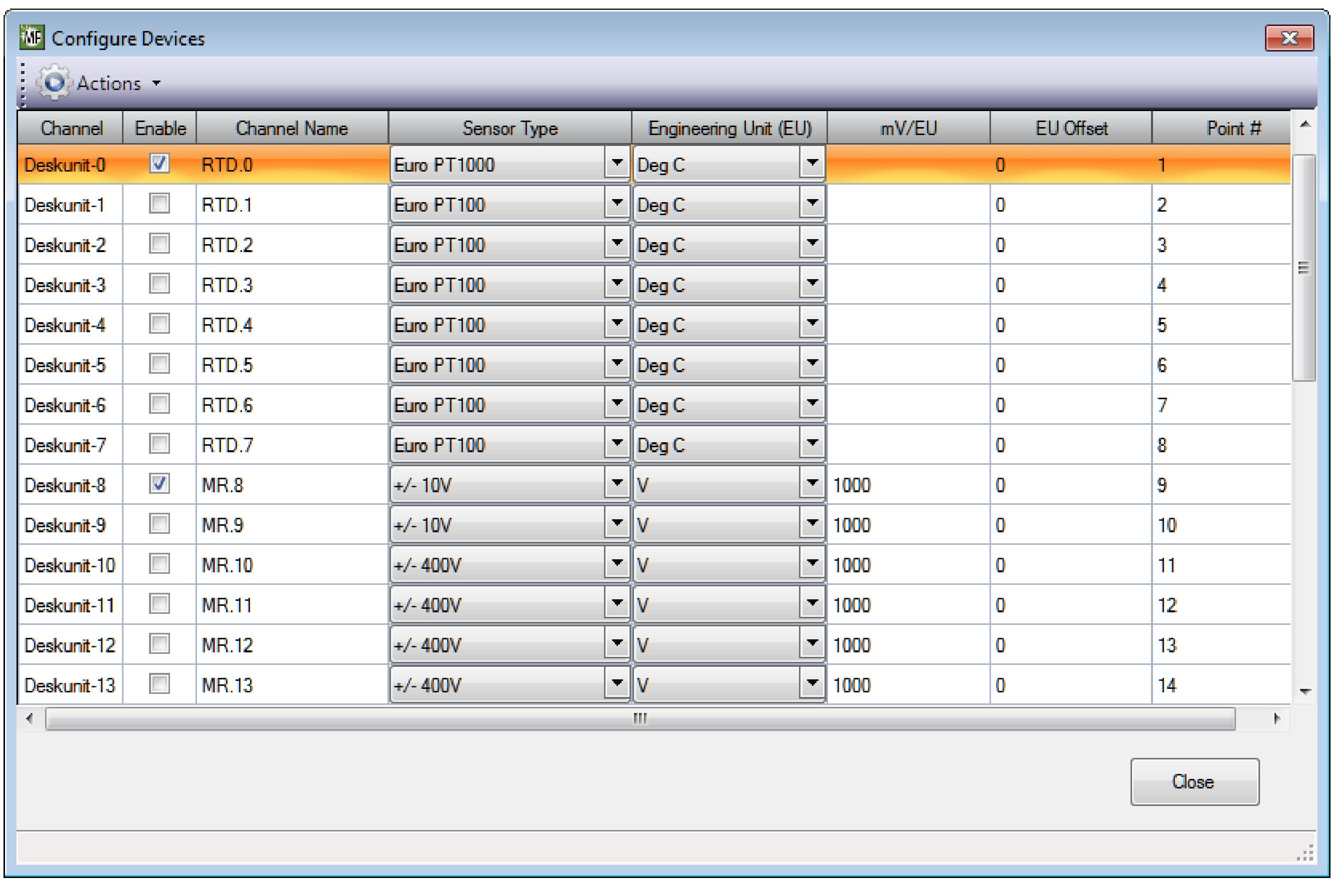
Notes: To configure the channels, you must have selected a supported device or have opened an .hpf file that contains configuration settings for a supported device.

You can change the following settings in an .hpf file: Channel Name, Ref/Resp (Advanced FFT Analysis option only), EU, mv/EU, and EU offset; once you've changed these settings, you can reprocess the data with the changes

If acquisition is in progress, you must choose **Stop** from the **Record** menu or click the **Stop** toolbar button (Stop) before you can change the configuration of a device.

Choose the **Input Channel Configuration** toolbar button ( ) or the **Input Channel Configuration** option from the **Configuration** menu to configure the channels from which to acquire data in the analog input data stream. The Configure Devices dialog appears.

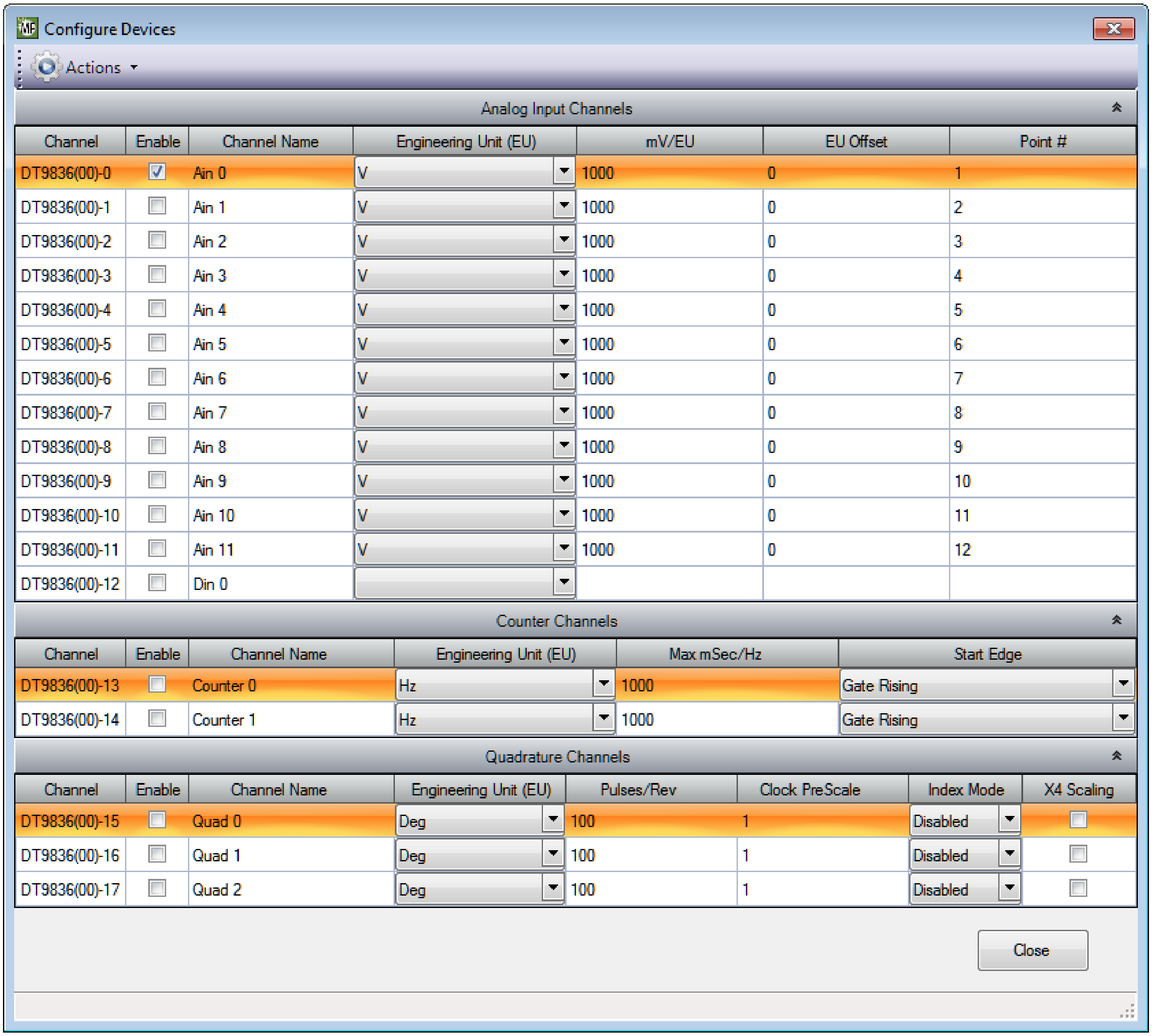
The elements of the Configure Devices dialog depend on the specified device. For example, Figure 10 shows the Configure Devices dialog for a MEASURpoint instrument:



Channel	Enable	Channel Name	Sensor Type	Engineering Unit (EU)	mV/EU	EU Offset	Point #
Deskunit-0	☑	RTD.0	Euro PT1000	Deg C		0	1
Deskunit-1	☐	RTD.1	Euro PT100	Deg C		0	2
Deskunit-2	☐	RTD.2	Euro PT100	Deg C		0	3
Deskunit-3	☐	RTD.3	Euro PT100	Deg C		0	4
Deskunit-4	☐	RTD.4	Euro PT100	Deg C		0	5
Deskunit-5	☐	RTD.5	Euro PT100	Deg C		0	6
Deskunit-6	☐	RTD.6	Euro PT100	Deg C		0	7
Deskunit-7	☐	RTD.7	Euro PT100	Deg C		0	8
Deskunit-8	☑	MR.8	+/- 10V	V	1000	0	9
Deskunit-9	☐	MR.9	+/- 10V	V	1000	0	10
Deskunit-10	☐	MR.10	+/- 400V	V	1000	0	11
Deskunit-11	☐	MR.11	+/- 400V	V	1000	0	12
Deskunit-12	☐	MR.12	+/- 400V	V	1000	0	13
Deskunit-13	☐	MR.13	+/- 400V	V	1000	0	14

Figure 10: Example Configuration of a MEASURpoint Instrument

Figure 11 shows an example configuration for the DT9836.



Analog Input Channels

Channel	Enable	Channel Name	Engineering Unit (EU)	mV/EU	EU Offset	Point #
DT9836(00)-0	☑	Ain 0	V	1000	0	1
DT9836(00)-1	☐	Ain 1	V	1000	0	2
DT9836(00)-2	☐	Ain 2	V	1000	0	3
DT9836(00)-3	☐	Ain 3	V	1000	0	4
DT9836(00)-4	☐	Ain 4	V	1000	0	5
DT9836(00)-5	☐	Ain 5	V	1000	0	6
DT9836(00)-6	☐	Ain 6	V	1000	0	7
DT9836(00)-7	☐	Ain 7	V	1000	0	8
DT9836(00)-8	☐	Ain 8	V	1000	0	9
DT9836(00)-9	☐	Ain 9	V	1000	0	10
DT9836(00)-10	☐	Ain 10	V	1000	0	11
DT9836(00)-11	☐	Ain 11	V	1000	0	12
DT9836(00)-12	☐	Din 0				

Counter Channels

Channel	Enable	Channel Name	Engineering Unit (EU)	Max mSec/Hz	Start Edge
DT9836(00)-13	☐	Counter 0	Hz	1000	Gate Rising
DT9836(00)-14	☐	Counter 1	Hz	1000	Gate Rising

Quadrature Channels

Channel	Enable	Channel Name	Engineering Unit (EU)	Pulses/Rev	Clock PreScale	Index Mode	X4 Scaling
DT9836(00)-15	☐	Quad 0	Deg	100	1	Disabled	☐
DT9836(00)-16	☐	Quad 1	Deg	100	1	Disabled	☐
DT9836(00)-17	☐	Quad 2	Deg	100	1	Disabled	☐

Figure 11: Example Configuration of a DT9836 Module

Figure 12 shows an example configuration for the DT9837C.

Note: The Ref/Resp heading appears only if you are using the Advanced FFT Analysis option of QuickDAQ.

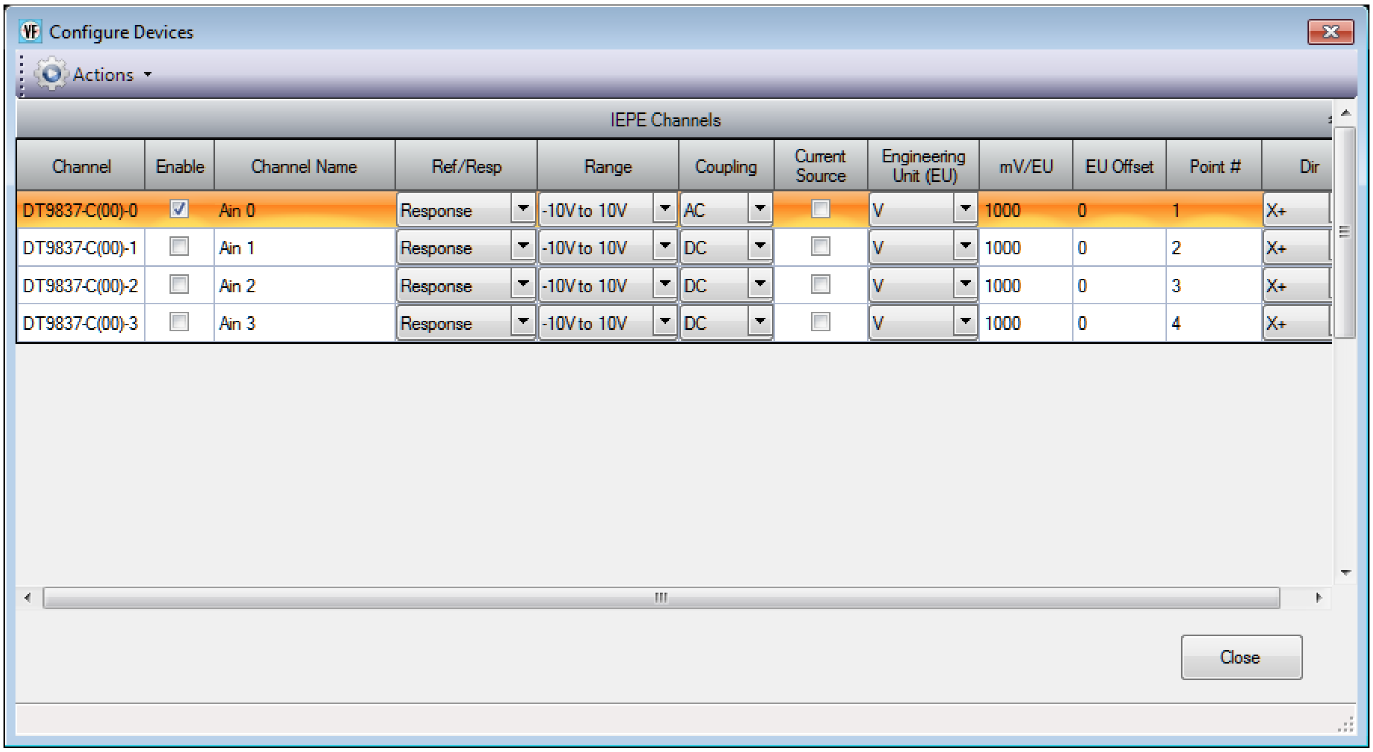


Figure 12: Example Configuration for the DT9837C

Figure 13 shows an example configuration for the DT9829.

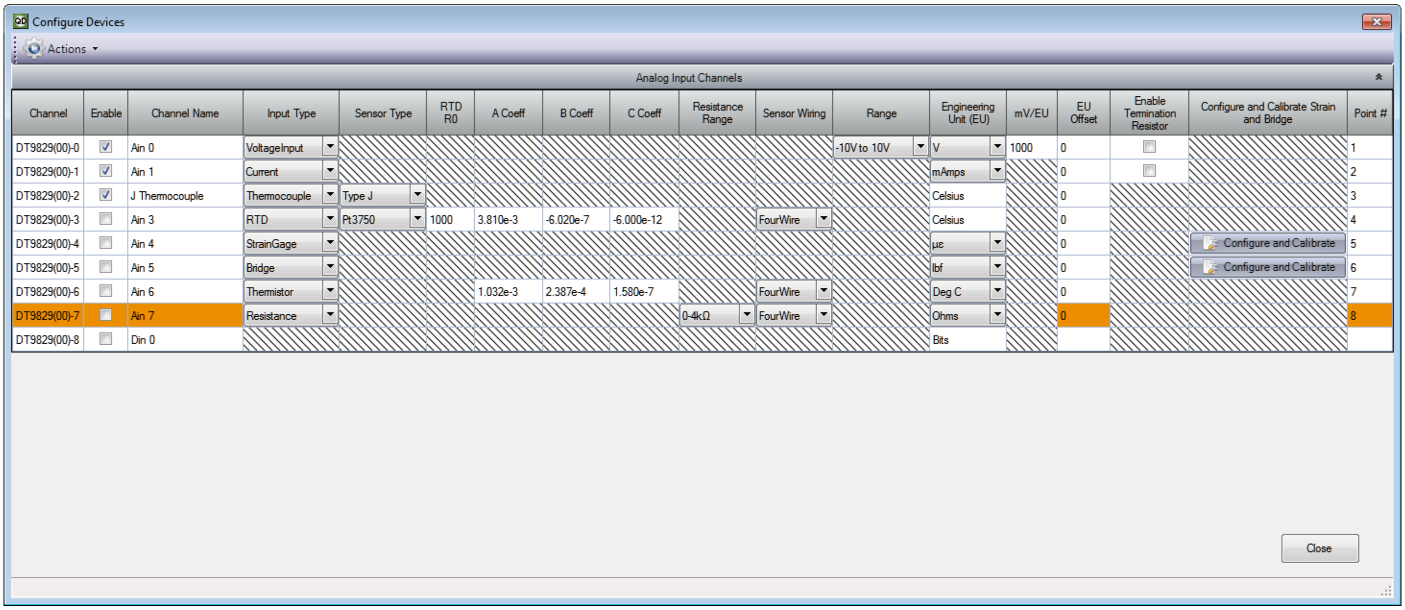
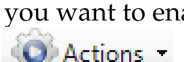


Figure 13: Example Configuration for the DT9829

Enabling Channels

Select the channels that you want to enable for acquisition by clicking the checkbox under the **Enable** column. Only data from the enabled channels is included in the analog input data stream.

Note: For the DT9805 and DT9806 modules, the CJC channel is enabled automatically if any of the analog input channels are configured for a temperature sensor type. You can disable this channel, if desired.

If you want to enable all the channels at once, click the **Multi-channel Actions** button () and then click **Enable all Channels**. If you want to enable several but not all channels at once, highlight the channels that you want to enable, click the **Multi-channel Actions** button, and then click **Enable all Selected Channels**.

If you want to disable all the channels at once, select the **Multi-channel Actions** button, and then click **Disable all Channels**. If you want to disable several but not all channels at once, highlight the channels that you want to disable, click the **Multi-channel Actions** button, and then click **Disable all Selected Channels**.

Channel Name

For each channel, you can specify a meaningful name for each channel, if desired, by highlighting the channel under the **Channel Name** column and entering a new name.

Input Type

For devices that support multiple sensor types for each channel, you can configure the type of measurement to perform for the channel using the **Input Type** column. If you are using the DT9829 module, for example, you can choose from one of the following selections: VoltageInput, Current, Thermocouple, RTD, Strain Gage, Bridge, Thermistor, or Resistance.

If your device supports dedicated measurement types for each channel, the **Input Type** column does not appear.

Ref/Resp

If you are using the Advanced FFT Analysis option of QuickDAQ and you want to use a two-channel FFT function, select **Reference** for the channel that will create the stimulus, and select **Response** for the channels that will measure the effect of the stimulus. You must select at least one enabled analog input channel as the **Reference** channel.

You can then compare the spectrum of the response channel to the spectrum of the reference channel by adding a two-channel FFT function to the list of channels. Refer to page 202 for more information about adding two-channel FFT functions.

Analog Input Channel Settings for Devices that Support Voltage

This section describes the settings that you can configure for the analog input channels on your device.

Sensor Type

This setting is only available for MEASURpoint instruments. To measure voltage from a thermocouple or RTD channel on a MEASURpoint instrument, select **Volts** as the Sensor Type.

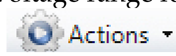
Range

If your device supports multiple input ranges, set the input range for each analog input channel.

For example, the following devices support multiple input ranges:

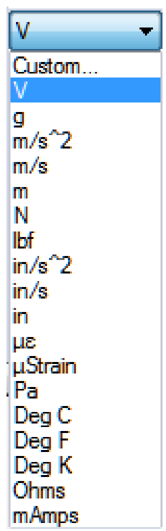
- DT8874, DT9847, DT8873, and DT9873 instruments: ±10 V, ±100 V, or ±400 V
- DT8875 instruments: ±10 V, ±100 V, or ±600 V
- DT8876 instruments: ±100 mV, ±1.0 V, or ±10 V
- DT8824 instrument modules: ±10 V, ±1.25 V, ±0.625 V, or ±0.3125 V

- DT8824-HV instrument modules: ±600 V, ±75 V, ±37.5 V, or ±18.75 V
- DT9829 modules: ±10 V, ±2 V, or ±0.2 V

If you want to specify the same voltage range for all multi-range channels, click the **Multi-channel Actions** button (), select **Set all Multi-Range Channels**, and select the specific voltage range.

Engineering Unit (EU)

For voltage inputs, you can choose **Volts** or one of the following engineering units to display the data, based on the sensor that is attached to the channel:



If you select **Custom**, you can define your own engineering units.

If you want to specify the same engineering units for all voltage channels, click the **Multi-channel Actions** button (), select **Set all Voltage Channel Units**, and select the specific engineering units to use from the list above.

If you want to set the engineering units for some but not all analog input channels at once, select the **Multi-channel Actions** button, click **Set all Selected Channel Units**, and select the desired engineering units.

mV/EU

Enter the number of millivolts per engineering unit (mV/EU) by which to scale the raw voltage based on the engineering units that you selected.

EU Offset

You can enter an offset (EU Offset) value to add to the scaled signal.

Enable Termination Resistor

Some devices support the option of enabling the bias return termination resistor for voltage inputs. Typically, this box should be checked (termination resistor is enabled) when your signal source is floating or grounded, and should be unchecked (termination resistor is disabled) when the signal source has a grounded reference.

Refer to your hardware documentation for more information on whether to enable or disable this resistor.

Point

If desired, you can enter a number representing the test point to which this channel corresponds. By default, values start at 1 and increment with each channel.

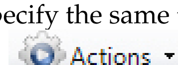
Analog Input Channel Settings for Devices that Support Thermocouples

This section describes the analog input settings that you can configure for devices, such as the MEASURpoint instruments, DT9829 modules, and DT9805/DT9806 modules, that support thermocouples.

Sensor Type

Select the type of thermocouple that is connected to each channel under the **Sensor Type** column. Values are as follows:

- Type B
- Type E
- Type J
- Type K
- Type N
- Type R
- Type S
- Type T
- Volts (if you want to measure voltage instead; refer to page 88 for more information on configuring channels for voltage measurements)

If you want to specify the same thermocouple type for all channels, click the **Multi-channel Actions** button (

 ), select **Set all Thermocouple Channels**, and select the specific thermocouple type.

Engineering Unit (EU)

For channels that are configured for thermocouple inputs, the data is returned in Celsius, Fahrenheit, or Kelvin. You select the engineering units using the Acquisition Config tab, described on page 127.

If you selected a **Sensor Type** of Volts, refer to page 89 for more information.

EU Offset

You can enter an offset (EU Offset) value to add to the scaled signal.

Point #

If desired, you can enter a number representing the test point to which this channel corresponds. By default, values start at 1 and increment with each channel.

Analog Input Channel Settings for Devices that Support RTDs

This section describes the analog input settings that you can configure for devices, such as the MEASURpoint instruments and DT9829 modules, that support RTDs.

Sensor Type

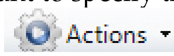
The available choices for **Sensor Type** depend on whether you are using RTDs on a MEASURpoint instrument (which uses an IVI-COM driver) or other devices, such as the DT9829 module, which use a DT-Open Layers device driver.

Sensor Type Selections for MEASURpoint Instruments

If you are using a MEASURpoint instrument, the following sensor types are available for RTD channels:

- Euro PT1000 (for 2- and 4-wire RTDs)
- Euro PT500 (for 2- and 4-wire RTDs)
- Euro PT100 (for 2- and 4-wire RTDs)
- American PT1000 (for 2- and 4-wire RTDs)
- American PT500 (for 2- and 4-wire RTDs)
- American PT100 (for 2- and 4-wire RTDs)
- Euro PT1000 3 Wire (for 3-wire RTDs)
- Euro PT500 3 Wire (for 3-wire RTDs)
- Euro PT100 3 Wire (for 3-wire RTDs)
- American PT1000 3 Wire (for 3-wire RTDs)
- American PT500 3 Wire (for 3-wire RTDs)
- American PT100 3 Wire (for 3-wire RTDs)

- Ohms (if you want to measure resistance instead; refer to page 95 for more information on configuring channels for resistance measurements)
- Volts (if you want to measure voltage instead; refer to page 88 for more information on configuring channels for voltage measurements)

If you want to specify the same RTD type for all channels, click the **Multi-channel Actions** button (

), select **Set all RTD Channels**, and select the specific RTD type.

Sensor Type Selections for the DT9829 and Other DT-Open Layers Devices

If you are using a DT9829 module or any other device that uses a DT-Open Layers device driver, the following sensor types are available for RTD channels:

- Custom
- Pt3750
- Pt3850
- Pt3911
- Pt3916
- Pt3920
- Pt3928

The **Sensor Type** identifies the Temperature Coefficient of Resistance (TCR) value that is used by the Callendar-Van Dusen transfer function to determine temperature for RTDs. The Callendar-Van Dusen transfer function is described as follows:

$$R_T = R_0[1 + AT + BT^2 + CT^3(T - 100)]$$

where,

- R_T is the resistance at temperature (TCR).
- R_0 is the resistance at 0° C.
- A, B, and C are the Callendar-Van Dusen coefficients for a particular RTD type. (The value of C is 0 for temperatures above 0° C.)

Table 1 lists the coefficients that are used by the Callendar-Van Dusen transfer function for each **Sensor Type**.

Table 1: Callendar-Van Dusen Coefficients Supported By RTD Channels

Sensor Type	Temperature Coefficient of Resistance (TCR)	R0 Coefficient	A Coefficient	B Coefficient	C Coefficient	Applicable Standards
Pt3750	0.003750 Ω / Ω /° C	1000 Ω	3.81×10^{-3}	-6.02×10^{-7}	-6.0×10^{-12}	Low Cost
Pt3850	0.003850 Ω / Ω /° C	100 Ω, 500 Ω, 1000 Ω	3.9083×10^{-3}	-5.775×10^{-7}	-4.183×10^{-12}	DIN/IEC 60751 ASTM-E1137
Pt3911	0.003911 Ω / Ω / ° C	100 Ω	3.9692×10^{-3}	-5.8495×10^{-7}	-4.233×10^{-12}	US Industrial Standard
Pt3916	0.003916 Ω / Ω /° C	100 Ω	3.9739×10^{-3}	-5.870×10^{-7}	-4.4×10^{-12}	Japanese JISC 1604-1989
Pt3920	0.003920 Ω / Ω /° C	98.129 Ω	3.9787×10^{-3}	-5.869×10^{-7}	-4.167×10^{-12}	SAMA RC21-4-1966
Pt3928	0.003928 Ω / Ω /° C	100 Ω	3.9888×10^{-3}	-5.915×10^{-7}	-3.85×10^{-12}	ITS-90

If you specify the **Sensor Type** as **Custom**, you must specify the values for the R0, A, B, and C coefficients used in the transfer function.

RTD R0

This setting is not available for MEASURpoint instruments. For all other devices, specify the appropriate R0 coefficient for the selected **Sensor Type**. This value is used by Callendar-Van Dusen transfer function to determine temperature.

For example, for a **Sensor Type** of Pt3929, select 100 Ω as the **RTD R0** value.

A Coeff

This setting is not available for MEASURpoint instruments. For all other devices, specify the appropriate A coefficient for the selected **Sensor Type**. This value is used by Callendar-Van Dusen transfer function to determine temperature.

For example, for a **Sensor Type** of Pt3929, select 39.888e-4 as the **A Coeff** value.

B Coeff

This setting is not available for MEASURpoint instruments. For all other devices, specify the appropriate B coefficient for the selected **Sensor Type**. This value is used by Callendar-Van Dusen transfer function to determine temperature.

For example, for a **Sensor Type** of Pt3929, select –59.15e-8 as the **B Coeff** value.

C Coeff

This setting is not available for MEASURpoint instruments. For all other devices, specify the appropriate C coefficient for the selected **Sensor Type**. This value is used by Callendar-Van Dusen transfer function to determine temperature.

For example, for a **Sensor Type** of Pt3929, select –38.5e-13 as the **C Coeff** value.

Sensor Wiring

This setting is not available for MEASURpoint instruments. For all other devices, specify the wiring configuration that is used to connect the RTD to the analog input channel. Available selections are TwoWire, ThreeWire, and FourWire.

Engineering Unit (EU)

For RTD inputs, the data is returned in Celsius, Fahrenheit, or Kelvin. You select the engineering units using the **Acquisition Config** tab, described on page 127.

If you selected a **Sensor Type** of Ohms for a MEASURpoint instrument, refer to page 95. If you selected a **Sensor Type** of Volts for a MEASURpoint instrument, refer to page 89.

EU Offset

You can enter an offset (EU Offset) value to add to the scaled signal.

Point

If desired, you can enter a number representing the test point to which this channel corresponds. By default, values start at 1 and increment with each channel.

Analog Input Channel Settings for Devices that Support Thermistors

This section describes the analog input settings that you can configure for devices, such as the DT9829 modules, that support thermistors.

A Coeff, B Coeff, and C Coeff

To determine the temperature of a thermistor, the software uses the Steinhart-Hart equation:

$$\frac{1}{T} = A + BlnR + Cln(R)^3$$

where,

- T is the temperature.
- R is the resistance at T, in ohms, that is supplied by the device.

- A, B, and C are the Steinhart-Hart coefficients for a particular thermistor type and value, and are supplied by the thermistor manufacturer.

Using software, you specify the A coefficient under the **A Coe**ff column, the B coefficient under the **B Coeff** column, and the C coefficient under the **C Coeff** column.

Sensor Wiring

Specify the wiring configuration that is used to connect the thermistor to the analog input channel. Available selections are TwoWire, ThreeWire, and FourWire.

Engineering Unit (EU)

For thermistors, the data is returned in Celsius, Fahrenheit, or Kelvin. You select the engineering units using the **Acquisition Config tab**, described on page 127.

EU Offset

You can enter an offset (EU Offset) value to add to the scaled signal.

Point

If desired, you can enter a number representing the test point to which this channel corresponds. By default, values start at 1 and increment with each channel.

Analog Input Channel Settings for Devices that Support Resistance

This section describes the analog input settings that you can configure for devices, such as MEASURpoint instruments and DT9829 modules, that support resistance.

Sensor Type

This setting is only available for MEASURpoint instruments.

To measure resistance from a channel on a MEASURpoint instrument, select **Ohms** as the Sensor Type.

Resistance Range

This setting is not available for MEASURpoint instruments. For all other devices, specify the range of the resistor that you are trying to measure.

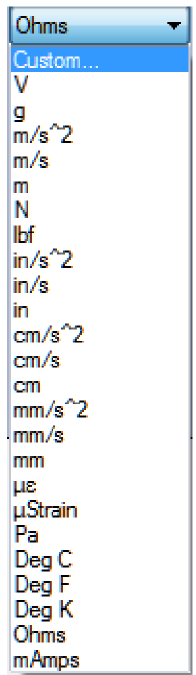
For example, the DT9829 module provides the following ranges: 0 to 4 kΩ and 4kW to 20 kΩ.

Sensor Wiring

This setting is not available for MEASURpoint instruments. For all other devices, specify the wiring configuration that is used to connect the resistor to the analog input channel. Available selections are TwoWire, ThreeWire, and FourWire.

Engineering Unit (EU)

For resistance measurements, you can choose **Ohms** or one of the following engineering units to display the data:



EU Offset

You can enter an offset (EU Offset) value to add to the scaled signal.

Point

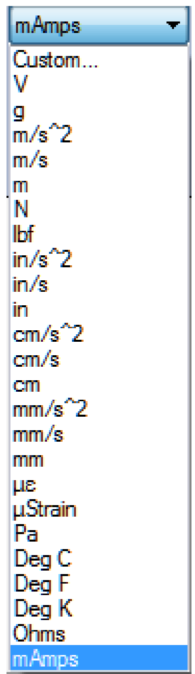
If desired, you can enter a number representing the test point to which this channel corresponds. By default, values start at 1 and increment with each channel.

Analog Input Channel Settings for Devices that Support Current

This section describes the analog input settings that you can configure for devices, such as DT9829 modules, that support current measurements.

Engineering Unit (EU)

For current measurements, you can choose **mAmps** or one of the following engineering units to display the data:



EU Offset

You can enter an offset (EU Offset) value to add to the scaled signal.

Enable Termination Resistor

Some devices support the option of enabling the bias return termination resistor for current inputs. Typically, this box should be checked (termination resistor is enabled) when your signal source is floating or grounded, and should be unchecked (termination resistor is disabled) when the signal source has a grounded reference.

Refer to your hardware documentation for more information on whether to enable or disable this resistor.

Point #

If desired, you can enter a number representing the test point to which this channel corresponds. By default, values start at 1 and increment with each channel.

Analog Input Channel Settings that Support IEPE Inputs

This section describes the settings that you can configure for devices, such as the DT8837, DT9837 Series, and DT9847 Series devices, that support IEPE inputs on the analog input channels.

Range

Set the input range for each analog input channel.

Coupling Type

Select **AC** (AC coupling) to remove any DC offset from your measurement, or select **DC** (DC coupling) if the signal source has no offset voltage or if the DC content of the acquired signal is important. Most IEPE input signals require AC coupling.

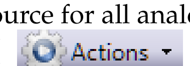
If you want to set the coupling type for all analog input channels at once, select the **multi-channel Actions** button (

), click **Set all Channel Coupling**, and select either **AC** or **DC**.

Note: On some devices, such as the DT9839 and DT9839E, AC or DC coupling is determined by the way the input is wired to the device; therefore, this software setting is not available for these devices.

Current Source

Enable use of the 4 mA current source by checking the **Current Source** checkbox, or disable use of the 4 mA current source by unchecking the **Current Source** checkbox. Most IEPE input signals require use of the current source.

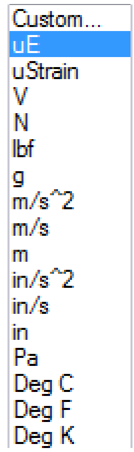
If you want to set the current source for all analog input channels at once, select the **multi-channel Actions** button (

), click **Set all Channel Current Source**, and select either **Enabled** or **Disabled**.

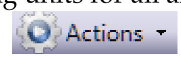
Note: On some devices, such as the DT9839 and DT9839E, use of the current source is determined by the way the input is wired to the device; therefore, this software setting is not available for these devices.

Engineering Unit (EU)

You can choose from one of the following engineering units to represent the data, based on the sensor that is attached to the channel:



If you select **Custom**, you can define your own engineering units.

If you want to set the engineering units for all analog input channels at once, select the **Multi-channel Actions** button (

 ), click **Set all Channel Units**, and select the desired engineering units.

If you want to set the engineering units for some but not all analog input channels at once, select the **Multi-channel Actions** button (

 ), click **Set all Selected Channel Units**, and select the desired engineering units.

mV/EU

Depending on the engineering units that you selected, you can enter the number of millivolts per engineering unit (mV/EU) by which to scale the raw voltage.

EU Offset

Depending on the engineering units that you selected, you can enter an offset (EU Offset) value to add to the scaled signal.

Point

If desired, you can enter a number representing the test point to which this channel corresponds. By default, values start at 1 and increment with each channel.

Dir

Specify one of the following sensor directions for the analog input channel, if desired: X+, X-, Y+, Y-, Z+, Z-, or Scalar (no direction).

Analog Input Channel Settings for Devices that Support Strain Gages and Bridges

This section describes the settings that you can configure for the analog input channels on devices, such as the DT9838 and DT9829 modules, that support strain gages and bridges.

Enable Shunt Resistor

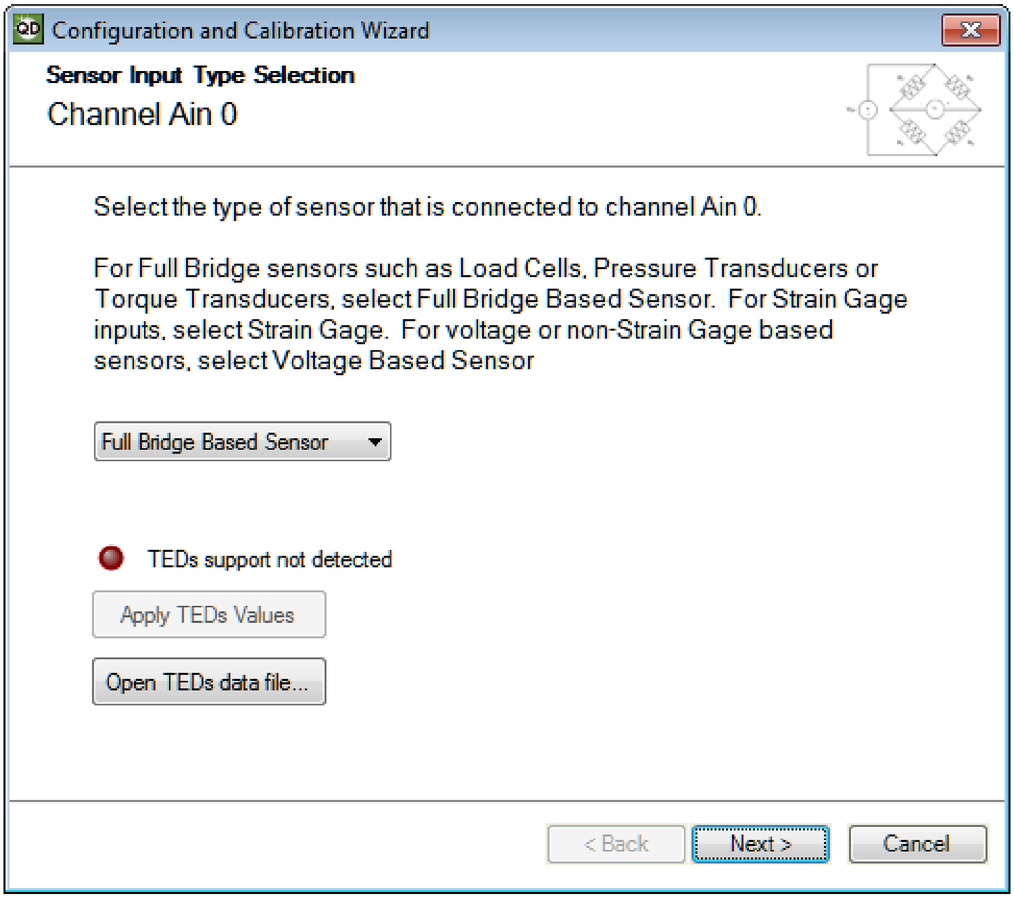
For devices that support an internal shunt resistor, you can quickly verify your wiring by enabling the internal shunt resistor and acquiring data from the strain gage or bridge. You can then determine whether the value that you measured is what you expected. (Be sure to disable the shunt resistor after you have verified your wiring.)

Note that this use of the shunt resistor is different from the use of the shunt resistor when performing shunt calibration. Shunt calibration is done using the Configuration and Calibration wizard, described next. During shunt calibration, the software automatically enables the shunt resistor for the calibration procedure and disables it once calibration has been performed.

Configuration and Calibration Wizard

You can use the **Configuration and Calibration** wizard to set up the parameters for the strain gage or bridge-based sensor and, if desired, calibrate it.

When you click the **Configure and Calibrate** button, a screen similar to the following is displayed:



Depending on your device, the following selections may be available:

- **Full Bridge Based Sensor** – for load cells, pressure transducers, torque transducers, and general-purpose bridge-based sensors
- **Strain Gage** – for strain gage sensors
- **Voltage Based Sensor** – for voltage inputs or non-strain gage inputs

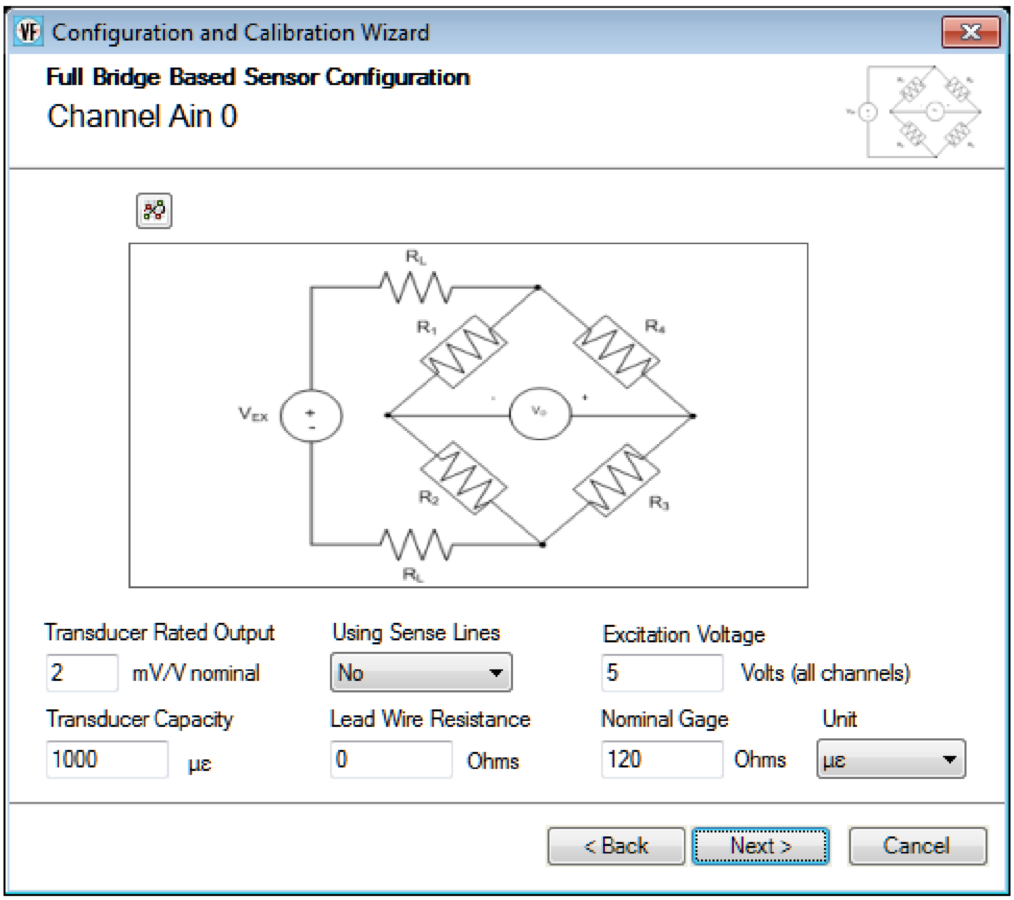
If your bridge-based sensor or strain gage supports TEDS (Transducer Electronic Data Sheet) data in hardware, the LED on this screen turns green. Click the **Apply TEDs Values** button to apply the TEDS data to the channel configuration.

If your sensor does not support TEDS data in hardware, the LED on this screen turns red. If a TEDS data file (virtual TEDS) is available for your sensor, click the **Open TEDs data file....** button, select the TEDS data file to read, and click **Open**. The TEDS values are applied to the channel configuration.

Click **Next** to continue to the next page of the wizard.

Parameters for Full Bridge Based Sensors

If the sensor type is **Full Bridge Based Sensor**, a screen similar to the following appears:

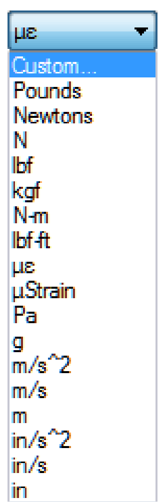


If you applied TEDS values from the sensor hardware or data file, the TEDS values are displayed in the fields on this screen.

You can configure the following parameters:

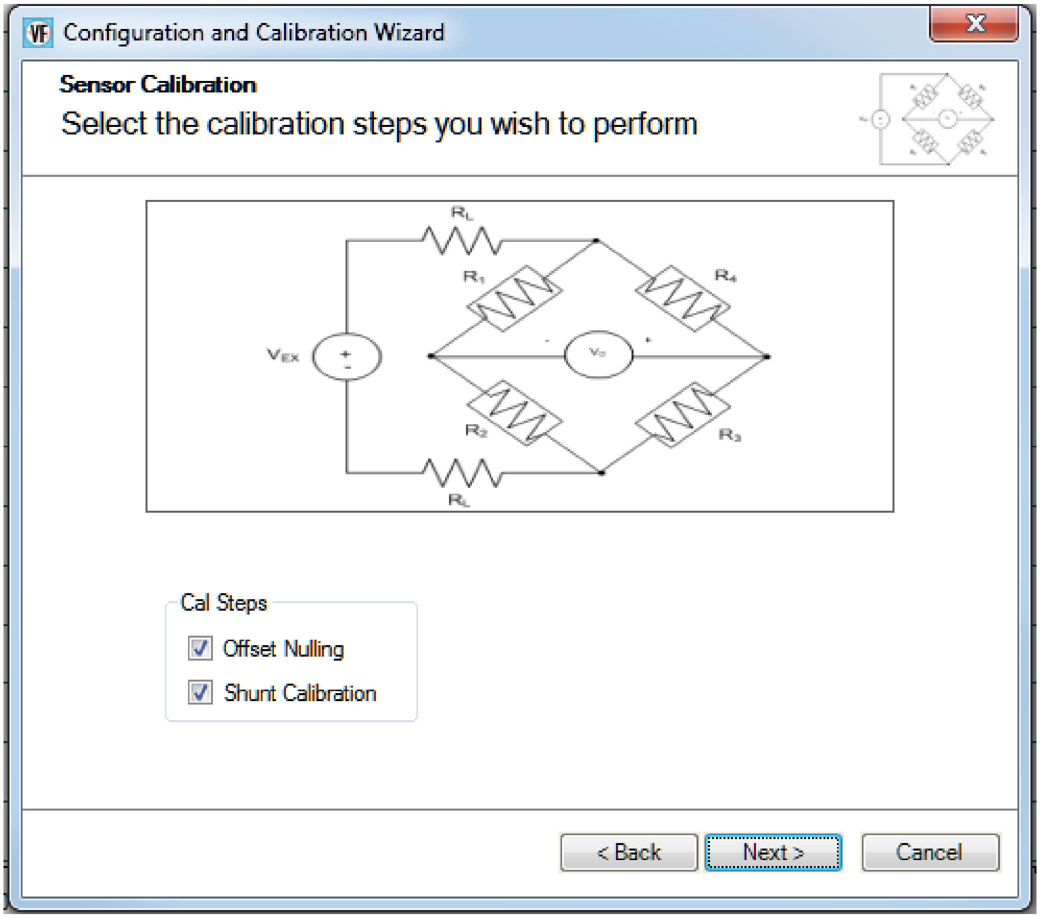
- **Transducer Rated Output** – Specify the rated output of the transducer in terms of mV/V excitation.
- **Transducer Capacity** – Specify the full-scale range of the transducer in its native engineering units.
- **Using Sense Lines** – For the selected analog input channel, enter **Yes** if remote sense lines are used or **No** if remote sense lines are not used in the wiring of the strain gage to the device.

  The wiring diagram changes depending on this selection. To see the wiring diagram of the strain gage to the device, click the  button.
- **Lead Wire Resistance** – For the selected analog input channel, specify the lead wire resistance, in ohms. Note that if remote sense lines are used, then this value is set to 0 automatically.
- **Excitation Voltage** – Specify the value of the excitation voltage that will be applied to all analog input channels on the device.
- **Nominal Gage** – For the selected analog input channel, enter the nominal gage resistance, in ohms, that is specified by the manufacturer of the sensor.
- **Unit** – For the selected analog input channel, select one of the following engineering units to represent the data, based on the sensor that is attached to the channel:



  If you select **Custom**, you can define your own engineering units.

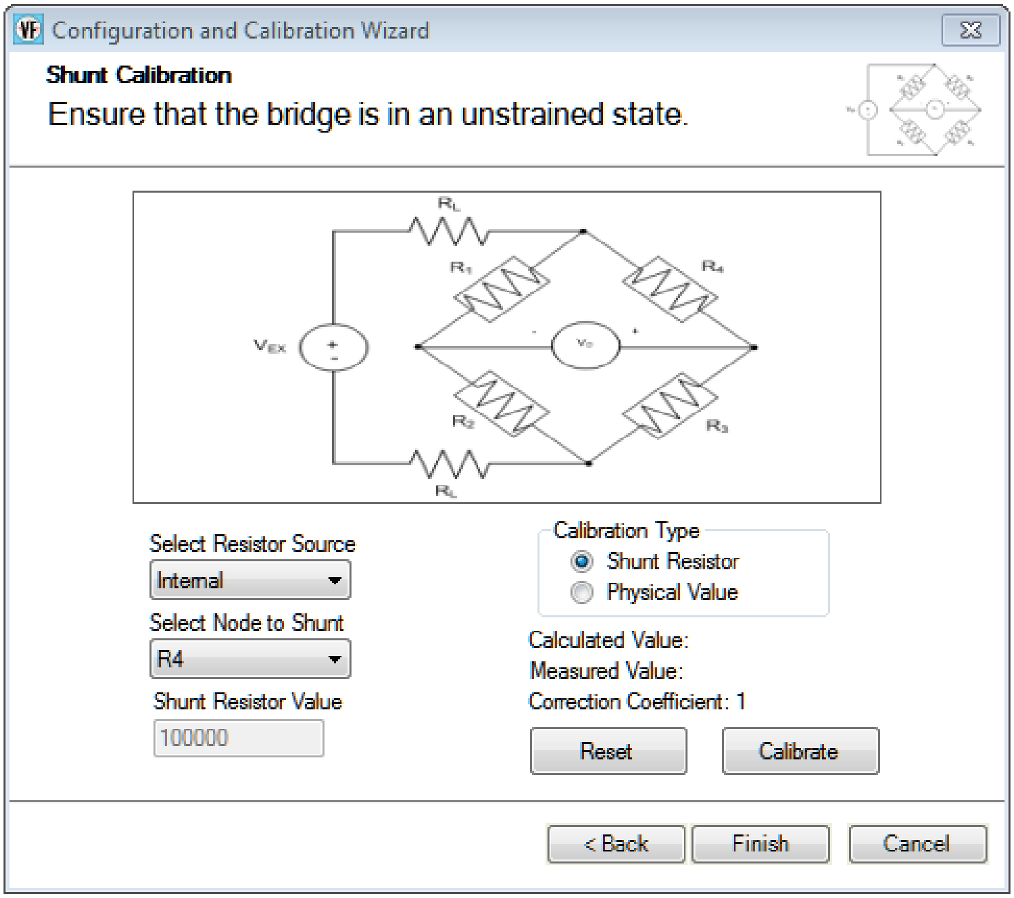
Click **Next** to continue to the calibration page of the wizard. A screen similar to the following appears:



A balanced bridge produces zero volts under ideal conditions with zero strain applied. In practice, however, the output of a bridge in an unstrained condition is offset from zero slightly due to imperfect matching of bridge resistances.

You can adjust the offset of the channel by performing offset nulling on the channel. To perform offset nulling, select the **Offset Nulling** checkbox, click **Next**, and then click **Calibrate**. Internally, this value is subtracted from all subsequent measurements before the voltage is converted to the selected engineering units. Note that you wish, you can reset the value by clicking the **Reset** button.

You can use shunt calibration to correct span errors in the measurement path. To perform shunt calibration, select the **Shunt Calibration** checkbox, and click **Next**. A screen similar to the following appears:



If you want to enter a value rather than use a shunt resistor to perform the shunt calibration procedure, select **Physical Value** as the **Calibration Type**. The page changes to allow you to enter a value. Enter the physical value to apply to the bridge, and click the **Calibrate** button.

If your device supports an internal shunt calibration resistor, you can use this resistor to perform shunt calibration as follows:

1. Select **Shunt Resistor** as the **Calibration Type**.
2. Select **Internal** for the **Select Resistor Source** combo box.
3. Ensure that the internal RSHUNT+ and RSHUNT– lines are connected across the gage and that no strain is applied to the specimen. (The internal shunt resistor is automatically enabled for this procedure and automatically disabled when the shunt calibration procedure is complete.)
4. For the **Select Node to Shunt** combo box, select the node (R1 to R4) that you want to shunt.
5. Click the **Calibrate** button.

If you want to use an external shunt calibration resistor to perform shunt calibration, do the following:

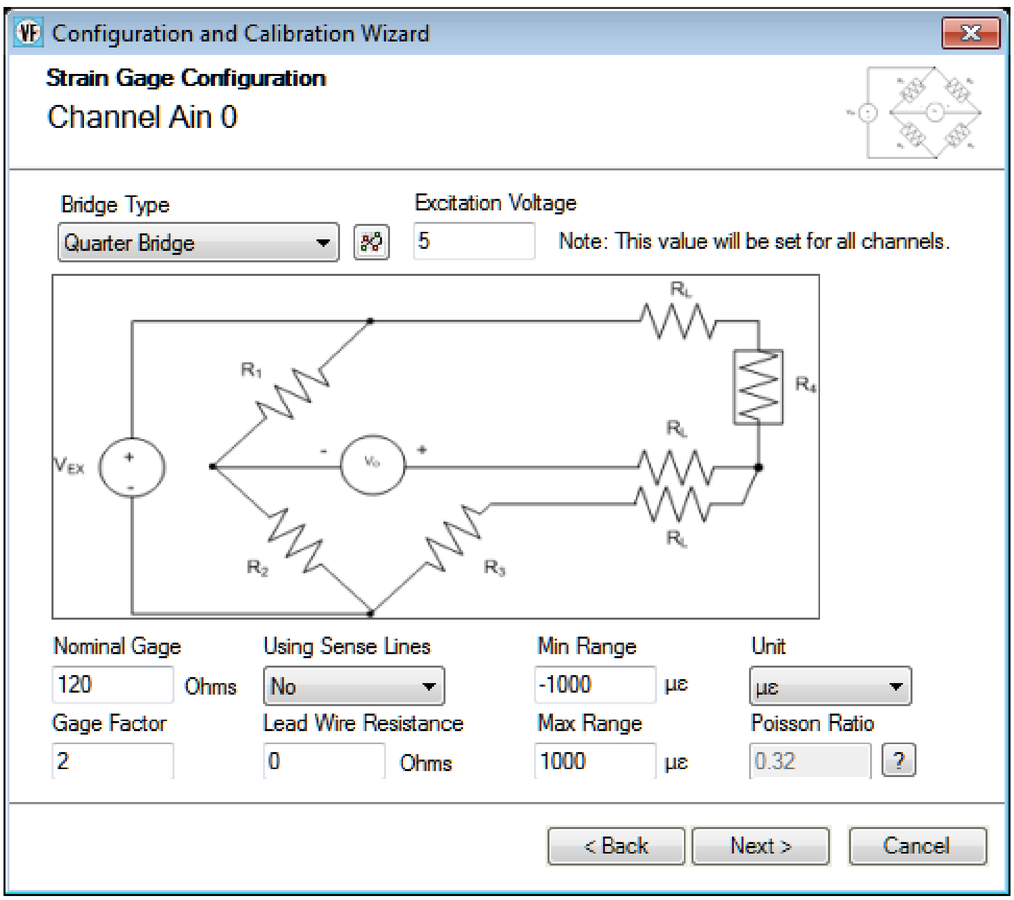
1. Select **Shunt Resistor** as the **Calibration Type**.
2. Select **External** for the **Select Resistor Source** combo box.
3. Ensure that the external shunt resistor is connected across the gage, and that no strain is applied to the specimen.
4. For the **Select Node to Shunt** combo box, select the node (R1 to R4) that you want to shunt.
5. Click the **Calibrate** button.

The calculated value, measured value, and correction coefficient are displayed. Note that you wish, you can reset the values by clicking the **Reset** button.

Click **Finish** when you are done using the wizard.

Parameters for Strain Gage Sensors

If the sensor type is **Strain Gage**, a screen similar to the following appears:



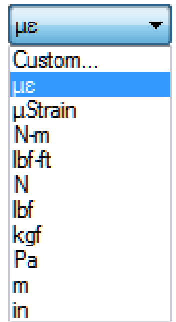
If you applied TEDS values from the sensor hardware or data file, the TEDS values are displayed in the fields on this screen.

You can configure the following parameters:

- **Bridge Type** – Select one of the following bridge configurations for each analog input channel; refer to the user's manual for your device for more information about each of these configuration types:
  - Quarter Bridge
  - Quarter Bridge Temp Comp
  - Full Bridge Axial Poisson
  - Full Bridge Bending
  - Full Bridge Bending Poisson
  - Half Bridge Bending
  - Half Bridge Poisson

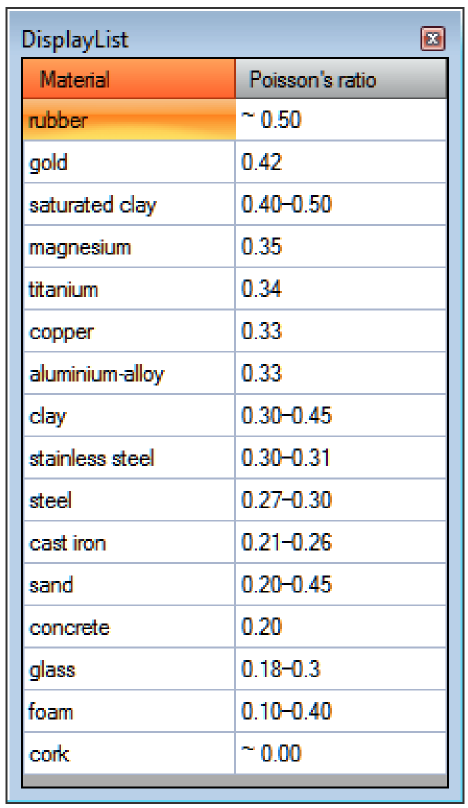
  A circuit diagram is shown for the selected configuration type. To see how to wire the strain gage to the device, click the button.
- **Excitation Voltage** – Specify the value of the excitation voltage that will be applied to all analog input channels on the device.
- **Nominal Gage** – For the selected analog input channel, enter the nominal gage resistance, in ohms, that is specified by the manufacturer of the strain gage.
- **Gage Factor** – For the selected analog input channel, enter the gage factor that is specified by the manufacturer of the strain gage.
- **Using Sense Lines** – For the selected analog input channel, enter **Yes** if remote sense lines are used or **No** if remote sense lines are not used in the wiring of the strain gage to the device.

  The wiring diagram changes depending on this selection. To see the wiring diagram of the strain gage to the device, click the button.
- **Lead Wire Resistance** – For the selected analog input channel, specify the lead wire resistance, in ohms. Note that if remote sense lines are used, then this value is set to 0 automatically.
- **Min Range** – For the selected analog input channel, specify the minimum strain value for the range.
- **Max Range** – For the selected analog input channel, specify the maximum strain value for the range.
- **Unit** – For the selected analog input channel, select one of the following engineering units to represent the data, based on the sensor that is attached to the channel:



If you select **Custom**, you can define your own engineering units.

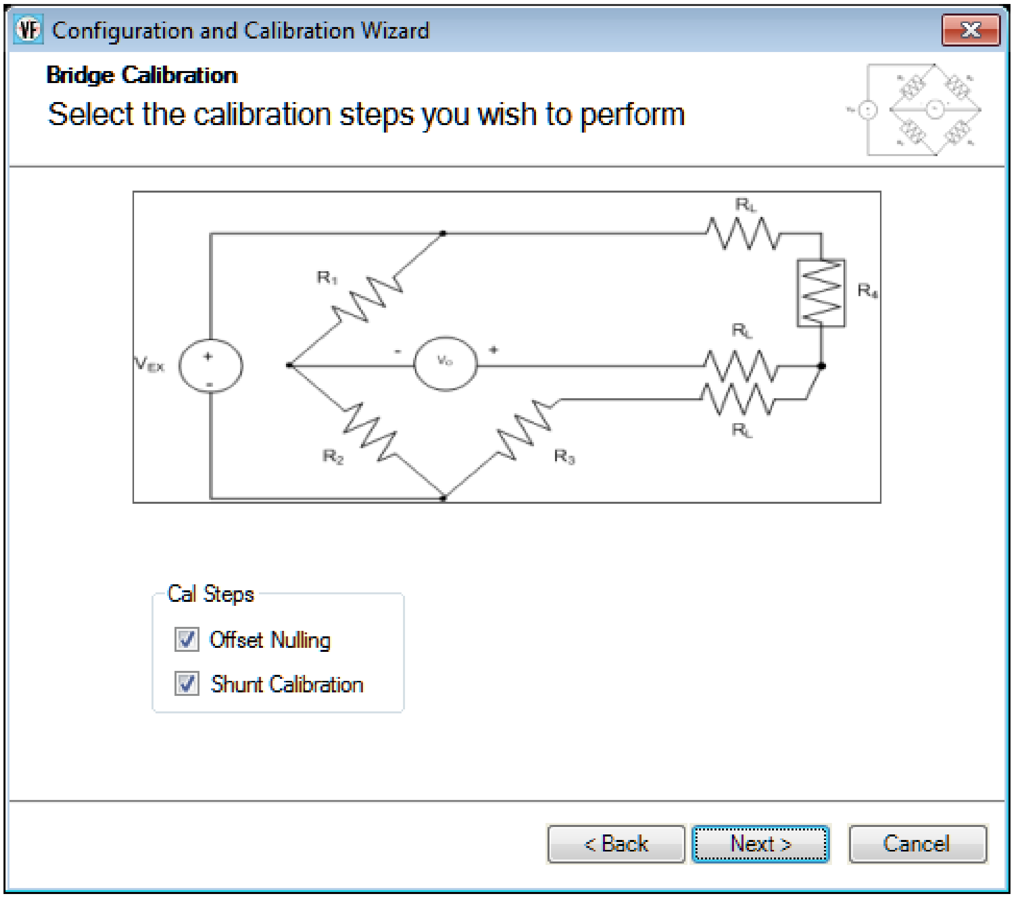
- **Poisson Ratio** – Depending on the bridge type, specify the Poisson ratio for each analog input channel. If you click the ? button, the following Poisson ratios are displayed for various materials:



DisplayList

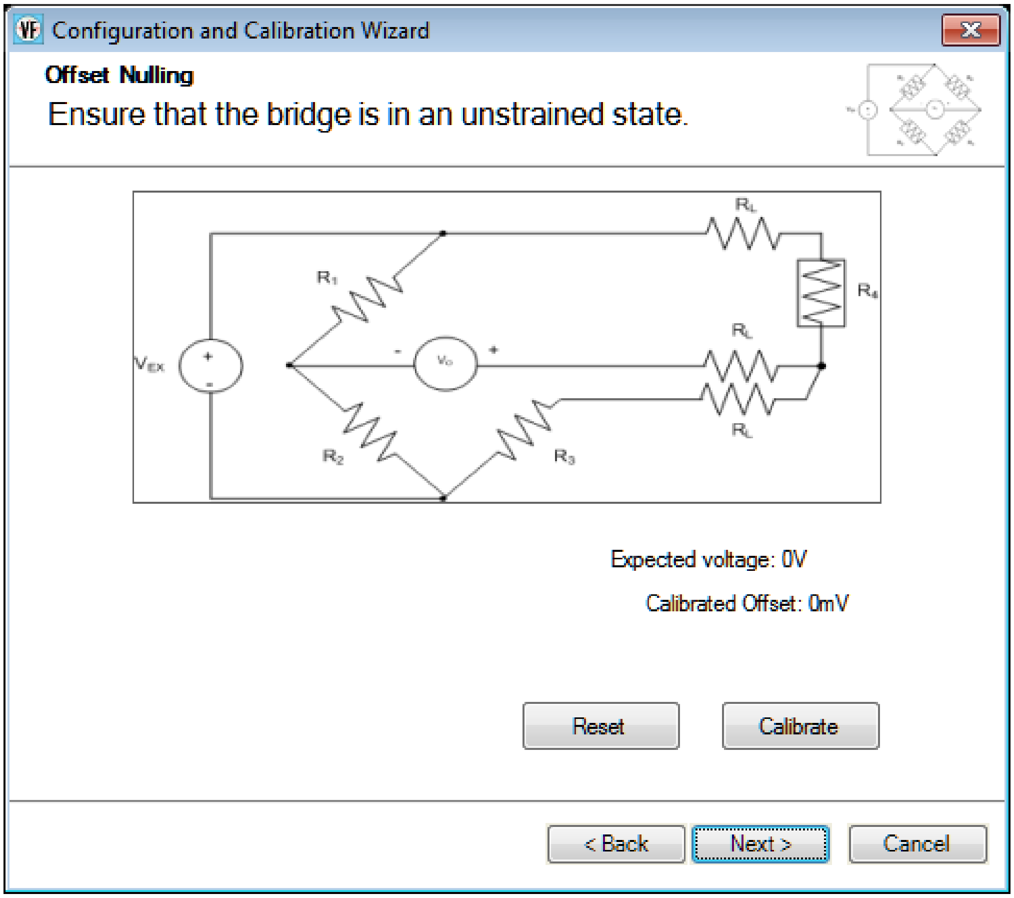
Material	Poisson's ratio
rubber	~ 0.50
gold	0.42
saturated clay	0.40–0.50
magnesium	0.35
titanium	0.34
copper	0.33
aluminium-alloy	0.33
clay	0.30–0.45
stainless steel	0.30–0.31
steel	0.27–0.30
cast iron	0.21–0.26
sand	0.20–0.45
concrete	0.20
glass	0.18–0.3
foam	0.10–0.40
cork	~ 0.00

Click **Next** to continue to the calibration page of the wizard. A screen similar to the following appears:



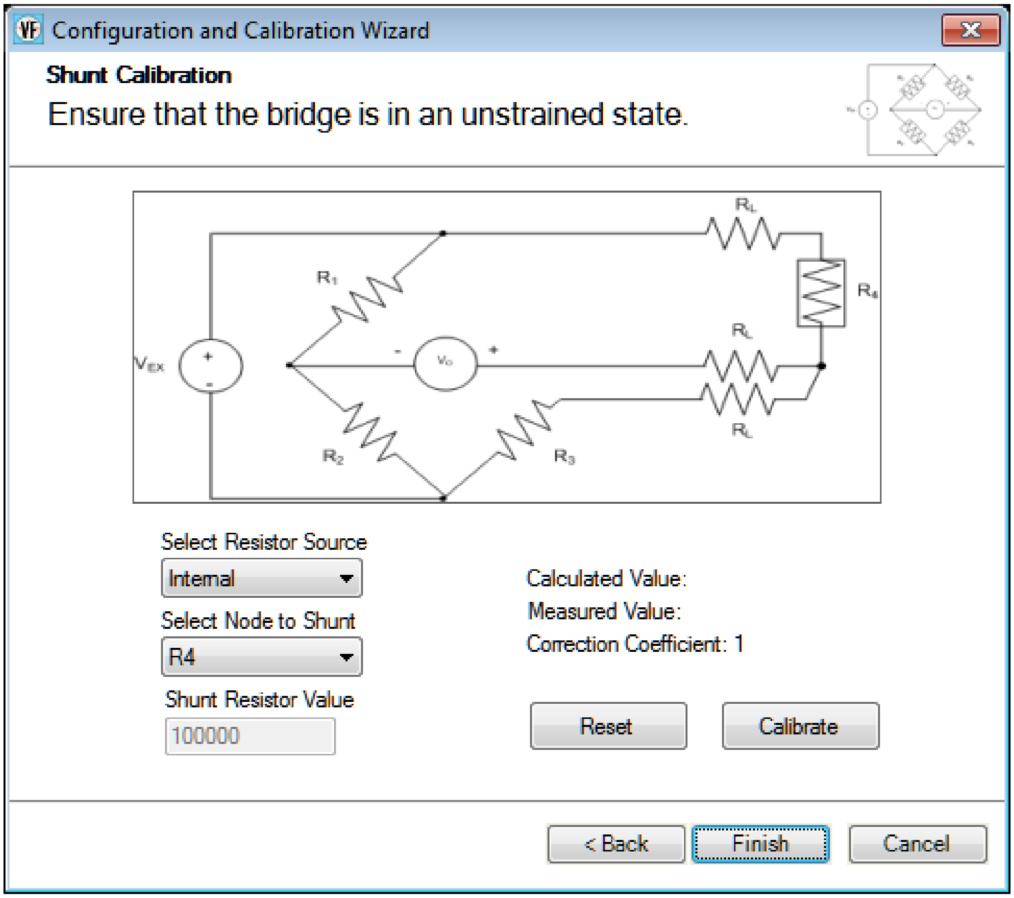
A balanced bridge produces zero volts under ideal conditions with zero strain applied. In practice, however, the output of a bridge in an unstrained condition is offset from zero slightly due to imperfect matching of bridge resistances.

You can adjust the offset of the channel by performing offset nulling on the channel. To perform offset nulling, select the **Offset Nulling** checkbox, and click **Next**. A screen similar to the following appears:



Click **Calibrate**. Internally, the calibrated offset is subtracted from all subsequent measurements before the voltage is converted to strain. Note that you wish, you can reset the value by clicking the **Reset** button.

You can use shunt calibration to correct span errors in the measurement path. To perform shunt calibration, select the **Shunt Calibration** checkbox, and click **Next**. A screen similar to the following appears:



If an internal shunt calibration resistor is provided by the device, you can use the internal shunt resistor to calibrate the strain gage. In this case, select **Internal** in the **Select Resistor Source** combo box, and ensure that the internal RSHUNT+ and RSHUNT– lines are connected across the gage and that no strain is applied to the specimen. (The internal shunt resistor is automatically enabled for this procedure and automatically disabled when the shunt calibration procedure is complete.) Then, for the **Select Node to Shunt** combo box, select the node that you want to shunt (the values depend on the bridge type that you selected).

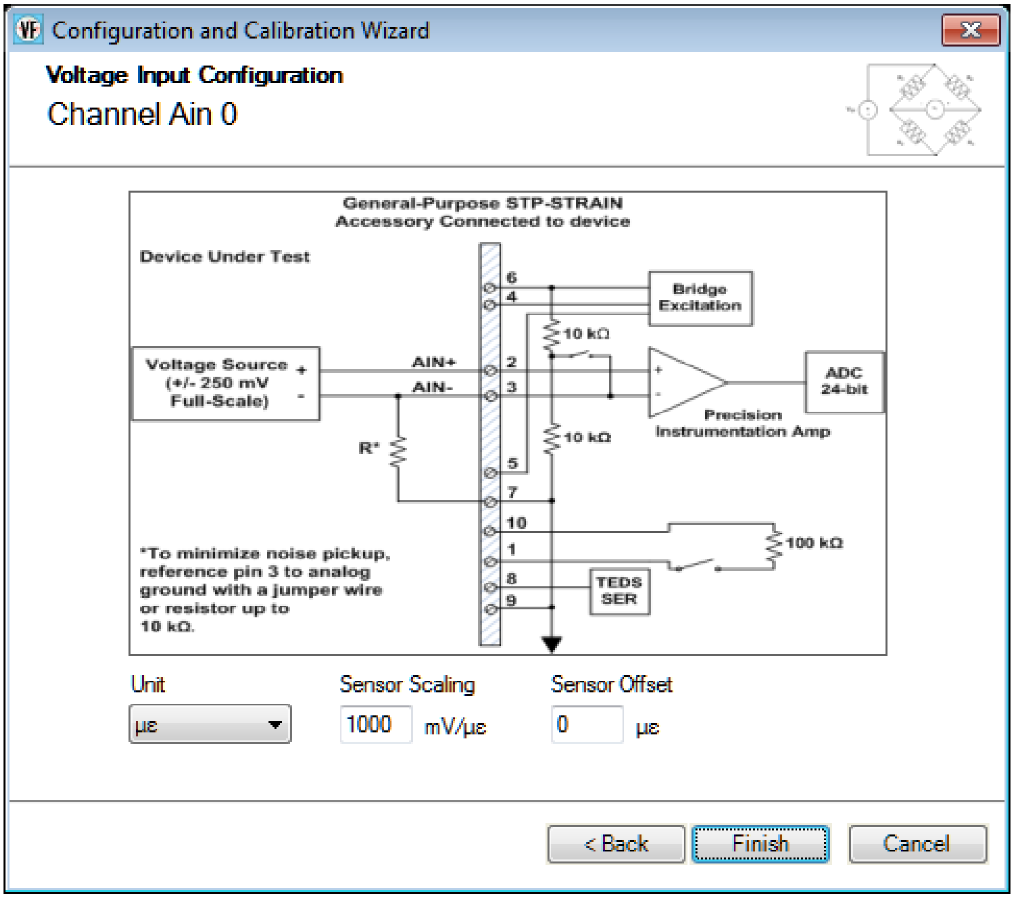
If you want to use a user-supplied external shunt calibration resistor, select **External** for the **Select Resistor Source** combo box, ensure that the external shunt resistor is connected across the gage, and that no strain is applied to the specimen. Then, for the **Select Node to Shunt** combo box, select the node that you want to shunt (the values depend on the bridge type that you selected).

Once the shunt calibration parameters have been configured, click **Calibrate**. The calculated value, measured value, and correction coefficient are displayed. Note that you wish, you can reset the values by clicking the **Reset** button.

Click **Finish** when you are done using the wizard.

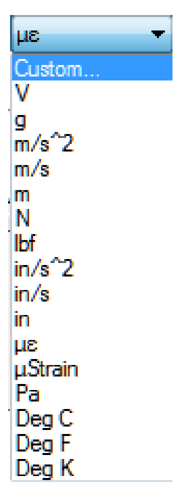
Parameters for Voltage Based Sensors

If the sensor type is **Voltage Based Sensor**, a screen similar to the following appears:



You can configure the following parameters:

- **Unit** – You can choose from one of the following engineering units to represent the data, based on the input that is attached to the channel:



If you select **Custom**, you can define your own engineering units.

- **Sensor Scaling** – Depending on the engineering units that you select, you can enter the number of millivolts per engineering unit (mV/EU) by which to scale the raw voltage.
- **Sensor Offset** – Depending on the engineering units that you select, you can enter an offset value to add to the scaled signal.

Click **Finish** when you are done using the wizard.

Point

If desired, you can enter a number representing the test point to which this channel corresponds. By default, values start at 1 and increment with each channel.

Digital Input Settings

If your device supports the ability to report the value of the digital input port in the analog input data stream, enable the digital input (DIN) channel.

The engineering units are automatically set to Bits.

Tachometer Settings

This section describes the settings that you can configure for tachometer channels that are supported in the analog input data stream.

Engineering Unit (EU)

For each tachometer channel, you can choose from one of the following engineering units to plot the data:

- **RPM** – Choose **RPM** if you want to plot the data from the tachometer in RPMs.
- **Sec** – Choose **Sec** if you want to plot the data from the tachometer in seconds.
- **Hz** – Choose **Hz** if you want to plot the data from the tachometer in Hertz.

Ticks/Rev

When you select **RPM** as the engineering unit for the tachometer, enter the number of ticks per revolution for your particular tachometer in the **Ticks/Rev** field. This allows the application to calibrate to the characteristics of your tachometer.

Max RPM

When you select **RPM** as the engineering unit for the tachometer, enter the maximum number of revolutions per minute for your particular tachometer in the **Max RPM** field.

This allows the application to calibrate to the characteristics of your tachometer.

RPM Multiplier

When you select **RPM** as the engineering unit for the tachometer, you can enter a value in the **RPM Multiplier** field to convert RPM into other units, such as Miles per Hour, Revolutions Per Second, or Hz. For example, to multiply by a factor of 2, enter 2 in the **RPM Multiplier** field. To divide by 2, enter a value of 0.5 in the **RPM Multiplier** field.

Max mSec/Hz

When you select **Sec** or **Hz** for the engineering units of the tachometer, you can enter the maximum value for the Y-axis in the **Max mSec/Hz** field.

Tach Edge

On the DT9837, the period of the tachometer input signal is not programmable; it is always measured from rising-to-rising edge. For other devices, select which period of the tachometer input signal to measure:

- **Rising –** Measures from the rising-to-rising edge of the tachometer input signal.
- **Falling –** Measures from the falling-to-falling edge of the tachometer input signal.

Note: Old tachometer values (from a previous tachometer operation) may be in the data stream until two tachometer edges of the specified type are detected.

Counter Settings

This section describes the settings that you can configure for counters that are supported in the analog input data stream.

Note that counters may be used differently depending on the device. For example, Table 2 describes how the counters are used the DT9837 Series devices. Refer to the documentation for your device for information on the counters provided by your device.

Table 2: Use of Counter 1 and Counter 2 on Each Device

Device	Counter 1 (Tach Counter)	Counter 2 (Gate Counter)
DT8837	Measure the frequency, period, or phase between two specified signals.	Measure the frequency, period, or phase between two specified signals.
DT9837	Not supported.	Not supported.
DT9837A	Measures the phase of the tachometer signal.	Not accessible on the device.
DT9837A-OEM	Measures the phase of the tachometer signal.	Measures the pulse width of the gate input signal or the phase between the gate signal and the A/D completion signal.
DT9837B	Measures the phase of the tachometer signal.	Measures the pulse width of the gate input signal or the phase between the gate signal and the A/D completion signal.
DT9837C	Not supported.	Not supported.
DT9838	Measure the frequency or period of the tachometer input signal.	Not supported.

Engineering Unit (EU)

For each counter, you can choose from one of the following engineering units to plot the data:

- **Sec** – Choose Sec if you want to plot the data from the counter in seconds.
- **Hz** – Choose Hz if you want to plot the data from the counter in Hertz.
- **Counts** – Some devices provide the **Counts** option. Choose **Counts** if you want to plot the number of counts that were counted by the counter/timer.

Max mSec/Hz

When you select **Sec** or **Hz** for the engineering units of the counter, you can enter the maximum value for the Y axis in the **Max mSec/Hz** field.

Start Edge

For each supported counter, select the signal edge that will start the measurement. You can choose from the following signals (note that not all options are provided by all devices):

- **A/D Complete**
- **Tach Rising** (on the DT9837 Series, this is supported by the Tach Counter only)
- **Tach Falling** (on the DT9837 Series, this is supported by the Tach Counter only)
- **Clock Rising**
- **Clock Falling**
- **Gate Rising** (on the DT9837 Series, this is supported on the Gate Counter only)
- **Gate Falling** (on the DT9837 Series, this is supported by the Gate Counter only)

Stop Edge

For some devices, you can specify the stop edge of the counter, which stops the measurement. You can choose from the following signals:

- **A/D Complete**
- **Tach Rising** (on the DT9837 Series, this is supported by the Tach Counter only)
- **Tach Falling** (on the DT9837 Series, this is supported by the Tach Counter only)
- **Gate Rising** (on the DT9837 Series, this is supported on the Gate Counter only)
- **Gate Falling** (on the DT9837 Series, this is supported by the Gate Counter only)

Quadrature Decoder Channel Settings

This section describes the settings that you can configure for quadrature decoder channels that are supported in the analog input data stream.

Engineering Unit (EU)

For each counter, you can choose from one of the following engineering units to plot the data:

- **Deg –** Choose **Deg** if you want to plot the data from the quadrature decoder in degrees.
- **Counts –** Choose **Counts** if you want to plot the number of counts that were counted by the quadrature decoder.

Pulses/Rev

Enter the number of pulses per revolution for your quadrature encoder. This number is device-specific; therefore, refer to the documentation for your quadrature encoder to determine this number.

Clock Prescale

Enter the pre-scale value that is used to filter the onboard clock. Using a clock pre-scale value can remove ringing edges and unwanted noise for more accurate data.

X4 Scaling

The scaling mode (X1 or X4 mode), determines the resolution of the quadrature encoder. If X4 Scaling is disabled, X1 mode is used. In X1 mode, the quadrature decoder counts the edges on the A signal input.

If X4 scaling mode is enabled, X4 scaling mode is used. In X4 mode, the quadrature decoder counts the edges on the A and B signal inputs. Therefore, if a quadrature encoder has 360 pulses per revolution, X1 mode yields 360 counts when the quadrature encoder is rotated 360 degrees. X4 mode yields four times the number of counts, or 1440 (360 x 4) when the quadrature encoder is rotated 360 degrees.

Index Mode

You can specify one of the following for the **Index Mode:**

- **High –** The Index input is enabled and the positional count is zeroed on the high edge of the Index signal.
- **Low** – The Index input is enabled and the positional count is zeroed on the low edge of the Index signal.
- **Disabled** – The Index input is disabled and the positional count is not zeroed. Note that if **X4 Scaling** is Enabled, **Index Mode** must be disabled.

Applying and Saving Changes

Click **Close** when you are through with this dialog box to apply the changes you made to the hardware and to reconfigure the application, as needed.

For information on saving and loading a configuration file, refer to page 43. For information about the backup configuration file, refer to page 44.

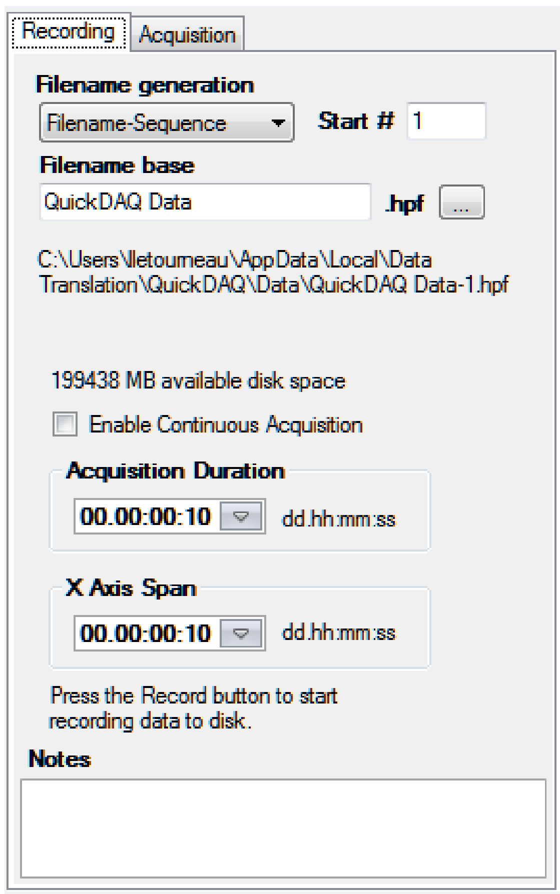
Configuring the Settings of the Recording Tab

Before starting acquisition, configure how the data is recorded using the Recording tab of the Acquisition Config window.

The Recording tab is different depending on the interface mode you are using, as described below.

Recording Tab When Using the Data Logger Interface

If you are using the Data Logger interface, the Recording tab appears as follows:



The Recording tab of the Data Logger interface contains these settings:

- **Filename generation** – Specify how to create the name of the file. You can choose one of the following options:
  - Filename – The specified filename is created when you click the **Record** toolbar button (). Each time you click the **Record** toolbar button, you will be prompted to overwrite the file. QuickDAQ Data.hpf is an example of a filename that was created using the Filename option.

- Filename-Sequence – The specified filename is created and a starting number that you define is appended to the filename when you click the **Record** toolbar button (). Each time you click the **Record** toolbar button, the number that is appended to the filename is incremented by one. QuickDAQ Data-1.hpf is an example of a filename that was created using the Filename-Sequence option.
- Filename-DateTime – The specified filename is created and the current date and time is appended to the filename when you click the **Record** toolbar button (). Each time you click the **Record** toolbar button, the filename with the current date and time is created. QuickDAQ Data-2012-11-15_02-15-31-PM.hpf is an example of a filename that was created using the Filename-DateTime option.

- **Filename base** – Specify the name of the High Performance binary file (.hpf) in which to store the data that is recorded when you click the **Record** toolbar button (). The default path is shown. You can change this path, if desired. The application saves the data file path and is uses this path by default.
- **Enable Continuous Acquisition** – Determines how data acquisition and processing stops.

  Check this checkbox to enable continuous acquisition. In this mode, you must click the **Stop** button or hold down the **Shift** key and press the **F5** key to stop acquisition/processing.

  If the **Enable Continuous Acquisition** checkbox is unchecked, you must specify how long to acquire data using the Acquisition Duration.

Note: Continuous acquisition is supported by DT8824, DT8837, and MEASURpoint devices when using the Data Logger interface.

If a device is acquiring data continuously and unread data in the hardware circular memory buffer starts to be overwritten, acquisition is stopped and an overrun error is reported.

- **Acquisition Duration** – (Available only if the **Enable Continuous Acquisition** checkbox is unchecked). You can either enter a value that specifies the number of days (dd), number of hours (hh), minutes (mm), and seconds (ss) that you want to acquire data from the device, or select one of the following predefined values from the combo box:
  - 10 Seconds
  - 1 Minute
  - 5 Minutes
  - 10 Minutes
  - 30 Minutes
  - 1 Hour
  - 5 Hours
  - 12 Hours
  - 1 Day
  - 2 Days

If the **Enable Continuous Acquisition** checkbox is unchecked, acquisition stops automatically when the specified duration elapses.

- **X Axis Span –** You can either enter a value that specifies the number of days (dd), number of hours (hh), minutes (mm), and seconds (ss) that you want to use for the span of the x-axis of the plots on the Channel Plot window, or select the default value from the combo box.

Note: If you specify a value for the x-axis span that is less than the acquisition duration, the x-axis scrolls using the span you specified.

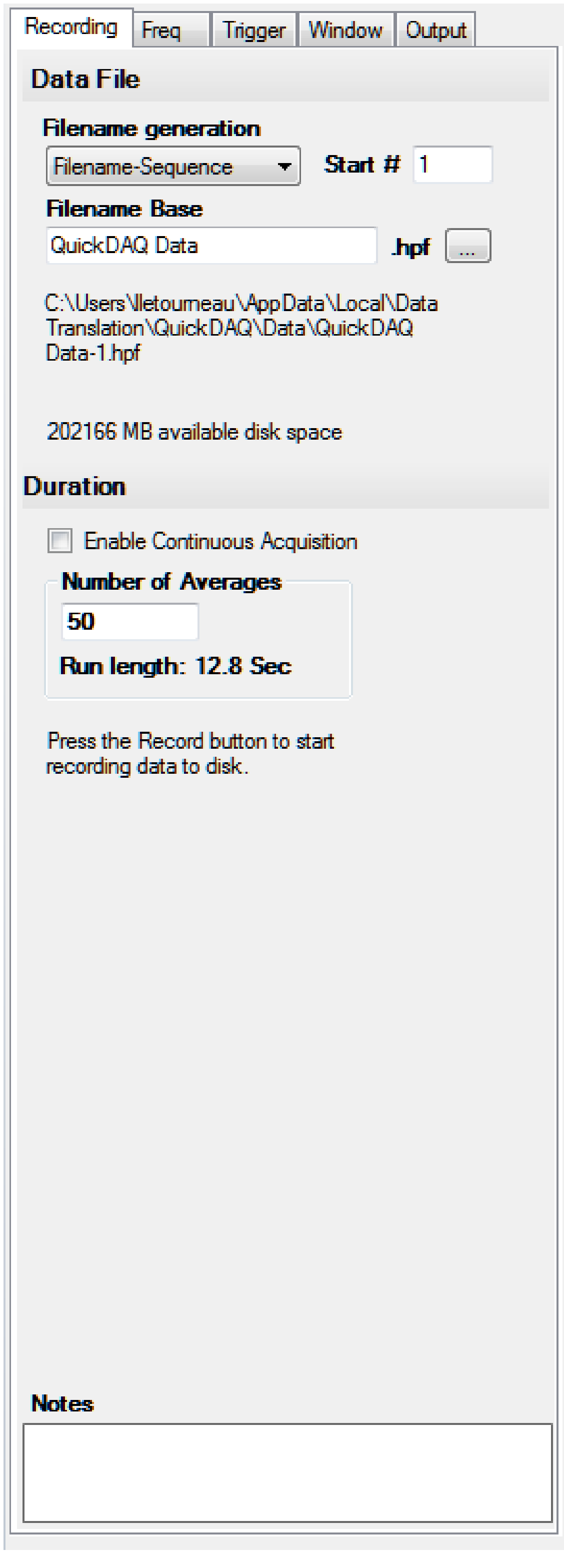
- **Notes –** Use this field to add any notes as desired. If you choose to export the data to ASCII, this information is also exported.

Based on the information that you enter for the **Recording** tab and the **Acquisition** tab (for the Data Logger interface), or **Freq** tab (for the FFT Analyzer interface), the software configures the hardware and displays the following configuration settings used by the device:

- The actual sample rate used by the device.
- The actual sample interval used by the device.
- The actual number of scans to acquire.

Recording Tab When Using the FFT Analyzer Interface

If you are using the FFT Analyzer interface, the Recording tab appears as follows:



The Recording tab of the FFT Analyzer interface contains these settings:

- **Filename generation** – Specify how to create the name of the file. You can choose one of the following options:
  - Filename – The specified filename is created when you click the **Record** toolbar button (Record). Each time you click the **Record** toolbar button, you will be prompted to overwrite the file. QuickDAQ Data.hpf is an example of a filename that was created using the Filename option.
  - Filename-Sequence – The specified filename is created and a starting number that you define is appended to the filename when you click the **Record** toolbar button (Record). Each time you click the **Record** toolbar button, the number that is appended to the filename is incremented by one. QuickDAQ Data-1.hpf is an example of a filename that was created using the Filename-Sequence option.
  - Filename-DateTime – The specified filename is created and the current date and time is appended to the filename when you click the **Record** toolbar button (Record). Each time you click the **Record** toolbar button, the filename with the current date and time is created. QuickDAQ Data-2012-06-25_02-39-59-PM.hpf is an example of a filename that was created using the Filename-DateTime option.
- **Filename base**– Specify the name of the High Performance binary file (.hpf) in which to store the data that is recorded when you click the **Record** toolbar button (Record). The default path is shown. You can change this path, if desired. The application saves the data file path and is uses this path by default.
- **Enable Continuous Acquisition** – Determines how data acquisition and processing stops.

  Check this checkbox (available on USB modules only) to enable continuous acquisition. In this mode, you must click the **Stop** button or hold down the **Shift** key and press the **F5** key to stop acquisition/processing.

  If the **Enable Continuous Acquisition** checkbox is unchecked, specify the number of averages that the application will compute before stopping acquisition/processing.

Note: When using the FFT Analyzer interface, continuous acquisition is not supported by DT8824 and DT8837 devices due to FFT data block size requirements and the possibility of network traffic preventing continuous streaming. Continuous acquisition is supported by MEASURpoint devices, however, due to their slow sampling rates.

If you need continuous acquisition on a DT8824 or DT8837 device, use the Data Logger interface.

If a device is acquiring data continuously and unread data in the hardware circular memory buffer starts to be overwritten, acquisition is stopped and an overrun error is reported.

- **Number of Averages –** (Available only if the **Enable Continuous Acquisition** checkbox is unchecked). Specify the number of records to acquire, process, and average.

  When the **Enable Continuous Acquisition** checkbox is unchecked, acquisition stops automatically when the specified number of averages has been reached. The software displays the total length of time for the specified number of records to be acquired, processed, and averaged), in seconds.

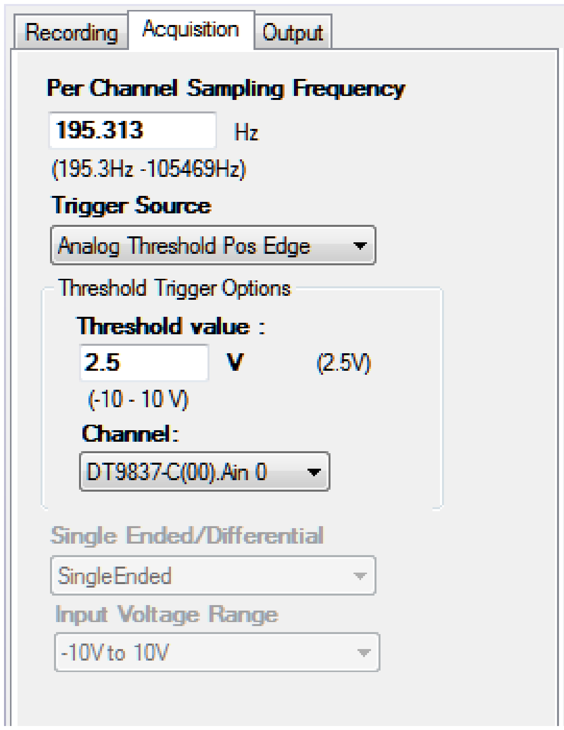
  How acquisition starts depends on the trigger settings, described on page 131.

- **Notes –** Use this field to add any notes as desired. If you choose to export the data to ASCII, this information is also exported.

Based on the settings of the Acquisition Config window, the available disk space is displayed and is updated as the recording progresses.

Configuring the Settings of the Acquisition Tab (Data Logger Interface Only)

If you are using the Data Logger interface, use the **Acquisition** tab of the Acquisition Config window to set up parameters that apply to the A/D subsystem of the device, including the sampling frequency, trigger source, filter type, temperature units, channel configuration, and/or input voltage range.



Sampling Frequency

Enter the sampling frequency per channel for the device. For simultaneous devices, the total throughput is the same as the per channel sampling frequency, since all channels are acquired at the same time.

For multiplexed devices, multiply the per channel sampling frequency by the number of enabled channels to determine the total throughput. For example, if five analog input channels are enabled on the multiplexed device and the per channel sampling frequency is 200 Hz, the total throughput is 1000 Hz (200 Hz x 5).

Refer to the user's manual for your device for information on the supported sampling frequencies.

Trigger Source

Specify the trigger source that starts acquisition on the device. Examples of trigger sources that may be supported for starting acquisition on the device include the following:

- **Software –** The device starts acquiring scans when you click the **Record** button or press the **F5** key. Once acquisition starts, all triggers are ignored. Acquisition stops when the conditions defined in the **Recording** tab, described on page 119, are met.
- **External TTL Pos Edge –** The device starts acquisition when it detects a rising-edge voltage on the external trigger line. Once triggered, the state of external trigger line is ignored. The process continues until the conditions defined in the **Recording** tab, described on page 119, are met.
- **External TTL Neg Edge –** The device starts acquisition when it detects a negative-edge voltage on the external trigger line. Once triggered, the state of external trigger line is ignored. The process continues until the conditions defined in the **Recording** tab, described on page 119, are met.
- **Analog Threshold Pos Edge –** The device starts acquisition when it detects a positive edge, analog threshold trigger event. You can specify the threshold value on which to trigger and the channel to use for the analog threshold trigger.

  You cannot use this trigger source if the following condition are true on the Output tab: the analog output channel is enabled and the start method is On Acquisition Start.
- **Analog Threshold Neg Edge –** The device starts acquisition when it detects a negative edge, analog threshold trigger event. You can specify the threshold value on which to trigger and the channel to use for the analog threshold trigger.

  You cannot use this trigger source if the following condition are true on the Output tab: the analog output channel is enabled and the start method is On Acquisition Start.

For example, assume that you specified the trigger source as **External TTL Pos Edge**, set the **Acquisition Duration** field on the **Recording** tab to **5 Seconds**, and clicked the **Record** toolbar button (

) or pressed the **F5** key. In this example, the device waits until it detects a rising-edge TTL pulse on external trigger line before acquiring data. When five seconds of data are acquired, acquisition stops.

Refer to the user's manual for your device for information on the supported trigger sources.

Filter Type

If supported by your device, you can configure one of the following filter types:

- **Moving average –** (The default filter setting.) This filter type removes unwanted noise from your measurements and provides a compromise of filter functionality and response time. This filter can be used in any application.

  This low-pass filter takes the previous 16 samples, adds them together, and divides by 16.
- **None –** No filter. Provides fast response times, but the data may be difficult to interpret. Use when you want to filter the data yourself.

The data is represented exactly as is when it comes out of the Delta-Sigma A/D converters. Note that Delta-Sigma converters provide substantial digital filtering above the Nyquist frequency.

Generally, the only time it is desirable to use the **None** filter setting is if you are using fast responding thermocouples, sampling them at higher speeds (> 1 Hz), and need as much response speed as possible.

Temperature Units

For thermocouple, RTD, and thermistor measurements, you can select the temperature units (Celsius, Fahrenheit, and Kelvin) into which the voltage data is converted. This setting is applied to all the temperature channels of the device.

You can see which temperature unit will be displayed for each channel in the Configure Devices dialog box, described on page 90.

Single-Ended or Differential Channel Configuration

If your device supports it, you can select the following channel configuration for the A/D subsystem:

- **Single-ended** – Choose this configuration when you want to measure high-level signals, noise is not significant, the source of the input is close to the device, and all the input signals are referred to the same common ground. Refer to the user’s manual for your device for wiring information. Refer to the user’s manual for your device for wiring information.
- **Differential** – Choose this configuration when you want to measure thermocouple or low-level signals, you are using an A/D converter with high resolution, noise is a significant part of the signal, or common-mode voltage exists. Refer to the user’s manual for your device for wiring information. Refer to the user’s manual for your device for wiring information.

Input Voltage Range

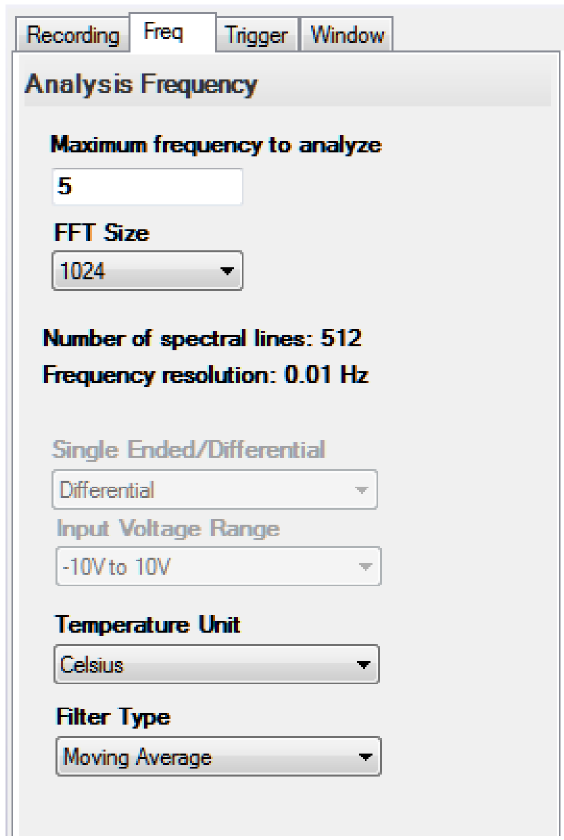
If your device supports multiple input ranges per channel, you configure the input range using the **Configure Devices** dialog, described on page 88.

If your device supports multiple input voltage ranges and/or gains that apply to the entire subsystem, you can select which effective input range to use for the A/D subsystem using the **Input Voltage Range** drop-down box. For example, if your device supports and input range of ±10 V with gains of 1 and 2, two effective input ranges are supported: ±10 V (±10 V divided by a gain of 1) and ±5 V (±10 V divided by a gain of 2).

Refer to the user’s manual for your device for specific information on the input voltage ranges and gains that are supported.

Configuring the Settings of the Freq Tab (FFT Analyzer Interface Only)

If you are using the FFT Analyzer interface, configure the settings of the **Freq** tab of the Acquisition Config window before starting acquisition; these settings are used to program the data acquisition device:



The settings of the **Freq** tab are described in the following subsections. Based on the information that you enter for the **Recording** tab and the **Freq** tab, the software configures the hardware and displays the following configuration settings used by the device:

- The actual sample rate used by the device.
- The actual sample interval used by the device.
- The actual number of scans to acquire.
- The actual number of averages to perform.

Maximum Frequency to Analyze

Specify the bandwidth or span of the input signals that you want to analyze. Internally, the software multiplies this value by 2 to set the sampling rate of the device.

FFT Size

Specify how many data points to use when generating the FFTs for the analog input channels.

The number of spectral lines and the frequency resolution are calculated and displayed when the maximum frequency to analyze and the FFT size are changed.

Single-Ended or Differential Channel Configuration

If your device supports it, you can select the following channel configuration for the A/D subsystem:

- **Single-ended** – Choose this configuration when you want to measure high-level signals, noise is not significant, the source of the input is close to the device, and all the input signals are referred to the same common ground.
- **Differential** – Choose this configuration when you want to measure thermocouple or low-level signals, you are using an A/D converter with high resolution, noise is a significant part of the signal, or common-mode voltage exists.

Input Voltage Range

If your device supports multiple input ranges per channel, you configure the input range using the **Configure Devices** dialog, described on page 88.

If your device supports multiple input voltage ranges and/or gains that apply to the entire subsystem, you can select which effective input range to use for the A/D subsystem using the **Input Voltage Range** drop-down box. For example, if your device supports and input range of ±10 V with gains of 1 and 2, two effective input ranges are supported: ±10 V (±10 V divided by a gain of 1) and ±5 V (±10 V divided by a gain of 2).

Refer to the user's manual for your device for specific information on the input voltage ranges and gains that are supported.

Temperature Units

For thermocouple, RTD, and thermistor measurements, you can select the temperature units (Celsius, Fahrenheit, and Kelvin) into which the voltage data is converted. This setting is applied to all the temperature channels of the device.

You can see which temperature unit will be displayed for each channel in the Configure Devices dialog box, described on page 90.

Filter Type

If supported by your device, you can configure one of the following filter types:

- **Moving average –** (The default filter setting.) This filter type removes unwanted noise from your measurements and provides a compromise of filter functionality and response time. This filter can be used in any application.

  This low-pass filter takes the previous 16 samples, adds them together, and divides by 16.

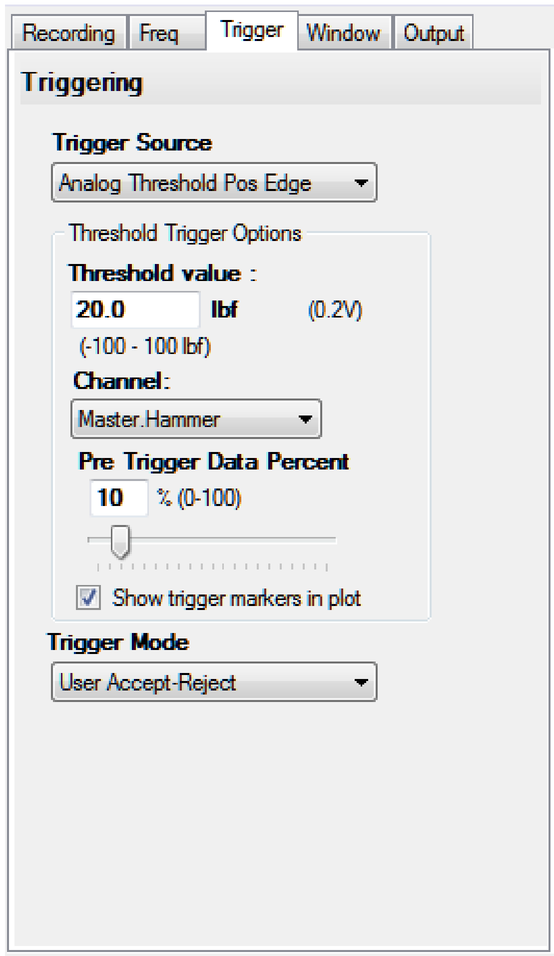
- **None –** No filter. Provides fast response times, but the data may be difficult to interpret. Use when you want to filter the data yourself.

  The data is represented exactly as is when it comes out of the Delta-Sigma A/D converters. Note that Delta-Sigma converters provide substantial digital filtering above the Nyquist frequency.

  Generally, the only time it is desirable to use the **None** filter setting is if you are using fast responding thermocouples, sampling them at higher speeds (> 1 Hz), and need as much response speed as possible.

Configuring the Settings of the Trigger Tab (FFT Analyzer Interface Only)

If you are using the FFT Analyzer interface, use the **Trigger** tab of the Acquisition Config window to set up the trigger source that starts acquisition on the device and the trigger mode.



Trigger Source

The following trigger sources are supported for starting acquisition on the device:

Note: In multiple device configurations, the trigger source is the trigger that starts acquisition on the master device.

- **Free Run** – The device starts acquiring contiguous, gap-free, records when you click the **Record** button or press the **F5** key.

  If the **Enable Continuous Acquisition** checkbox on the **Recording** tab, described on page 119, is checked, you must click the **Stop** button or hold down the **Shift** key and press the **F5** key to stop acquisition/processing.

  If the **Enable Continuous Acquisition** checkbox on the **Recording** tab is unchecked, acquisition stops automatically when the specified number of records have been averaged.

- **External TTL Pos Edge** – The device acquires one record each time it detects a rising-edge transition on the external digital (TTL) signal connected to the Trigger In input. The process continues until the conditions defined in the **Recording** tab, described on page 119, are met.

  This trigger provides a threshold voltage of 2 V with 0.5 V of hysteresis.

- **Analog Threshold Pos Edge** – The device acquires one record each time it detects a positive edge, analog threshold trigger event. Refer to page 133 or the threshold trigger options.

  You cannot use this trigger source if the following condition are true on the Output tab: the analog output channel is enabled and the start method is On Acquisition Start.

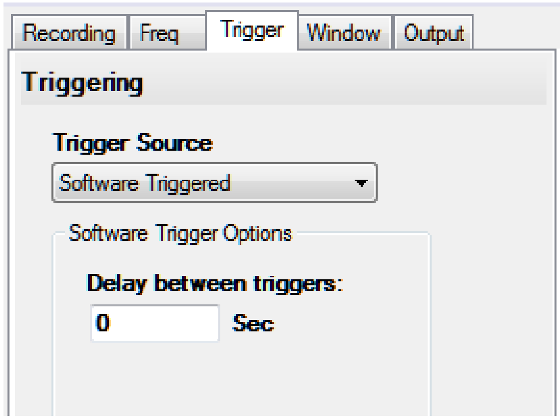
- **Analog Threshold Neg Edge** – The device acquires one record each time it detects a negative edge, analog threshold trigger event. Refer to page 133 or the threshold trigger options.

  You cannot use this trigger source if the following condition are true on the Output tab: the analog output channel is enabled and the start method is On Acquisition Start.

- **LXI0 to LXI5** – Supported on DT8837 devices only, the device acquires one record each time it detects the specified LXI trigger on the Trigger Bus. (This trigger source applies to the master device only.)

- **LAN0 to LAN7** – Supported on DT8837 devices only, the device acquires one record each time it detects the specified LAN trigger event.

- **Software Triggered** – The device starts acquiring records based on an internal software trigger. If Auto Retrigger as the trigger mode, you can also specify a delay, in seconds, to wait before the next software triggered record/frame is acquired. By default, the delay is 0 s, which is equivalent to no delay. Refer to page 134 for more information on Auto Retrigger mode.



This mode is typically used in conjunction with the **Output** tab when you want to use an analog output signal to generate burst random and burst chirp operations; in these applications, the full output frequency range and duration is contained within the time of a single analysis frame or record.

Note: When you use the software triggered option, the horizontal axis of the Channel Plot shows relative time within the average, not absolute time, as some overhead is required to start each acquisition when the trigger occurs.

If the **Enable Continuous Acquisition** checkbox on the **Recording** tab, described on page 122, is checked, you must click the **Stop** button or hold down the **Shift** key and press the **F5** key to stop acquisition/processing.

If the **Enable Continuous Acquisition** checkbox on the **Recording** tab is unchecked, acquisition stops automatically when the specified number of records have been averaged.

Threshold Trigger Options

If you selected the analog threshold trigger as the trigger source, the following options are available for configuring the threshold trigger:

- **Threshold Value** – Enter the user-specified threshold value in the units of the threshold channel that you select. (QuickDAQ supports a threshold voltage between 0.8 V and 9.8 V for the DT8837, and –9.8 V to +9.8 V for the DT9837 Series modules.)
- **Channel** – Specify the channel to use for the analog threshold trigger. For the DT8837, this is analog input channel 0. For the DT9837 Series modules, this can be any enabled analog input channel.

- **Pre Trigger Data Percent –** Specify the percentage of data (0 to 100) that you want to acquire prior to the start of the threshold trigger event. This allows you to acquire the entire pulse and get a reliable trigger. If you specify 0% as the pre-trigger data, only the data after the trigger is captured.

  Note that an entire record of data is always captured when the threshold trigger event occurs; however, the percentage of pre-trigger data that you requested is not guaranteed, as the trigger event may occur earlier than expected.

- **Show trigger markers in plot –** Select this checkbox if you want to see when the trigger occurred in the trace for the stimulus (reference) signal. The trigger marker is shown in red if you do not get all the pre-trigger data that you specified. If you get all the pre-trigger data that you specified, the trigger marker is displayed in the same color as the trace color.

Note: Ensure that the trace color is not red if you want to use the color of trigger marker to indicate whether or not you captured all the pre-trigger data.

Trigger Mode

If the trigger source is not **Free Run**, you can specify how you want to accept each average by choosing one of the following trigger modes:

- **Auto Retrigger –** Select this trigger mode if you want the application to accept the data after each average automatically.
- **User Accept-Reject –** Select this trigger mode if you want to determine whether to keep the data after each average.

  Once you start recording when using this trigger mode, you must click the **Accept** button (to keep the data) or click the **Reject** button (if you do not want to keep the data) after each average. The **Accept** and **Reject** buttons are located in the Acquisition Status window under the Channel Plot and Channel Display windows.

Note: Once a record has been acquired and the software is waiting for you to click the **Accept** or **Reject** button, the Channel Plot window controls are enabled. This allows you to review the data in detail before you accept or reject it.

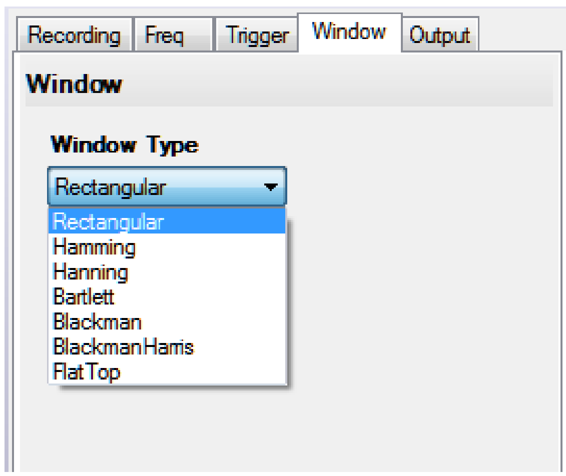
To start the next average, ensure that you provide the appropriate trigger.

Configuring the Settings of the Window Tab (FFT Analyzer Interface Only)

If you are using the FFT Analyzer interface, specify the windowing function to apply to the FFT channels to reduce spectral leakage errors using the **Window** tab of the Acquisition Config window. The contents of the Window tab are different for the FFT Analysis and Advanced FFT Analysis options.

Window Tab for the FFT Analysis Option

If you are using the FFT Analysis option, the Window tab appears as follows:



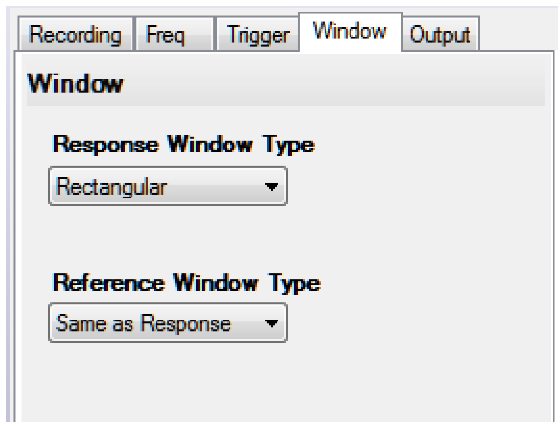
The following windowing types are available when generating FFTs; this is a global setting that is applied to all FFT channels:

- **Rectangular –** No window. Recommended for use when your signal content contains random noise (white noise) or closely spaced sine waves.
- **Hamming –** Bell-shaped cosine window optimized to minimize the maximum (nearest) side lobe. Recommended for use when your signal content consists of closely spaced sine waves.
- **Hanning –** Bell-shaped cosine window typically used to analyze continuous signals. Recommended for use when your signal content contains sine waves, a combination of sine waves, vibration data consisting of narrowband random signals, or unknown content.
- **Bartlett –** Triangular window.
- **Blackman –** Similar to the Hanning and Hamming windows, this window has slightly wider central lobes and less sideband leakage.
- **BlackmanHarris –** Similar to Hanning window, this window has a slightly wider main lobe but much better dynamic range. It is optimized to provide the minimum side lobe level.

- **FlatTop –** Optimized for measuring peak amplitudes of discrete spectral components that are separated by several spectral lines. Recommended for use when your signal content consists of sine waves and amplitude accuracy is important.

Window Tab for the Advanced FFT Analysis Option

If you are using the Advanced FFT Analysis option, the Window tab appears as follows:



The following windowing types are available when generating FFTs for channels that were configured as response channels; this is a global setting that is applied to the FFTs of all enabled response channels:

- **Rectangular –** No window. Recommended for use when your signal content contains random noise (white noise) or closely spaced sine waves.
- **Hamming –** Bell-shaped cosine window optimized to minimize the maximum (nearest) side lobe. Recommended for use when your signal content consists of closely spaced sine waves.
- **Hanning –** Bell-shaped cosine window typically used to analyze continuous signals. Recommended for use when your signal content contains sine waves, a combination of sine waves, vibration data consisting of narrowband random signals, or unknown content.
- **Bartlett –** Triangular window.
- **Blackman –** Similar to the Hanning and Hamming windows, this window has slightly wider central lobes and less sideband leakage.
- **BlackmanHarris –** Similar to Hanning window, this window has a slightly wider main lobe but much better dynamic range. It is optimized to provide the minimum side lobe level.
- **FlatTop –** Optimized for measuring peak amplitudes of discrete spectral components that are separated by several spectral lines. Recommended for use when your signal content consists of sine waves and amplitude accuracy is important.

- **Exponential –** This window is equal to 1.0 at the beginning of the record and decays exponentially to a smaller value at the end of the record. You can specify a value between 0 and 1 as the *Exp Factor*. This value indicates how much to decay the signal, allowing you to tailor the resultant FFT to the relevant data in the record.

These windowing types are available when generating FFTs for channels that were configured as reference channels; this is a global setting that is applied to the FFTs of all reference channels:

- **Same as Response –** Applies the same windowing function that was selected for the response channels to the FFTs for the reference channels.
- **Force –** This window allows you to specify the percentage of the signal period to which the rectangular window is applied. Data outside the selected percentage is forced to zero.

  In the **Start %** field, enter the percentage of the period at which the rectangular window starts. In the **End %** field, enter the percentage of the signal period at which the rectangular window stops.

  By applying this window to the output of a transducer, you can avoid collecting extraneous signals, such as forces from the retraction of a hammer, that are not important to the measurement.
- **Cosine Taper –** This window allows you to specify the percentage of the period to allow the signal to come through unaltered before smoothing the signal ends to zero.

Configuring the Settings of the Output Tab

If your device support analog output channels, you can configure the analog output signal of your device to provide a stimulus to a system under test using the **Output** tab of the Acquisition Config window.

Figure 14 shows the appearance of the Output tab when using the Data Logger interface.

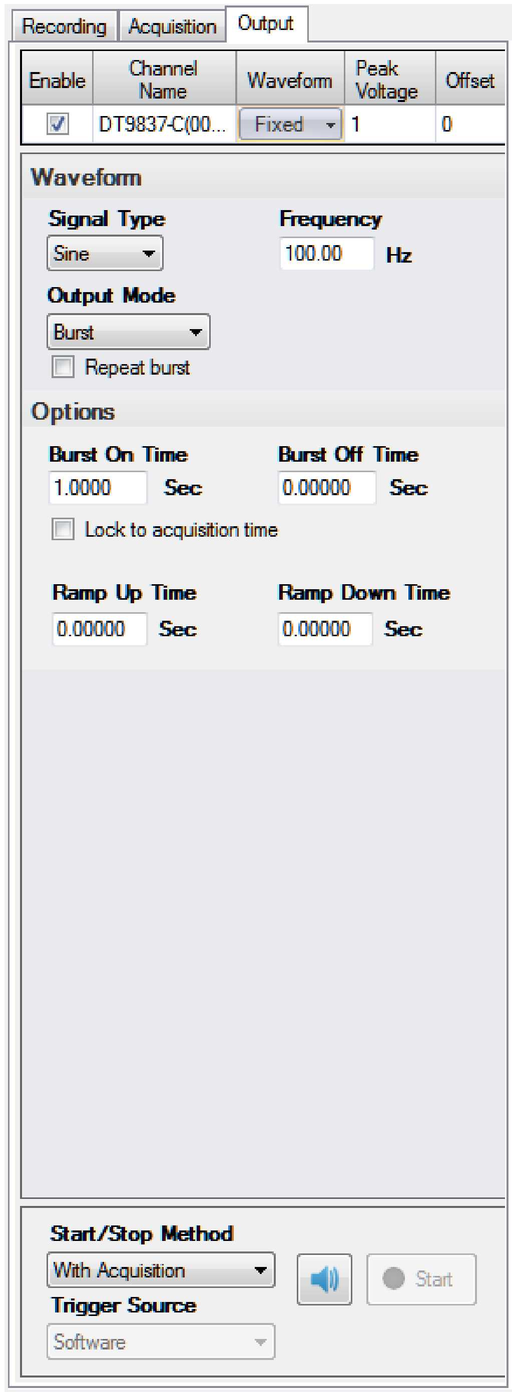


Figure 14: Output Tab When Using the Data Logger Interface

Figure 15 shows the appearance of the Output tab when using the FFT Analyzer interface.

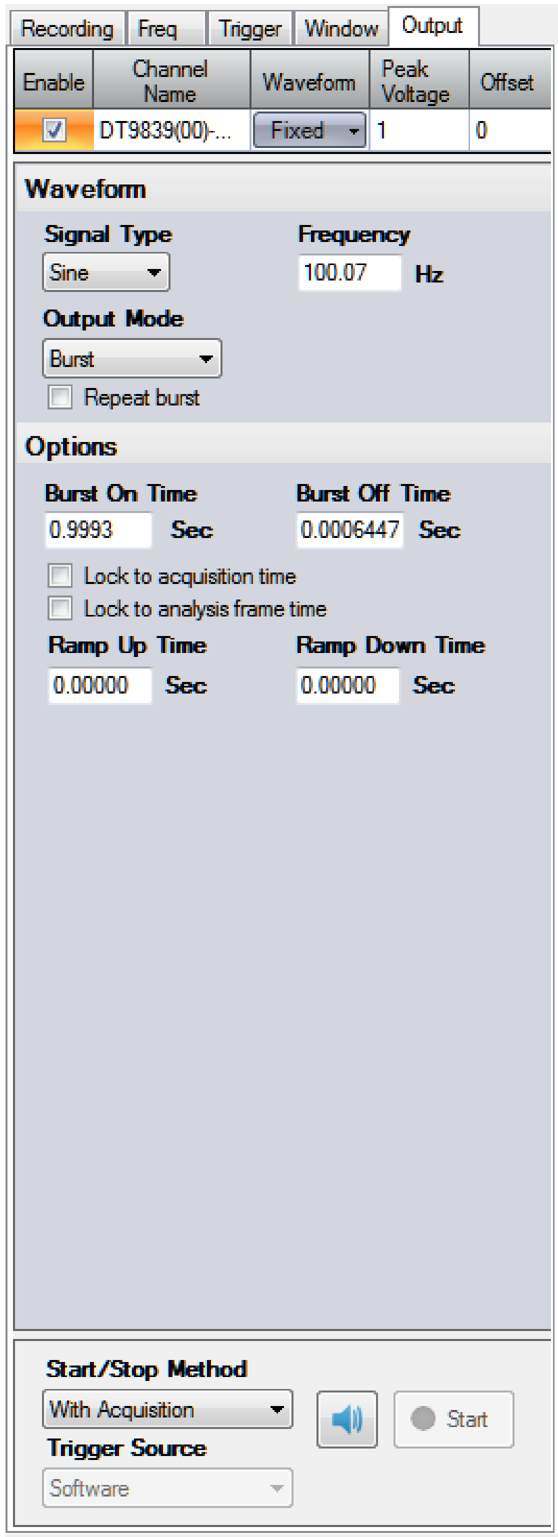


Figure 15: Output Tab When Using the FFT Analyzer Interface

Enabling Channels

If you want to output an analog signal, enable the analog output channel on the module by clicking the checkbox under the **Enable** column of the **Output** tab.

Note: Only one analog output channel can be enabled at a time.

Note that if you are using multiple devices in a Master/Slave arrangement, only the analog output channel on the master device is available to be enabled.

Channel Name

The name of the channel is displayed under the **Channel Name** column of the **Output** tab. It cannot be changed.

Peak Voltage

Enter the voltage of the largest peak to output, measured from 0 V. The output will be ± the peak voltage.

Offset

If desired, enter a value to offset the output signal from the default baseline of 0 V. The default, is 0 V. Choosing a positive value, moves the signal up (above 0 V). Choosing a negative value, moves the signal down (under 0 V).

Waveform Type

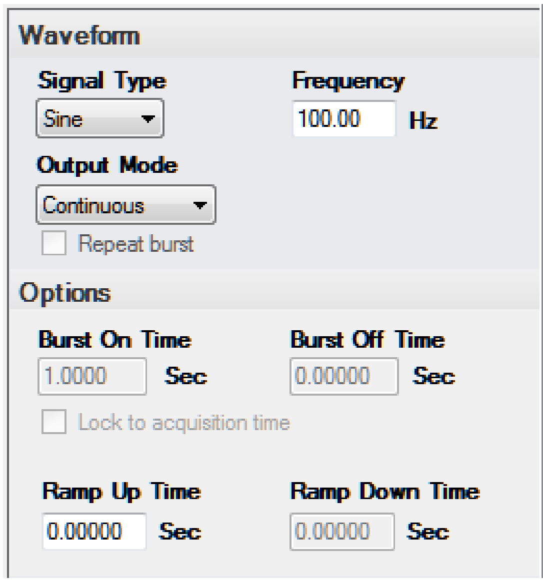
You can choose one of the following waveform types to output:

- **Fixed –** A known output pattern with a fixed frequency.
- **Sweep –** A sine wave whose frequency changes over time.
- **Noise –** A variable waveform that carries a range of frequency components.

Specific parameters are configurable based on the waveform type that you choose. The waveform types and their configurable parameters are described in detail in the following sections.

Fixed Waveforms

A fixed waveform has a known pattern with a fixed frequency. When you choose a fixed waveform type, the following options are available:



Note: You can change these parameters when the analog output operation is running to change the output signal in real-time.

Signal Type

Specify the pattern of the waveform to output by choosing one of the following signal types:

- **Sine** – A sinusoidal, periodic waveform.
- **Square** – A non-sinusoidal waveform that is shaped like a square, alternating between two levels. (Note that due to the nature of Sigma-Delta converters used on the supported data acquisition devices, you may see variations of the signal at each level.)
- **Ramp** – A non-sinusoidal waveform that ramps up and then drops sharply. This is sometimes called a sawtooth waveform.
- **Triangle** – A non-sinusoidal waveform that is shaped like a triangle.

Output Frequency

Specify the frequency at which to output the waveform. For fixed waveforms, this frequency is constant for the duration of the waveform.

Output Mode

You can choose one of the following output modes:

- **Continuous** – In continuous mode, the waveform is output repetitively until acquisition stops if the Start/Stop method is **With Acquisition** or until you stop it with the **Stop** button.
- **Burst** – In burst mode, the waveform is output for a specified amount of time. If the **Repeat Burst** checkbox is unchecked, the waveform is output once. If the **Repeat Burst** checkbox is checked, the burst waveform is output repetitively.

  You specify how long the signal is on by specifying the **Burst On Time**. You specify how long the signal is off (signal is at 0 V) by specifying the **Burst Off Time**.

  If you are using the Data Logger interface and the **Lock to acquisition time** checkbox is checked, the waveform is output in the time that it takes to acquire the data (specified in the **Recording** tab).

  If you are using the FFT Analyzer interface and the **Lock to acquisition time** checkbox is checked, the waveform is output in the time that it takes to do all the averages (an acquisition of all the records specified in the **Recording** tab).

  If you are using the FFT Analyzer interface and the **Lock to analysis frame time** checkbox is checked, the waveform is output in the time that it takes to do one average (an acquisition of one record). This option is useful in applications where repeatability is required, such as outputting a repeatable chirp signal.

Notes: If you are using the FFT Analyzer interface, the **Lock to acquisition time** checkbox is checked, and the Start Method on the **Output** tab is **With Acquisition**, the analog input trigger source should be Freerun or you will be prompted to change it to this setting. This is to guarantee that the analog output data is correlated properly with the analog input data for each record when many averages are performed.

If the **Lock to analysis frame time** checkbox is checked and the Start Method on the **Output** tab is **With Acquisition**, the analog input trigger source should be Software Triggered or you will be prompted to change it to this setting. This is to guarantee that the analog output data is correlated properly with the analog input data for each record when many averages are performed.

If you make any changes to the parameters on the Recording tab, Acquisition tab, or Freq tab, the values are automatically updated on the Output tab.

Ramp Up Time and Ramp Down Time

If desired, you can determine how long it takes for the signal to ramp up at the start of the waveform to the specified peak voltage by specifying the **Ramp Up Time**.

If the output mode is **Burst**, you can also specify how long it takes for the signal to ramp down to 0 V at the end of the waveform by specifying the **Ramp Down Time**.

Example

Figure 16 shows a sine wave with a fixed frequency of 10 Hz, a peak voltage of 3 V, an offset of 0 V, a Burst On Time of 3 seconds, a Ramp Up time of 1.0 second and a Ramp Down time of 1.0 second:

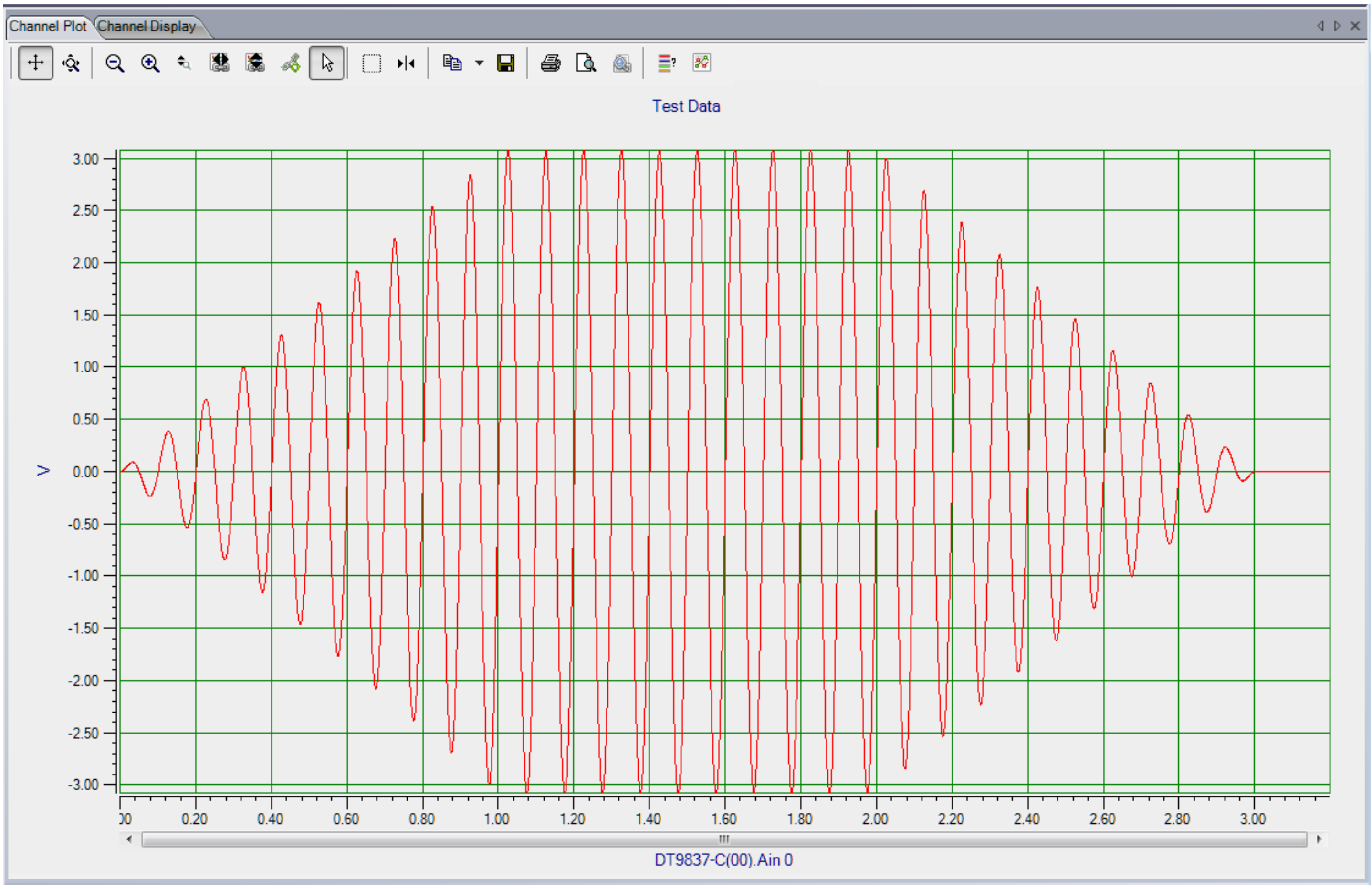


Figure 16: Fixed Sine Wave with Modified Burst On Time and Ramp Up/Ramp Down Time

Sweep Waveforms

A sweep waveform is a sine wave whose frequency changes over time. Swept sine waves allow you to test a device over a wide frequency range to determine if any resonances occur.

If you are using the Data Logger interface, the following options are available when you choose the Sweep waveform type:

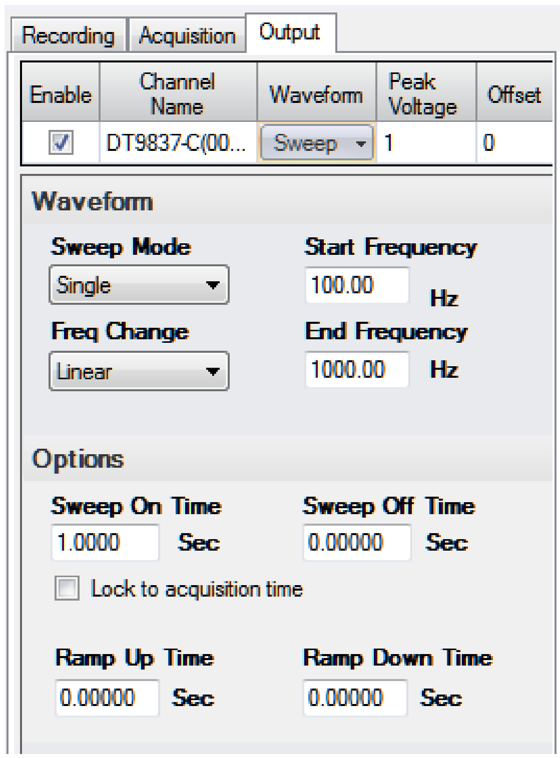


Figure 17: Sweep Waveform Options for the Data Logger Interface

If you are using the FFT Analyzer interface, the following options are available when you choose the Sweep waveform type:

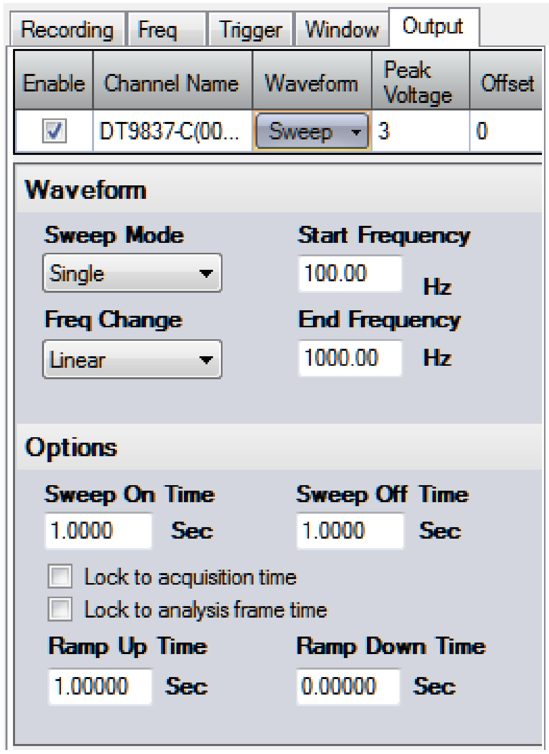


Figure 18: Sweep Waveform Options for the FFT Analyzer Interface

---

Note: You can change these parameters when the analog output operation is running to change the output signal in real-time.

Sweep Mode

Specify one of the following sweep modes:

- **Loop** – In loop mode, the swept sine wave repeats continuously.
- **Single** – In single mode, the swept sine wave is output once.

Frequency Change

Specify how you want the frequency of the waveform to change over time by selecting one of the following values for the **Freq Change** combo box:

- **Linear** – The frequency increases at a fixed rate of frequency per time (Hz/s).
- **Log** – In this mode, the frequency increases at a fixed rate of octave change per time (Octave/s). This is also called an exponential sweep.

Start and End Frequencies

In the **Start Frequency** field, specify the frequency of the waveform when it starts.

In the **End Frequency** field, specify the frequency of the waveform when it ends.

If you want a swept sine wave that increases in frequency over time, ensure that the ending frequency is greater than the starting frequency; this is sometimes called a chirp. If you want a swept sine wave that decreases in frequency over time, ensure that the ending frequency is less than the starting frequency.

Sweep On Time and Sweep Off Time

You specify how long the signal is on by specifying the **Sweep On Time**. You specify how long the signal is off (signal is at 0 V) by specifying the **Sweep Off Time**.

If you are using the Data Logger interface and the **Lock to acquisition time** checkbox is checked, the waveform is output in the time that it takes to acquire the data (specified in the **Recording** tab).

If you are using the FFT Analyzer interface and the **Lock to acquisition time** checkbox is checked, the waveform is output in the time that it takes to do all the averages (an acquisition of all the records specified in the **Recording** tab).

If you are using the FFT Analyzer interface and the **Lock to analysis frame time** checkbox is checked, the waveform is output in the time that it takes to do one average (an acquisition of one record). This option is useful in applications where repeatability is required, such as outputting a repeatable chirp signal.

Notes: If you are using the FFT Analyzer interface, the **Lock to acquisition time** checkbox is checked, and the Start Method on the **Output** tab is **With Acquisition**, the analog input trigger source should be Freerun or you will be prompted to change it to this setting. This is to guarantee that the analog output data is correlated properly with the analog input data for each record when many averages are performed.

If the **Lock to analysis frame time** checkbox is checked and the Start Method on the **Output** tab is **With Acquisition**, the analog input trigger source should be Software Triggered or you will be prompted to change it to this setting. This is to guarantee that the analog output data is correlated properly with the analog input data for each record when many averages are performed.

If you make any changes to the parameters on the Recording tab, Acquisition tab, or Freq tab, the values are automatically updated on the Output tab.

Ramp Up Time and Ramp Down Time

If desired, you can determine how long it takes for the signal to ramp up at the start of the waveform to the specified peak voltage by specifying the **Ramp Up Time**.

You can also specify how long it takes for the signal to ramp down to 0 V at the end of the waveform by specifying the **Ramp Down Time**.

Examples

Figure 19 shows a linear swept sine wave from 1 to 30 Hz.

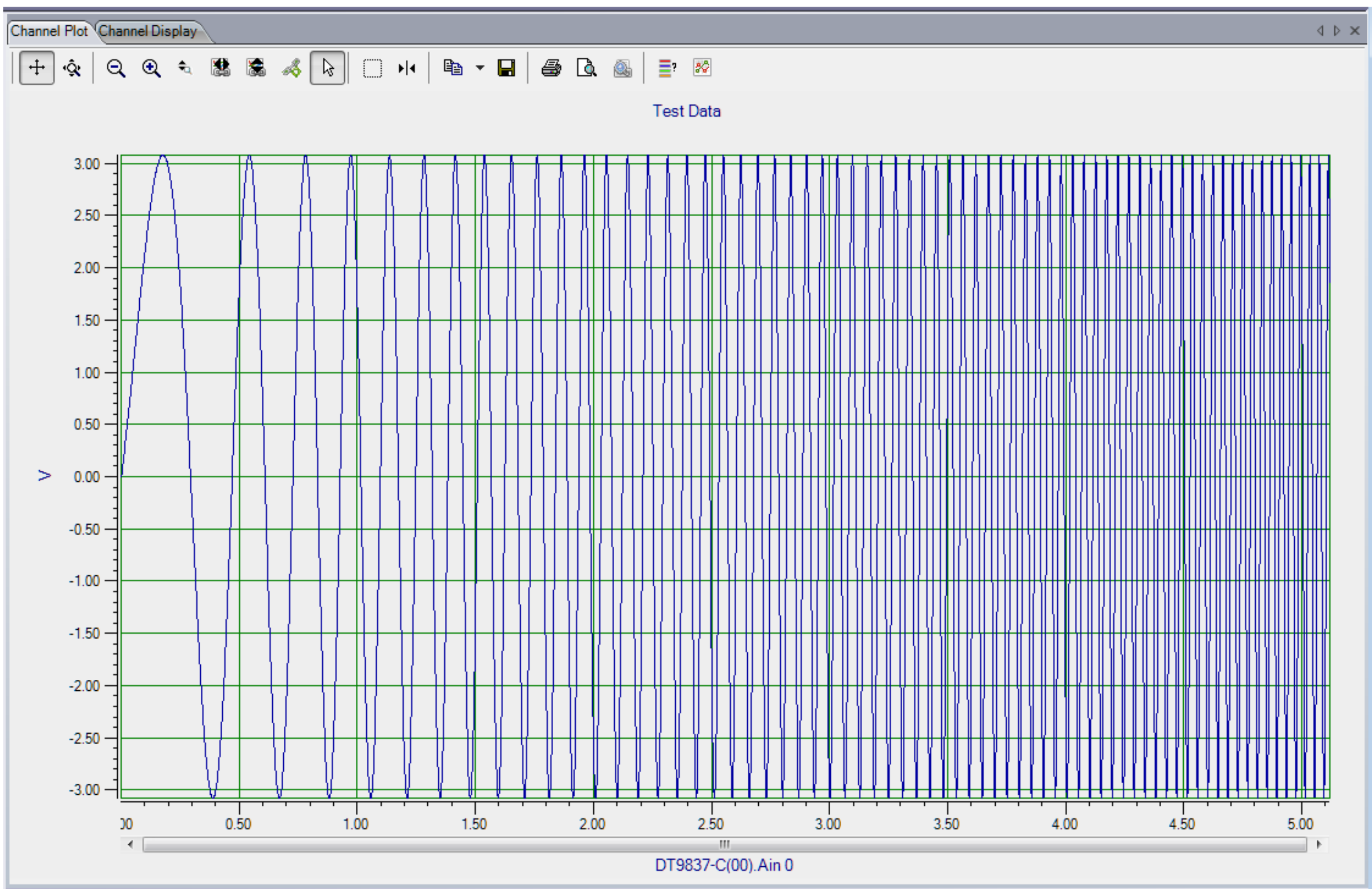


Figure 19: Linear Swept Sine Wave from 1 to 30 Hz

Figure 20 shows a logarithmic swept sine wave from 1 to 30 Hz.

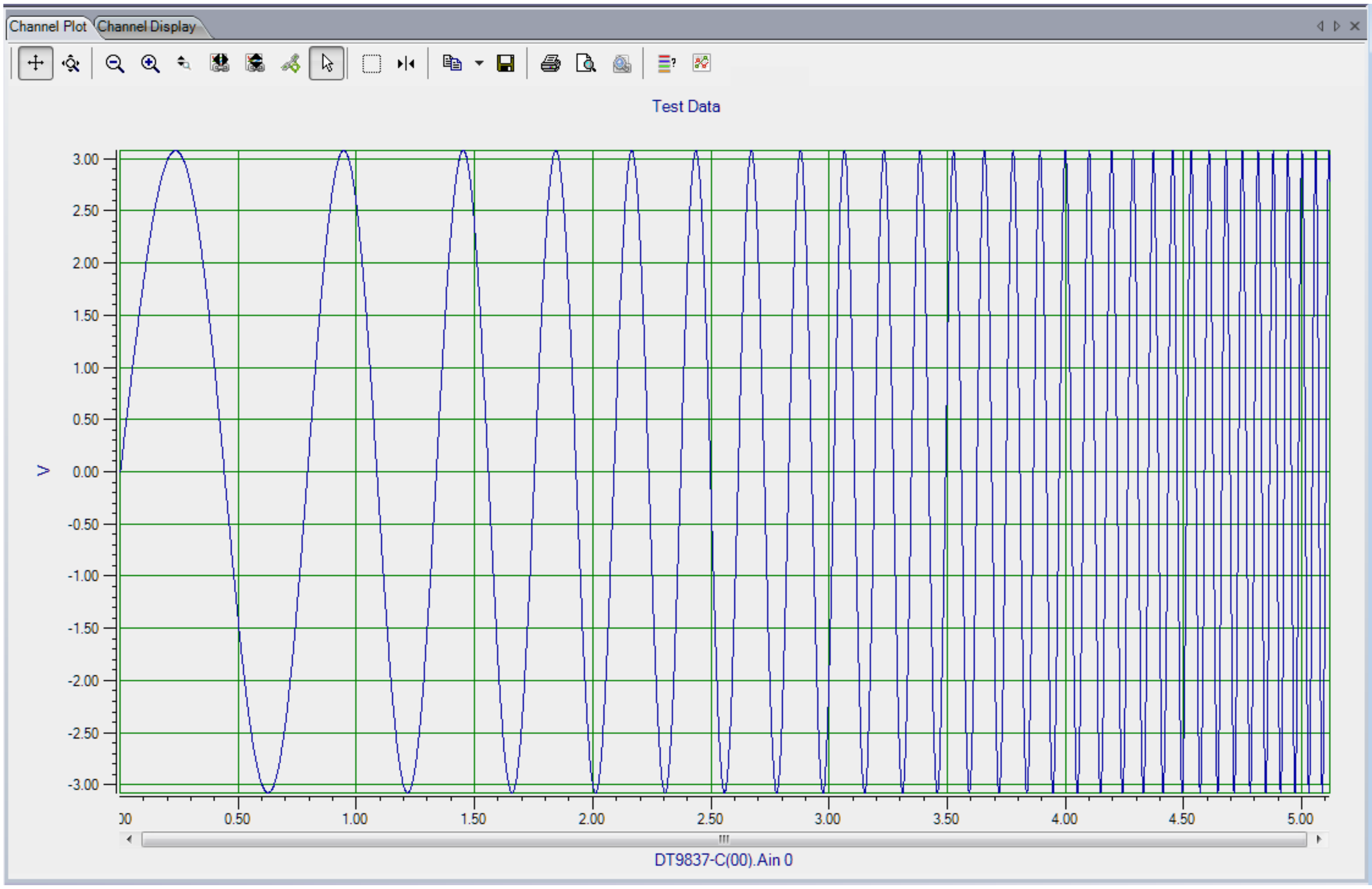


Figure 20: Logarithmic Swept Sine Wave from 1 to 30 Hz

Figure 21 shows a logarithmic swept sine wave from 1 to 100 Hz with a Sweep On time of 3.0 s, a Sweep Off time of 2.0 s, a Ramp Up time of 1.0 s, and a Ramp Down time of 2.0 s.

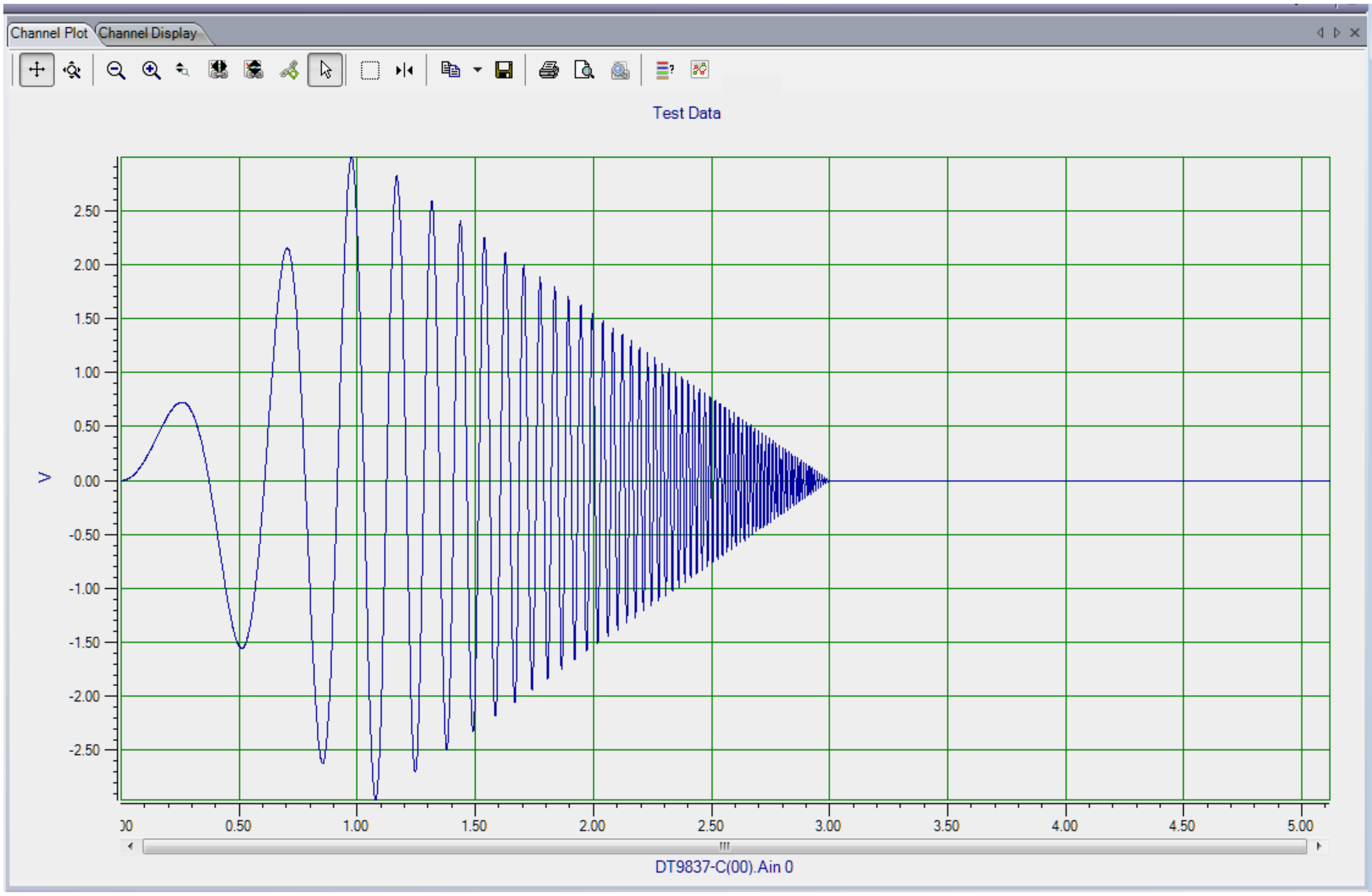


Figure 21: Logarithmic Swept Sine Wave from 1 to 100 Hz with Modified Sweep On/Off Times and Ramp Up/Down Times

Noise Waveforms

A noise waveform is a variable waveform that carries a range of frequency components, allowing you to uncover resonances at a variety of frequencies.

If you are using the Data Logger interface and choose a noise waveform type, the following options are available:

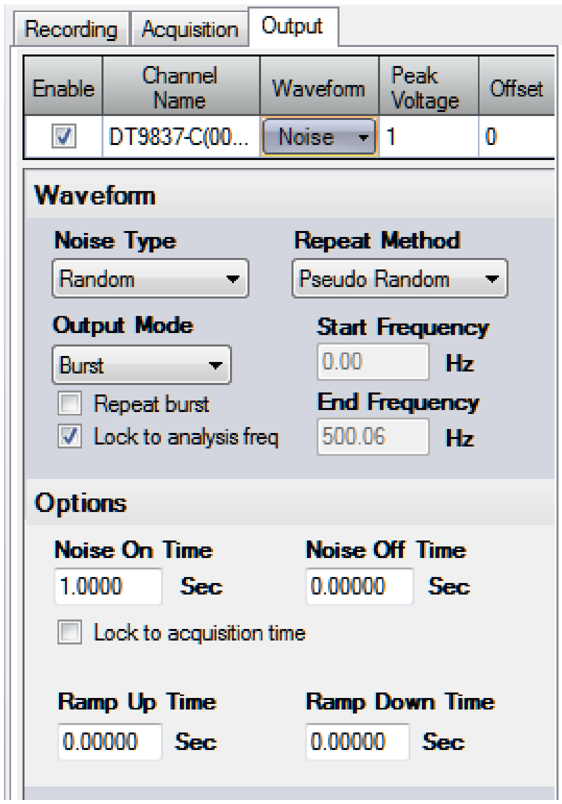


Figure 22: Noise Waveform Options Using the Data Logger Interface

If you are using the FFT Analyzer interface and choose a noise waveform type, the following options are available:

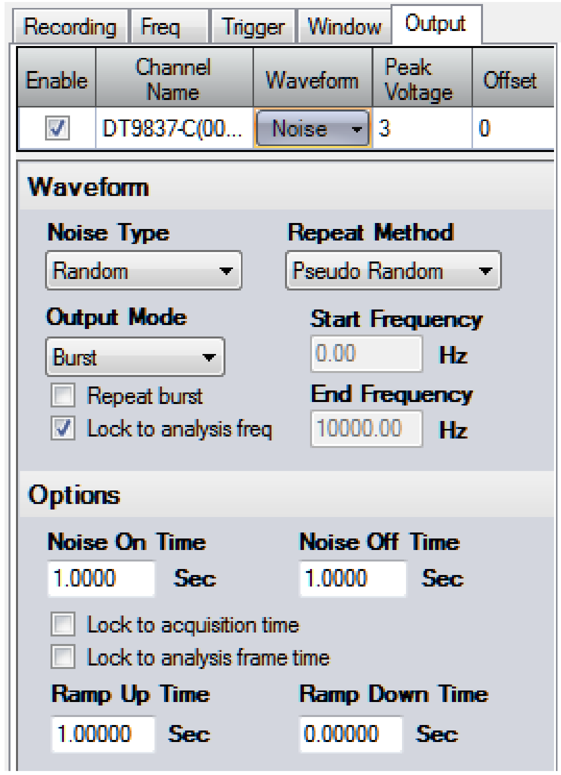


Figure 23: Noise Waveform Options Using the FFT Analyzer Interface

---

Note: You can change these parameters when the analog output operation is running to change the output signal in real-time.

Noise Type

Specify the type of noise waveform to output by choosing one of the following signal types:

- **Random –** A noise signal with a power spectral density that remains constant over the entire noise bandwidth and an amplitude histogram that exhibits a constant distribution.

  Figure 24 shows a random noise signal (from 0 Hz to 200 Hz) in the time domain.

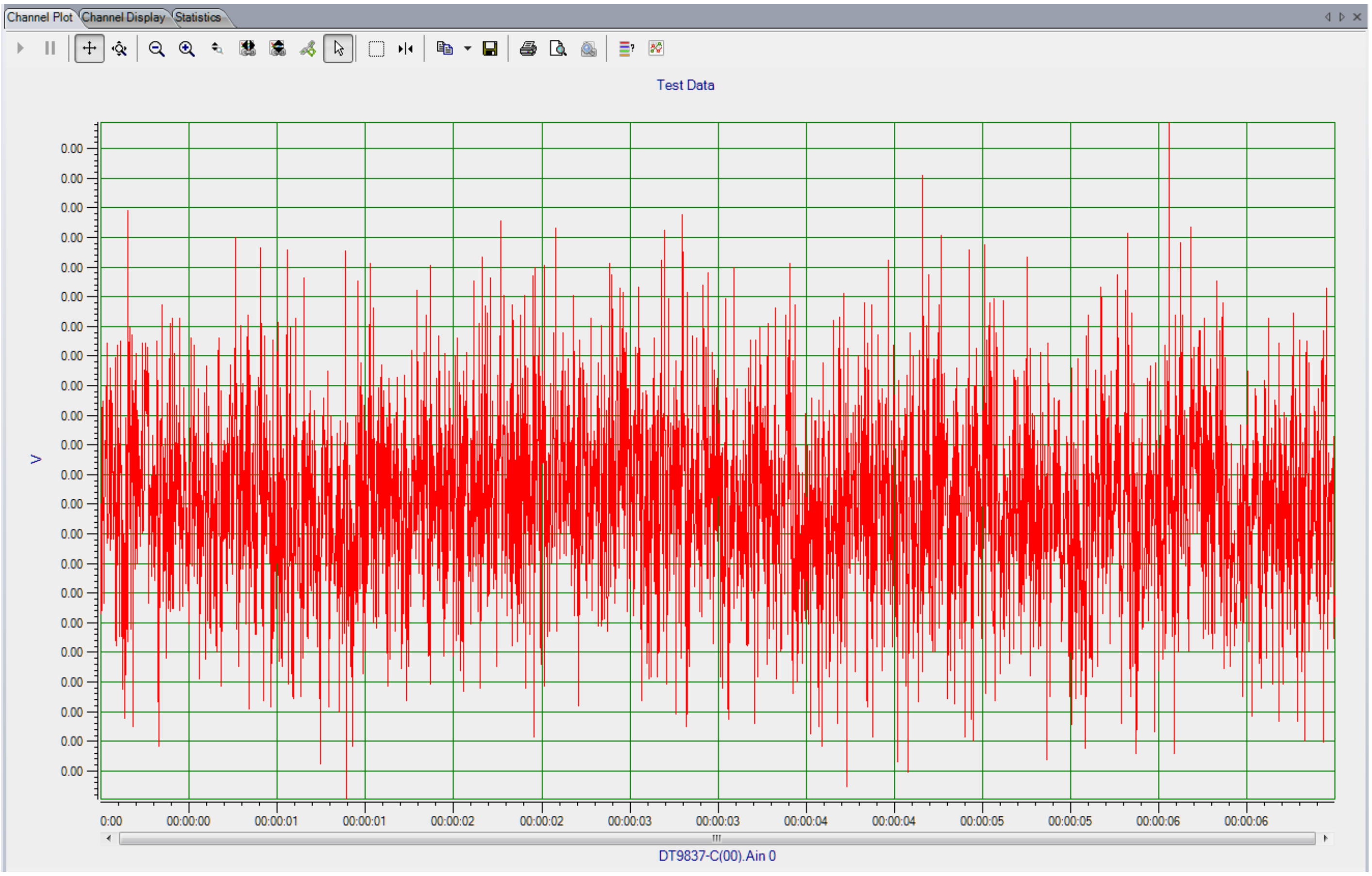


Figure 24: Random Noise Signal

Figure 25 shows a histogram of the random noise in the time domain. As you can see, the amplitudes in the time domain form a flat histogram.

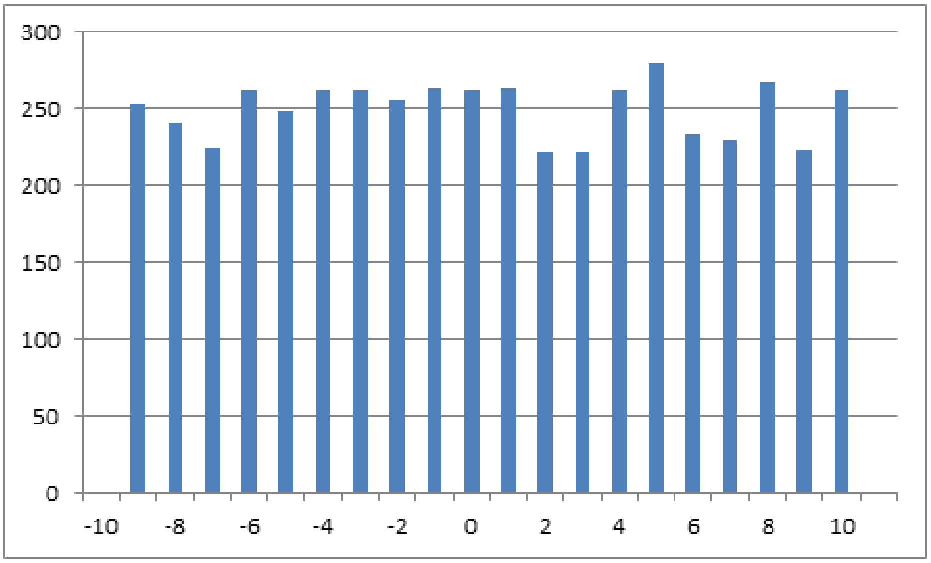


Figure 25: Histogram of Random Noise in the Time Domain Shows a Flat Distribution

- **Random Normal –** A noise signal with a power spectral density that remains constant over the entire noise bandwidth and an amplitude histogram that exhibits a Gaussian distribution.

  The software creates random normal noise by implementing the Box-Muller transform.

  Figure 26 shows a random normal noise signal (from 0 Hz to 200 Hz) in the time domain.

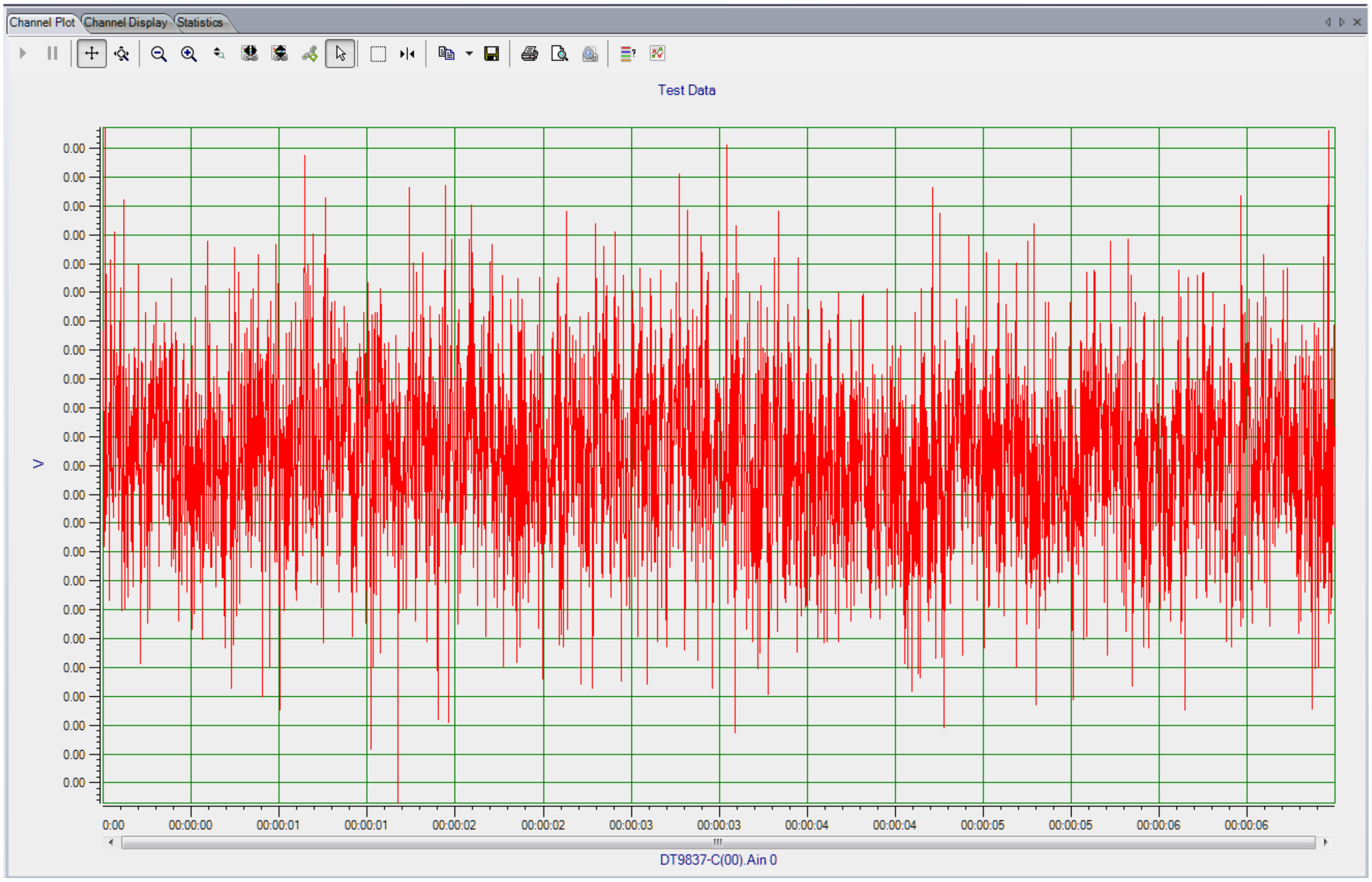


Figure 26: Random Normal Noise Signal

Figure 27 shows a histogram of the random normal noise in the time domain. As you can see, the amplitudes in the time domain form a Gaussian histogram.

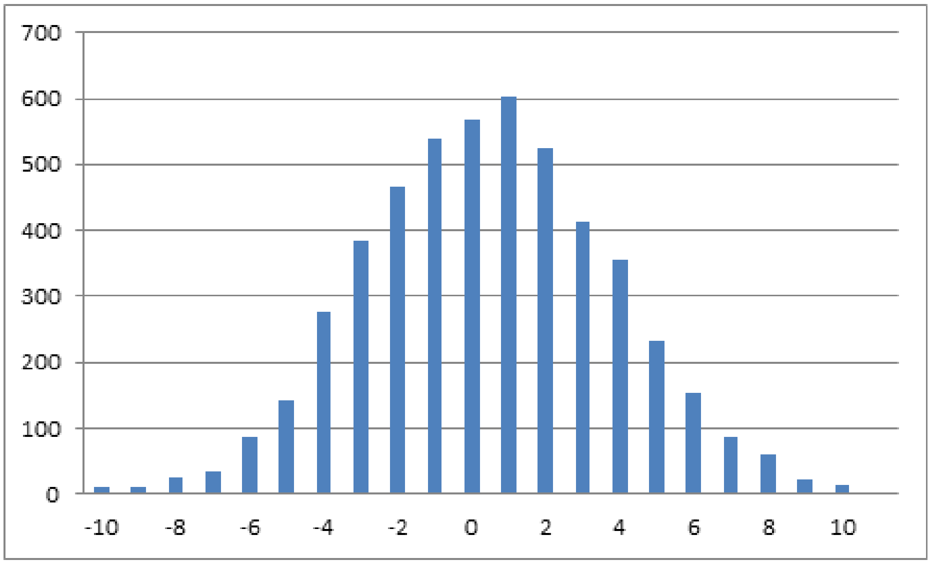


Figure 27: Histogram of Random Normal Noise in the Time Domain Shows a Gaussian Distribution

- **Pink (1/f Noise) –** A noise signal with a power spectral density that is inversely proportional to the frequency. Each octave contains an equal amount of noise power.

  Figure 28 shows a pink noise signal (from 0 Hz to 10 kHz) in the time domain.

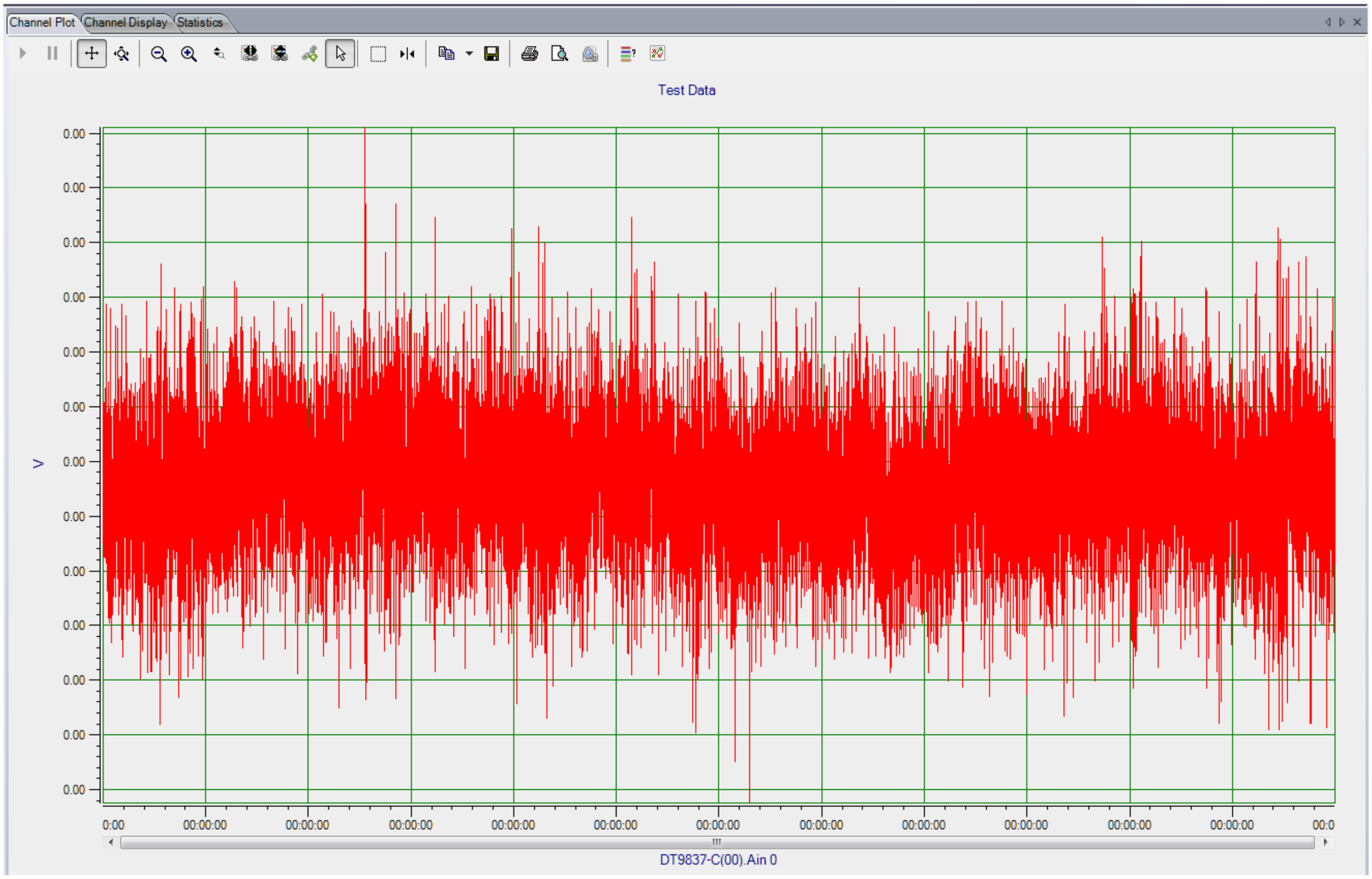


Figure 28: Pink Noise Signal

Figure 29 shows a histogram of the pink noise in the time domain.

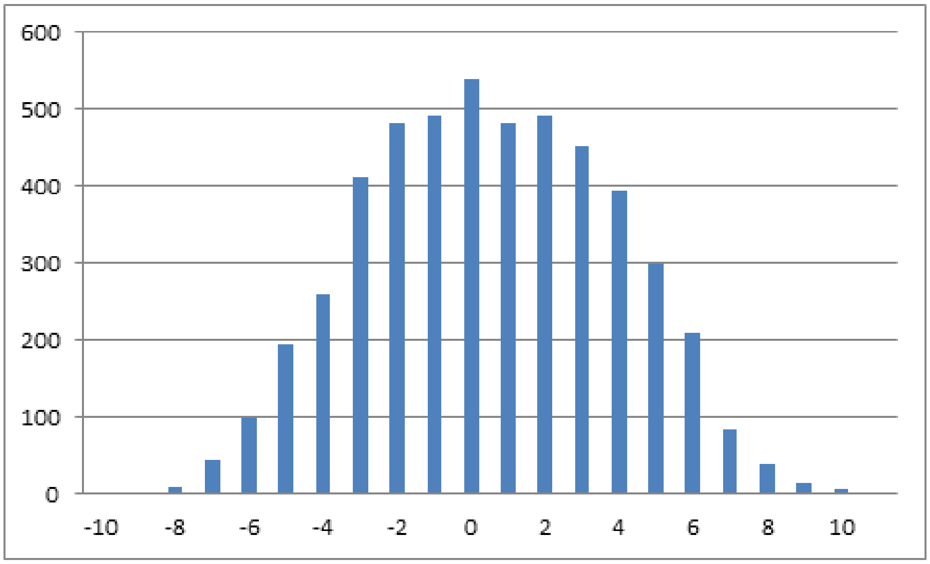


Figure 29: Histogram of Pink Noise in the Time Domain

Output Mode

You can choose one of the following output modes for noise waveforms:

- **Continuous** – In continuous mode, the waveform is output repetitively until acquisition stops if the Start/Stop method is **With Acquisition** or until you stop it with the **Stop** button.
- **Burst** – In burst mode, the waveform is output for a specified amount of time. If the **Repeat Burst** checkbox is unchecked, the waveform is output once. If the **Repeat Burst** checkbox is checked, the burst waveform is output repetitively.

  You specify how long the signal is on by specifying the **Burst On Time**. You specify how long the signal is off (signal is at 0 V) by specifying the **Burst Off Time**.

  If you are using the Data Logger interface and the **Lock to acquisition time** checkbox is checked, the waveform is output in the time that it takes to acquire the data (specified in the **Recording** tab).

  If you are using the FFT Analyzer interface and the **Lock to acquisition time** checkbox is checked, the waveform is output in the time that it takes to do all the averages (an acquisition of all the records specified in the **Recording** tab).

  If you are using the FFT Analyzer interface and the **Lock to analysis frame time** checkbox is checked, the waveform is output in the time that it takes to do one average (an acquisition of one record). This option is useful in applications where repeatability is required, such as outputting a repeatable chirp signal.

Notes: If you are using the FFT Analyzer interface, the **Lock to acquisition time** checkbox is checked, and the Start Method on the **Output** tab is **With Acquisition**, the analog input trigger source should be Freerun or you will be prompted to change it to this setting. This is to guarantee that the analog output data is correlated properly with the analog input data for each record when many averages are performed.

If the **Lock to analysis frame time** checkbox is checked and the Start Method on the **Output** tab is **With Acquisition**, the analog input trigger source should be Software Triggered or you will be prompted to change it to this setting. This is to guarantee that the analog output data is correlated properly with the analog input data for each record when many averages are performed.

If you make any changes to the parameters on the Recording tab, Acquisition tab, or Freq tab, the values are automatically updated on the Output tab.

Ramp Up Time and Ramp Down Time

If desired, you can determine how long it takes for the signal to ramp up at the start of the waveform to the specified peak voltage by specifying the **Ramp Up Time**.

If the output mode is **Burst**, you can also specify how long it takes for the signal to ramp down to 0 V at the end of the waveform by specifying the **Ramp Down Time**.

Output Frequency

If you check the **Lock to analysis freq** checkbox, the maximum frequency of the noise waveform is locked to the maximum frequency to analyze (specified in the **Acquisition** tab for the Data Logger interface or the **Freq** tab for the FFT Analyzer interface).

If the **Lock to analysis freq** checkbox is unchecked, you can specify the starting frequency component and the ending frequency component in the generated signal.

Repeat Method

You can choose one of the following repeat methods for noise waveforms:

- **True Random –** Generates a new noise signal of the specified waveform pattern (Random, Random Normal, or Pink) for each burst period.

  A true random signal is always nonperiodic during the total acquisition time. Therefore, a time domain window, such as a Hanning window, must be used with this repeat mode to minimize leakage.

- **Pseudo Random –** Repeats the last output signal for the specified noise waveform type (Random, Random Normal, or Pink).

  A pseudo random signal does not excite nonlinearities differently between spectrum averages; therefore, spectrum averaging will not remove nonlinearities from FRF measurements.

  However, the excitation signal is completely contained in the total acquisition time, and is therefore, periodic during the total acquisition time and without leakage.

Start/Stop Methods

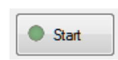
To start the waveform generation operation, you can choose one of the following options:

- **With Acquisition –** Choose this option if you want the device to output the specified waveform when acquisition starts.

Note: If you choose **With Acquisition** and the analog output channel is enabled, ensure that you select an analog input trigger source other than a threshold trigger, or you will be prompted to change the analog input trigger source.

If you are using the FFT Analyzer interface and you choose **With Acquisition** when the **Lock to acquisition time** checkbox is selected, the analog input trigger source should be Freerun or you will be prompted to change it to this setting. This is to guarantee that the analog output data is correlated properly with the analog input data for each record when many averages are performed.

If you are using the FFT Analyzer interface and you choose **With Acquisition** when the **Lock to analysis frame time** checkbox is checked, the analog input trigger source should be Software Triggered or you will be prompted to change it to this setting. This is to guarantee that the analog output data is correlated properly with the analog input data for each record when many averages are performed.

- **Manual –** Choose this option if you want the device to output the specified waveform when one of the following trigger conditions is detected:
  - **Software –** The operation starts immediately when the **Start** button 

 on the **Output** tab is clicked.
  - **Ext TTL Pos Edge –** After the **Start** button 

 on the **Output** tab is clicked, the operation starts when the device detects a positive edge on the external digital trigger line.

You can stop the waveform output operation by stopping acquisition or by clicking the **Stop** button 

 on the **Output** tab. When the waveform is stopped, the specified offset value is output.

Muting and Unmuting the Output Signal

If your device supports it, you can mute the output in hardware by clicking the **Unmute** button . This forces the output to 0 V, as shown in Figure 30.

Note: The number of samples it takes for the mute process to complete is hardware dependent. Refer to the user's manual for your device for more information.

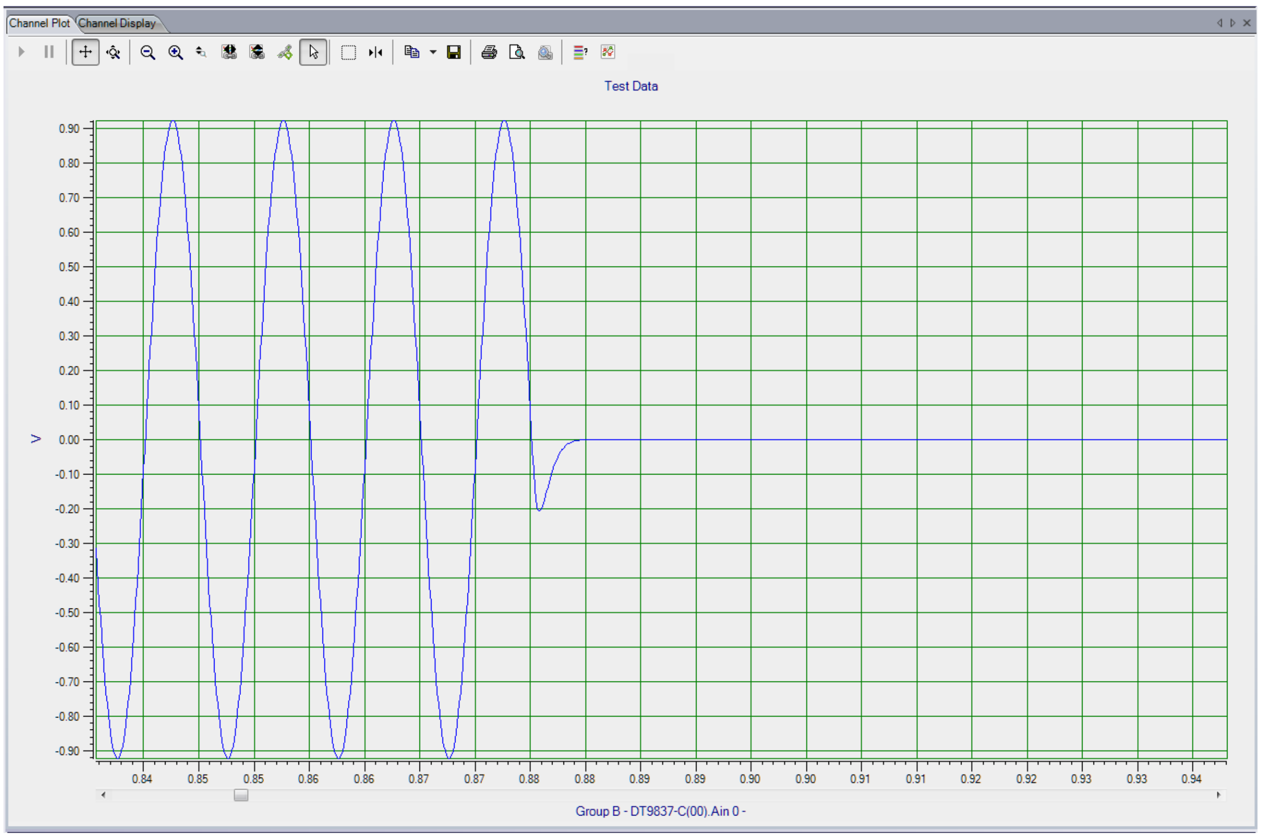


Figure 30: Muting the Output Signal

Note: Mute is also activated if you click the **Stop** button on the main interface or on the Output tab, causing the analog output operation to stop.

To unmute the output, click the **Mute** button . The output pattern that you selected resumes at the current level, as shown in Figure 31.

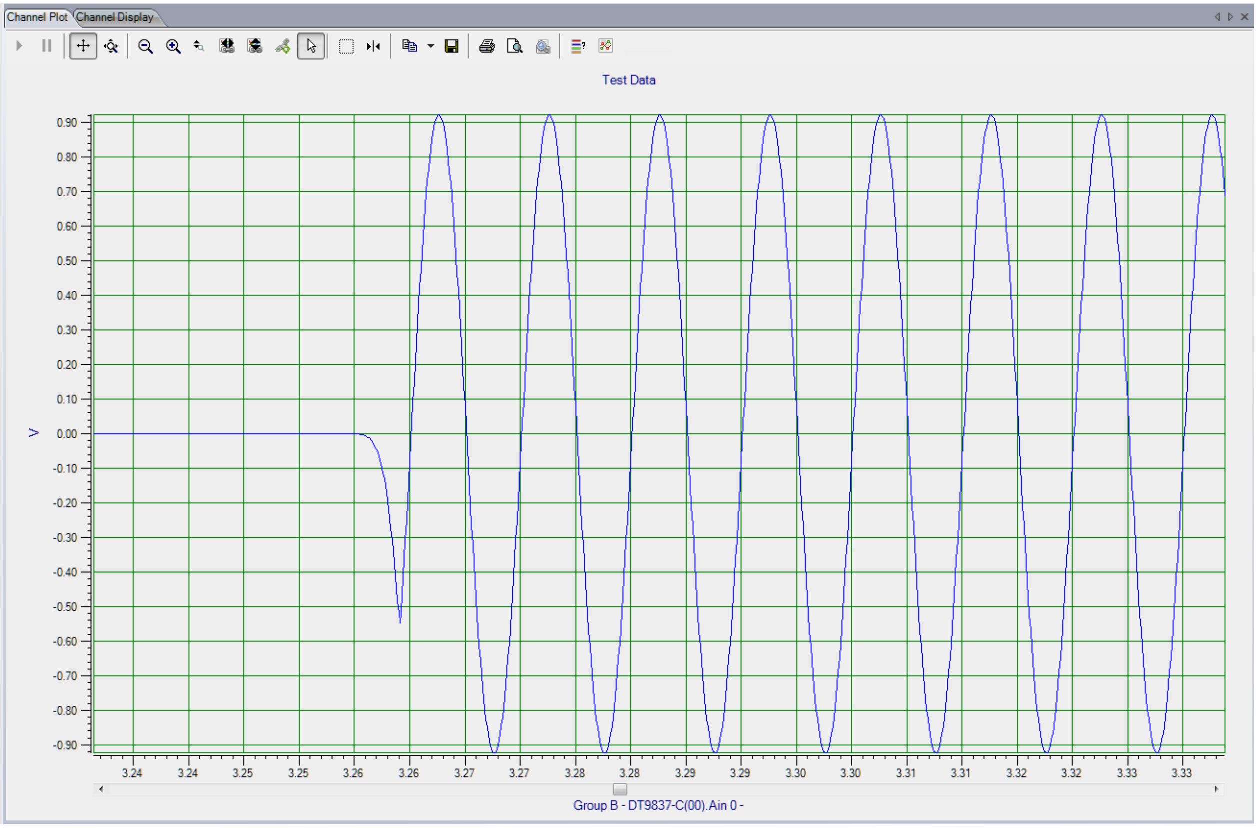


Figure 31: Unmuting the Output Signal

Configuring Digital Filters

Notes: To configure digital filters, you must have purchased the **FFT Analysis Option** or the **Advanced FFT Analysis Option** of QuickDAQ.

Choose the **Digital Filter Configuration** toolbar button ( ) or the **Digital Filter Configuration** option from the **Configuration** menu to configure IIR (infinite impulse response) filters for the analog input channels. The Digital Filter Configuration dialog appears. Figure 32 shows an example of the Digital Filter Configuration dialog.

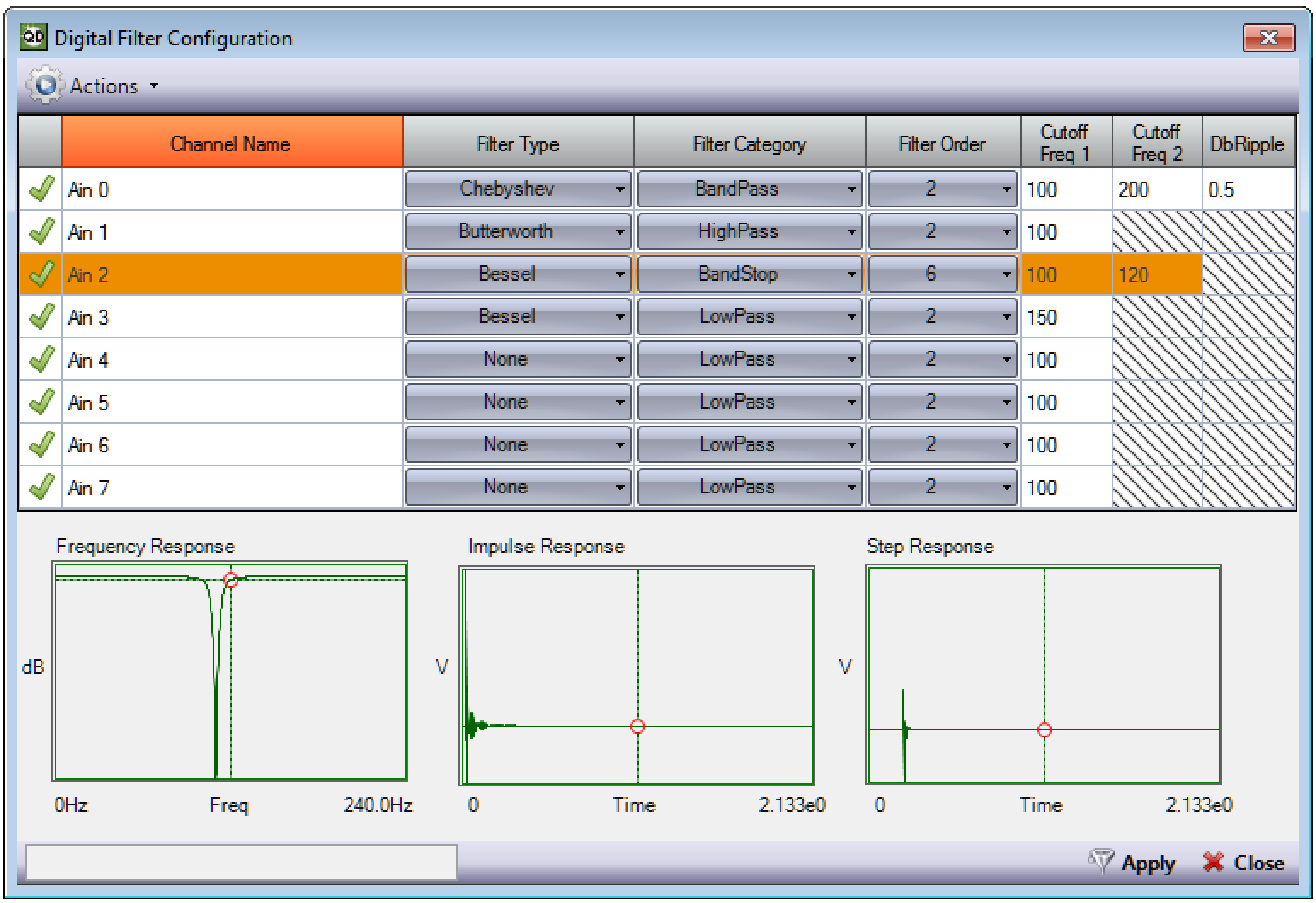


Figure 32: Digital Filter Configuration Dialog

Note: The settings used by the Digital Filter Configuration dialog depend on the sampling or analysis frequency. Therefore, it is recommended that you set up the sampling/analysis frequency before using the Digital Filter Configuration dialog.

If you later change the sampling/analysis frequency, the application may notify you that one or more filters have been invalidated. If this occurs, it is recommended that you change your filter settings based on the new sampling/analysis frequency.

If you are using the Data Logger interface, refer to page 125 for more information on setting the sampling frequency. If you are using the FFT Analyzer interface, refer to page 128 for more information on setting the analysis frequency.

Channels and Filter Status Indicators

All the analog input channels that are supported by your device, whether they are enabled or not in the Configure Devices dialog, are shown in the Digital Filter Configuration dialog.

By default, the status indicator for each channel is shown as Stable .

As you choose various filter settings for the channel (before you click the **Apply** button), the indicator changes to In Process .

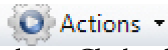
Once you apply the changes, the status indicator either indicates that the filter is Stable , or Unstable .

A stable filter assures that every limited input signal produces a limited filter response. An unstable filter cannot be qualified, and in some situations may prove useless or even harmful. Therefore, it is important that you choose the settings for your filters carefully to avoid instability.

Filter Types

You can select one of the following filter types to apply to each analog input channel:

- None – No digital filter.
- Bessel – A linear filter with a maximally flat group delay (maximally linear phase response). Bessel filters are often used in audio applications. Bessel filters are characterized by almost constant group delay across the entire passband, thus preserving the wave shape of filtered signals in the passband.
- Butterworth – A maximally flat magnitude filter, with no ripples in the passband or stopband.
- Chebyshev – This filter type minimizes the error between the idealized and the actual filter characteristic over the range of the filter, but with ripples in the passband. It has a steeper roll-off and more passband ripple or stopband ripple than Butterworth filters.

If you want to set the same filter type for all channels at once, select the **Multi-channel Actions** button (

 ), choose **Set all Filter Types**, and select the filter type (None, Bessel, Butterworth, or Chebyshev) that you want to use.

Filter Category and Cutoff Frequency

You can further refine the type of filter to apply by selecting one of the following filter categories:

- **LowPass –** A low-pass filter is used to eliminate unwanted high-frequency signals by passing low-frequency signals and attenuating (reducing the amplitude of) signals with frequencies higher than the cutoff frequency. The actual amount of attenuation for each frequency varies depending on the specific filter type.

  When you select **LowPass**, enter the cutoff frequency above which signals will be attenuated.

- **HighPass –** A high-pass filter is used to eliminate any unwanted low frequency components by passing high-frequency signals and attenuating signals with frequencies lower than the cutoff frequency. The actual amount of attenuation for each frequency varies depending on the specific filter type.

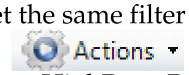
  When you select **HighPass**, enter the cutoff frequency below which signals will be attenuated.

- **BandPass –** A band-pass filter is a combination of a low-pass and a high-pass filter. It passes frequencies within a certain range and rejects (attenuates) frequencies outside of that range.

  When you select **BandPass**, enter the cutoff frequency below which (**Cutoff Freq 1**) and above which (**Cutoff Freq 2**) signals will be attenuated.

- **BandStop –** A band-stop filter passes most frequencies unaltered, but attenuates those in a specific range to very low levels. It is the opposite of a band-pass filter. A very narrow band-stop filter is known as a notch filter.

  When you select **BandStop**, enter the cutoff frequency below which (**Cutoff Freq 1**) and above which (**Cutoff Freq 2**) signals will be attenuated.

If you want to set the same filter category for all channels at once, select the **Multi-channel Actions** button (

 ), choose **Set all Filter Categories**, and select the filter category (LowPass, HighPass, BandPass, or BandStop) that you want to use.

Filter Order

The filter order (also known as the number of poles) determines the rejection characteristics of the filter. The greater the filter order, the greater the rejection, yielding a steeper rolloff. You can select a number from 1 to 10 for the filter order.

If you want to set the filter order for all channels at once, select the **Multi-channel Actions** button (

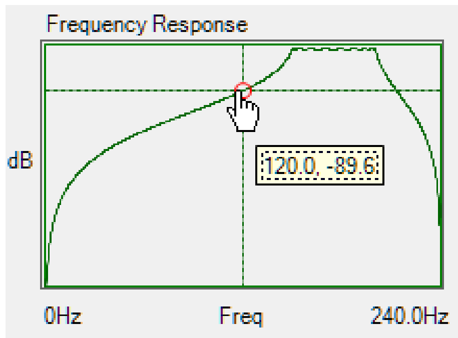
 ), choose **Set all Filter Orders**, and select the filter order (1 to 10) that you want to use.

dB Ripple

For Chebyshev filters, specify the ripple, in dB, that you want in the passband.

Frequency Response Graph

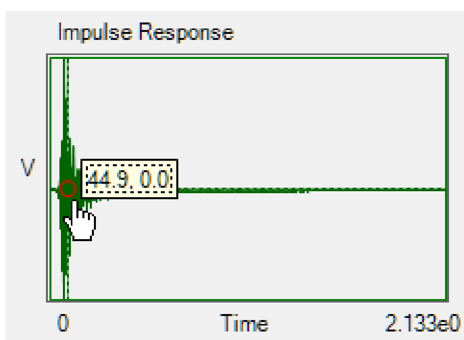
The frequency response graph shows the output of the specified filter as a function of frequency. When you select a channel, you can see the frequency response graph based on the specified filter settings, where dB is plotted on the y-axis and frequency is plotted on the x-axis.



Using your mouse, you can use the left-click button to move along the frequency response graph to see the frequency and dB values.

Impulse Response Graph

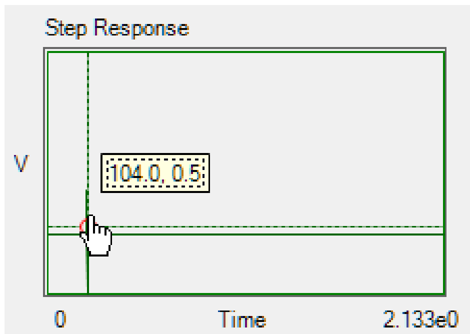
The impulse response graph shows the reaction of the specified filter as a function of time. When you select a channel, you can see the impulse response graph based on the specified filter settings, where voltage is plotted on the y-axis and time, in seconds, is plotted on the x-axis.



Using your mouse, you can use the left-click button to move along the impulse response graph to see the time and voltage values.

Step Response Graph

The step response graph shows the time behavior of the filter when its inputs change from zero to one in a very short time. When you select a channel, you can see the step response graph based on the specified filter settings, where voltage is plotted on the y-axis and time, in seconds, is plotted on the x-axis.



Using your mouse, you can use the left-click button to move along the step response graph to see the time and voltage values.

5

Configuring the Display of the Data Logger Interface

Introduction

This chapter describes how to configure the display when using the Data Logger interface. If you are using the FFT Analyzer interface, refer to Chapter 6 starting on page 189.

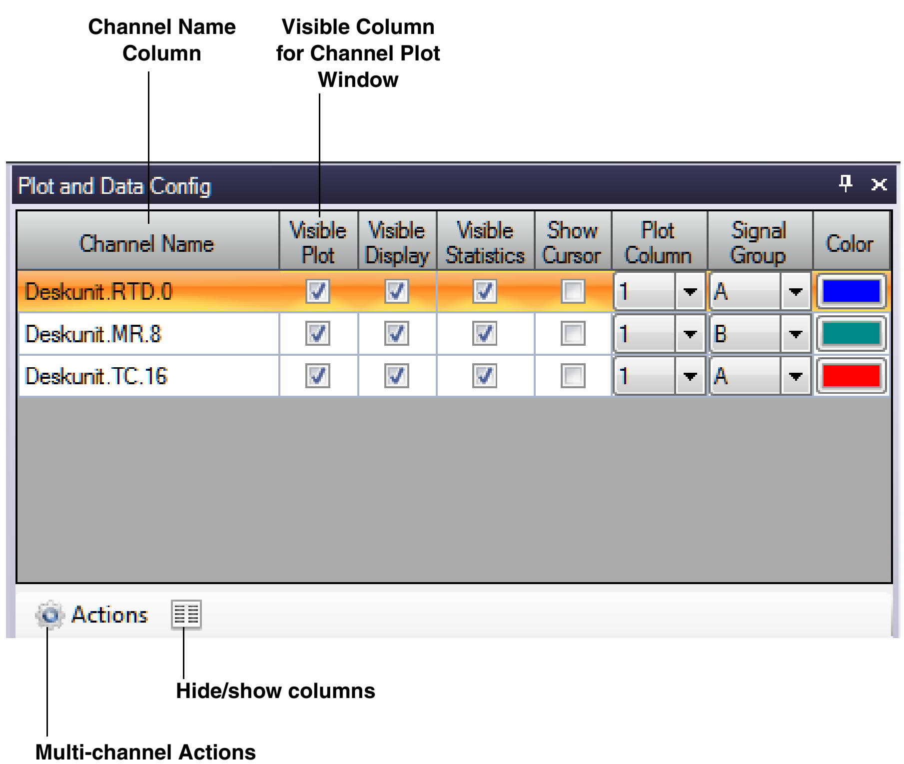
Configuring the Appearance of the Channel Plot Window

This section describes the following aspects related to the appearance of the Channel Plot window when using the Data Logger interface:

- Displaying channels in the Channel Plot window, described on this page
- Displaying data cursors in the Channel Plot window, described on page 171
- Specifying the columns of the Channel Plot window, described on page 172
- Grouping signals in bands on the Channel Plot window, described on page 174
- Specifying the trace color on the Channel Plot window, described on page 176
- Controlling how data is plotted the Channel Plot window, described on page 177
- Specifying the style of the plots in the Channel Plot window, described on page 179

Displaying Channels in the Channel Plot Window

You can determine which channels are displayed in the Channel Plot window by using the following controls in the Plot and Data Config window:



You can see the names of all enabled channels under the **Channel Name** column in the Plot and Data Config window. Only data from the list of enabled channels is acquired.

Note: If desired, you can change the name of a channel using the **Input Channel Configuration** menu option, described on page 84.

For each enabled channel listed under the **Channel Name** column, you can control which channels are displayed in the Channel Plot window using the **Visible Plot** checkbox. If a box is checked under the **Visible Plot** column, a plot for the associated channel is displayed in the Channel Plot window. If a box under the **Visible Plot** column is unchecked, the plot for the associated channel is not displayed in the Channel Plot window.

Note: Data for all enabled channels is recorded regardless of whether the channel is visible.

Making All Selected Channels Visible or Hidden in the Channel Plot Window at Once

If you want to make all selected channels visible in the Channel Plot window at once, select the **Multi-channel Actions** button (

), and choose **Selected channels visible in Channel Plot**.

If you want to hide all selected channels in the Channel Plot window at once, select the **Multi-channel Actions** button (

), and choose **Selected channels hidden in Channel Plot**.

Showing or Hiding the Visible Plot Column

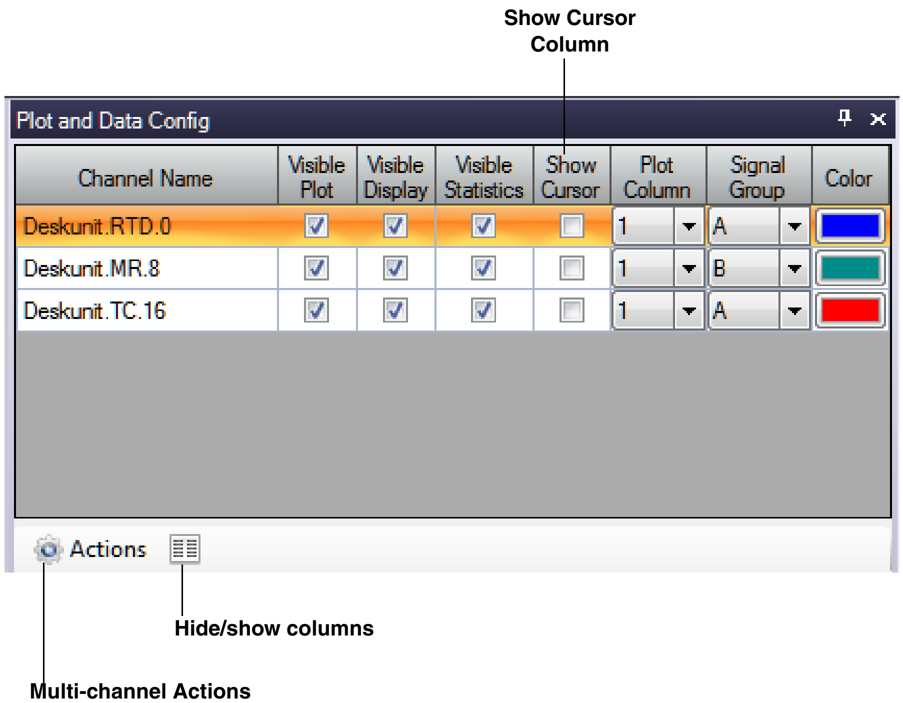
If you wish, you can change whether the **Visible Plot** column is shown or hidden by selecting the **Hide/show columns** () button, and selecting **Visible Plot**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Visible Plot Column

If you wish, you can move the position of the **Visible Plot** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Displaying Data Cursors in the Channel Plot Window

You can determine whether data cursors are displayed in the Channel Plot window by using the following controls in the Plot and Data Config window:



For each of the enabled channels, you can choose to display a data cursor in the Channel Plot window using the **Show Cursor** column in the Plot and Data Config window.

If a box is checked under the **Show Cursor** column, a data cursor for the associated channel is displayed in the Channel Plot window. If a box under the **Show Cursor** column is unchecked, a data cursor for the associated channel is not displayed in the Channel Plot window.

You can enable the capability of moving the data cursor using the data-cursor control ( ), described on page 74. Various options are available for changing the cursor style, line style, and cursor color.

Making All Selected Data Cursors Visible or Hidden at Once

If you want to make all selected data cursors visible in the Channel Plot window at once, select the **Multi-channel Actions** button (Actions), and choose **Selected data cursors visible**.

If you want to hide all selected data cursors in the Channel Plot window at once, select the **Multi-channel Actions** button (Actions), and choose **Selected data cursors hidden**.

Showing or Hiding the Show Cursor Column

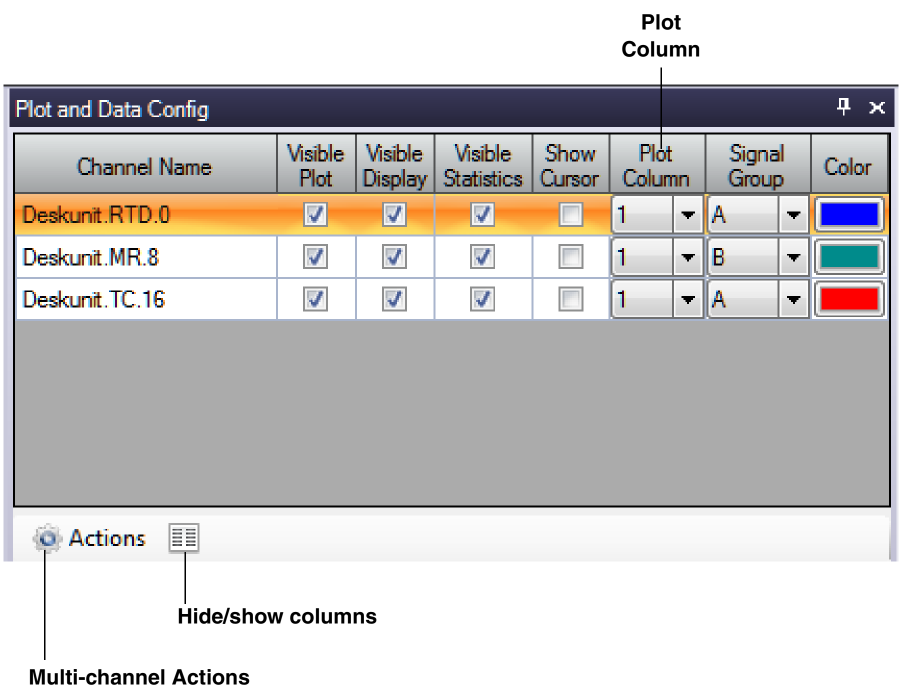
If you wish, you can change whether the **Show Cursor** column is shown or hidden by selecting the **Hide/show columns** ( ) button, and selecting **Show Cursor**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Show Cursor Column

If you wish, you can move the position of the **Show Cursor** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

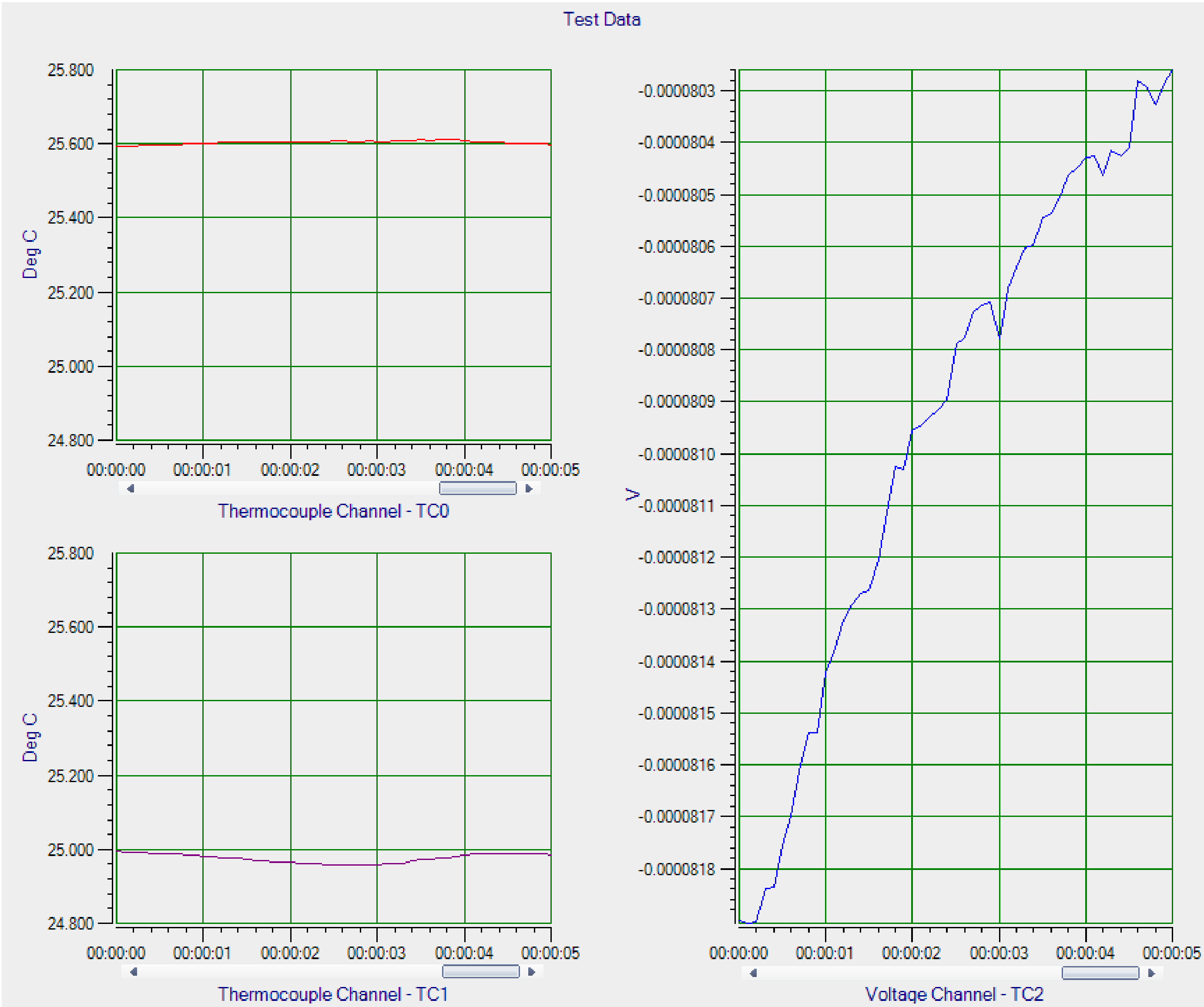
Specifying the Columns of the Channel Plot Window

You can determine how many columns are displayed in the Channel Plot window by using the following controls in the Plot and Data Config window:



You can specify up to six columns to view in the Channel Plot window.

Under **Plot Column**, select the column (1 to 6) in which to display the selected channel. The following example shows a display of two plot columns, where the thermocouple channels are displayed in column 1 and voltage channels are displayed in column 2:



Changing the Plot Column for All Selected Channels at Once

If you want to change the plot column for all selected channels at once, do the following:

1. Highlight the channels in the Plot and Data Config window that you want to change.
2. Select the **Multi-channel Actions** button ().
3. Choose **Selected channels Plot Column**.
4. Enter the number of plot columns to use.

Showing or Hiding the Plot Column

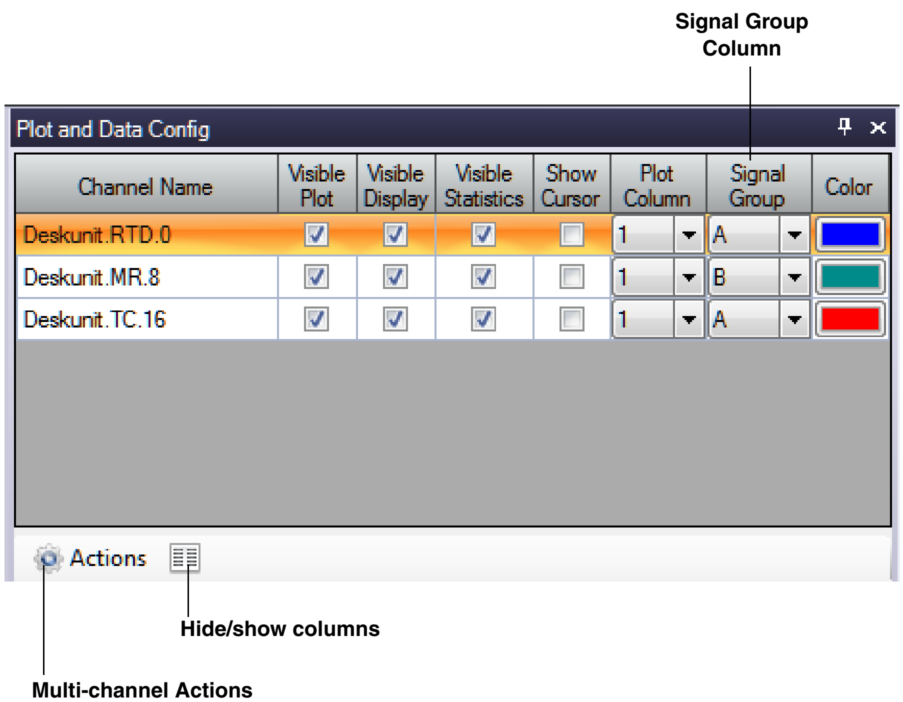
If you wish, you can change whether the **Plot Column** is shown or hidden by selecting the **Hide/show columns** ( ) button, and selecting **Plot Column**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Plot Column

If you wish, you can move the position of the **Plot Column** by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Grouping Signals in Bands on the Channel Plot Window

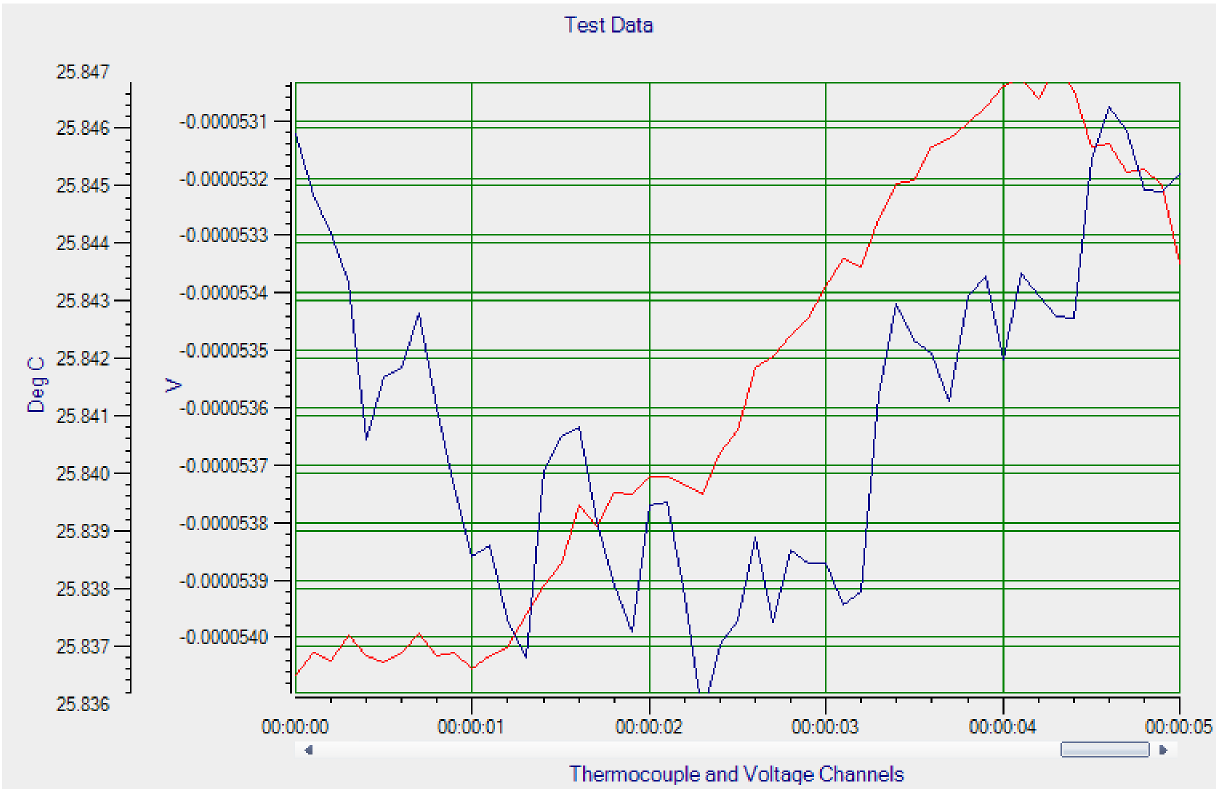
You can group signals in bands in the Channel Plot window by using the following controls in the Plot and Data Config window:



You can choose to display multiple signals in the same band by assigning the same **Signal Group** letter to them. You can specify a value of **A** through **T** for the Signal Group.

For example, to see the data from analog input signals 1 and 2 in the same band, you could set the **Signal Group** for both channels to **A**.

If two analog input channels have a different plot data type (for example, one is temperature and the other is voltage), two Y-axes are displayed for the band, as shown in the following example:



Changing the Signal Group for All Selected Channels at Once

If you want to change the value of the signal group for all selected channels at once, do the following:

1. Highlight the channels in the Plot and Data Config window that you want to change.
2. Select the **Multi-channel Actions** button (Actions).
3. Choose **Selected channels Signal Group**.
4. Enter the value of the signal band to use.

Showing or Hiding the Signal Group Column

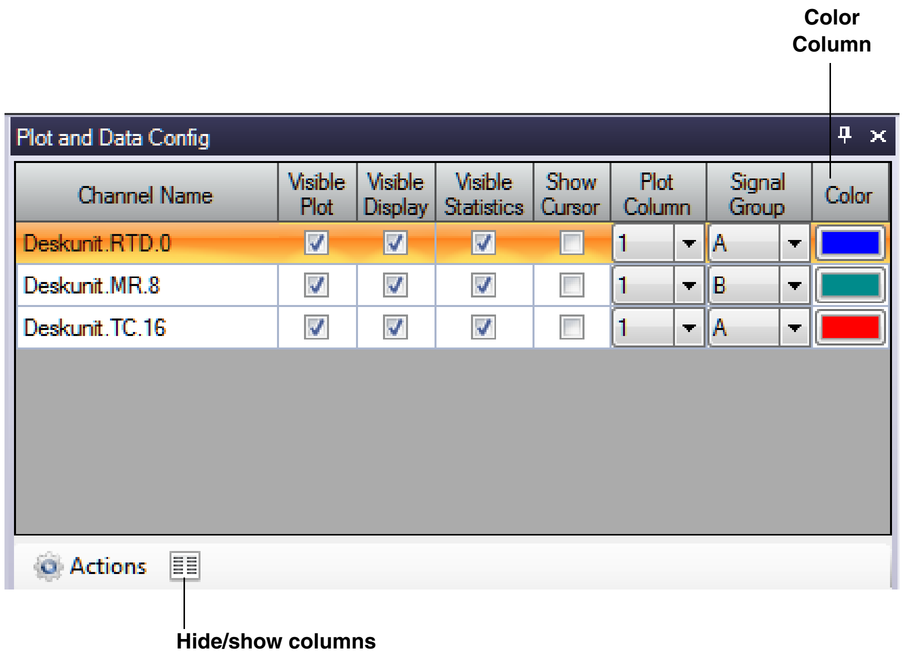
If you wish, you can change whether the **Signal Group** column is shown or hidden by selecting the **Hide/show columns** ( ) button, and selecting **Signal Group**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Signal Group Column

If you wish, you can move the position of the **Signal Group** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Specifying the Trace Color on the Channel Plot Window

You can specify the color of the trace on the Channel Plot window by using the following controls in the Plot and Data Config window:



For each channel, select the **Color** column, and then select the color that you want to display for the trace in the Channel Plot window.

Showing or Hiding the Color Column

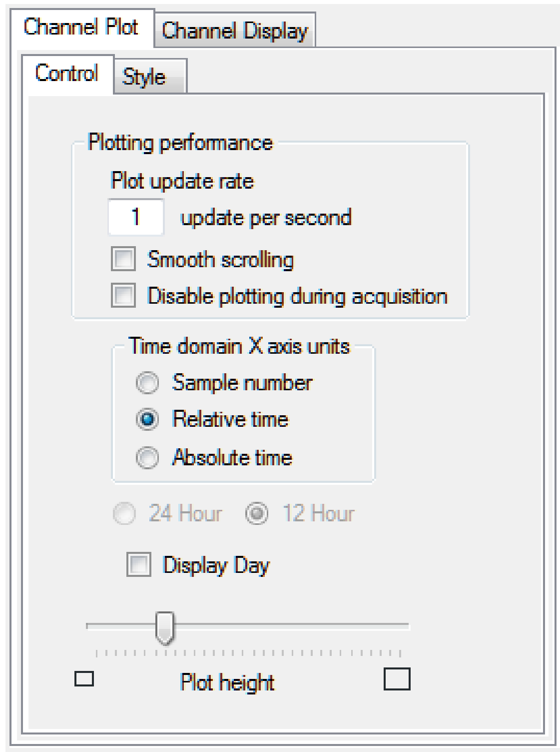
If you wish, you can change whether the **Color** column is shown or hidden by selecting the **Hide/show columns** (▤) button, and selecting **Color**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Color Column

If you wish, you can move the position of the **Color** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Controlling How Data is Plotted in the Channel Plot Window

Using the Channel Plot Control tab in the Plot and Data Config window, you can control how data is plotted in the Channel Plot window.



You can configure the following settings:

- **Plot update rate**, – Enter a value between 1 and 50 to indicate how often you want the display to be refreshed per second.
- **Smooth scrolling –** You can adjust how the plot scrolls to the left using the **Smooth scrolling** checkbox. If **Smooth scrolling** is checked, the plot attempts to scroll after each data point is added. However, scrolling is limited by the actual update rate. For example, if the **Plot update rate** is set to 50, and 100 points are added per second (determined by the **Recording** settings), the plot scrolls to the left by 2 samples, 50 times per second.

  If the **Smooth scrolling** checkbox is unchecked, the plot does not attempt to scroll after every point is added; instead, it scrolls after each buffer is plotted. The buffer size is set to the FFT size.
- **Disable plotting during acquisition** – This checkbox is provided for fast data acquisition boards, such as the DT9832, which acquire data faster than the application can display it.

  If the **Disable plotting during acquisition** checkbox is checked, no data is displayed in the Channel Plot window during acquisition. When acquisition is finished, the data is displayed in the Channel Plot window.

  If the **Disable plotting during acquisition** checkbox is unchecked, the data is displayed in the Channel Plot window as the data is acquired.

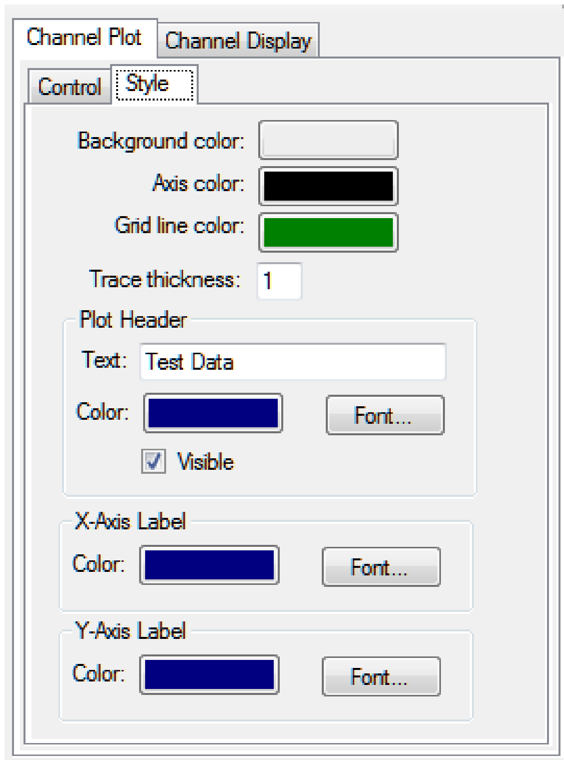
- Under **Time domain X axis units**, select the units for the x-axis as one of the following:
  - **Sample number –** The units for the x-axis are represented as the number of samples (0 to the maximum number of samples acquired).
  - **Relative time –** The units for the x-axis are represented as the recording time from 0 to the total recording time.
  - **Absolute time –** The units for the x-axis are represented as the actual time of acquisition based on the system clock. If you select absolute time, you can select whether to represent the time in 24-hour (military) format or 12-hour format.
- **Display Day** – If **Absolute Time** is selected and you check the **Display Day** checkbox, the x-axis shows dd:hh:mm:ss, where *dd* is the day of the month, *hh* is hours, *mm* is minutes, and *ss* is seconds. If **Relative Time** is selected, the value for day starts at 0 and increments by one for each subsequent day. You can also set the value for day in the x-axis options if the **Display Day** checkbox is selected.

Note: You can change the resolution or precision of the x-axis, as described on page 69.

- Use the **Plot height** slider bar, to determine the relative height of each plot in the Channel Plot window. Slide the control to the right to increase the height; slide the control to the left to decrease the height. Note that the entire screen is always filled, so, typically, this setting is useful when you want to view many plots at once.

Specifying the Style of the Plots in the Channel Plot Window

The Channel Plot Style tab allows you to specify the style of the plots that are displayed in the Channel Plot window.



You can configure the following settings:

- Use the **Background Color** box to choose a different color for the background of the Channel Plot window.
- Use the **Axis color** box to choose a different color for the axes of the Channel Plot window.
- Use the **Grid line color** box to choose a different color for the grid lines of the Channel Plot window.
- In the **Trace thickness** box, enter the value between 1 and 5 to set the thickness of the line used by the trace on the Channel Plot window, where 1 is the thinnest line and 5 is the thickest line.
- Under **Plot Header**, click the **Visible** checkbox if you want a header displayed for the data on the plot, and then enter the text for the header. By default, the header is called *Test Data*. You can also select the color for the header and the font in which to display the header.
- Under **X-Axis Label**, you can select the color for the label of the x-axis and the font in which to display the x-axis label.
- Under **Y-Axis Label**, you can select the color for the label of the y-axis and the font in which to display the y-axis label.

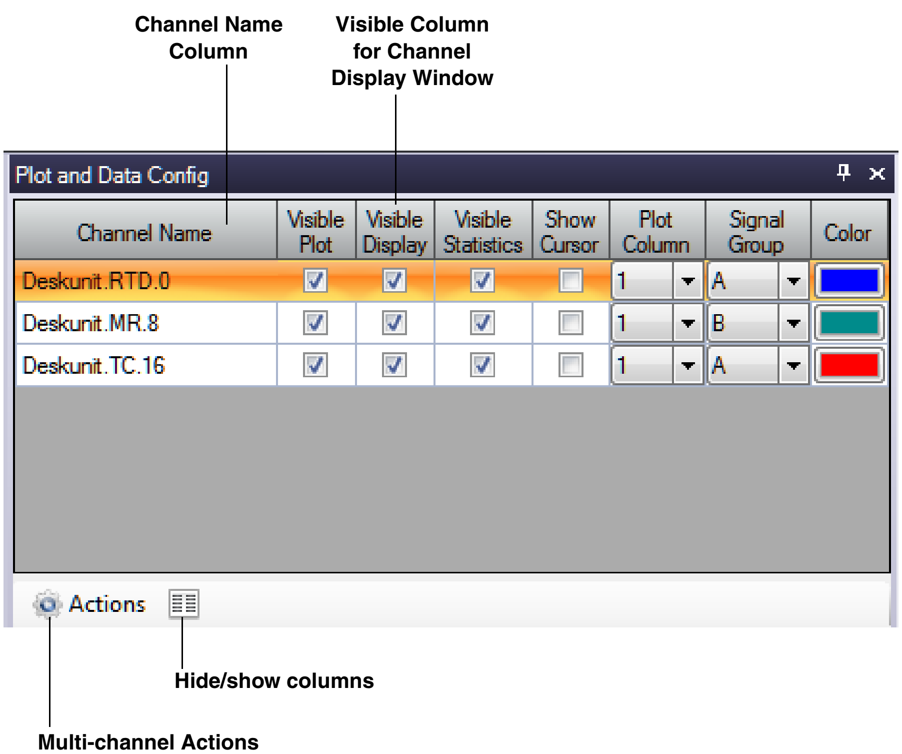
Configuring the Appearance of the Channel Display Window

This section describes the how to configure the appearance of the Channel Display window, when using the Data Logger interface, and includes the following topics:

- Displaying channels in the Channel Display window, described on this page
- Controlling how data is displayed in the Channel Display window, described on page 182
- Specifying the style of the Channel Display window, described on page 184

Displaying Channels in the Channel Display Window

You can determine which channels are displayed in the Channel Display window by using the following controls in the Plot and Data Config window:



You can see the names of all enabled channels under the **Channel Name** column in the Plot and Data Config window. Only data from the list of enabled channels is acquired.

Note: If desired, you can change the name of a channel using the **Input Channel Configuration** menu option, described on page 84.

For each enabled channel, you can control which channels are displayed in the Channel Display window using the **Visible Display** checkbox. If a box is checked under the **Visible Display** column, the data for the associated channel is displayed in the Channel Display window. If a box under the **Visible Display** column is unchecked, the data for the associated channel is not displayed in the Channel Display window.

Note: Data for all enabled channels is recorded regardless of whether the channel is visible.

Making All Selected Channels Visible or Hidden in the Channel Display Window at Once

If you want to make all selected channels visible in the Channel Display window at once, select the **Multi-channel Actions** button (

 ), and choose **Selected channels visible in Channel Display**.

If you want to hide all selected channels in the Channel Display window at once, select the **Multi-channel Actions** button (

 ), and choose **Selected channels hidden in Channel Display**.

Showing or Hiding the Visible Display Column

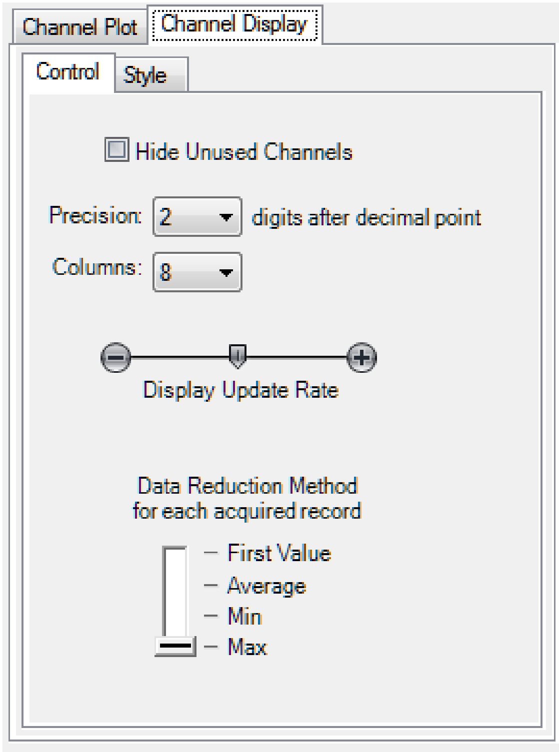
If you wish, you can change whether the **Visible Display** column is shown or hidden by selecting the **Hide/show columns** () button, and selecting **Visible Display**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Visible Display Column

If you wish, you can move the position of the **Visible Display** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Controlling How Data is Displayed in the Channel Display Window

Using the **Channel Display - Control** tab in the Plot and Data Config window, you can control how data is displayed in the Channel Display window.



You can configure the following settings:

- **Hide Unused Channels** – By default, the Channel Display window resembles the front panel of the device and shows up to 48 channels. However, if you want your display to show only the enabled channels rather than all the channels, check the **Hide Unused Channels** box. The following example illustrates how the display looks when three channels are enabled and the **Hide Unused Channels** box is checked.

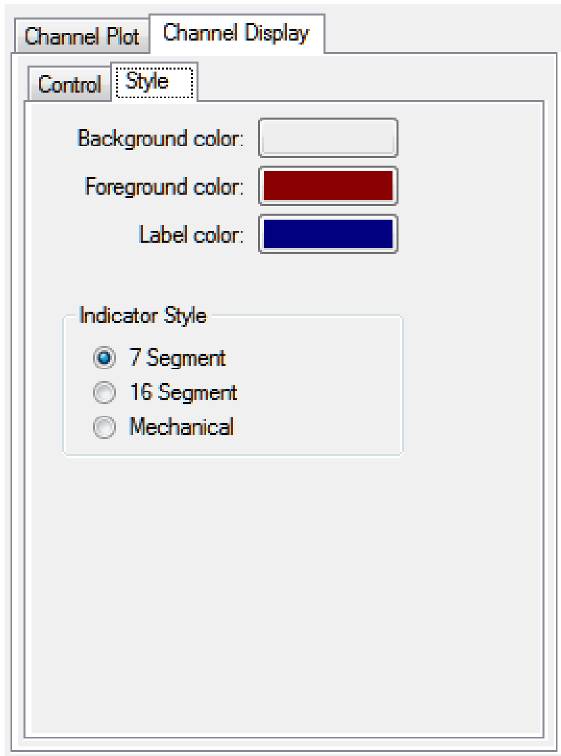
- **Precision** – You can select the number of significant digits after the decimal point to display; values range from 1 to 8 (2 is the default).

- **Columns –** You can specify how many columns to use when displaying the channels; values range from 1 to 16 (8 is the default). For example, by choosing 1, shown below, you can display each channel, one on top of the other (note that the lowest numbered channel is always at the bottom of the display and the highest numbered channel is at the top of the display):

- **Display Update Rate –** You can choose how fast you want the display of the Channel Display window to be updated. Slide the bar to the left for faster update rates, or slide the bar to the right for slower update rates.
- **Data Reduction Method –** You can choose which value to display in the Channel Display window:
  - **First Value –** The first value in the most recent buffer is displayed for each channel.
  - **Average –** The average value in the most recent buffer is displayed for each channel.
  - **Min –** The minimum value in the most recent buffer is displayed for each channel.
  - **Max –** The maximum value in the most recent buffer is displayed for each channel.

Specifying the Style of the Channel Display Window

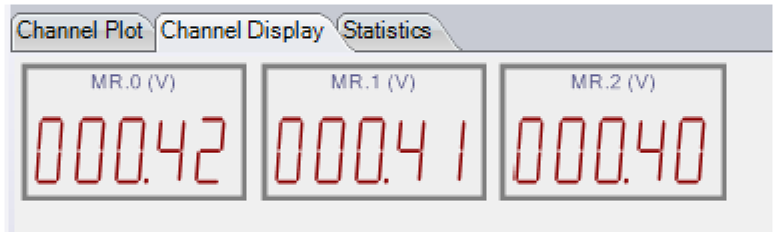
Using the Channel Display Style tab, you can specify the style of the Channel Display window.



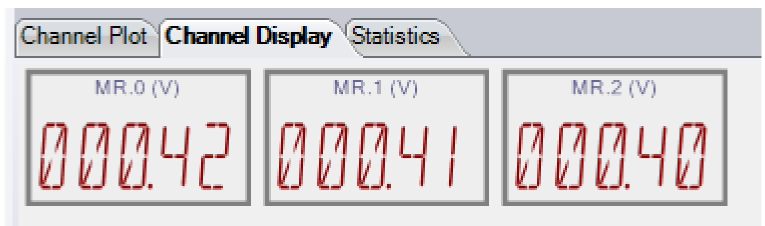
You can configure the following parameters:

- **Background color** – Allows you to choose a specific color for the background of the Channel Display window. You can select from a number of predefined colors or you can define your own color for the background.
- **Foreground color** – Allows you to choose a specific color for the foreground of the Channel Display window. You can select from a number of predefined colors or you can define your own color for the foreground.
- **Label color** – Allows you to choose a specific color for the label used on the Channel Display window. You can select from a number of predefined colors or you can define your own color for the label.

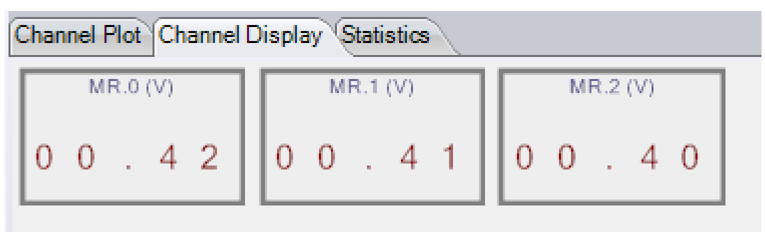
- **Indicator style –** Allows you to specify how the numbers and letters that represent the data appear. The following choices are available:
  - **7 Segment –** The numbers and letters that represent the data are composed of 7-segment lines.



  - **16 Segment –** The numbers and letters that represent the data are composed of 16-segment lines.

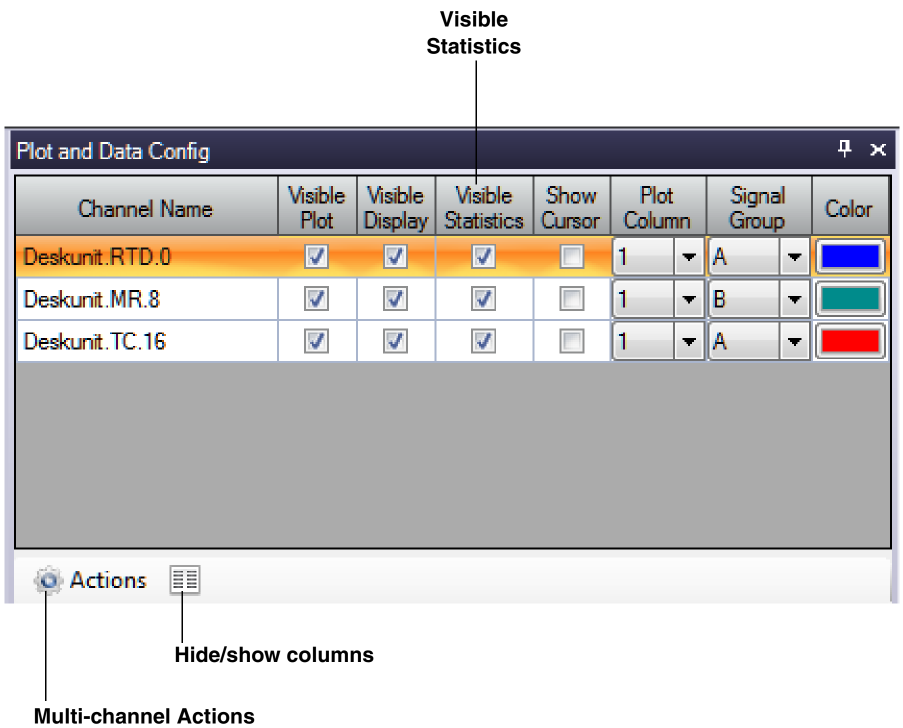


  - **Mechanical –** The numbers and letters that represent the data appear to be formed by a contiguous line.



Configuring the Appearance of the Statistics Window

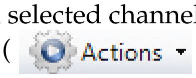
You can determine whether statistics are displayed in the Statistics window by using the following controls in the Plot and Data Config window:



For each enabled channel listed under the **Channel Name** column, you can control whether statistics are displayed in the Statistics window using the **Visible Statistics** checkbox. If a box is checked under the **Visible Statistics** column, the statistics for the associated channel are listed in the Statistics window. If a box under the **Visible Statistics** column is unchecked, the statistics for the associated channel are not displayed in the Statistics window.

Making Statistics for All Selected Channels Visible or Hidden in the Statistics Window at Once

If you want to make statistics for all selected channels visible in the Statistics window at once, select the **Multi-channel Actions** button (), and choose **Selected channels visible in Statistics Display**.

If you want to hide statistics for all selected channels in the Statistics window at once, select the **Multi-channel Actions** button (), and choose **Selected channels hidden in Statistics Display**.

Showing or Hiding the Visible Statistics Column

If you wish, you can change whether the **Visible Statistics** column is shown or hidden by selecting the **Hide/show columns** ( ) button, and selecting **Visible Statistics**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Visible Statistics Column

If you wish, you can move the position of the **Visible Statistics** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

6

Configuring the Display of the FFT Analyzer Interface

Introduction

This chapter describes how to configure the display when using the FFT Analyzer interface. If you are using the Data Logger interface, refer to Chapter 5 starting on page 167.

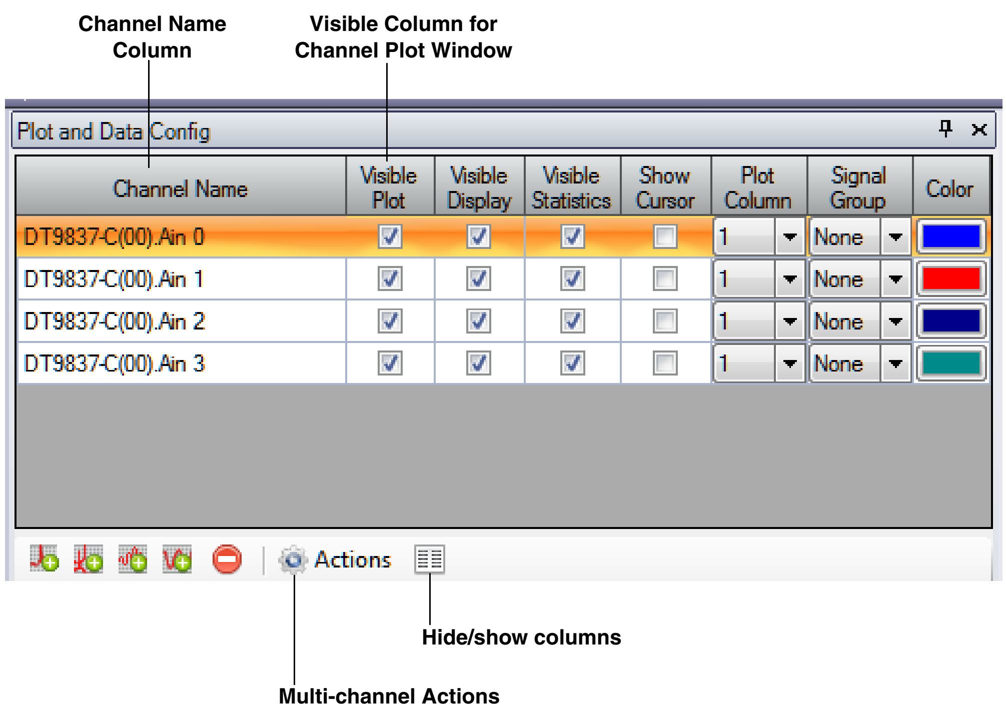
Configuring the Appearance of the Channel Plot Window

This section describes the following aspects related to the appearance of the Channel Plot window when using the FFT Analyzer interface:

- Displaying channels in the Channel Plot window, described on this page
- Displaying data cursors in the Channel Plot window, described on page 193
- Specifying the columns of the Channel Plot window, described on page 194
- Grouping signals in bands on the Channel Plot window, described on page 196
- Specifying the trace color on the Channel Plot window, described on page 198
- Controlling how data is plotted the Channel Plot window, described on page 199
- Controlling the style of the plots in the Channel Plot window, described on page 200

Displaying Channels in the Channel Plot Window

You can determine which channels are displayed in the Channel Plot window by using the following controls in the Plot and Data Config window:



You can see the names of all enabled channels under the **Channel Name** column in the Plot and Data Config window. Only data from the list of enabled channels is acquired.

Note: If desired, you can change the name of a channel using the **Input Channel Configuration** menu option or toolbar button, described on page 88.

For each enabled channel listed under the **Channel Name** column, you can control which channels are displayed in the Channel Plot window using the **Visible Plot** checkbox. If a box is checked under the **Visible Plot** column, a plot for the associated channel is displayed in the Channel Plot window. If a box under the **Visible Plot** column is unchecked, the plot for the associated channel is not displayed in the Channel Plot window.

Note: Data for all enabled channels is recorded regardless of whether the channel is visible.

Making All Selected Channels Visible or Hidden in the Channel Plot Window at Once

If you want to make all selected channels visible in the Channel Plot window at once, select the **Multi-channel Actions** button (Actions), and choose **Selected channels visible in Channel Plot**.

If you want to hide all selected channels in the Channel Plot window at once, select the **Multi-channel Actions** button (Actions), and choose **Selected channels hidden in Channel Plot**.

Showing or Hiding the Visible Plot Column

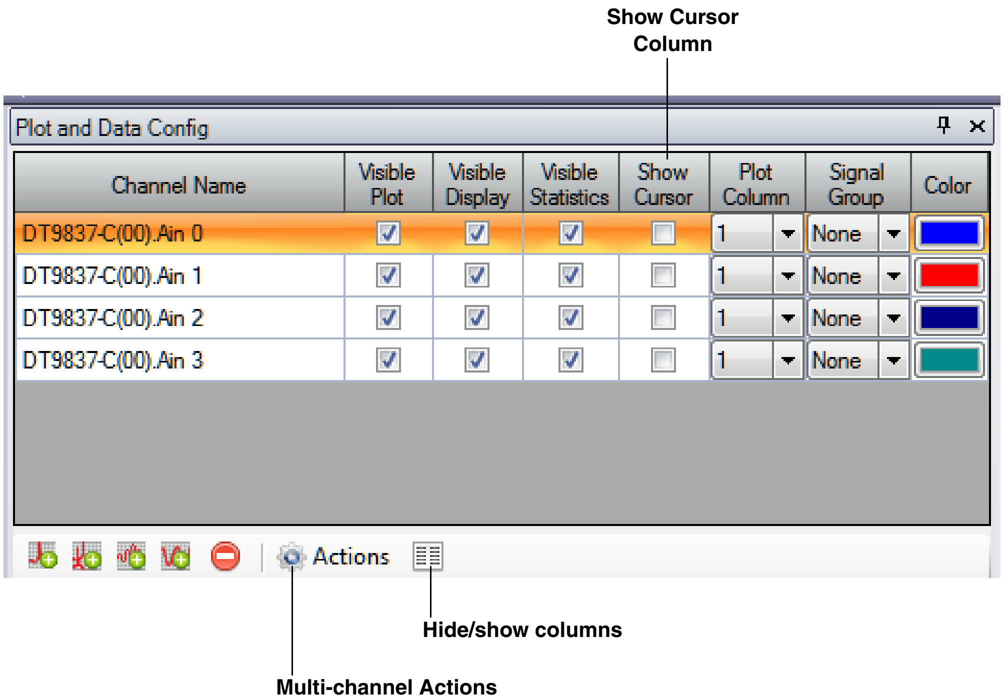
If you wish, you can change whether the **Visible Plot** column is shown or hidden by selecting the **Hide/show columns** ( ) button, and selecting **Visible Plot**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Visible Plot Column

If you wish, you can move the position of the **Visible Plot** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Displaying Data Cursors in the Channel Plot Window

You can determine whether data cursors are displayed in the Channel Plot window by using the following controls in the Plot and Data Config window:



For each of the enabled channels, you can choose to display a data cursor in the Channel Plot window using the **Show Cursor** column in the Plot and Data Config window.

If a box is checked under the **Show Cursor** column, a data cursor for the associated channel is displayed in the Channel Plot window. If a box under the **Show Cursor** column is unchecked, a data cursor for the associated channel is not displayed in the Channel Plot window.

You can enable the capability of moving the data cursor using the data-cursor control ( ), described on page 74. Various options are available for changing the cursor style, line style, and cursor color.

Making All Selected Data Cursors Visible or Hidden at Once

If you want to make all selected data cursors visible in the Channel Plot window at once, select the **Multi-channel Actions** button (Actions), and choose **Selected data cursors visible**.

If you want to hide all selected data cursors in the Channel Plot window at once, select the **Multi-channel Actions** button (Actions), and choose **Selected data cursors hidden**.

Showing or Hiding the Show Cursor Column

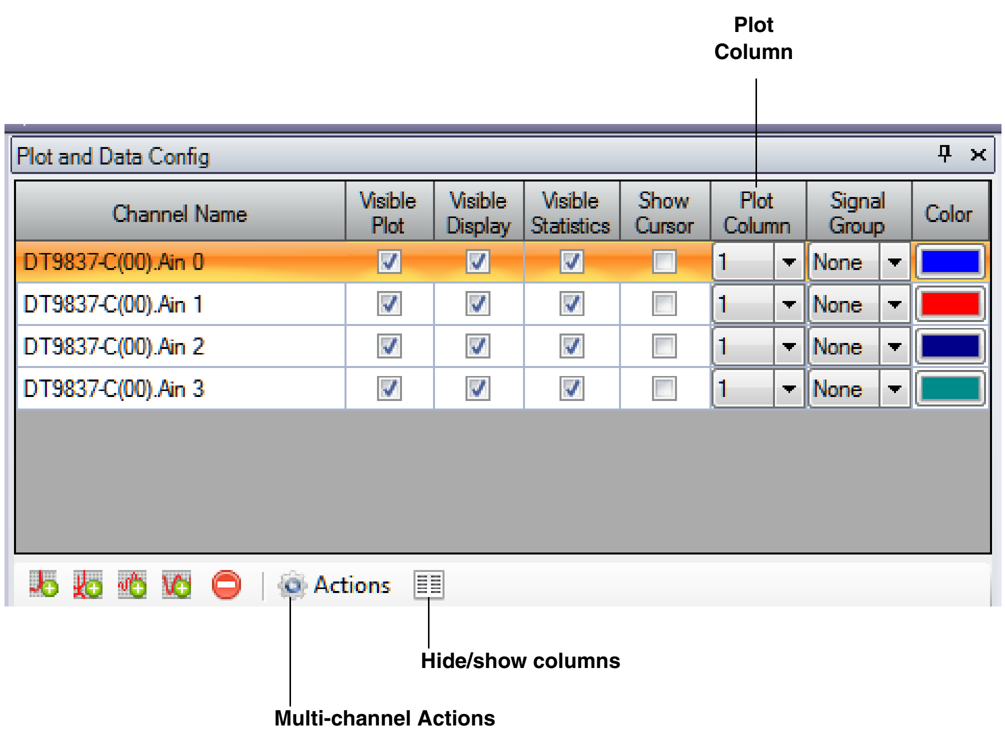
If you wish, you can change whether the **Show Cursor** column is shown or hidden by selecting the **Hide/show columns** ( ) button, and selecting **Show Cursor**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Show Cursor Column

If you wish, you can move the position of the **Show Cursor** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

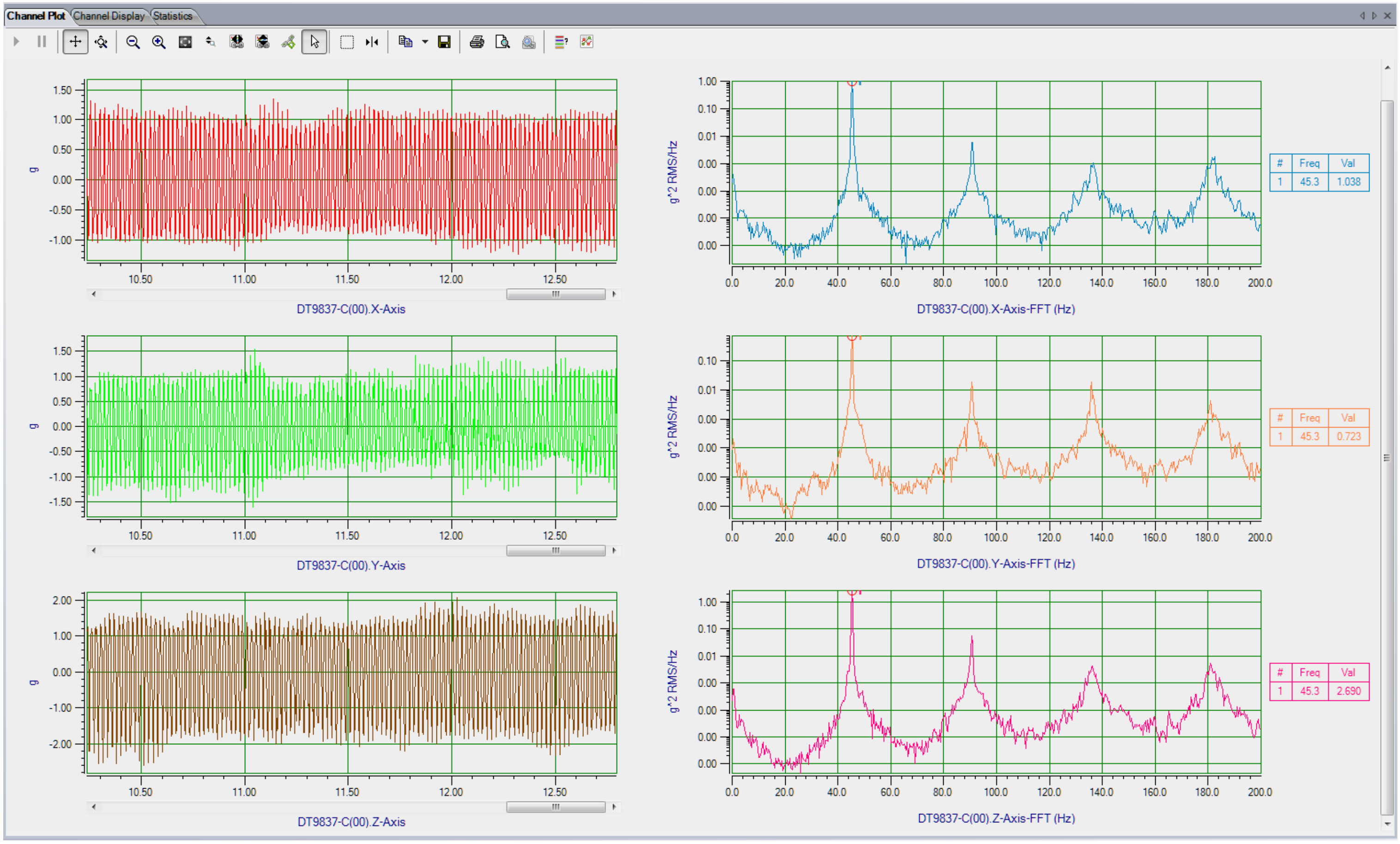
Specifying the Plot Columns of the Channel Plot Window

You can determine how many columns are displayed in the Channel Plot window by using the following controls in the Plot and Data Config window:



You can specify up to six columns to view in the Channel Plot window.

Under **Plot Column**, select the column (1 to 6) in which to display the selected channel. The following example shows a display of two plot columns, where the time domain channels are displayed in column 1 and the FFT channels are displayed in column 2:



Changing the Plot Column for All Selected Channels at Once

If you want to change the plot column for all selected channels at once, do the following:

1. Highlight the channels in the Plot and Data Config window that you want to change.
2. Select the **Multi-channel Actions** button (Actions).
3. Choose **Selected channels Plot Column**.
4. Enter the number of the plot column to use.

Showing or Hiding the Plot Column

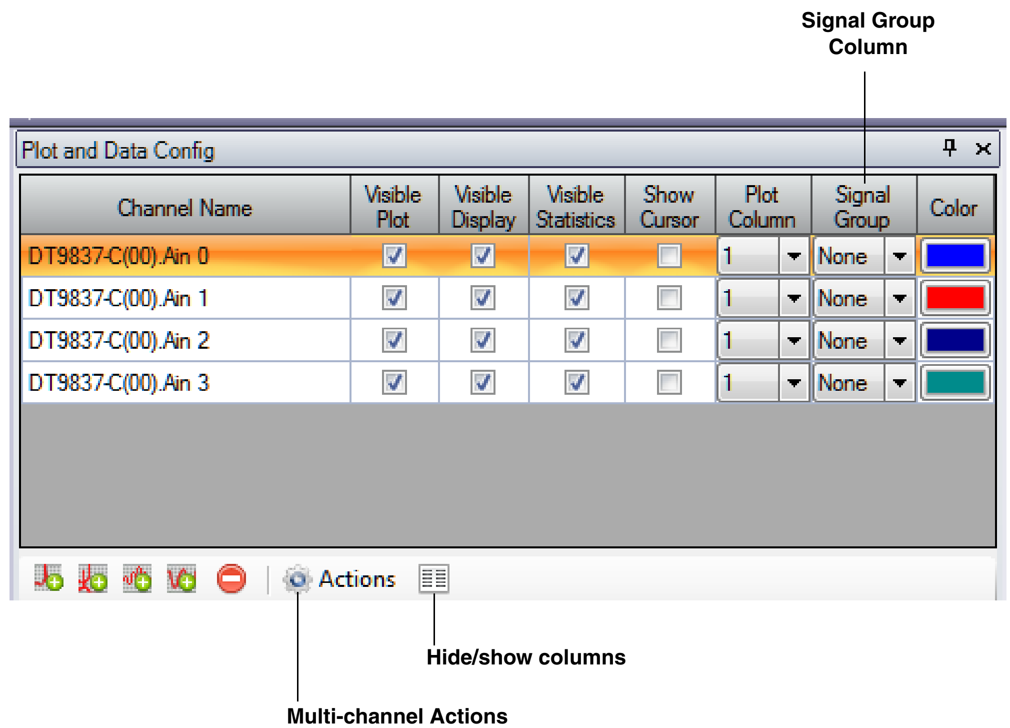
If you wish, you can change whether the **Plot Column** is shown or hidden by selecting the **Hide/show columns** ( ) button, and selecting **Plot Column**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Plot Column

If you wish, you can move the position of the **Plot Column** by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Grouping Signals in Bands on the Channel Plot Window

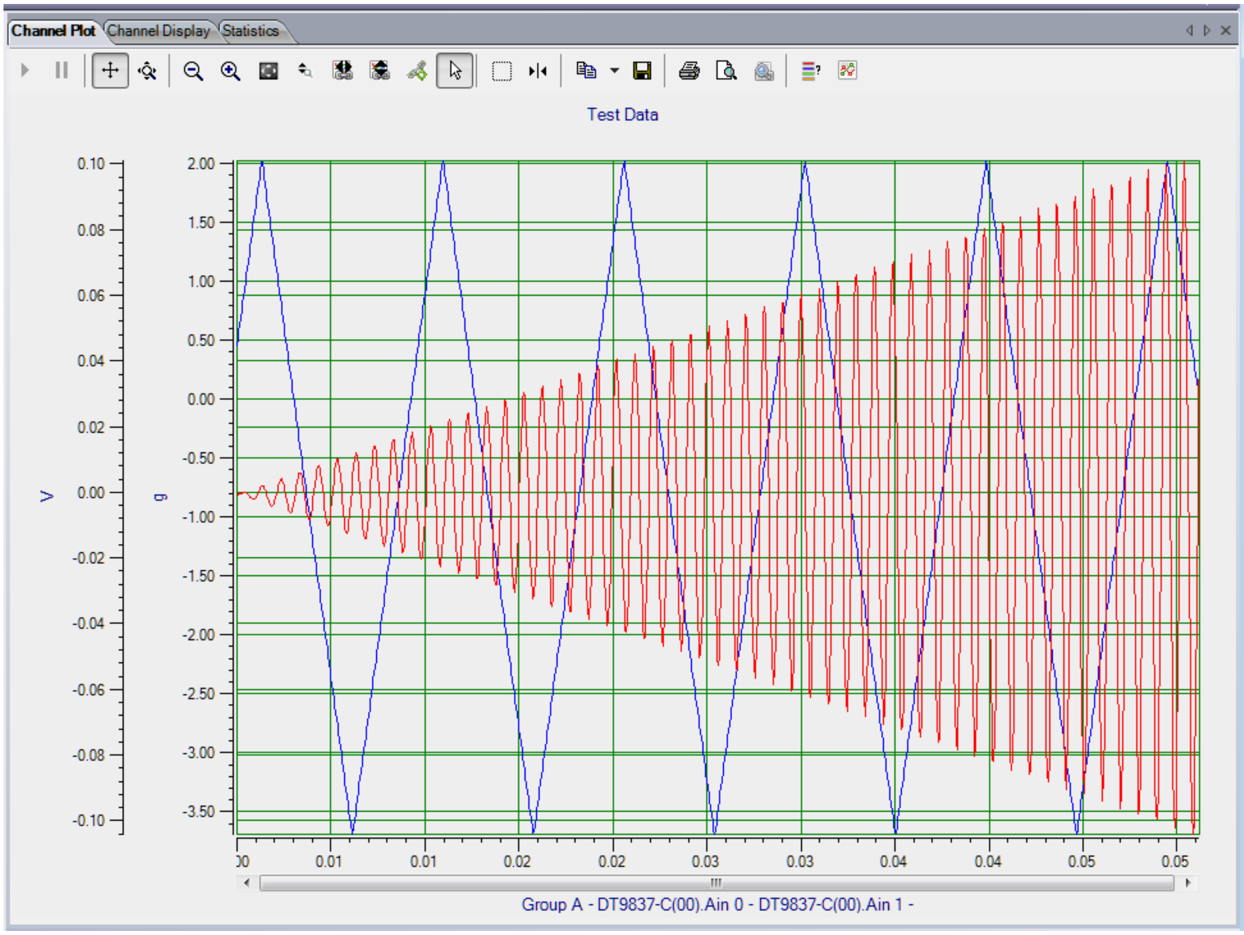
You can group signals in bands in the Channel Plot window by using the following controls in the Plot and Data Config window:



You can choose to display multiple signals in the same band by assigning the same **Signal Group** letter to them. You can specify a value of **A** through **T** for the Signal Group.

For example, to see the data from analog input signals 1 and 2 (in the time domain) in the same band, you could set the **Signal Group** for both channels to **A**.

If two analog input channels have a different plot data type (for example, one is voltage and the other is a sensor), two Y-axes are displayed for the band, as shown in the following example:



Likewise, if two FFT channels have different parameters, two y-axis are displayed for the band.

Note that the **Signal Group** letter is shown in the legend beneath the x-axis.

Changing the Signal Group for All Selected Channels at Once

If you want to change the value of the signal group for all selected channels at once, do the following:

1. Highlight the channels in the Plot and Data Config window that you want to change.
2. Select the **Multi-channel Actions** button ().
3. Choose **Selected channels Signal Group**.
4. Enter the value of the signal band to use.

Showing or Hiding the Signal Group Column

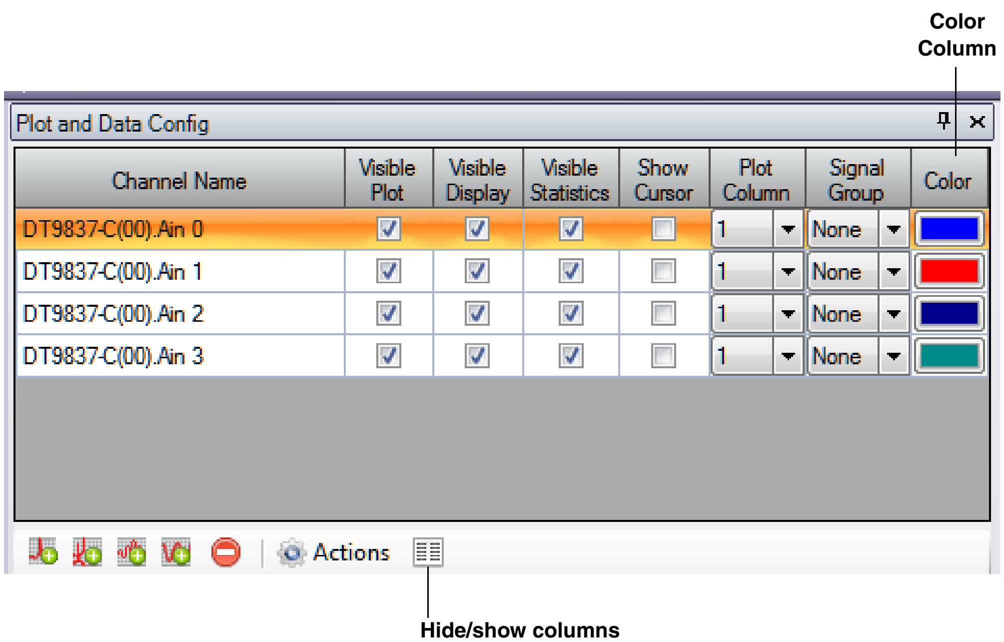
If you wish, you can change whether the **Signal Group** column is shown or hidden by selecting the **Hide/show columns** () button, and selecting **Signal Group**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Signal Group Column

If you wish, you can move the position of the **Signal Group** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Specifying the Trace Color on the Channel Plot Window

You can specify the color of the trace on the Channel Plot window by using the following controls in the Plot and Data Config window:



For each channel, select the **Color** column, and then select the color that you want to display for the trace in the Channel Plot window.

Showing or Hiding the Color Column

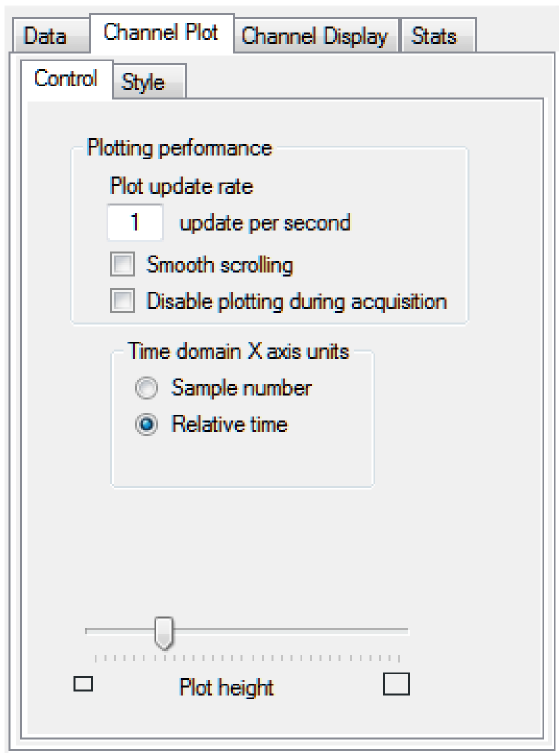
If you wish, you can change whether the **Color** column is shown or hidden by selecting the **Hide/show columns** ( ) button, and selecting **Color**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Color Column

If you wish, you can move the position of the **Color** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Controlling How Data is Plotted in the Channel Plot Window

Using the Channel Plot Control tab in the Plot and Data Config window, you can control how data is plotted in the Channel Plot window.



You can configure the following settings:

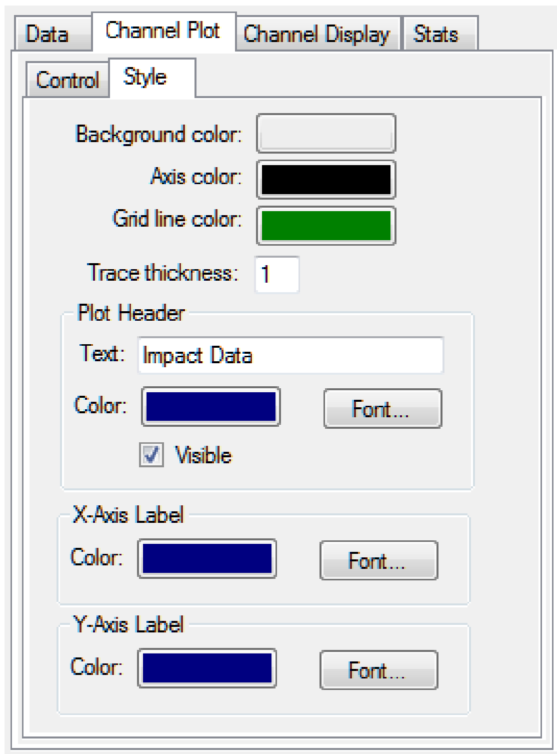
- **Plot update rate**, – Enter a value between 1 and 50 to indicate how often you want the display to be refreshed per second.
- **Smooth scrolling –** You can adjust how the plot scrolls to the left using the **Smooth scrolling** checkbox. If **Smooth scrolling** is checked, the plot attempts to scroll after each data point is added. However, scrolling is limited by the actual update rate. For example, if the **Plot update rate** is set to 50, and 100 points are added per second (determined by the **Recording** settings), the plot scrolls to the left by 2 samples, 50 times per second.

  If the **Smooth scrolling** checkbox is unchecked, the plot does not attempt to scroll after every point is added; instead, it scrolls after each buffer is plotted. The buffer size is set to the FFT size.
- **Disable plotting during acquisition**, – You can determine whether the Channel Plot display updates during acquisition using the **Disable plotting during acquisition checkbox**. When checked, the Channel Plot display is updated only after acquisition has stopped. When unchecked, the Channel Plot display during acquisition.
- Under **Time domain X axis units**, select the units for the x-axis as either the number of samples (0 to the maximum number of samples acquired) or the relative time (0 to the total recording time).

- Use the **Plot height** slider bar, to determine the relative height of each plot in the view. Slide the control to the right to increase the height; slide the control to the left to decrease the height. Note that the entire screen is always filled, so, typically, this setting is useful when you want to view many plots at once.

Specifying the Style of the Plots in the Channel Plot Window

The Channel Plot Style tab allows you to specify the style of the plots that are displayed in the Channel Plot window.



You can configure the following settings:

- Use the **Background Color** box to choose a different color for the background of the Channel Plot window.
- Use the **Axis color** box to choose a different color for the axes of the Channel Plot window.
- Use the **Grid line color** box to choose a different color for the grid lines of the Channel Plot window.
- In the **Trace thickness** box, enter the value between 1 and 5 to set the thickness of the line used by the trace on the Channel Plot window, where 1 is the thinnest line and 5 is the thickest line.
- Under **Plot Header**, click the **Visible** checkbox if you want a header displayed for the data on the plot, and then enter the text for the header. By default, the header is called *Test Data*. You can also select the color for the header and the font in which to display the header.

- Under **X-Axis Label**, you can select the color for the label of the x-axis and the font in which to display the x-axis label.
- Under **Y-Axis Label**, you can select the color for the label of the y-axis and the font in which to display the y-axis label.

Adding Channels to the Plot and Data Config Window

If you are using the FFT Analyzer interface, buttons are provided under the channel selection area of the Plot and Data Config window to add channels to the Plot and Data Config window.

If you are using the FFT Analysis option of QuickDAQ, you can add a single FFT channel, a windowed time domain channel, or a time domain channel to the Plot and Data Config window, as shown in Figure 33.

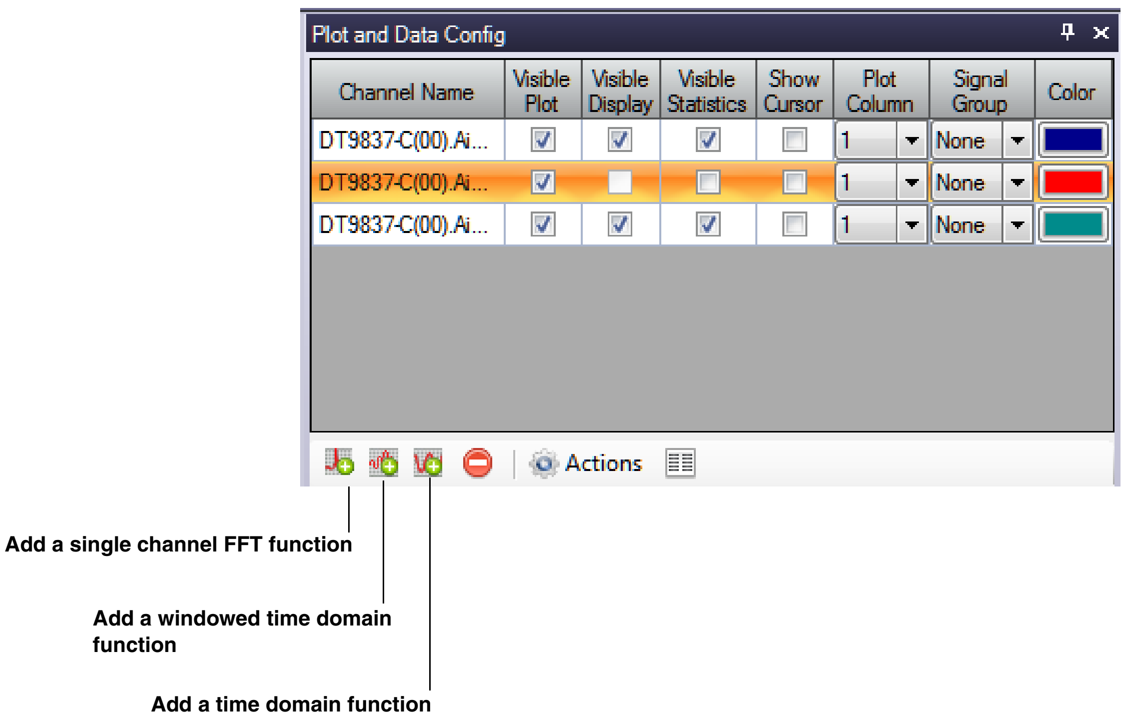


Figure 33: Buttons for Adding Channels in the FFT Analysis Option

If you are using the Advanced FFT Analysis option of QuickDAQ, you can add a single FFT channel, a two-channel FFT function, a windowed time domain channel, or a time domain channel to the Plot and Data Config window, as shown in Figure 34.

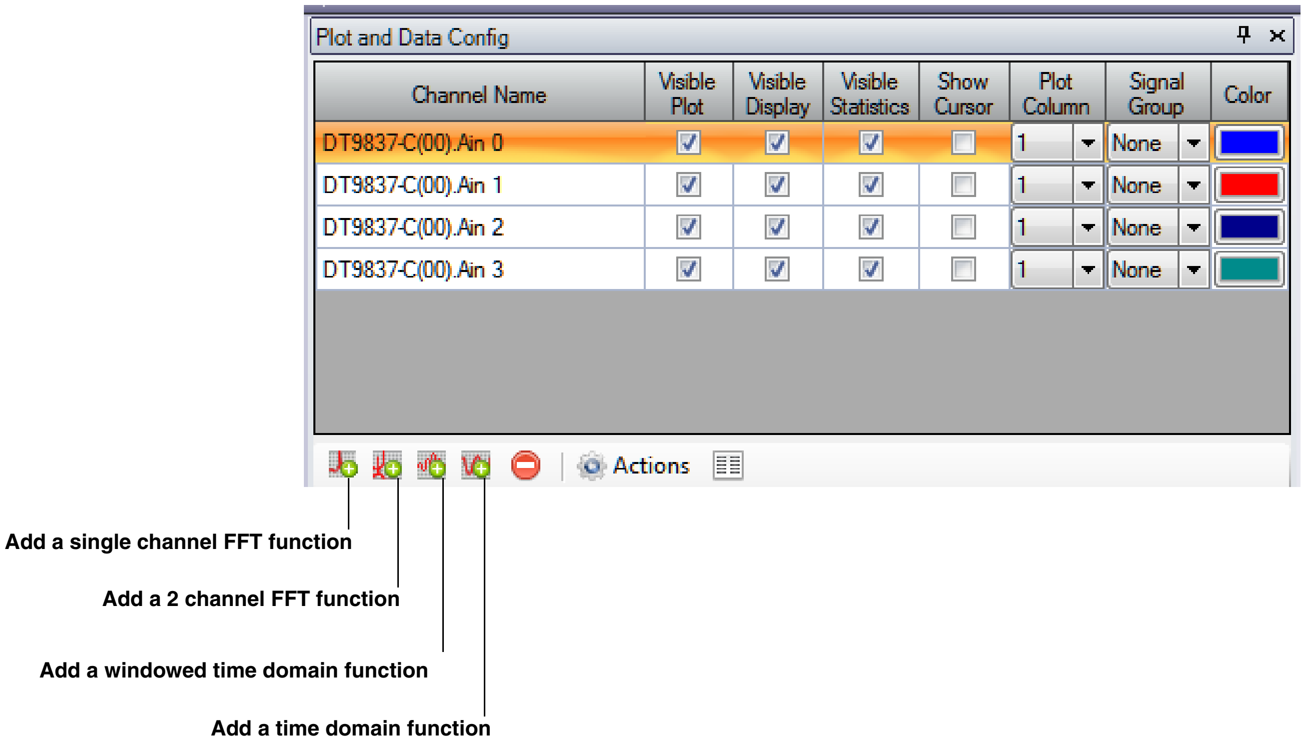


Figure 34: Buttons for Adding Channels in the Advanced FFT Analysis Option

The following subsections describe how to add channel to the Plot and Data Config window.

Adding a Single Channel FFT Function to the Channel List

If you want to add a single channel FFT function to the list of channels shown under **Channel Name**, click the **Add single channel FFT function** ( ) button. Then, choose the analog input channel for which to generate the FFT.

Refer to page 207 for information on configuring the FFT channel once it is added.

Adding a Two-Channel FFT Function to the Channel List

If you are using the Advanced FFT Analysis option of QuickDAQ, you can add two-channel FFT functions to the channel list. Two-channel FFT functions are particularly useful when you want to apply a stimulus to a device and compare the response of the device to a known signal.

Before you can add a two-channel FFT function, you must configure one analog input channel as a response channel (the channel to which the stimulus is applied), and another analog input channel as the reference channel (the channel to which the response channel is compared). Refer to page 84 for more information about configuring channels.

To add a two-channel FFT function to the list of channels shown under **Channel Name**, click the **Add 2 channel FFT function** ( ) button. The following options are available:

- **FRF** – Choose this option to add a frequency response function (FRF) channel. A FRF channel represents the complex ratio between the output (response) signal and the input (reference) signal as a function of frequency.

  When you select **FRF**, you must choose an enabled response channel to compare to the reference channel; the resultant FRF channel is created and listed in the channel list.

  Refer to page 213 for information on configuring a FRF channel once you have added it.

- **Cross Spectrum** – Choose this option if you want to add a Cross Spectrum channel. A cross spectrum channel contains a building block that is used in FRF and other more advanced functions.

  When you select **Cross Spectrum**, you must choose an enabled response channel to compare to the reference channel; the resultant Cross Spectrum channel is created and listed in the channel list.

  Refer to page 218 for information on configuring a Cross Spectrum channel once you have added it.

- **Coherence** – Choose this option if you want to add a coherence channel. A Coherence function evaluates the linearity between the input (reference) signal and the output (response) signal at each frequency, and generates a value between 0.0 and 1.0, where 0 is consistently not linear and 1.0 is consistently linear.

  When you select **Coherence**, you must choose an enabled response channel to compare to the reference channel; the resultant Coherence channel is created and listed in the channel list.

  Refer to page 222 for information on configuring a Coherence channel once you have added it.

Adding a Windowed Time Domain Channel to the List

If you want to see the effect of a window on a time domain channel before doing any FFT processing on the data, click the **Add windowed time domain channel** ( ) button. Then, select the time domain channel to which to apply the window function and from the list of enabled channels. The resulting channel is added to the list of channels shown under **Channel Name**.

You select the window to apply to the channel using the **Window** tab of the Acquisition Config window, described on page 135.

No additional configuration is required for a windowed time domain channel. Therefore, all the options under the **Data** tab are grayed out for windowed time domain channels.

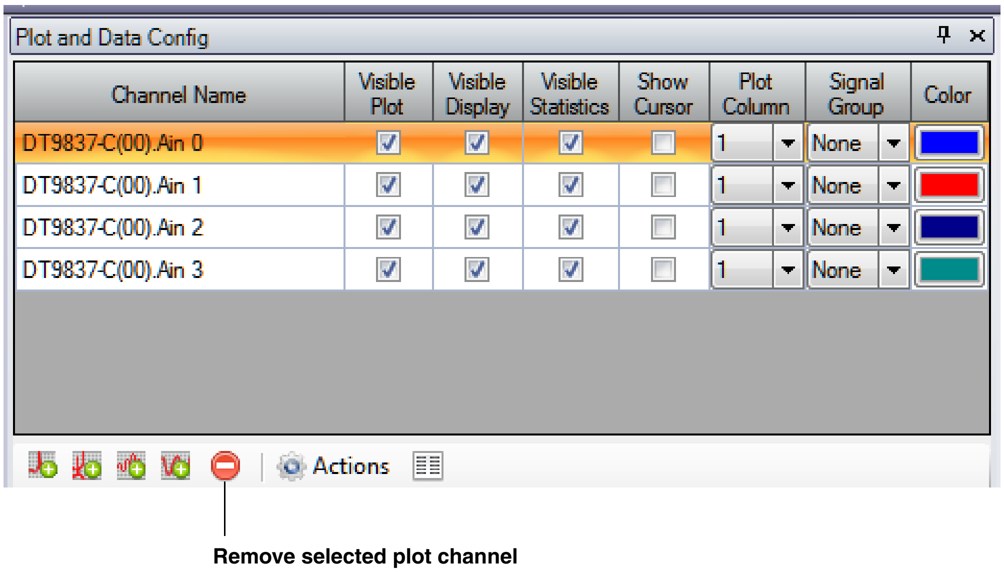
Adding a Time Domain Channel to the List

If you want to add a time domain channel to the list of channels shown under **Channel Name**, click the **Add time domain channel** () button. Then, select a time domain channel to add from the list of enabled channels. The resulting channel is added to the list of channels shown under **Channel Name**.

No additional configuration is required for a time domain channel. Therefore, all the options under the **Data** tab are grayed out for time domain channels.

Removing Channels from the Plot and Data Config Window

If you are using the FFT Analyzer interface, a button is provided under the channel selection area of the Plot and Data Config window to remove a channel from the Plot and Data Config window:

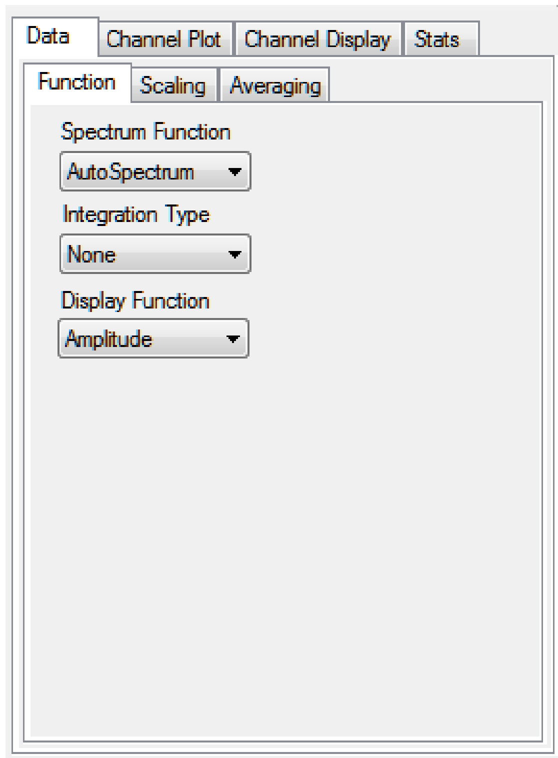


To remove a channel from the list of channels shown under **Channel Name**, click the **Remove selected plot channel** () button, and then click **OK**.

Alternatively, you can select the channel from the list of channels shown under **Channel Name**, click the **Delete** key, and then click **OK** to remove the channel from the Plot and Data Config window.

Configuring Parameters for a Single FFT Channel

If you are using the FFT Analyzer interface, you can add an FFT channel, if desired, as described on page 203. Once you have selected an FFT channel under the **Channel Name** column, you can configure the parameters of the FFT channel using the **Data** tab options:



This section describes the options that appear under the **Data** tab if you select a single channel FFT channel in the Plot and Data Config window.

Note: Once you have acquired data, you can change the parameters under the **Data** tab and see the effect of your changes immediately by checking the **Auto-reprocess on parameter change** checkbox. Or, if this checkbox is not checked, click the **Apply settings to current data** button to see the results.

Function Tab

The **Function** tab provides the following selections: **Spectrum Function**, **Integration Type**, and **Display Function**. These options are described in the following subsections.

Spectrum Function

Under **Spectrum Function**, choose one of the following settings:

- **Spectrum –** Plots the magnitude of a periodic frequency component at a discrete frequency. It is computed as the square root of the average auto spectrum. For an accelerometer, for example, the units for **Spectrum** are represented as *g*. Typically, **Spectrum** is best for narrowband, deterministic signals.
- **Auto Spectrum** – Plots the power of a periodic frequency component at a discrete frequency. For an accelerometer, for example, the units for **Auto Spectrum** are represented as g^2. Typically, **Auto Spectrum** is best for narrowband, deterministic signals.
- **PSD** – (Power Spectral Density) Plots the power of random vibration intensity as "mean-square acceleration per frequency unit." The spectrum is computed by squaring the magnitude of each frequency component in the FFT, and dividing this number by the change in frequency multiplied by the equivalent noise bandwidth of the windowing function. For an accelerometer, for example, the units for PSD are represented as g^2/Hz. Typically, **PSD** is best for wideband, continuous signals.

Integration Type

Primarily used for accelerometer sensor types, choose one of the following integration types:

- **None –** No integration; acceleration data is represented as acceleration.
- **Single** – Data is integrated once; acceleration data is represented as velocity.
- **Double** – Data is integrated twice; acceleration data is represented as displacement.

Display Function

If you are using the FFT Analysis option, the following settings are available for **Display Function**:

- **Amplitude** – Plots the length of the frequency response.
- **Phase** – Plots the frequency response phase shift (angle).

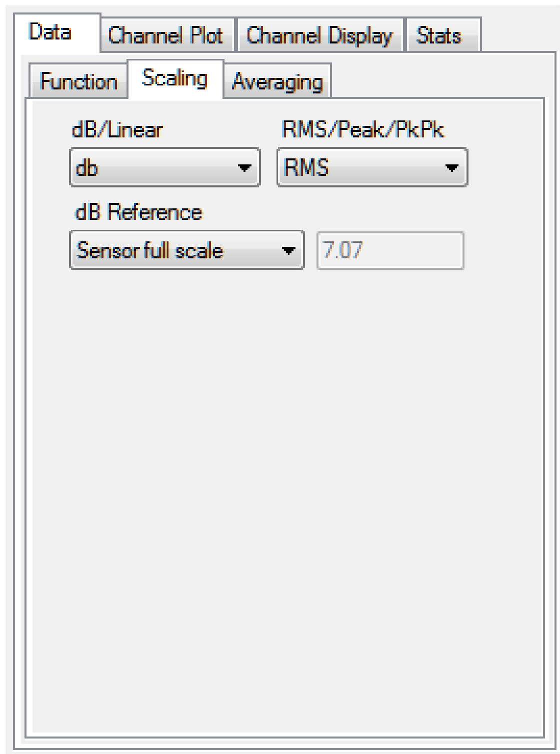
If you are using the Advanced FFT Analysis option of QuickDAQ, the following settings are available for **Display Function**:

- **Amplitude** – Plots the length of the frequency response.
- **Phase** – Plots the frequency response phase shift (angle).
- **Real** – Plots the real component of the complex resultant data on the y-axis against frequency on the x-axis.
- **Imaginary** – Plots the imaginary component of the complex resultant data on the y-axis against frequency on the x-axis.
- **Nyquist** – Plots the complex resultant data with the real components on the x-axis and imaginary components on the y-axis; the consecutive points are joined by line segments.

Scaling Tab

The **Scaling** tab allows you specify how you want to scale the data for the FFT channel when it is plotted.

Note: This tab is not used if you select the Real, Imaginary, or Nyquist display function.



- **dB** or **Linear** – Select **dB** for decibels or **Linear** for the engineering units of the sensor, such as *g* for an accelerometer.

Note: If you select **Linear** scaling for the FFT channel, you can right-click on the axis of the plot, and select **Y Axis Setup** (for the y-axis) or **X Axis Setup** (for the x-axis) to change the scaling of the axis to **Linear** or **Log10**. If you select **dB** scaling for the channel, the Y-axis is set to **Linear** automatically.

If a channel is configured for a strain gage or bridge-based sensor, you cannot choose dB scaling for the corresponding FFT channel.

- **dB Reference** – If you select **dB**, you can select one of the following dB reference values to use when computing dB: **Sensor full-scale** (the full-scale range of the sensor that you defined is used), **Acceleration (1e-3)**, **Velocity (1e-6)**, **Displacement (1e-9)**, or a **User-Defined** value. If you select **User-Defined**, change the default reference value to the value that you desire; this value must be greater than 0.
- **RMS**, **Peak**, or **Pk-Pk** – Allows you to scale the FFT data using **RMS** (the square root of the average of the square of the value of the function taken throughout one period, as shown in Figure 35), **Peak** (zero to Peak, as shown in Figure 36), or **PeakPeak** (the absolute value of the data from the negative peak to the positive peak, as shown in Figure 37).

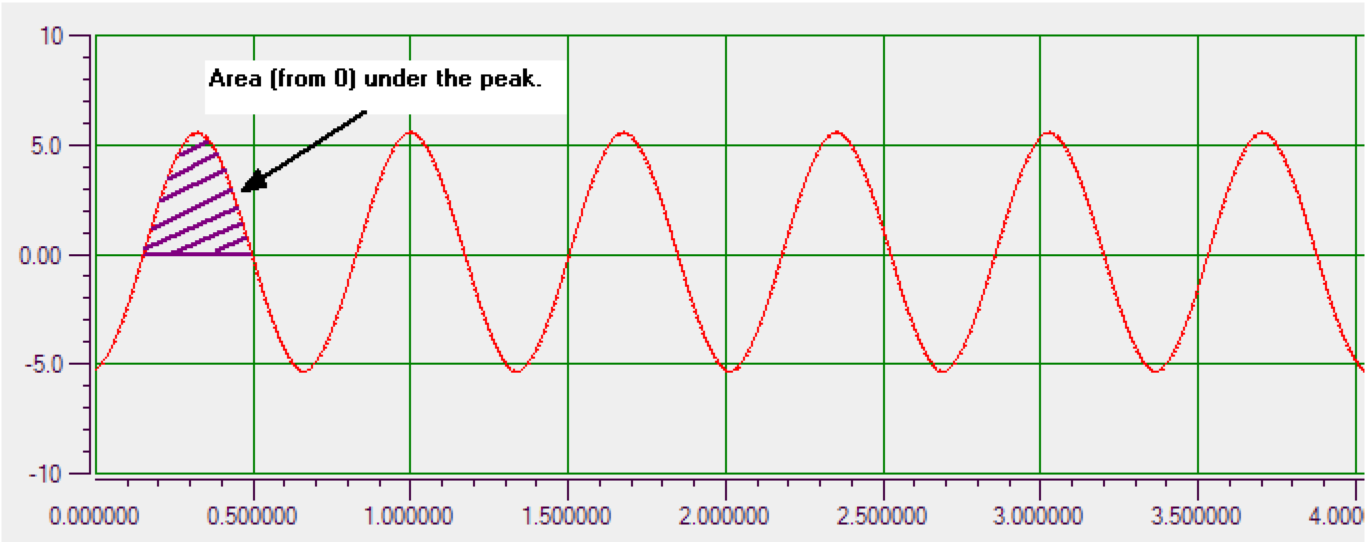


Figure 35: RMS (Square Root of the Average of the Squares)

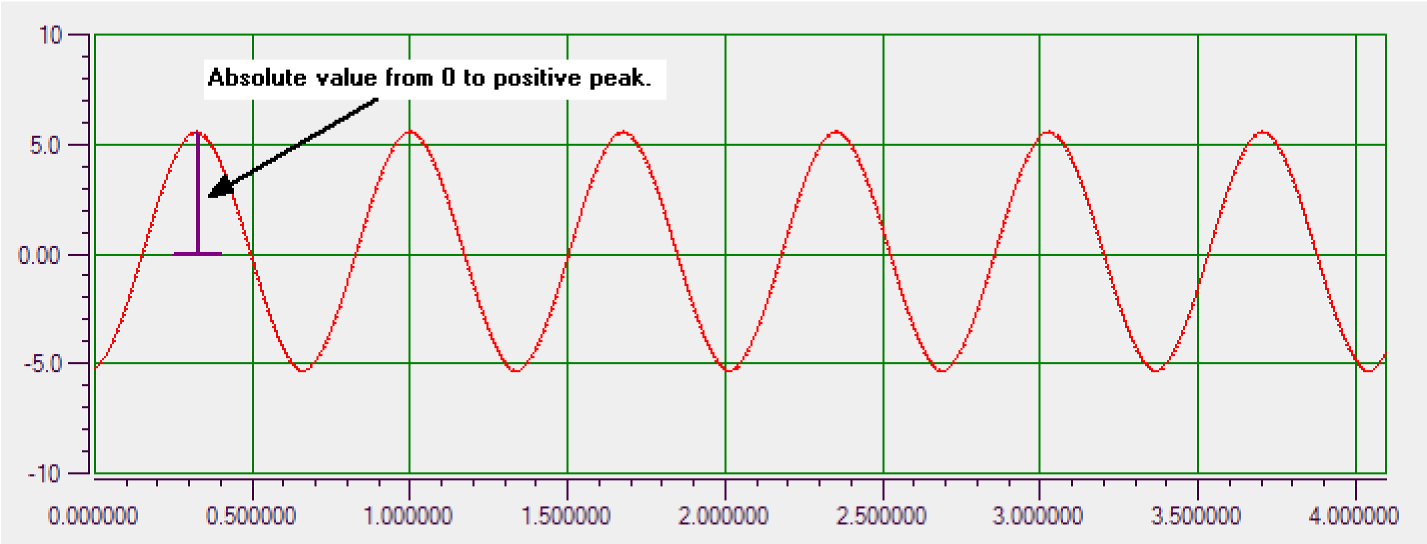


Figure 36: Peak (Zero to Peak)

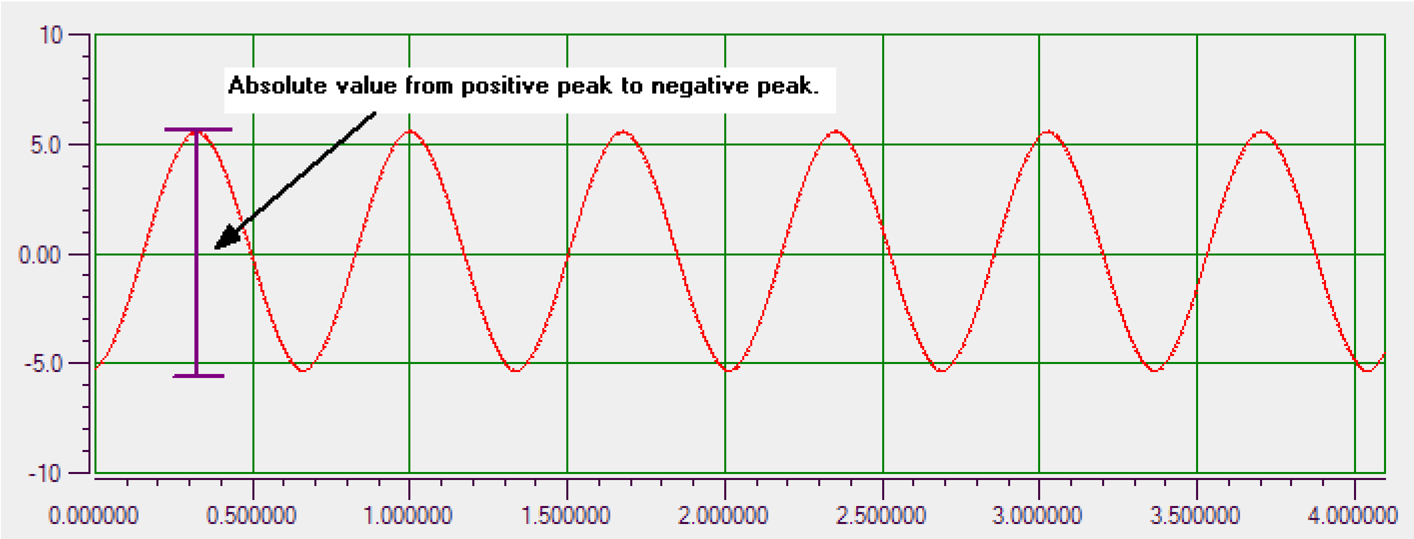
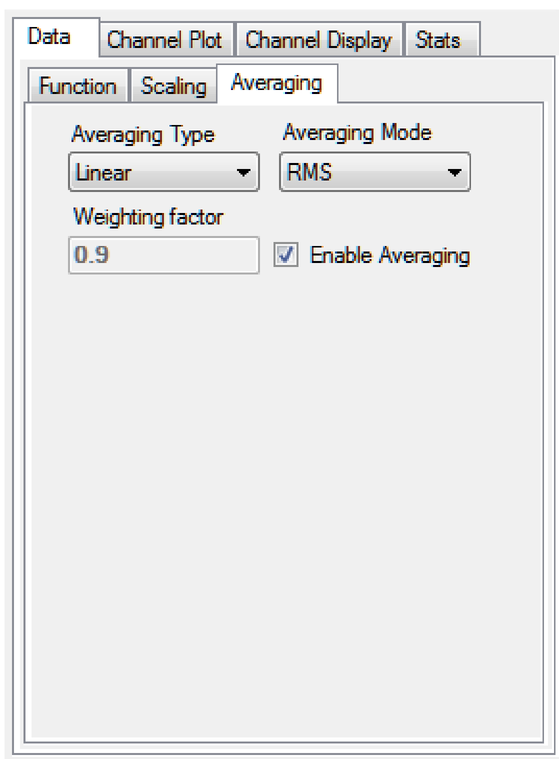


Figure 37: PeakPeak (Negative Peak to Positive Peak)

Averaging Tab

The **Averaging** tab allows you specify how you want to average the data for the FFT channel.



- **Enable Averaging –** Enable averaging by selecting this checkbox, or disable averaging by unchecking this checkbox.

  When this box is checked, the FFT plot shows the accumulated average result of the data at the completion of each acquired record.

- **Averaging Type** – If you want to average FFTs rather than showing an FFT for each individual record, choose one of the following spectral averaging types:
  - **Linear –** Data from each FFT is averaged with the data from the other FFTs; all data contributes equally to the average.
  - **Exponential –** New FFT data is weighted differently than older FFT data. You specify the weighting factor to apply, which either increases or decreases the effect of the new FFT data on the average. This is a continuous averaging type.

- **Weighting Factor** – If you choose **Exponential** as the spectral averaging type, enter the weighting factor to apply. The weighting factor either increases or decreases the effect of new FFT data on the resultant average using on the following formula:

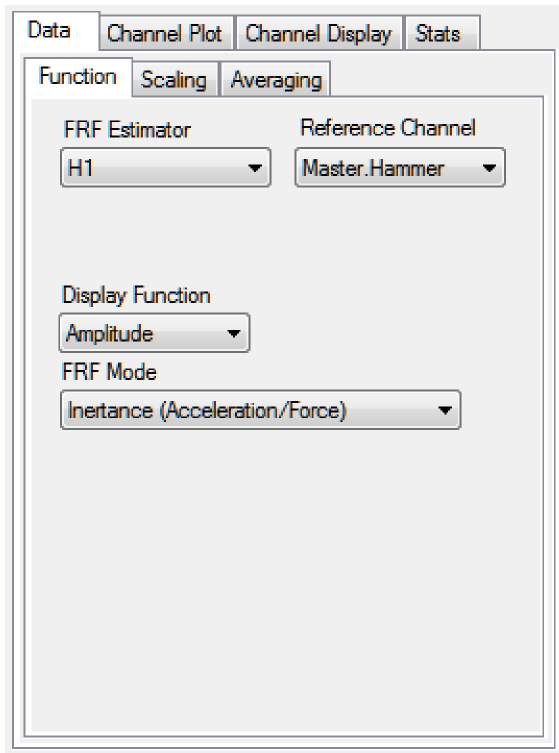
  Result = ((New FFT Data) * Weighting Factor) + (Old Average * (1- Weighting Factor))

- **Averaging Mode** – Choose one of the following averaging mode to use:
  - **RMS** – RMS (root mean squared) averaging averages the power of the signal in real numbers directly. For single channel measurements, this results in zero phase. RMS averaging reduces signal fluctuations but not the noise floor.
  - **Vector –** Vector (angle) averaging computes the average of complex numbers directly. The real part of the number is averaged separately from the imaginary part of the number, reducing the noise floor for random signals. A trigger is usually required for this averaging mode.
  - **PeakHold –** With PeakHold averaging, the averaged data is a collection of points that represent the peak amplitude for each point in the FFT. With each new FFT, the current data is compared with the new data on a point by point basis. The highest amplitude for each point in the FFT is retained.

Configuring Parameters for a Two-Channel FRF Function

This section applies to the Advanced FFT Analysis option of QuickDAQ.

If you have added a two-channel FRF (Frequency Response Function) channel, as described on page 204, and selected it under the **Channel Name** column, you can configure the parameters of the FRF channel using the **Data** tab options:



This section describes the options that appear under the **Data** tab if you select a two-channel FRF channel in the Plot and Data Config window.

Note: Once you have acquired data, you can change the parameters under the **Data** tab and see the effect of your changes immediately by checking the **Auto-reprocess on parameter change** checkbox. Or, if this checkbox is not checked, click the **Apply settings to current data button** to see the results.

Function Tab

The **Function** tab provides the following selections: **FRF Estimator**, **Reference Channel**, **Display Function**, and **FRF Mode**.

FRF Estimator

The **FRF Estimator** determines how the frequency response function is calculated. Choose one of the following settings for **FRF Estimator**:

- **H1 –** Reduces noise on the response channel. It is calculated as follows:

  $H_1 = S_{xf}/S_{ff}$

  where,

  S_{xf} represents the average cross power spectrum between the response and the reference signals.

  S_{ff} represents the average auto power spectrum of the reference signal.

- **H2** – Reduces noise on the reference channel. It is calculated as follows:

  $H_2 = S_{xx}/S_{fx}$

  where,

  S_{xx} represents the average auto spectrum of the response signal.

  S_{fx} represents the average cross power spectrum between the response and the reference signals.

- **H3** – Reduces noise on both the reference and response channels. H3 is computed as follows:

  $H_3 = (H_1 + H_2)/2$

Reference Channel

From the list of reference channels that you configured using the **Input Channel Configuration** menu option, described on page 84, select the reference channel that you want to compare with the selected response channel.

Display Function

Under **Display Function**, choose one of these settings:

- **Amplitude –** Plots the magnitude of the frequency response gain.
- **Phase** – Plots the frequency response phase shift (angle).
- **Real** – Plots the real component of the complex resultant data on the y-axis against frequency on the x-axis.
- **Imaginary –** Plots the imaginary component of the complex resultant data on the y-axis against frequency on the x-axis.
- **Nyquist –** Plots the complex resultant data with the real components on the x-axis and imaginary components on the y-axis; the consecutive points are joined by line segments.

FRF Mode

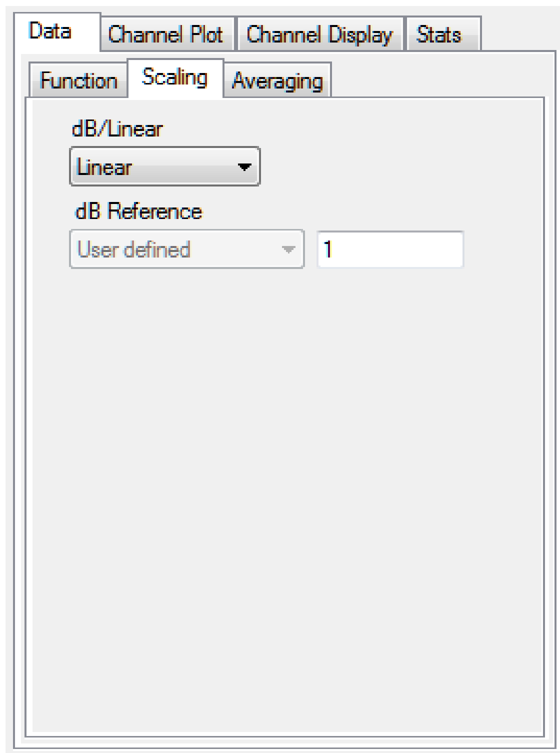
Choose one of the following FRF modes; based on the mode selected, the application sets the integration type for the response channel accordingly:

- **Inertance –** Calculated as Acceleration of Response Channel/Force of Reference Channel.
- **Mobility –** Calculated as Velocity of Response Channel/Force of Reference Channel.
- **Compliance –** Calculated as Displacement of Response Channel/Force of Reference Channel.
- **Apparent Mass –** Calculated as Force of the Reference Channel/Acceleration of Response Channel.
- **Impedance –** Calculated as Force of Reference Channel/Velocity of the Response Channel.
- **Dynamic Stiffness –** Calculated as Force of Reference Channel/Displacement of Response Channel
- **Custom –** When you select this option, you can enter a integration type for the response channel and an integration type for the reference channel, if desired. The following selections are available:
  - **None –** No integration; acceleration data is represented as acceleration.
  - **Single –** Data is integrated once; acceleration data is represented as velocity.
  - **Double –** Data is integrated twice; acceleration data is represented as displacement.

Scaling Tab

The **Scaling** tab allows you specify how you want to scale the data of the FRF channel when it is plotted.

Note: The Scaling tab is not used if you select the Real, Imaginary, or Nyquist display function.



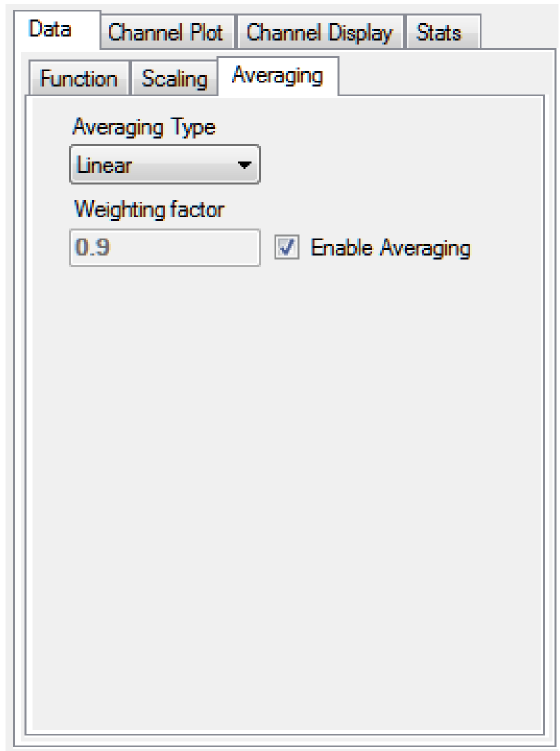
Select **Linear** for the engineering units of the sensor, such as *g* for an accelerometer, or **dB** for decibels.

Note: If you select **Linear** scaling for the channel, you can right-click on the axis of the plot, and select **Y Axis Setup** (for the y-axis) or **X Axis Setup** (for the x-axis) to change the scaling of the axis to **Linear** or **Log10**. If you select **dB** scaling for the channel, the Y-axis is set to **Linear** automatically.

If you select **dB**, **User-Defined** is selected by default for **dB Reference**.You can change the default reference value to a value that you desire; this value must be greater than 0.

Averaging Tab

The **Averaging** tab allows you specify how you want to average the data for the FRF channel.



- **Enable Averaging –** Enable averaging by selecting this checkbox, or disable averaging by unchecking this checkbox.

  When this box is checked, the FRF plot shows the accumulated average result of the data at the completion of each acquired record.

- **Averaging Type –** If you want to average FFTs rather than showing an FFT for each individual record, choose one of the following spectral averaging types:
  - **Linear –** Data from each FFT is averaged with the data from the other FFTs; all data contributes equally to the average.
  - **Exponential –** New FFT data is weighted differently than older FFT data. You specify the weighting factor to apply, which either increases or decreases the effect of the new FFT data on the average. This is a continuous averaging type.

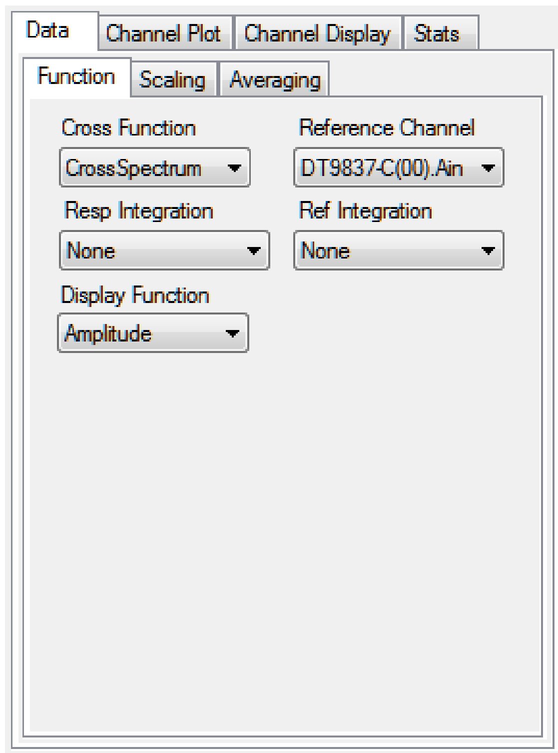
- **Weighting Factor –** If you choose **Exponential** as the spectral averaging type, enter the weighting factor to apply. The weighting factor either increases or decreases the effect of new FFT data on the resultant average using on the following formula:

  Result = ((New FFT Data) * Weighting Factor) + (Old Average * (1- Weighting Factor))

Configuring Parameters for a Two-Channel Cross Spectrum Function

This section applies to the Advanced FFT Analysis option of QuickDAQ.

Once you have added a two-channel Cross Spectrum channel, as described on page 204, and selected it under the **Channel Name** column, you can configure the parameters of the Cross Spectrum channel using the **Data** tab options:



This section describes the options that appear under the **Data** tab if you select a two-channel Cross Spectrum channel in the Plot and Data Config window.

Note: Once you have acquired data, you can change the parameters under the **Data** tab and see the effect of your changes immediately by checking the **Auto-reprocess on parameter change** checkbox. Or, if this checkbox is not checked, click the **Apply settings to current data button** to see the results.

Function Tab

The **Function** tab provides the following selections: **Cross Function**, **Reference Channel**, and **Resp Integration, Ref Integration, and Display Function**.

Cross Function

Under **Cross Function**, choose one of the following settings:

- **Cross Spectrum –** A building block used in FRF and other more advanced functions. The cross-spectrum is defined as the product of the complex Fourier spectrum of the response channel multiplied by the complex conjugate of the Fourier spectrum of the reference channel.
- **Cross PSD** – (Cross Power Spectral Density) The distribution of power per unit frequency.

Reference Channel

From the list of reference channels that you configured using the **Input Channel Configuration** menu option, described on page 84, select the reference channel that you want to compare with the selected response channel.

Resp Integration

For the selected response channel, choose one of the following integration types:

- **None** – No integration; acceleration data is represented as acceleration.
- **Single** – Data is integrated once; acceleration data is represented as velocity.
- **Double** – Data is integrated twice; acceleration data is represented as displacement.

Ref Integration

For the selected reference channel, choose one of the following integration types:

- **None** – No integration; acceleration data is represented as acceleration.
- **Single** – Data is integrated once; acceleration data is represented as velocity.
- **Double** – Data is integrated twice; acceleration data is represented as displacement.

Display Function

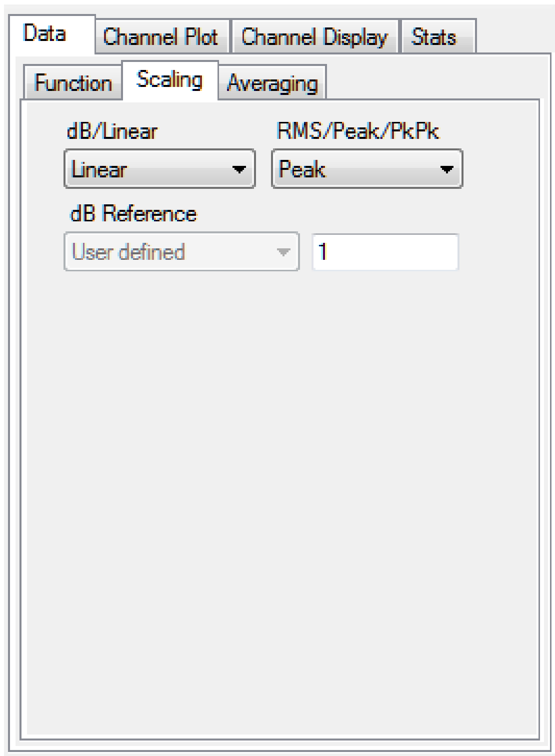
Under **Display Function**, choose one of these settings:

- **Amplitude** – Plots the magnitude of the frequency response gain.
- **Phase** – Plots the frequency response phase shift (angle).
- **Real** – Plots the real component of the complex resultant data on the y-axis against frequency on the x-axis.
- **Imaginary** – Plots the imaginary component of the complex resultant data on the y-axis against frequency on the x-axis.
- **Nyquist** – Plots the complex resultant data with the real components on the x-axis and imaginary components on the y-axis; the consecutive points are joined by line segments.

Scaling Tab

The **Scaling** tab allows you specify how you want to scale the data when it is plotted.

Note: The Scaling tab is not used if you specify the Real, Imaginary, or Nyquist display function.



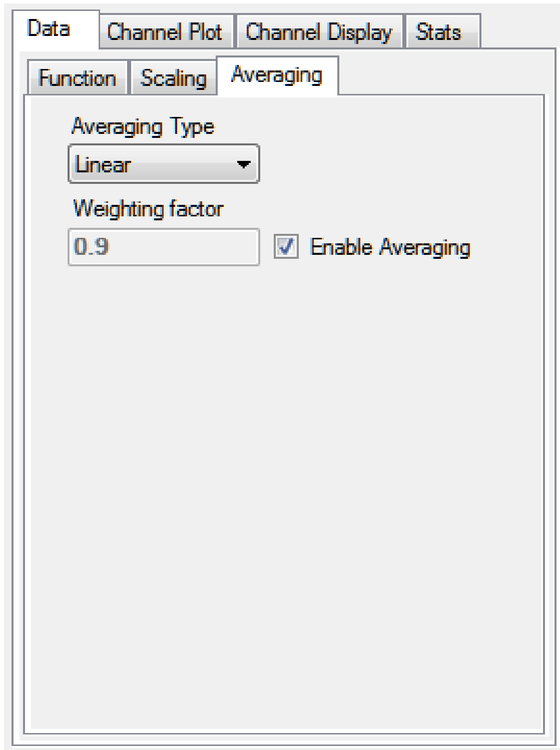
- **dB** or **Linear** – Select **dB** for decibels or **Linear** for the engineering units of the sensor, such as *g* for an accelerometer.

Note: If you select **Linear** scaling for the FFT channel, you can right-click on the axis of the plot, and select **Y Axis Setup** (for the y-axis) or **X Axis Setup** (for the x-axis) to change the scaling of axis to **Linear** or **Log10**. If you select **dB** scaling for the channel, the Y-axis is set to **Linear** automatically.

- **dB Reference** – If you select **dB**, **User-Defined** is selected by default. You can change the default reference value to a value that you desire; this value must be greater than 0.
- **RMS**, **Peak**, or **Pk-Pk** – Allows you to scale the FFT data using **RMS** (the square root of the average of the square of the value of the function taken throughout one period, as shown in Figure 35 on page 210), **Peak** (zero to Peak, as shown in Figure 36 on page 210), or **PeakPeak** (the absolute value of the data from the negative peak to the positive peak, as shown in Figure 37 on page 211).

Averaging Tab

The **Averaging** tab allows you specify how you want to average the data.



- **Enable Averaging –** Enable averaging by selecting this checkbox, or disable averaging by unchecking this checkbox.

  When this box is checked, the Cross Spectrum plot shows the accumulated average result of the data at the completion of each acquired record.

- **Averaging Type** – If you want to average FFTs rather than showing an FFT for each individual record, choose one of the following spectral averaging types:
  - **Linear –** Data from each FFT is averaged with the data from the other FFTs; all data contributes equally to the average.
  - **Exponential –** New FFT data is weighted differently than older FFT data. You specify the weighting factor to apply, which either increases or decreases the effect of the new FFT data on the average. This is a continuous averaging mode.

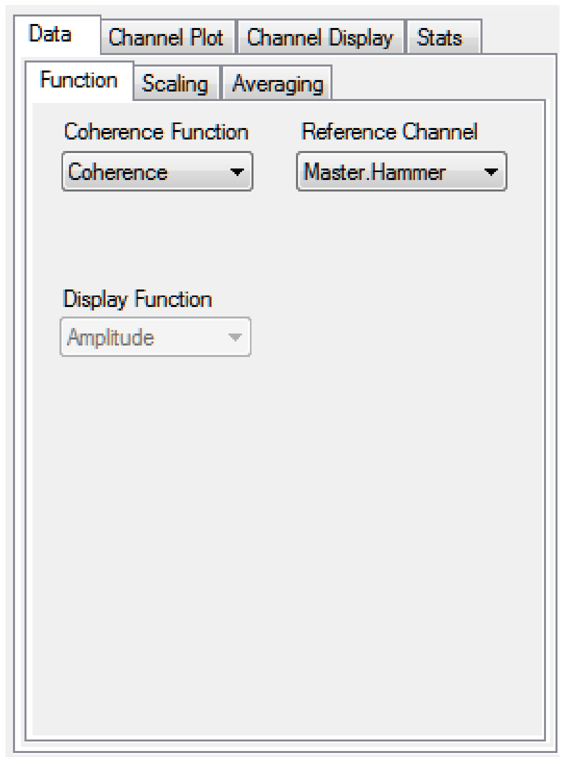
- **Weighting Factor** – If you choose **Exponential** as the spectral averaging type, enter the weighting factor to apply. The weighting factor either increases or decreases the effect of new FFT data on the resultant average using on the following formula:

  Result = ((New FFT Data) * Weighting Factor) + (Old Average * (1- Weighting Factor))

Configuring Parameters for a Two-Channel Coherence Function

This section applies to the Advanced FFT Analysis option of QuickDAQ.

Once you have added a two-channel Coherence channel, as described on page 204, and selected it under the **Channel Name** column, you can configure the parameters of the Coherence channel using the **Data** tab options:



This section describes the options that appear under the **Data** tab if you select a two-channel Coherence channel in the Plot and Data Config window.

Note: Once you have acquired data, you can change the parameters under the **Data** tab and see the effect of your changes immediately by checking the **Auto-reprocess on parameter change** checkbox. Or, if this checkbox is not checked, click the **Apply settings to current data button** to see the results.

Function Tab

The **Function** tab provides the following selections: **Coherence Function** and **Reference Channel**. These options are described in the following subsections.

Coherence Function

Under **Coherence Function**, choose one of the following settings:

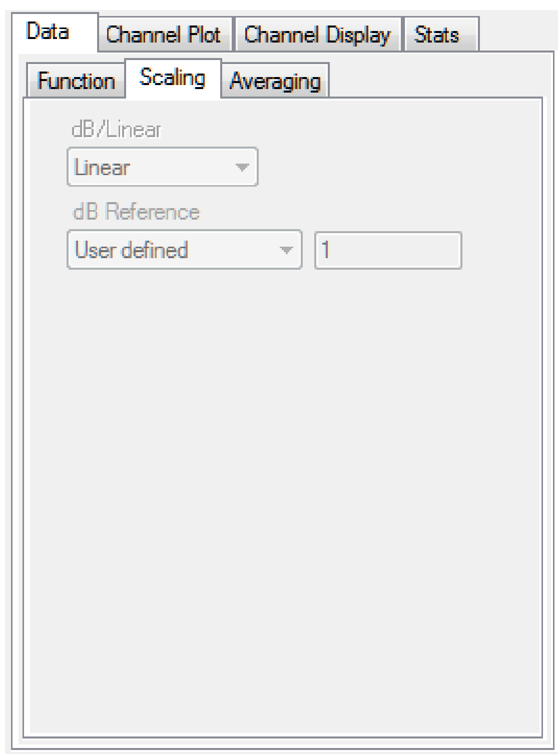
- **Coherence –** Evaluates the linearity between the reference signal and the response signal at each frequency, and generates a value between 0.0 and 1.0, where 0 is consistently not linear and 1.0 is consistently linear.
- **COP** – (Coherent Output Power) The COP function is the power of the response channel multiplied, at each frequency, by the coherence between the reference and the response channels.

Reference Channel

From the list of reference channels that you configured using the **Input Channel Configuration** menu option, described on page 84, select the reference channel that you want to compare with the selected response channel.

Scaling Tab

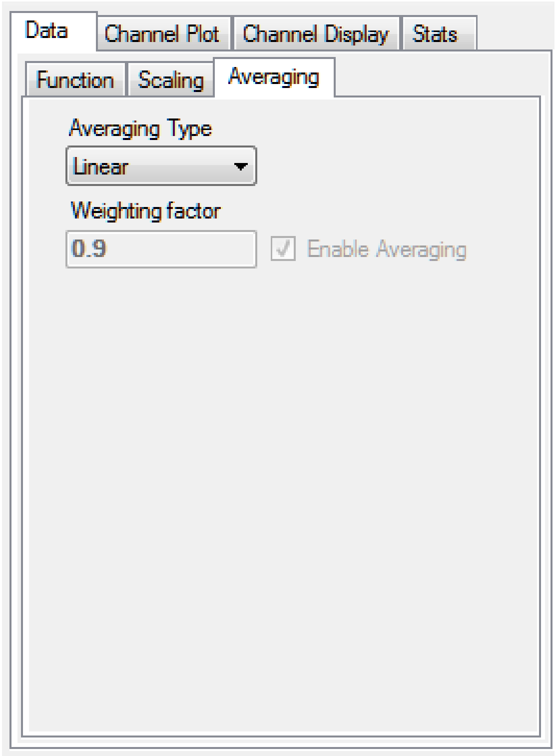
For Coherence channels, **Linear** scaling is automatically selected for the **Scaling** tab; therefore, all choices for this tab are disabled.



Note: You can right-click on the axis of the plot, and select **Y Axis Setup** (for the y-axis) or **X Axis Setup** (for the x-axis) to change the scaling of axis to **Linear** or **Log10**.

Averaging Tab

The **Averaging** tab allows you specify how you want to average the data for the Coherence channel.



- **Enable Averaging –** Averaging is always enabled for Coherence channels; therefore, this checkbox is disabled.

  The Coherence plot shows the accumulated average result of the data at the completion of each acquired record.

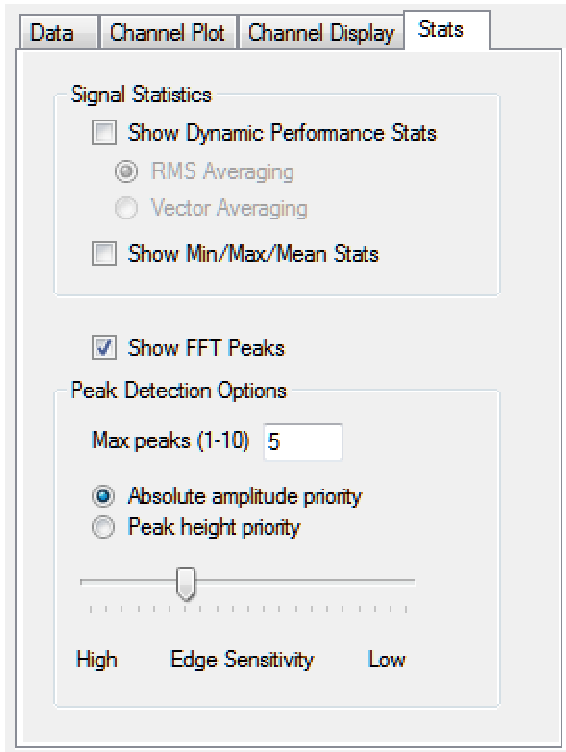
- **Averaging Type –** If you want to average FFTs rather than showing an FFT for each individual record, choose one of the following spectral averaging modes:
  - **Linear –** Data from each FFT is averaged with the data from the other FFTs; all data contributes equally to the average.
  - **Exponential –** New FFT data is weighted differently than older FFT data. You specify the weighting factor to apply, which either increases or decreases the effect of the new FFT data on the average. This is a continuous averaging mode.

- **Weighting Factor –** If you choose **Exponential** as the spectral averaging type, enter the weighting factor to apply. The weighting factor either increases or decreases the effect of new FFT data on the resultant average using on the following formula:

  Result = ((New FFT Data) * Weighting Factor) + (Old Average * (1- Weighting Factor))

Configuring Statistics for FFT Plots Using the Stats Tab

If you are using the FFT Analyzer interface, the **Stats** tab allows you to set up statistics for FFT and time-domain channels. This section describes how to set up statistics for FFT plots. Refer to page 235 for more information on configuring statistics for the Statistics window.



If you select **Show FFT Peaks**, a graphic similar to the following appears next to each FFT plot:

#	Freq	Val
1	78.1	-39.451
2	136.7	-39.877
3	449.2	-43.353

The peaks are ordered by height, and the frequency and values for the detected peaks are displayed.

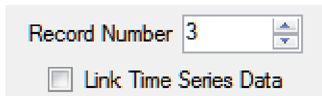
You can configuring the following attributes for FFT peak detection:

- **Max peaks –** Select the number of peaks (1 to 10) that you want the application to detect and display in the FFT plot.
- **Absolute amplitude priority –** Determine whether the application detects the peak by Absolute amplitude or by Peak height.
- **Edge sensitivity –** Determine the sensitivity (high to low) with which the application detects the peak by adjusting the Edge sensitivity slider bar.

Scrolling through FFTs for an Analog Input Channel

If you are using the FFT Analyzer interface and averaging is disabled for an FFT channel, you can use the **Record Number** box in the Acquisition Status window to scroll through the records that were acquired for the corresponding analog input channel once data is acquired.

For example, if 10 records were acquired for the analog input channels (0 to 9), and you are interested in looking at record number 3, set the Record Number to 3 and the plot for that FFT is displayed in the Channel Plot window.



If you want to see the actual analog input data that corresponds to a particular Record Number, click the **Link Time Series Data** checkbox. The data plot for the corresponding analog input channels is then displayed for the selected record number.

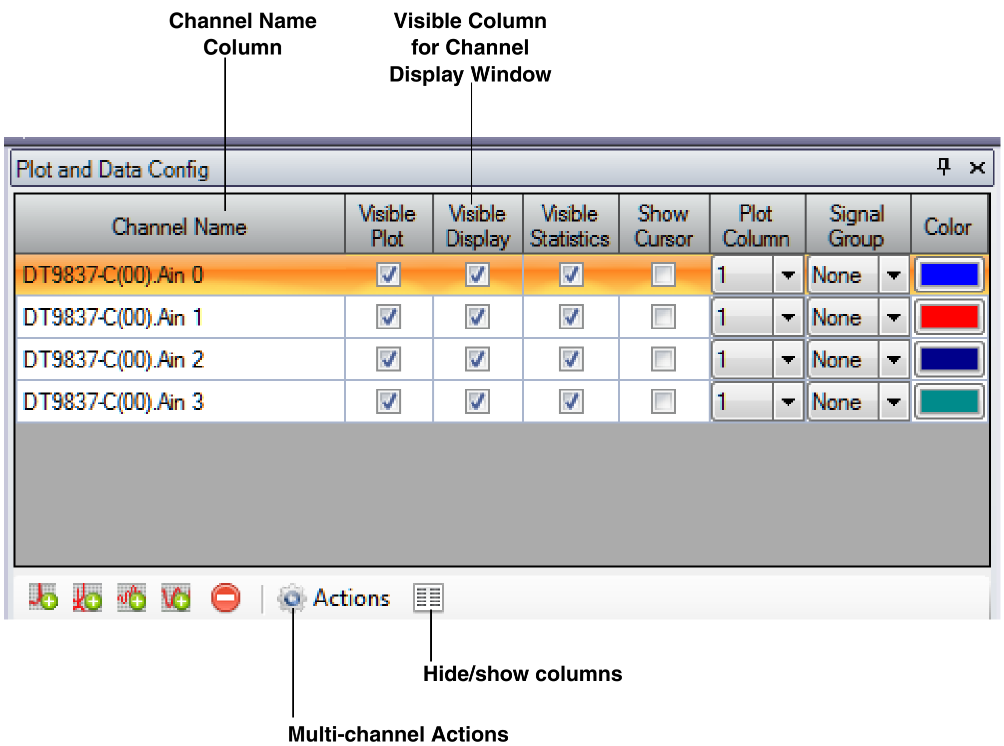
Configuring the Appearance of the Channel Display Window

This section describes the how to configure the appearance of the Channel Display window when using the FFT Analyzer interface, and includes the following topics:

- Displaying channels in the Channel Display window, described on this page
- Controlling how data is displayed in the Channel Display window, described on page 230
- Specifying the style of the Channel Display window, described on page 232

Displaying Channels in the Channel Display Window

You can determine which channels are displayed in the Channel Display window by using the following controls in the Plot and Data Config window:



You can see the names of all enabled channels under the **Channel Name** column in the Plot and Data Config window. Only data from the list of enabled channels is acquired.

Note: If desired, you can change the name of a channel using the **Input Channel Configuration** menu option or toolbar button, described on page 84.

For each enabled time-domain channel, you can control which channels are displayed in the Channel Display window using the **Visible Display** checkbox. If a box is checked under the **Visible Display** column, the data for the associated channel is displayed in the Channel Display window. If a box under the **Visible Display** column is unchecked, the data for the associated channel is not displayed in the Channel Display window.

Note: FFT channels cannot be displayed in the Channel Display window.

Data for all enabled time-domain channels is recorded regardless of whether the channel is visible.

Making All Selected Channels Visible or Hidden in the Channel Display Window at Once

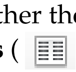
If you want to make all selected channels visible in the Channel Display window at once, select the **Multi-channel Actions** button (

) button, and choose **Selected channels visible in Channel Display**.

If you want to hide all selected channels in the Channel Display window at once, select the **Multi-channel Actions** (

) button, and choose **Selected channels hidden in Channel Display**.

Showing or Hiding the Visible Display Column

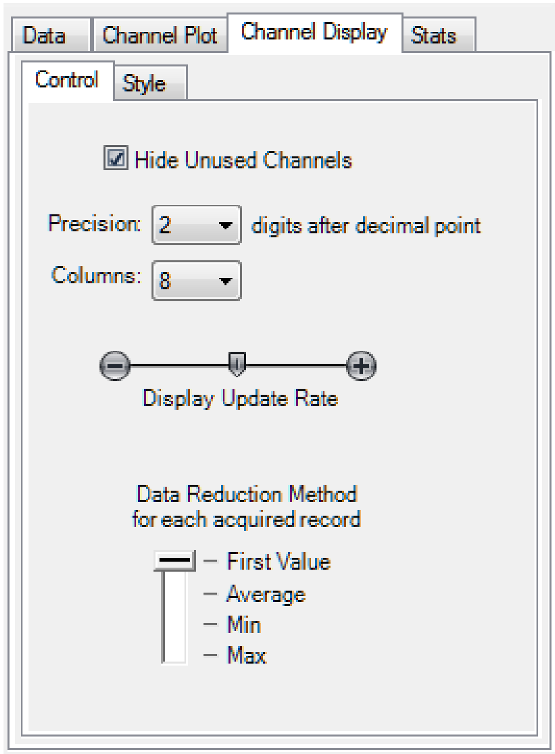
If you wish, you can change whether the **Visible Display** column is shown or hidden by selecting the **Hide/show columns** () button, and selecting **Visible Display**. A checkmark means the column is shown; no checkmark means the column is hidden.

Moving the Visible Display Column

If you wish, you can move the position of the **Visible Display** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

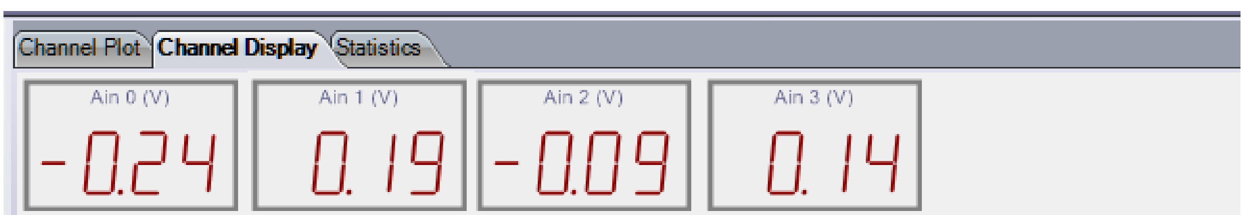
Controlling How Data is Displayed in the Channel Display Window

Using the **Channel Display - Control** tab in the Plot and Data Config window, you can control how data is displayed in the Channel Display window.

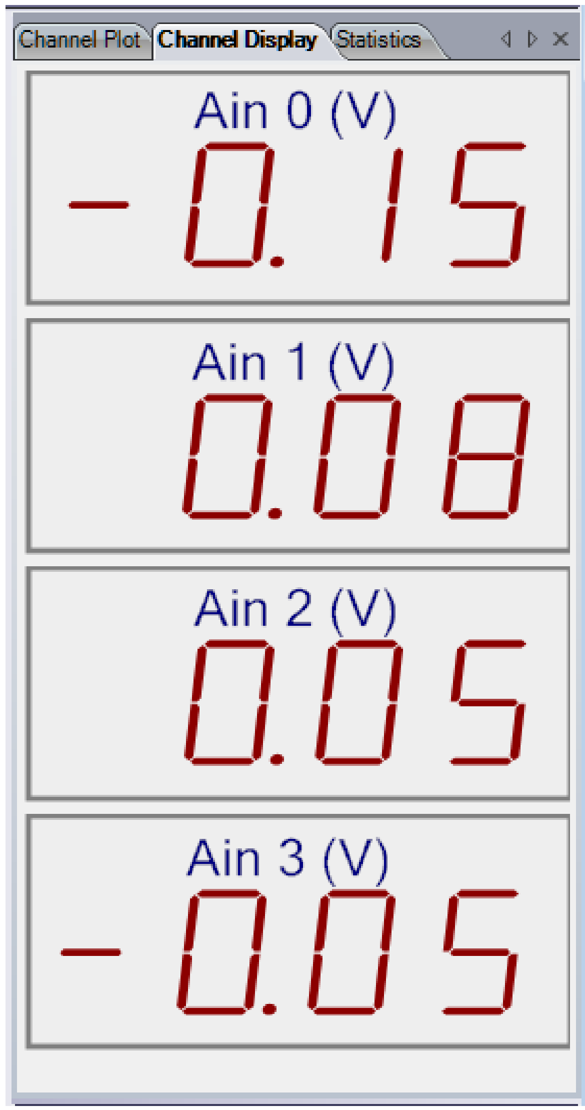


You can configure the following settings:

- **Hide Unused Channels** – The Channel Display window shows the channels on the device. By default, the **Hide Unused Channels** checkbox is checked; therefore, only the enabled channels are shown. If you want your display to show all the channels rather than just the enabled channels, uncheck the **Hide Unused Channels** box. The following example illustrates how the display looks when the **Hide Unused Channels** box is unchecked to show all the channels.



- **Precision** – You can select the number of significant digits after the decimal point to display; values range from 1 to 8 (2 is the default).
- **Columns** – You can specify how many columns to use when displaying the channels; values range from 1 to 16 (8 is the default). For example, by choosing 1, shown below, you can display each channel, one on top of the other (note that the lowest numbered channel is always at the top of the display and the highest numbered channel is at the bottom of the display):

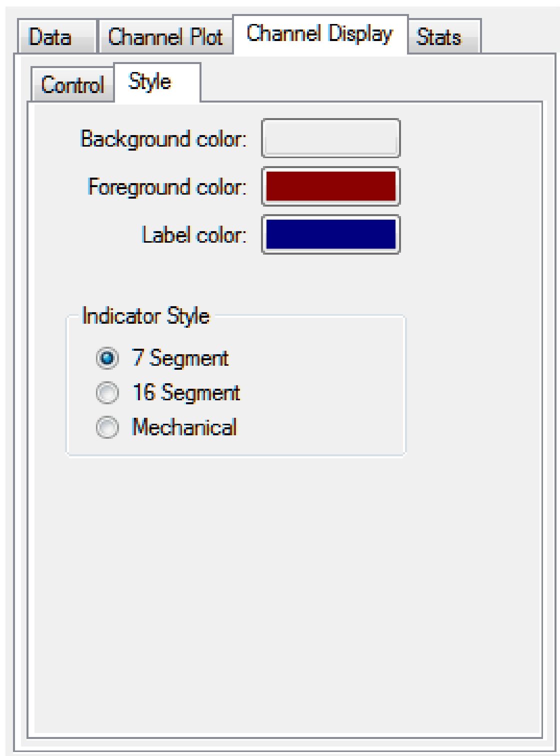


- **Display Update Rate** – You can choose how fast you want the display of the Channel Display window to be updated. Slide the bar to the left for faster update rates, or slide the bar to the right for slower update rates.

- **Data Reduction Method –** You can choose which value to display in the Channel Display window:
  - **First Value –** The first value in the most recent record is displayed for each channel.
  - **Average –** The average value in the most recent record is displayed for each channel.
  - **Min –** The minimum value in the most recent record is displayed for each channel.
  - **Max –** The maximum value in the most recent record is displayed for each channel.

Specifying the Style of the Channel Display Window

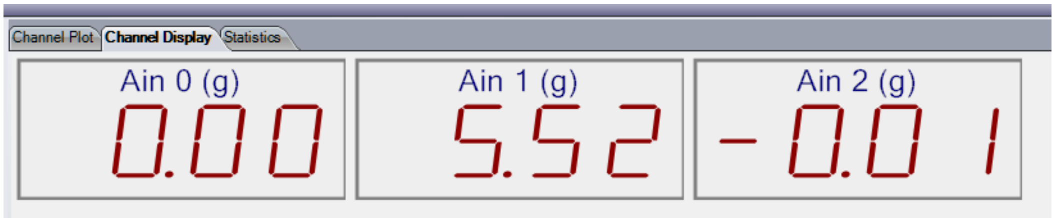
Using the Channel Display - Style tab, you can specify the style of the Channel Display window.



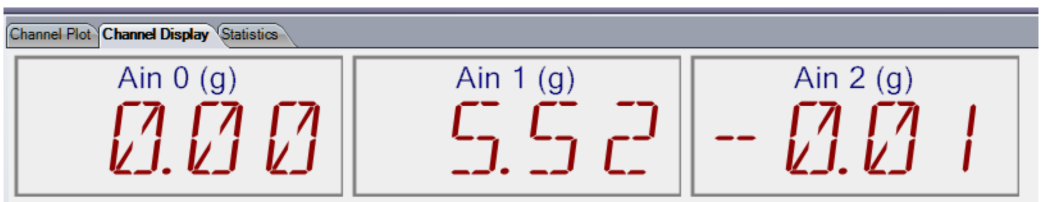
You can configure the following parameters:

- **Background color –** Allows you to choose a specific color for the background of the Channel Display window. You can select from a number of predefined colors or you can define your own color for the background.
- **Foreground color –** Allows you to choose a specific color for the foreground of the Channel Display window. You can select from a number of predefined colors or you can define your own color for the foreground.
- **Label color –** Allows you to choose a specific color for the label used on the Channel Display window. You can select from a number of predefined colors or you can define your own color for the label.

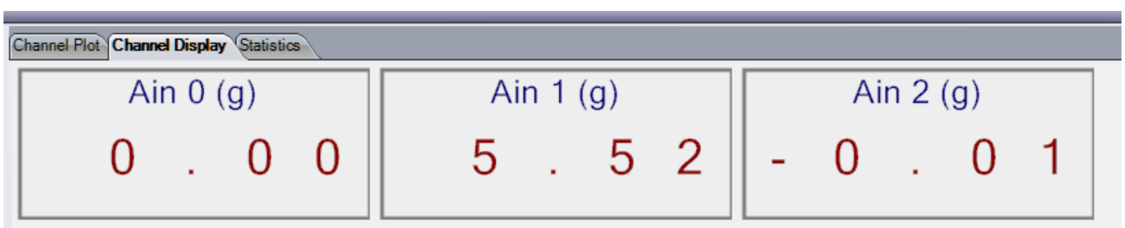
- **Indicator style –** Allows you to specify how the numbers and letters that represent the data appear. The following choices are available:
  - **7 Segment –** The numbers and letters that represent the data are composed of 7-segment lines.



  - **16 Segment –** The numbers and letters that represent the data are composed of 16-segment lines.



  - **Mechanical –** The numbers and letters that represent the data appear to be formed by a contiguous line.



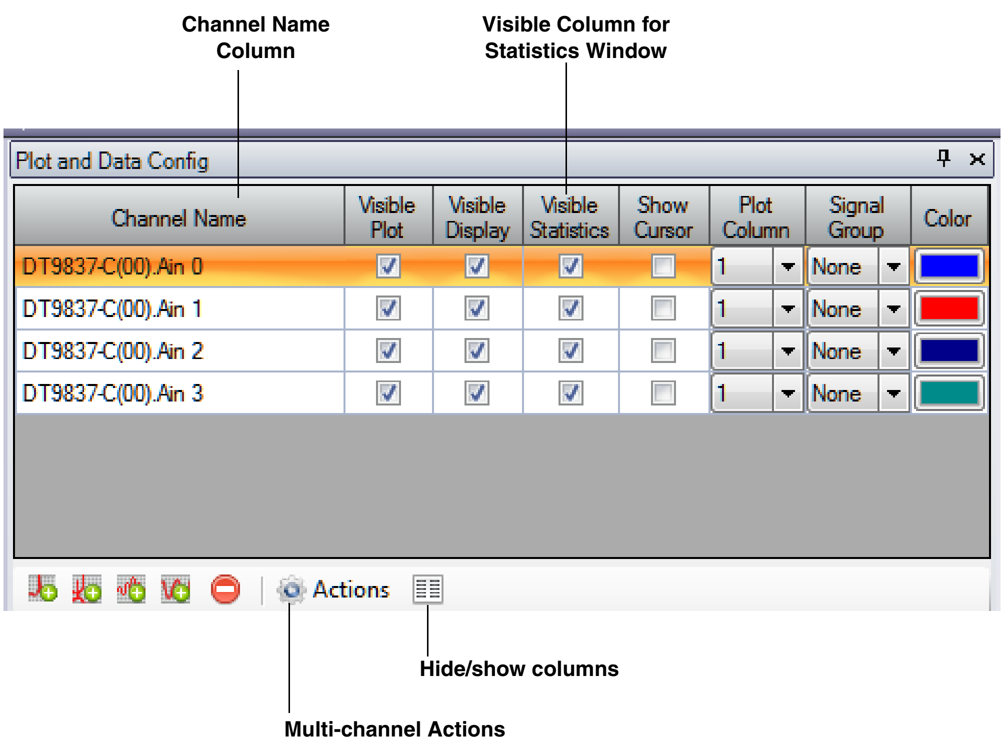
Configuring the Appearance of the Statistics Window

This section describes the how to configure the appearance of the Statistics window when using the FFT Analyzer interface, and includes the following topics:

- Displaying channels in the Statistics window, described on this page
- Controlling how data is displayed in the Statistics window, described on page 235

Displaying Channels in the Statistics Window

You can determine which channels are displayed in the Statistics window by using the following controls in the Plot and Data Config window:



You can see the names of all enabled channels under the **Channel Name** column in the Plot and Data Config window. Only data from the list of enabled channels is acquired.

Note: If desired, you can change the name of a channel using the **Input Channel Configuration** menu option or toolbar button, described on page 84.

For each enabled time-domain channel, you can control which channels are displayed in the Statistics window using the **Visible Statistics** checkbox. If a box is checked under the **Visible Statistics** column, the data for the associated channel is displayed in the Statistics window. If a box under the **Visible Statistics** column is unchecked, the data for the associated channel is not displayed in the Statistics window.

Note: Statistics are not created for FFT channels; therefore, statistics for FFT channels are not displayed in the Statistics window.

Making All Selected Channels Visible or Hidden in the Statistics Window at Once

If you want to make all selected channels visible in the Statistics window at once, select the **Multi-channel Actions** button ( ) button, and choose **Selected channels visible in Statistics Display**.

If you want to hide all selected channels in the Channel Display window at once, select the **Multi-channel Actions** ( ) button, and choose **Selected channels hidden in Statistics Display**.

Showing or Hiding the Visible Statistics Column

If you wish, you can change whether the **Visible Statistics** column is shown or hidden by selecting the **Hide/show columns** ( ) button, and selecting **Visible Statistics**. A checkmark means the column is shown; no checkmark means the column is hidden.

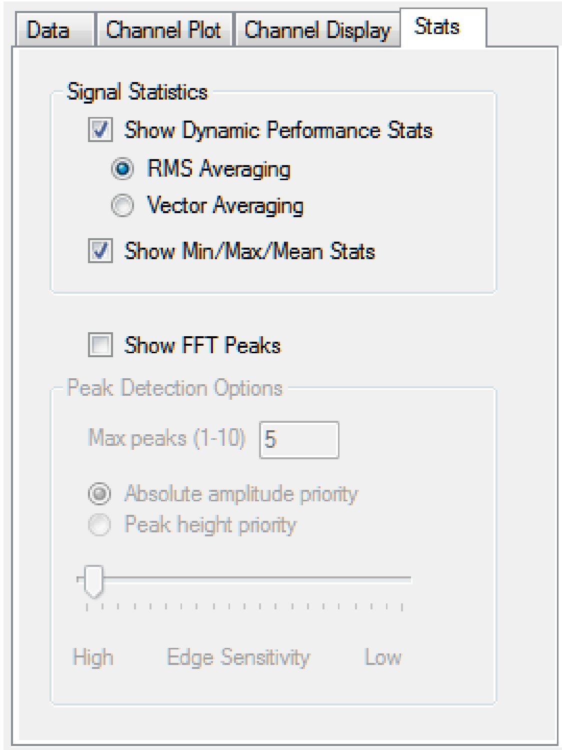
Moving the Visible Statistics Column

If you wish, you can move the position of the **Visible Statistics** column by clicking the left-mouse button to select it, dragging it to the right or left, and letting go of the left-mouse button.

Controlling How Data is Displayed in the Statistics Window

Using the **Stats** tab in the Plot and Data Config window, you can control how data is displayed in the Statistics window.

Note: Refer to page 225 for information on using the Stats tab to configure statistics for FFT plots. Statistics for FFT plots are displayed in the Channel Plot window for the FFT channel.



Showing Dynamic Performance Statistics

By selecting **Show Dynamic Performance Stats**, you can quantify the analog characteristics of the data that is acquired through the A/D converter on the device.

When the **Show Dynamic Performance Stats** option is selected, an FFT is performed on the data for the selected time-domain channels (but you do not see the FFT plot.) The returned FFT data contains the DC amplitude, main signal amplitude, harmonic amplitude, and noise amplitude of the signal at specific frequencies. You can select one of the following averaging types to compute the FFT components:

- **RMS** – (Root Mean Squared) – Averages the power of the signal in real numbers directly. For single channel measurements, this results in zero phase. RMS averaging reduces signal fluctuations but not the noise floor.
- **Vector** – (Angle) – Averaging computes the average of complex numbers directly. The real part of the number is averaged separately from the imaginary part of the number, reducing the noise floor for random signals.

  Note that to use this type of averaging, the trigger source (specified on the Trigger tab), must be either an external TTL trigger, threshold trigger, or software trigger (if you are also using the analog output channel as the input source). It cannot be a Free Run trigger.

Once the FFT is performed, additional mathematical operations are done on the data to compute the following statistics:

- IBF – Input Below Full-Scale
- THD – Total Harmonic Distortion

- SFDR – Spurious Free Dynamic Range
- SINAD – Signal-to-Noise And Distortion
- SNR – Signal-to-Noise Ratio
- ENOB – Effective Number of Bits

The following sections describe the dynamic performance statistics in detail.

Note: The FFT Analysis and Advanced FFT Analysis options of QuickDAQ always use a Blackman Harris window to compute dynamic performance statistics due to its steep side lobe roll-off characteristics. The user-selected FFT window is still applied to FFT plots and time-domain window plots, but is not used to compute dynamic signal statistics.

IBF – Input Below Full-Scale

The IBF value, returned in dB, represents the level of the main signal component relative to the full-scale voltage, as shown in Figure 38.

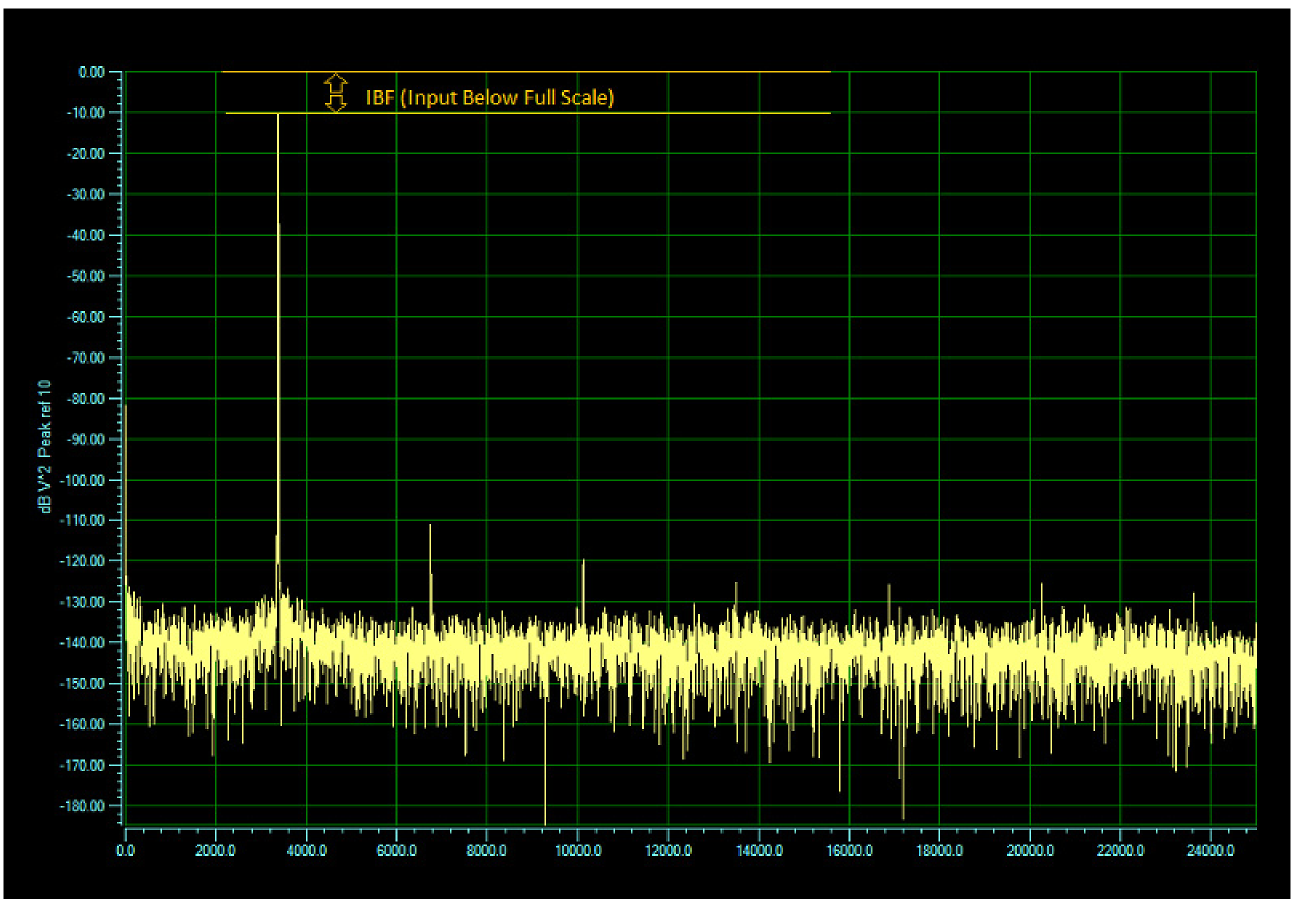


Figure 38: IBF (Input Below Full-Scale)

IBF is computed using the following equation:

$$IBF = 20 \cdot \log_{10}\left(\frac{V_s}{V_f}\right)$$

where V_s is the main signal component and V_f is the full-scale voltage.

For example, if a ±1 V sine wave is applied to an analog input channel that is configured for a ±10 V range, IBF is computed as follows:

$$IBF = 20 \cdot \log_{10}\left(\frac{1}{10}\right) = -20dB$$

THD – Total Harmonic Distortion

The THD value, returned in dB and %, represents the level of the harmonic distortion of the first five harmonics relative to the main signal component, as shown in Figure 39.

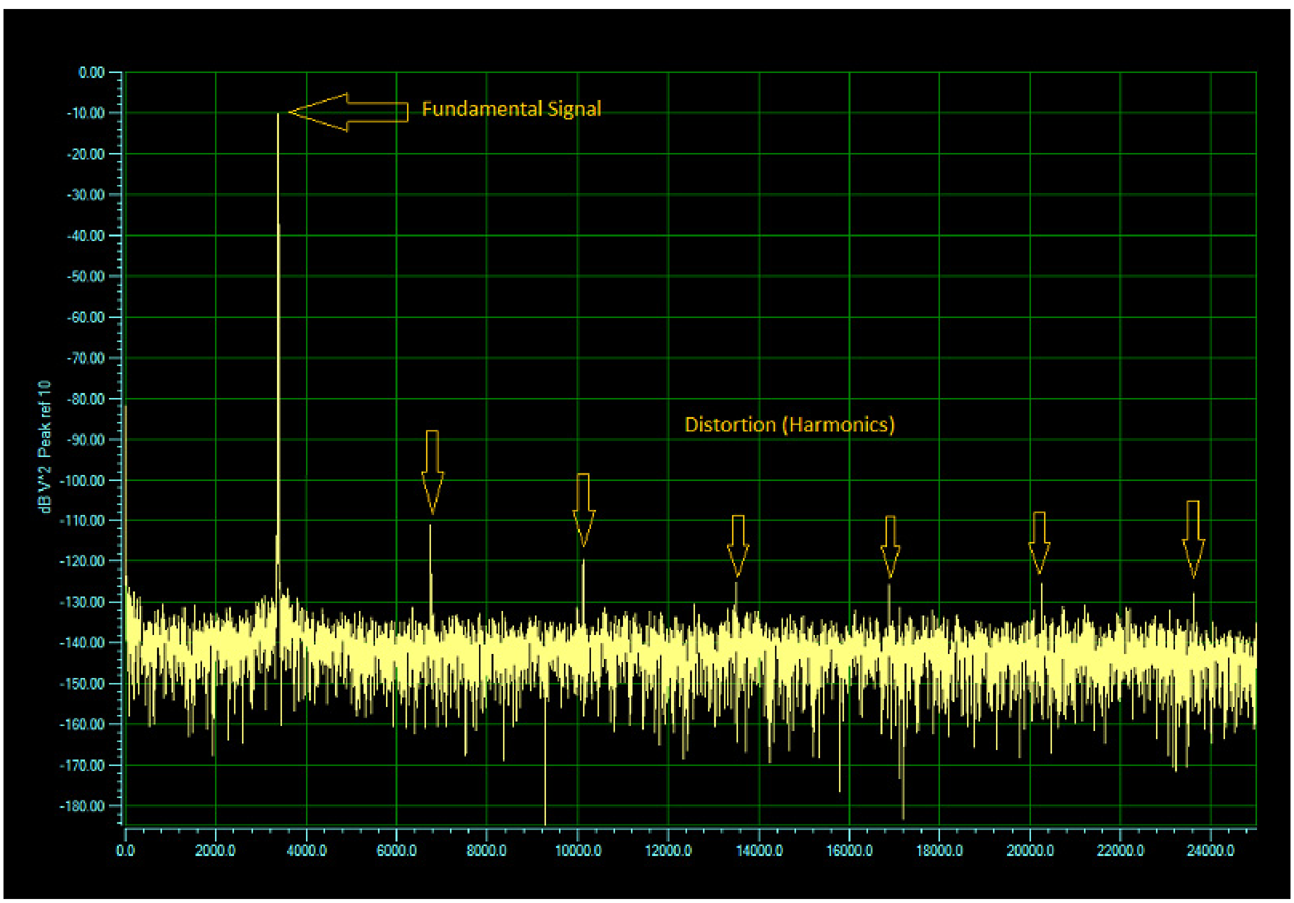


Figure 39: THD (Total Harmonic Distortion)

THD is computed using the following equation:

<u>*dB:*</u>

$$THD = 20 \cdot \log_{10}\left(\frac{\sqrt{h_1^2 + h_2^2 + h_3^2 + h_4^2 + h_5^2}}{V_s}\right)$$

<u>*%:*</u>

$$THD = \left(\frac{\sqrt{h_1^2 + h_2^2 + h_3^2 + h_4^2 + h_5^2}}{V_s}\right) \cdot 100$$

For example, if a 7.07 V_{RMS} sine wave is applied to an analog input channel that is configured for a ±10 V range and the distortion (first five harmonics) is 14 μV_{RMS}, THD is computed as follows:

$$THD = 20 \cdot \log_{10}\left(\frac{0.00001414}{7.07}\right) = -114dB$$

SFDR – Spurious Free Dynamic Range

The SFDR value, returned in dB, represents the level of the full-scale range to the greatest noise or distortion component, as shown in Figure 40.

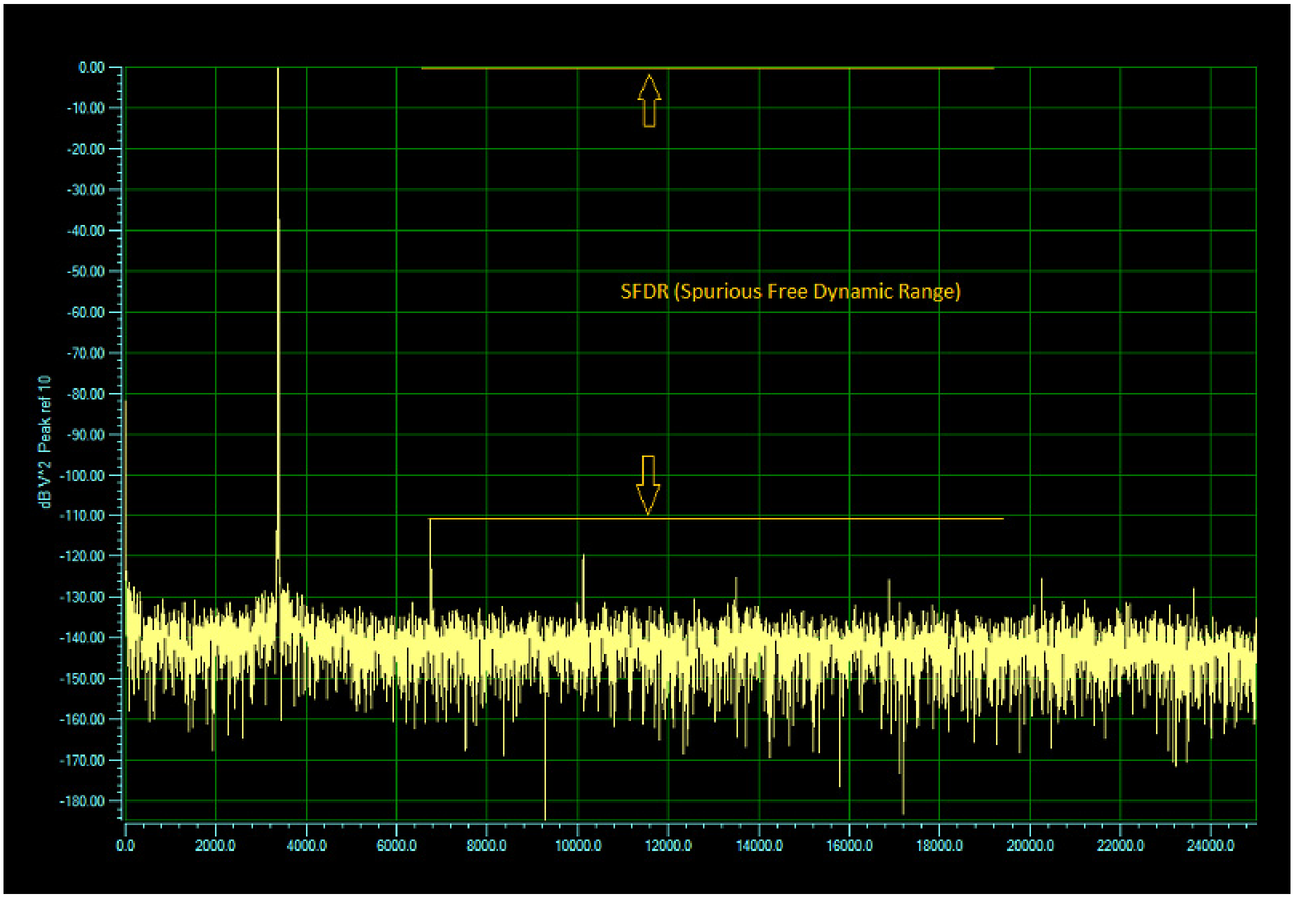


Figure 40: SFDR (Spurious Free Dynamic Range)

SFDR is computed using the following equation:

$$SFDR = 20 \cdot \log_{10}\left(\frac{V_f}{V_a}\right)$$

where V_f is the full-scale range and V_a is the greatest noise component.

For example, if the analog input channel is configured for a ±10 V range, and the greatest noise or distortion component is 7.07 V_{RMS}, SFDR is computed as follows:

$$SFDR = 20 \cdot \log_{10}\left(\frac{7.07}{0.00000707}\right) = 120dB$$

SINAD – Signal-to-Noise and Distortion Ratio

The SINAD value, returned in dB, represents the level of the main signal component relative to the total noise, V_{RMS}, including harmonics, as shown in Figure 41.

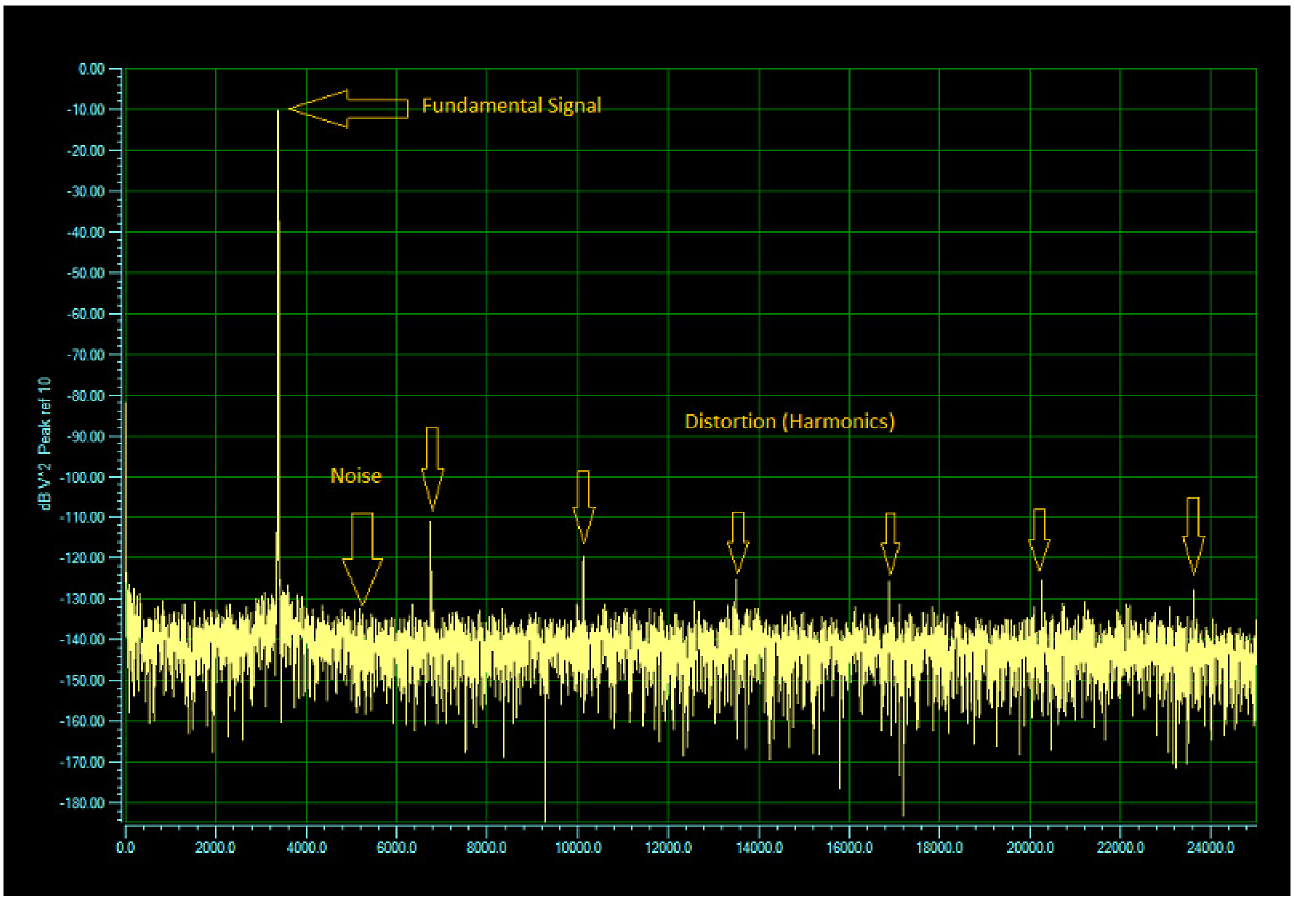


Figure 41: SINAD (Signal-to-Noise And Distortion) Statistic

SINAD is computed using the following equation:

$$SINAD = 20 \cdot \log_{10}\left(\frac{V_s}{V_{n+d}}\right)$$

where V_s is the main signal component and V_{n+d} is the total noise plus distortion.

For example, if a 7.07 V_{RMS} sine wave is applied to an analog input channel that is configured for a ±10 V range and the total noise and harmonics (distortion) is 14.14 μV_{RMS}, SINAD is computed as follows:

$$SINAD = 20 \cdot \log_{10}\left(\frac{7.07}{0.00001414}\right) = 114dB$$

SNR – Signal-to-Noise Ratio

The SNR value, returned in dB, represents the level of the main signal component relative to the total noise, V_{RMS}, as shown in Figure 42. Note that harmonics are not included in the noise.

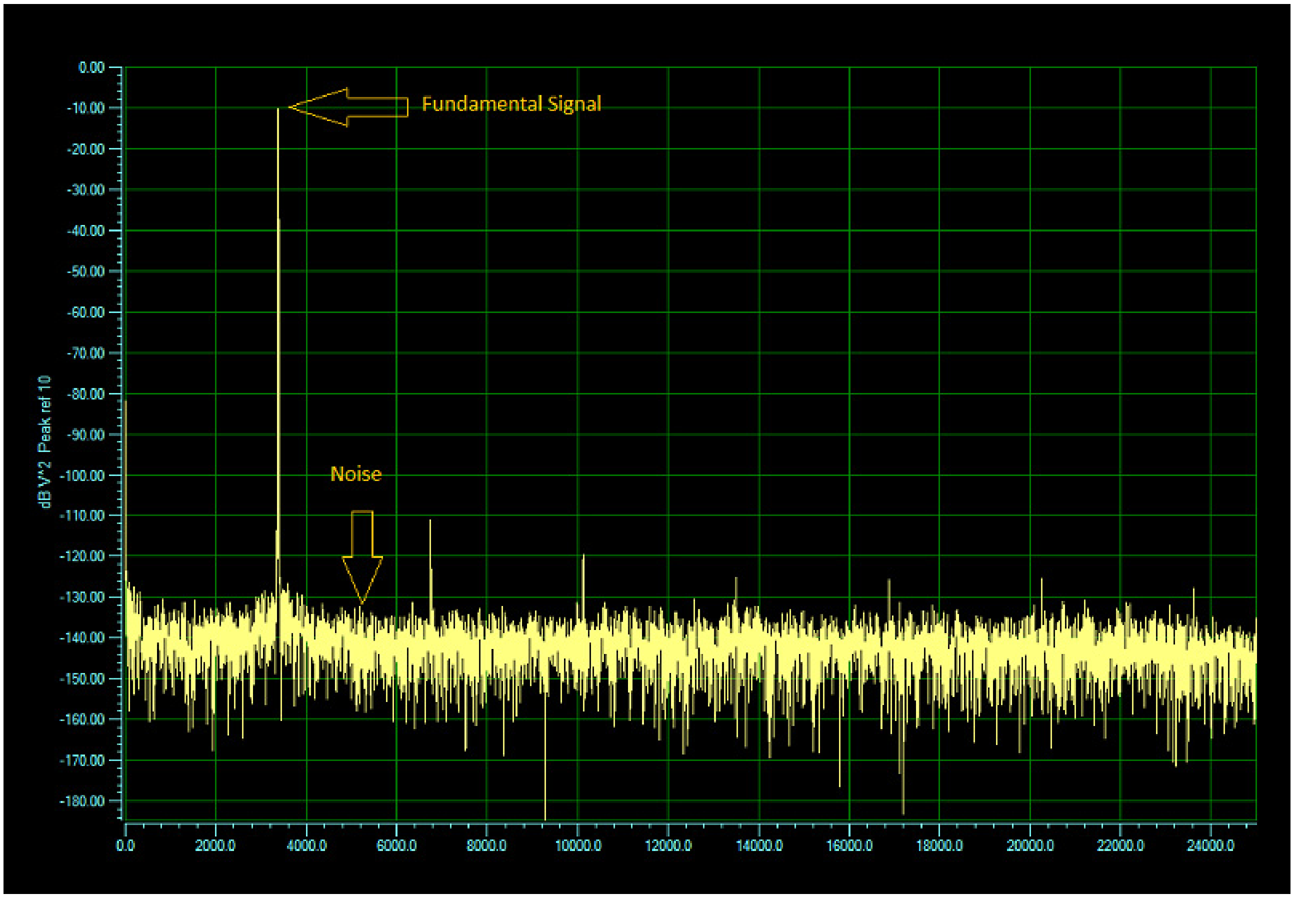


Figure 42: SNR (Signal-to-Noise Ratio) Statistic

SNR is computed using the following equation:

$$SNR = 20 \cdot \log_{10}\left(\frac{V_s}{V_n}\right)$$

where V_s is the main signal component and V_n is the total noise.

For example, if a 7.07 V_{RMS} sine wave is applied to an analog input channel that is configured for a ±10 V range and the total noise is 7.07 μV_{RMS}, SNR is computed as follows:

$$SNR = 20 \cdot \log_{10}\left(\frac{7.07}{0.00000707}\right) = 120dB$$

ENOB – Effective Number of Bits

The ENOB value, returned in bits, represents the quality of a digitized signal. The input signal is typically sinusoidal and should be near full-scale to properly characterize the A/D converter and/or system.

ENOB is computed using the following equation:

$$ENOB = \frac{SINAD - 1.76 - IBF}{6.02}$$

For example, if a ±9 V sine wave is applied to an analog input channel that is configured for a ±10 V range and the average noise floor plus distortion (harmonics) is 30 µV peak, the ENOB is computed as follows:

$$ENOB = \frac{109.54 - 1.76 - (-0.92)}{6.02} = 18.06bits$$

Showing Min/Max/Mean Statistics

If you select **Show Min/Max/Mean Stats**, you can view the following statistics for the selected time-domain channels:

- Min – The minimum value in the engineering units for that channel.
- Max – The maximum value in the engineering units for that channel.
- Delta – The difference between the minimum and maximum value in the engineering units for that channel.
- Mean – The average value for the channel in the engineering units for that channel.
- Std Dev – The standard deviation for the channel.

7

Examples Using the Data Logger Interface

Example of Measuring Temperature and Voltage

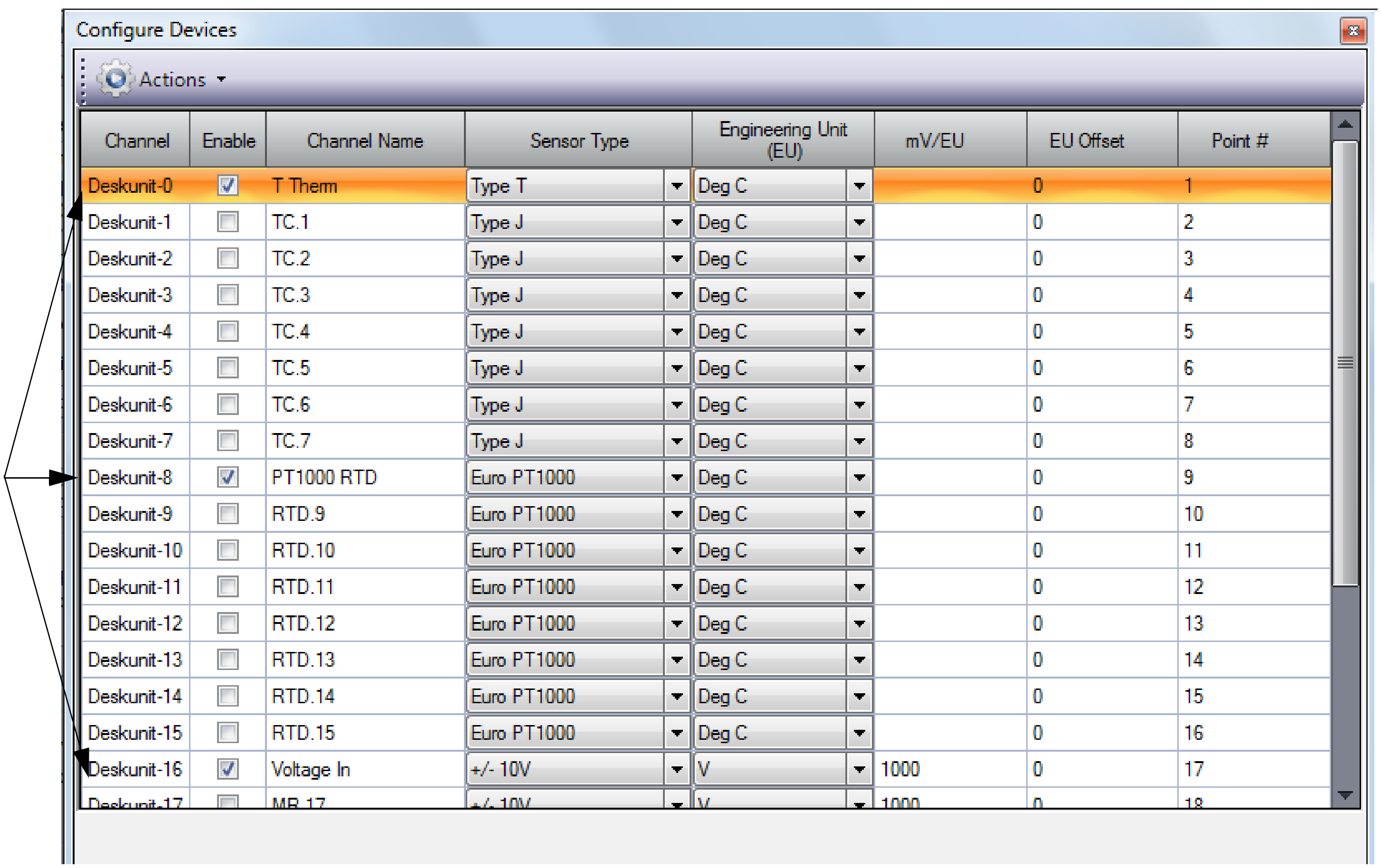
In this example, the Data Logger interface of QuickDAQ is used to configure, measure, and display temperature and voltage data from a data acquisition device. For this example, the following sensors are connected to a MEASURpoint instrument:

- T thermocouple connected to analog input channel 0
- 4-wire, Euro PT1000 RTD connected to analog input channel 8
- ±10 V input signal connected to analog input channel 16

Configure the Channels

Configure the channels as follows:

1. Configure each analog input channel by clicking the **Configuration** menu, and clicking **Input Channel Configuration**, or by clicking the **Input Channel Configuration** toolbar button ( ).
2. Enable analog input channels 0, 8, and 16 by clicking the checkbox under the **Enable** column.



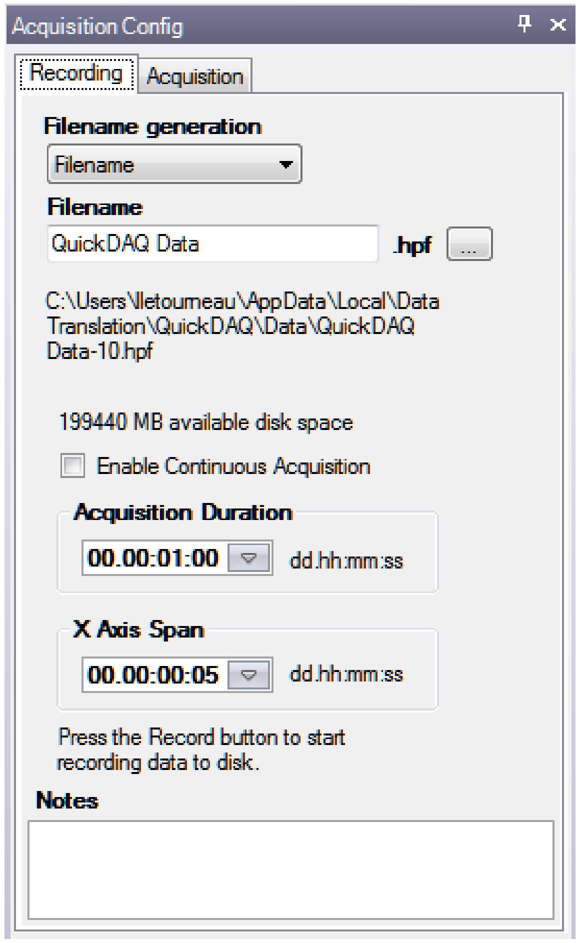
Channel	Enable	Channel Name	Sensor Type	Engineering Unit (EU)	mV/EU	EU Offset	Point #
Deskunit-0	☑	T Therm	Type T	Deg C		0	1
Deskunit-1	☐	TC.1	Type J	Deg C		0	2
Deskunit-2	☐	TC.2	Type J	Deg C		0	3
Deskunit-3	☐	TC.3	Type J	Deg C		0	4
Deskunit-4	☐	TC.4	Type J	Deg C		0	5
Deskunit-5	☐	TC.5	Type J	Deg C		0	6
Deskunit-6	☐	TC.6	Type J	Deg C		0	7
Deskunit-7	☐	TC.7	Type J	Deg C		0	8
Deskunit-8	☑	PT1000 RTD	Euro PT1000	Deg C		0	9
Deskunit-9	☐	RTD.9	Euro PT1000	Deg C		0	10
Deskunit-10	☐	RTD.10	Euro PT1000	Deg C		0	11
Deskunit-11	☐	RTD.11	Euro PT1000	Deg C		0	12
Deskunit-12	☐	RTD.12	Euro PT1000	Deg C		0	13
Deskunit-13	☐	RTD.13	Euro PT1000	Deg C		0	14
Deskunit-14	☐	RTD.14	Euro PT1000	Deg C		0	15
Deskunit-15	☐	RTD.15	Euro PT1000	Deg C		0	16
Deskunit-16	☑	Voltage In	+/- 10V	V	1000	0	17
Deskunit-17	☐	MR.17	+/- 10V	V	1000	0	18

3. Under the **Channel Name** column, enter a meaningful name for each channel. For this example, enter the following names:
   - For analog input channel 0, enter **T Therm** as the channel name.
   - For analog input channel 8, enter **PT1000 RTD** as the channel name.
   - For analog input channel 16, enter **Voltage In** as the channel name.
4. Under the **Sensor** column, select the following sensor types for the analog input channels:
   - For analog input channel 0, select **Type T** as the thermocouple type.
   - For analog input channel 8, select **Euro PT1000** as the RTD type.
   - For analog input channel 16, select **+/-10V** as the input voltage range.
5. Under the **Engineering Units** column, **Deg C** is selected by default. If you want to change this setting, change the temperature units under the **Acquisition** tab.
   *In this example, **Deg C** is used.*
6. If you know the offset for your calibrated thermocouple and RTD, enter it under the **EU Offset** column. For this example, leave **0** as the EU offset for the thermocouple and RTD channels and **1000** as the EU offset for the voltage input channel.
7. Leave the test point values for each channel unchanged.
8. Click **Close** to close the Channel Configuration dialog box.

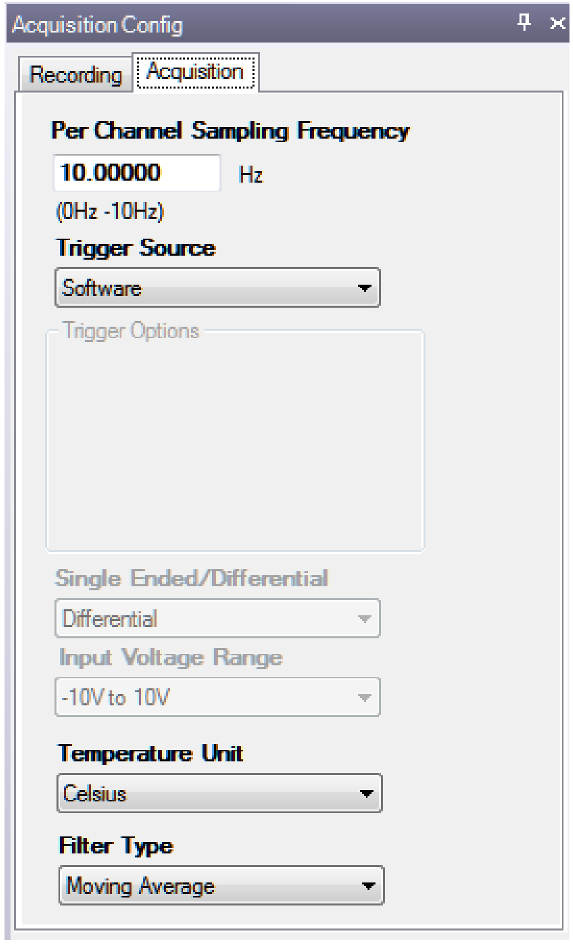
Configure the Parameters of the Acquisition Config Window

For this example, set the Acquisition Config parameters as follows:

1. Click the **Recording** tab.



2. For **Filename generation**, select **Filename**.
3. Enter a meaningful name for the data file.
   In this example, QuickDAQ Data.hpf is used.
4. Leave the **Enable Continuous Acquisition** checkbox unchecked.
5. For **Acquisition Duration**, select **1 minute** as the time to acquire the measurement data.
   The amount of available disk space is shown; in addition, the number of scans in the Acquisition Info area is updated based on the acquisition duration that is selected.
6. For **X Axis Span**, enter **5 seconds** as the span for the x-axis.
7. Click the **Acquisition** tab.

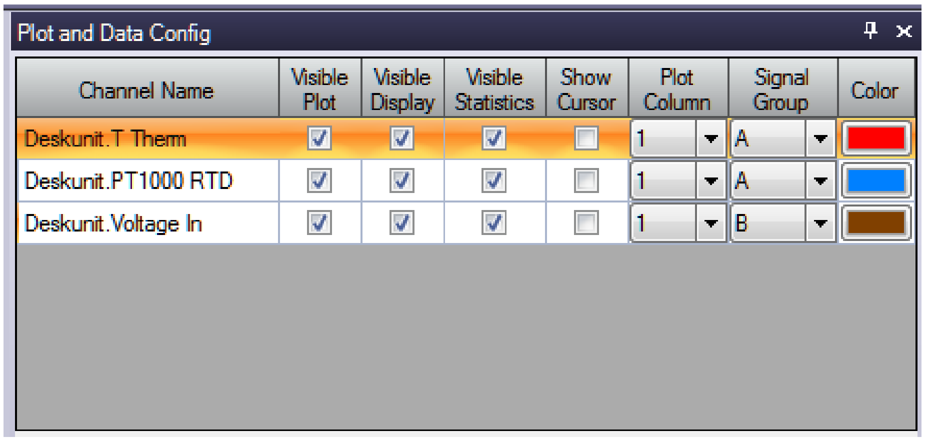


8. For this example, ensure that the following default settings are used:
   - **Sampling Frequency:** 10 Hz
   - **Trigger Source:** Software
   - **Temperature Unit:** Celsius
   - **Filter Type:** Moving Average
9. If desired, hide the **Acquisition Config** window by clicking the **Auto-Hide** pin ( ) in the top, right corner of the window.

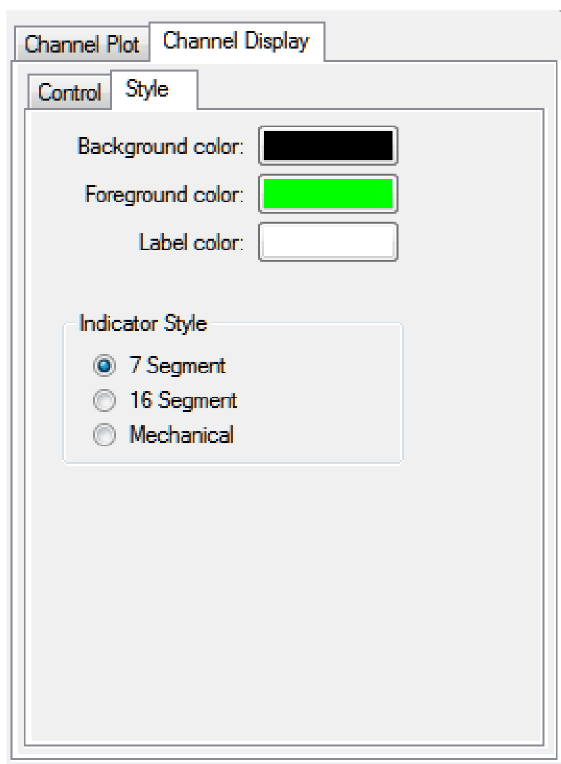
Configure the Appearance of the Channel Display Window

Configure the appearance of the Channel Display window as follows:

1. Ensure that the **Visible Display** column in the **Plot and Data Config** window is checked for all three enabled channels.



2. Click the **Channel Display - Style** tab, and select the color scheme that you want for the Channel Display window.
In this example, black is used for the background color, green is used for the foreground color, and white is used for the label color.

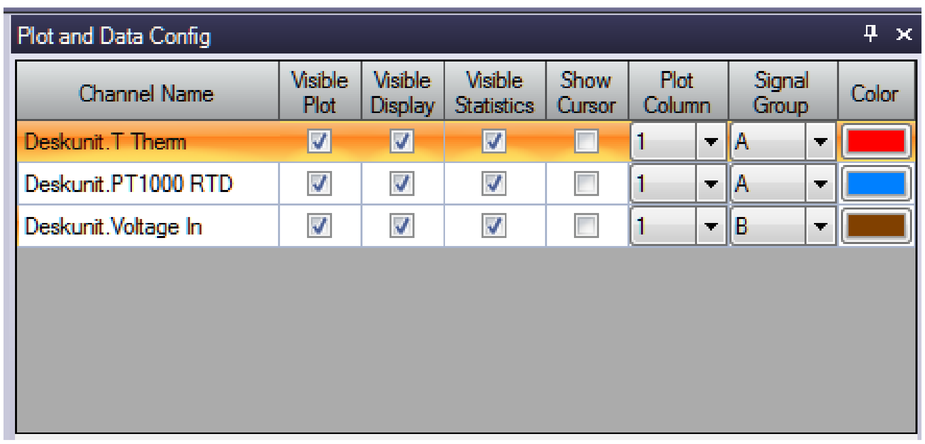


3. Leave the indicator style as **7 segment**.

Configure the Appearance of the Channel Plot Window

Configure the appearance of the Channel Plot window as follows:

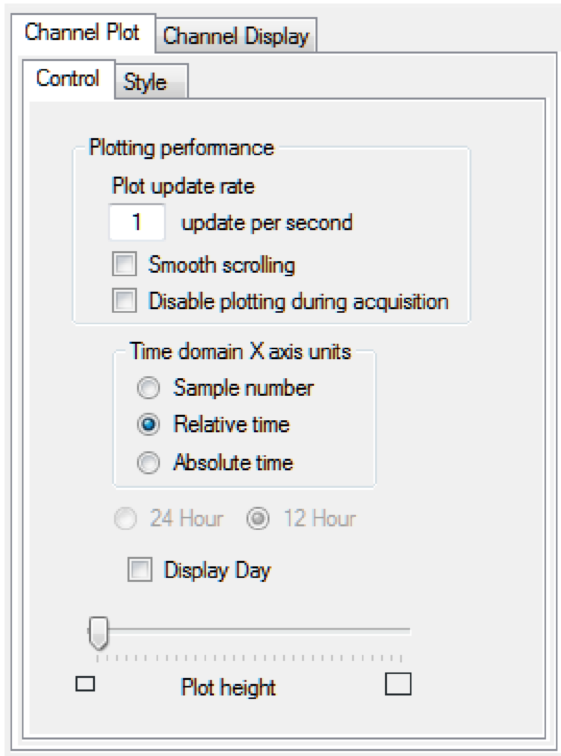
1. In the **Plot and Data Config** window, set up the following parameters:
   a. Ensure that the **Visible Plot** column is checked for all three enabled channels.
   b. Leave the **Show Cursor** column unchecked for all three enabled channels.
   c. Under **Plot Column**, use the default plot column setting of 1 for all three enabled channels.
   d. Under the **Signal Group** column, select **A** for the thermocouple and RTD channels, and select **B** for the voltage input channel.
   e. Under the **Color** column, assign a unique color to each trace.



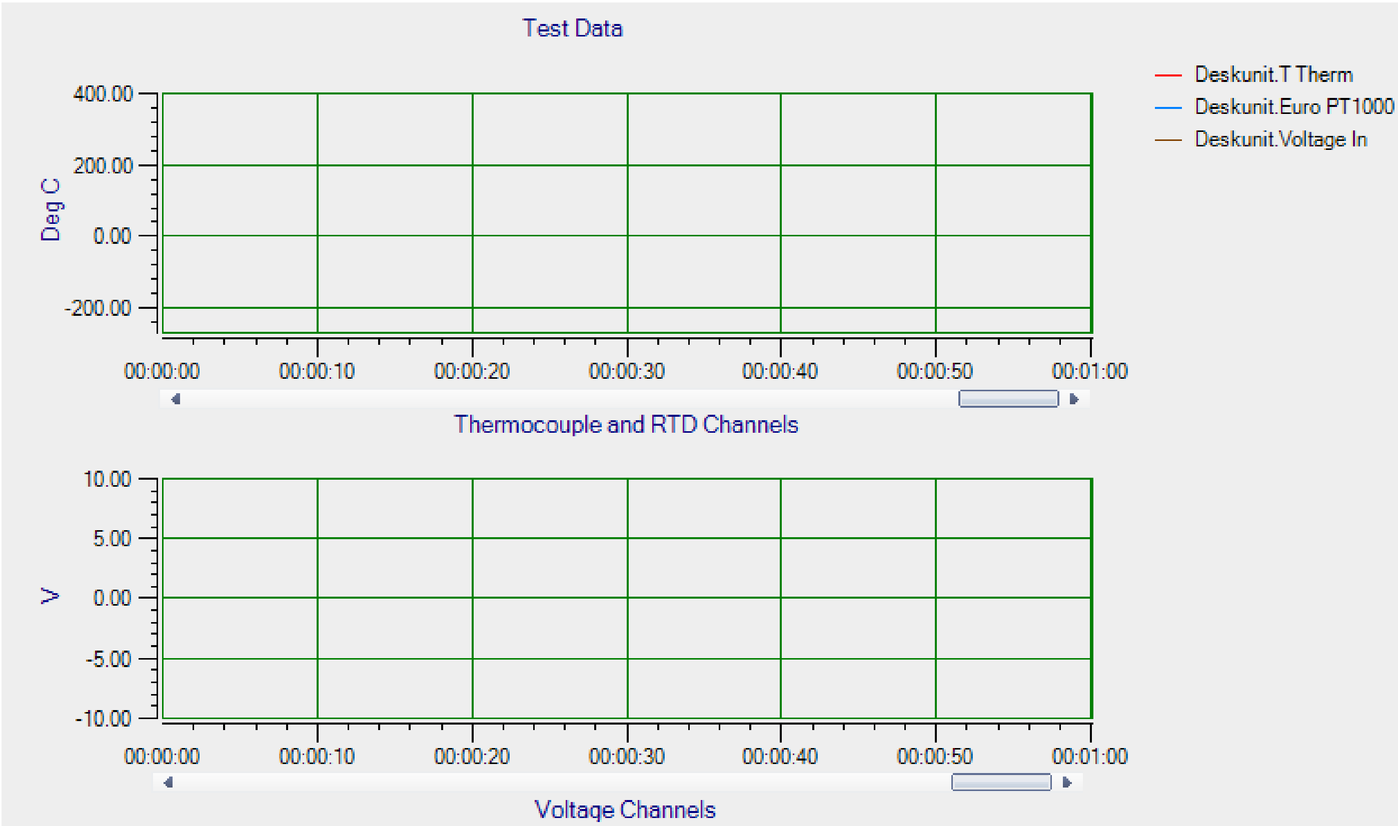
Plot and Data Config

Channel Name	Visible Plot	Visible Display	Visible Statistics	Show Cursor	Plot Column	Signal Group	Color
Deskunit.T Therm	☑	☑	☑	☐	1	A	
Deskunit.PT1000 RTD	☑	☑	☑	☐	1	A	
Deskunit.Voltage In	☑	☑	☑	☐	1	B	

2. Click the **Channel Plot - Control** tab, and slide the **Plot height** bar to the left until you can see both plots on the screen at once.



3. Leave the following settings unchanged:
   - **Plot update rate:** 1
   - **Smooth scrolling:** unchecked
   - **Disable during acquisition:** unchecked
   - **Time domain X axis units:** Relative time
   - **Display Days**: unchecked
4. In the display area, click the tab for the **Channel Plot** window.
5. Click the **Show Legend** control () on the toolbar.
6. For the top plot in this window, change the text for the label on the x-axis, by doing the following:
   a. Right-click on the label.
   b. Select **Edit Label**.
   c. Enter the following text: **Thermocouple and RTD Channels**
7. For the bottom plot in this window, change the text for the label on the x-axis, by doing the following:
   a. Right-click on the label.
   b. Select **Edit Label**.
   c. Enter the following text: **Voltage Channels**
   The Channel Plot window should appear as follows:

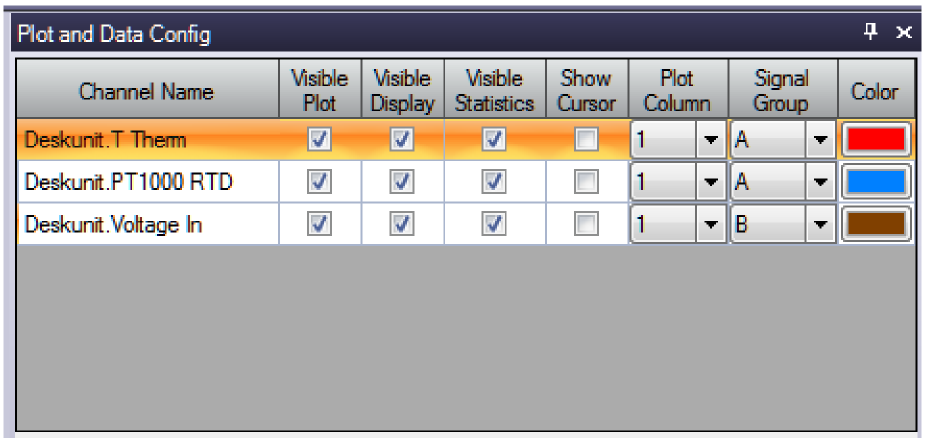


Test Data
Deskunit.T Therm
Deskunit.Euro PT1000
Deskunit.Voltage In
Deg C
400.00
200.00
0.00
-200.00
00:00:00
00:00:10
00:00:20
00:00:30
00:00:40
00:00:50
00:01:00
Thermocouple and RTD Channels
V
10.00
5.00
0.00
-5.00
-10.00
00:00:00
00:00:10
00:00:20
00:00:30
00:00:40
00:00:50
00:01:00
Voltage Channels

Configure the Appearance of the Statistics Window

Configure the appearance of the Statistics window as follows:

1. Ensure that the **Visible Statistics** column in the **Plot and Data Config** window is checked for all three enabled channels:



Plot and Data Config

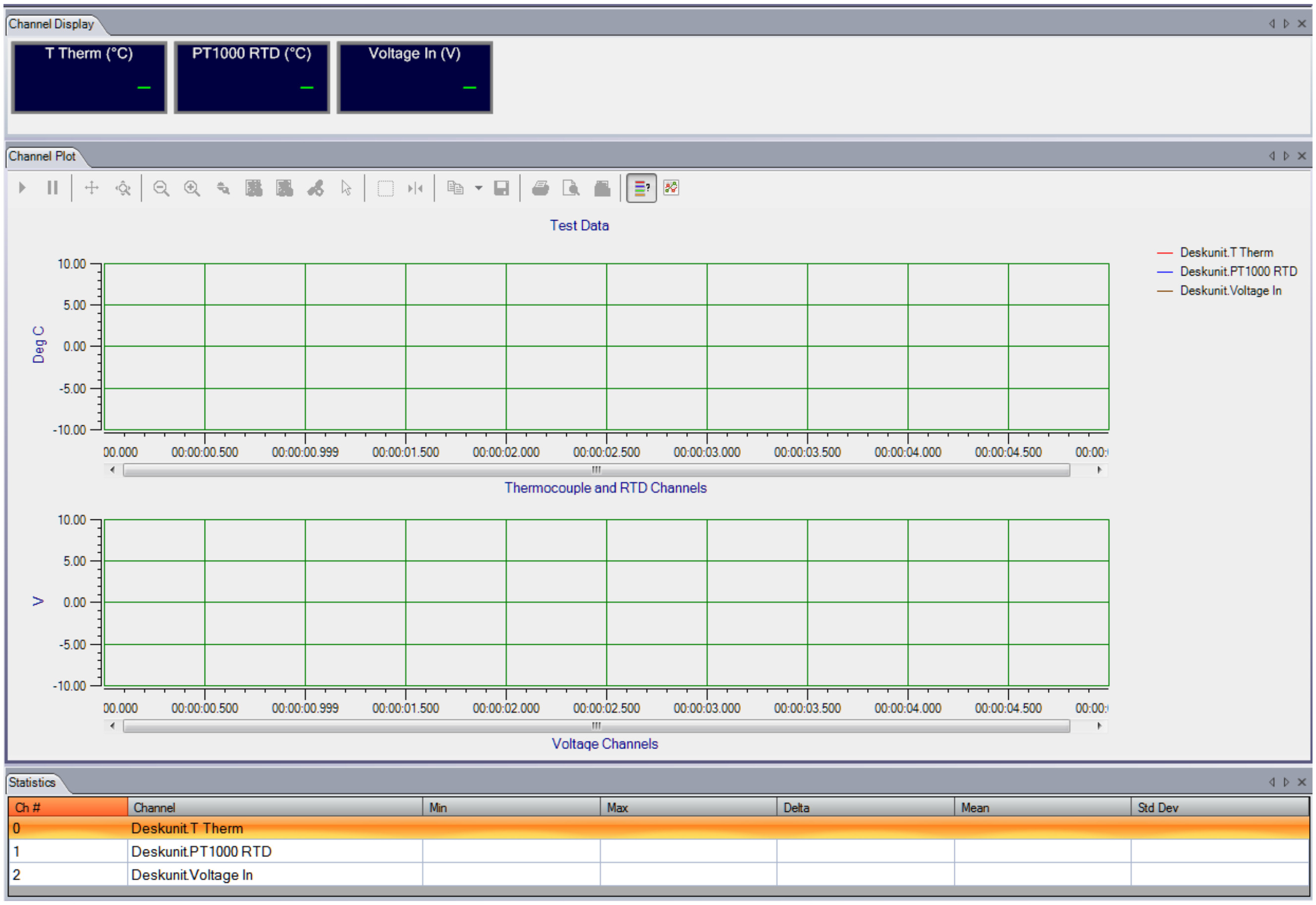
Channel Name	Visible Plot	Visible Display	Visible Statistics	Show Cursor	Plot Column	Signal Group	Color
Deskunit.T Therm	☑	☑	☑	☐	1	A	
Deskunit.PT1000 RTD	☑	☑	☑	☐	1	A	
Deskunit.Voltage In	☑	☑	☑	☐	1	B	

2. If desired, hide the **Plot and Data Config** window by clicking the **Auto-Hide** pin ( ) in the top, right corner of the window.

Position the Windows

If you want see the data that is displayed in the Channel Display, Channel Plot, and Statistics windows at once, you need to move the windows to different locations in the display area.

In the following example, the Channel Display window is located at the top of the display area, the Channel Plot window is located in the middle of the display area, and the Statistics window is located at the bottom of the display area:



Perform the following steps to position the Channel Display window at the top of the display area, the Channel Plot window in the middle of the display area, and the Statistics window at the bottom of the display area:

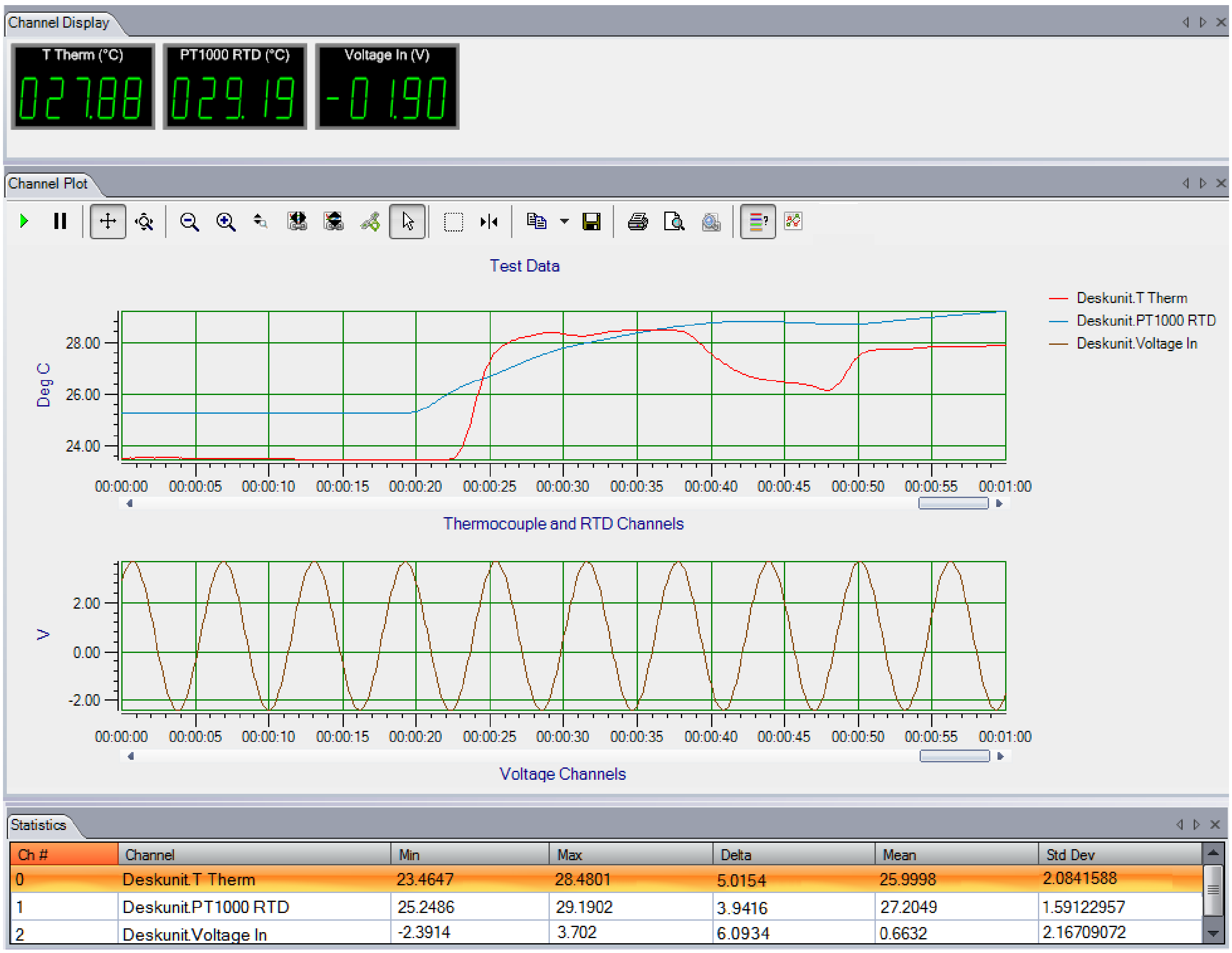
1. Click the tab for the **Statistics** window, drag the window toward the middle of the display area, move the mouse over the guide on the bottom of the guide diamond, and then release the mouse button.
   The Statistics window is now placed at the bottom of the display area.

2. Click the tab for the **Channel Plot** window, drag the window toward the middle of the display area, move the mouse over the guide on the bottom of the guide diamond, and then release the mouse button.
   The Channel Plot window is now placed in the middle of the display area, revealing the Channel Display window at the top of the display area.

3. Resize each window, as desired.

Start the Measurement

Once you have configured the channels and the display area, start acquisition and log data to disk by clicking the **Record** toolbar button (Record).
Results similar to the following are displayed in the display area.



If desired, you can view the data in Excel by clicking the **Open Current Data in Excel** toolbar button ( ).

Example of Monitoring an Analog Output Signal

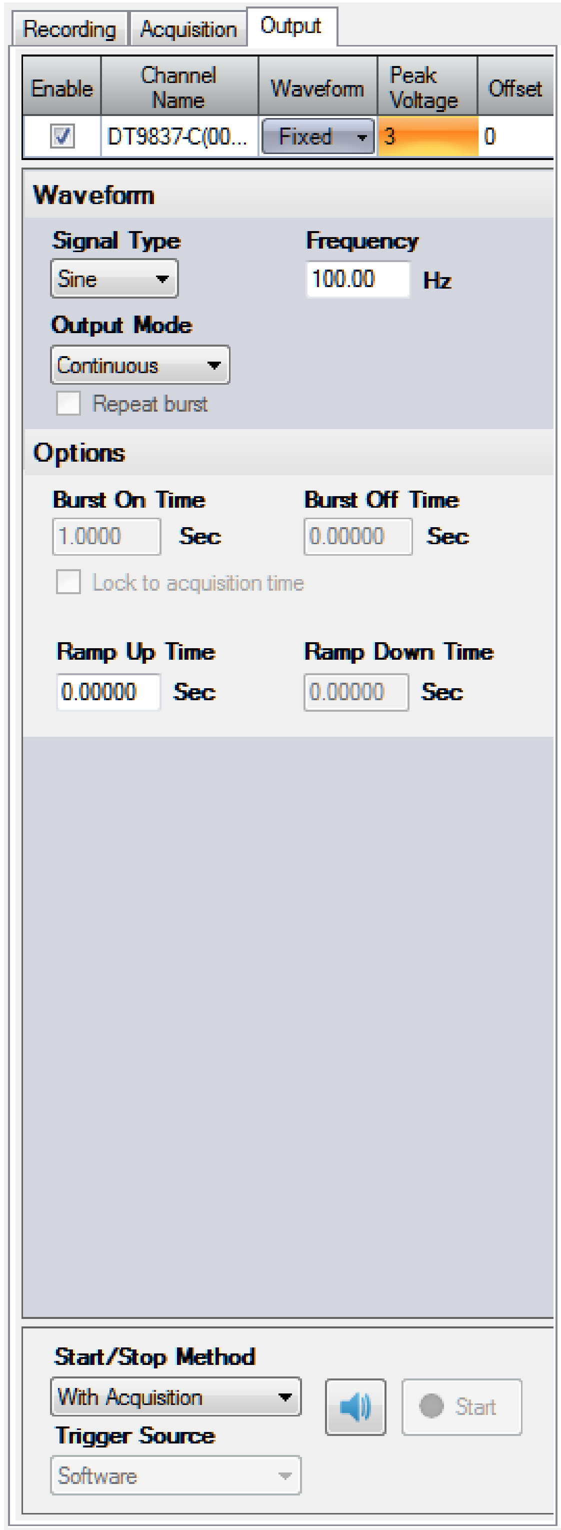
The following steps describe how to use the Data Logger interface of QuickDAQ to monitor the output of an analog output signal of a device using one of its analog input channels.

In this example, a 3 V sine wave is output on analog output channel 0 of a DT9837C module. The analog output channel is connected to analog input channel 0 on the DT9837C module.

Configure the Analog Output Settings

For this example, configure the analog output channel as follows:

1. Click the **Output** tab of the Acquisition Config window.



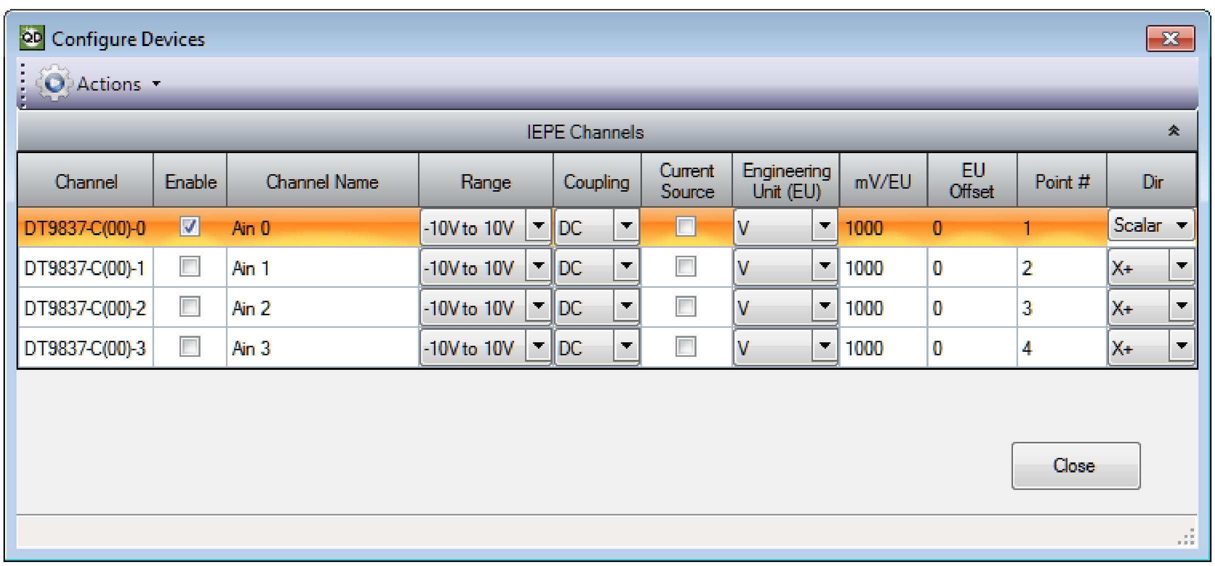
2. Select the **Enable** checkbox to enable the analog output channel on the module.
3. For the **Waveform** type, select **Fixed**.
4. For **Peak Voltage,** enter **3** to output a ±3 V signal.

5. For **Offset**, enter **0**.
6. For **Signal Type**, select **Sine** to output a sine wave.
7. For **Output Mode**, select **Continuous** to output a waveform that repeats continuously.
8. For **Frequency**, select **100 Hz**; this is the frequency of the output waveform.
9. For **Ramp Up Time**, enter **0**.
10. For the **Start/Stop Method**, select **With Acquisition**.
    The analog output operation will start when you start acquisition.
11. If desired, hide the Acquisition Config window by clicking the Auto-Hide pin ( ) in the top, right corner of the window.

Configure the Analog Input Channel

Configure the analog input channel as follows:

1. Ensure that the output from analog output channel 0 is connected to the input of analog input channel 0.
2. Configure the analog input channel by clicking the **Input Channel Configuration** toolbar button ( ) or by clicking the **Configuration** menu and clicking **Input Channel Configuration**.
3. Enable analog input channel 0 by clicking the **Enable** checkbox next to the channel.



4. Under the **Channel Name** column, leave the default channel name as Ain 0.
5. Under the **Range** column, select the input range for analog input channel.
   In this example, ***±10 V*** *is used.*
6. Under the **Coupling** column, select the coupling type (AC or DC) for your sensor.
   Since this example is monitoring a voltage input, ***DC*** *is used.*

7. Under the **Current Source** column, select whether to enable or disable use of the current source on the data acquisition device.
   *Since this example is monitoring a voltage input, the **Current Source** checkbox is not checked (disabled) for the analog input channel.*

8. Under the **Engineering Units** column, select the engineering units for the input.
   *In this example, **V** is used.*

9. Enter the number of mV per engineering unit in the **mV/EU** field.
   *In this example, **1000** is used.*

10. If an offset is specified for the input, enter the value in the **EU Offset** field.
    *In this example, no offset (**0**) is specified.*

11. If desired, enter a test point value for the channel.
    *In this example, **0** is used.*

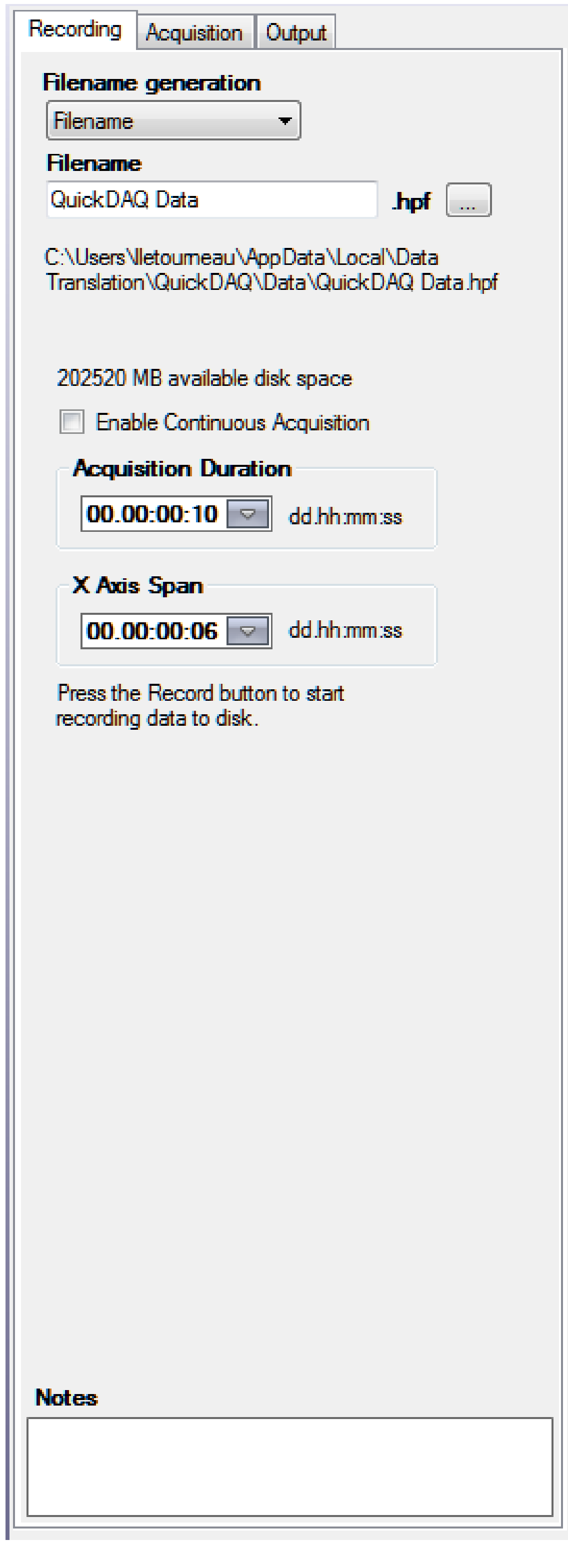
12. If desired, enter a sensor direction for each channel.
    *In this example, **Scalar** is used.*

13. Click **Close** to close the Configure Devices dialog box.

Configure the Recording Settings

For this example, configure the recording settings as follows:

1. Click the **Recording** tab of the Acquisition Config window.

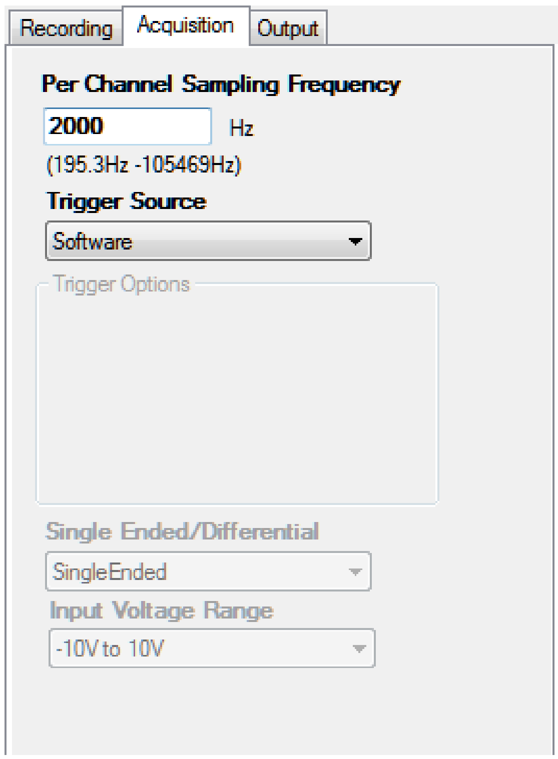


2. For **Filename generation**, use the **Filename** option.
3. For **Filename**, use the default name for the data file.
4. Leave the **Enable Continuous Acquisition** checkbox unchecked.
5. For **Acquisition Duration**, select **10 Seconds** as the time to acquire data.

Configure the Acquisition Settings

For this example, configure the acquisition settings as follows:

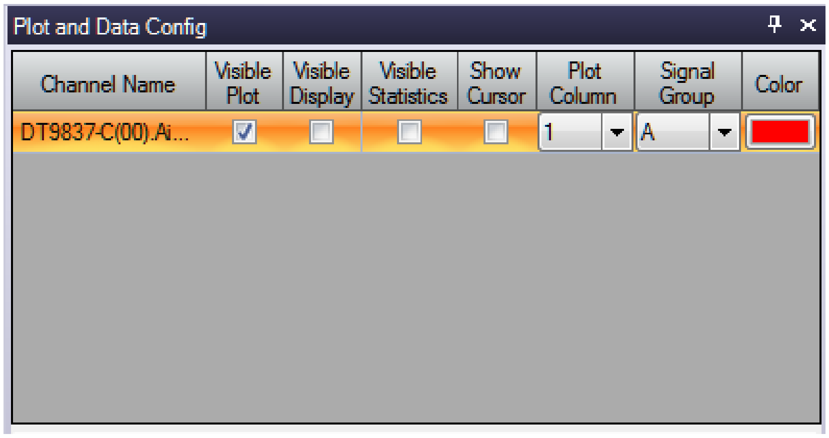
1. Click the **Acquisition** tab of the Acquisition Config window.



2. In the **Maximum frequency to analyze** text box, enter **2000** as the bandwidth or span, in Hz, of the input signals that you want to analyze
Internally, the sampling rate of the data acquisition device is set to twice the bandwidth, or 4000 Hz in this example.
3. For the **Trigger Source**, select **Software** to ensure that the measurement starts as soon as the **Record** button is clicked.
4. If desired, hide the Acquisition Config window by clicking the Auto-Hide pin ( ) in the top, right corner of the window.

Configure the Appearance of the Display

In the Plot and Data Config window, ensure that the **Visible Plot** checkbox is checked.

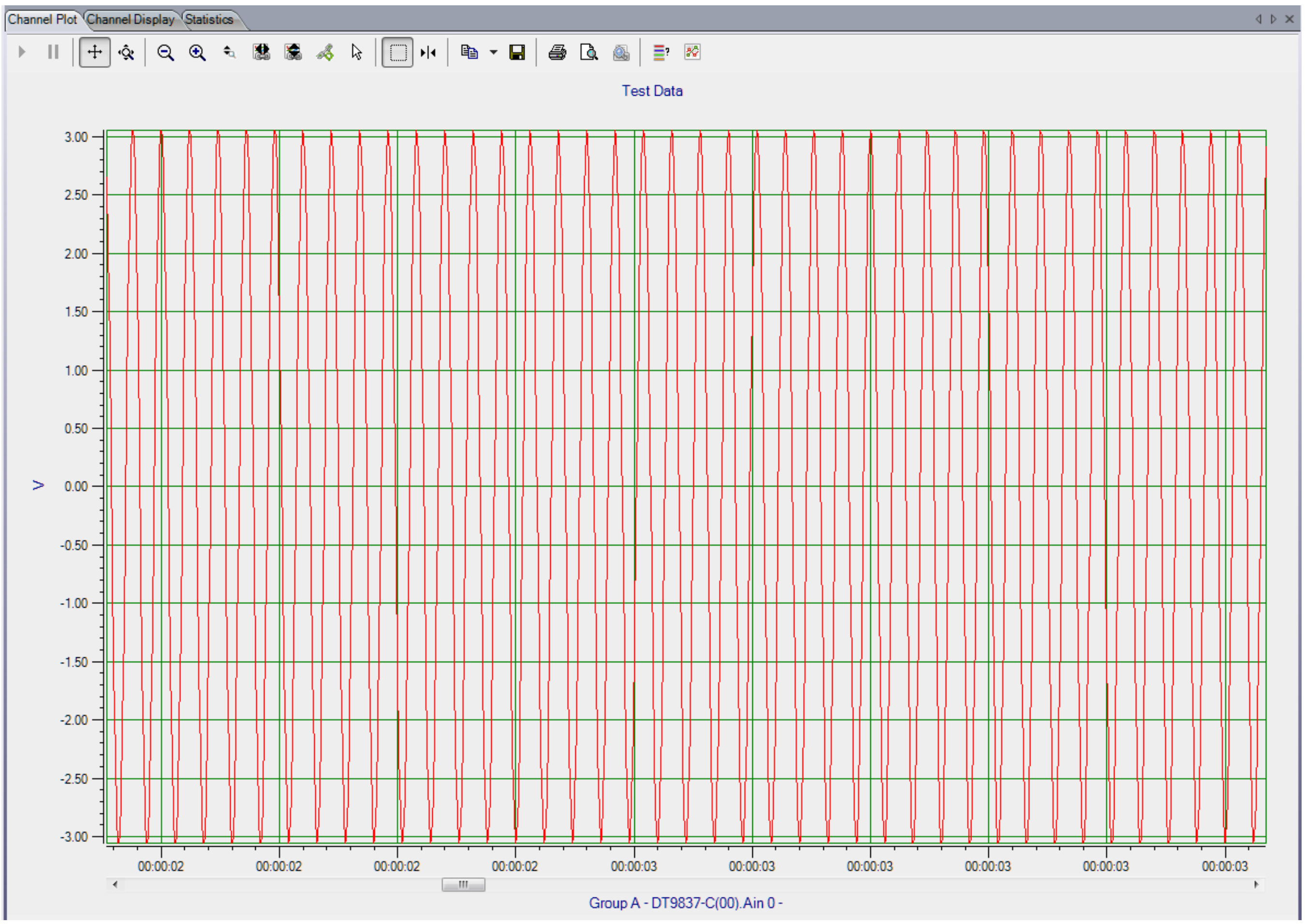


Start the Operation

Once you have configured the channels and the application parameters, click the **Record** toolbar button (

) or press the **F5** key to start the operation.

Results similar to the following are displayed in the Channel Plot window, showing the output of analog output channel 0 as measured by analog input channel 0.



8

Examples Using the FFT Analyzer Interface

Acceleration Measurement Example

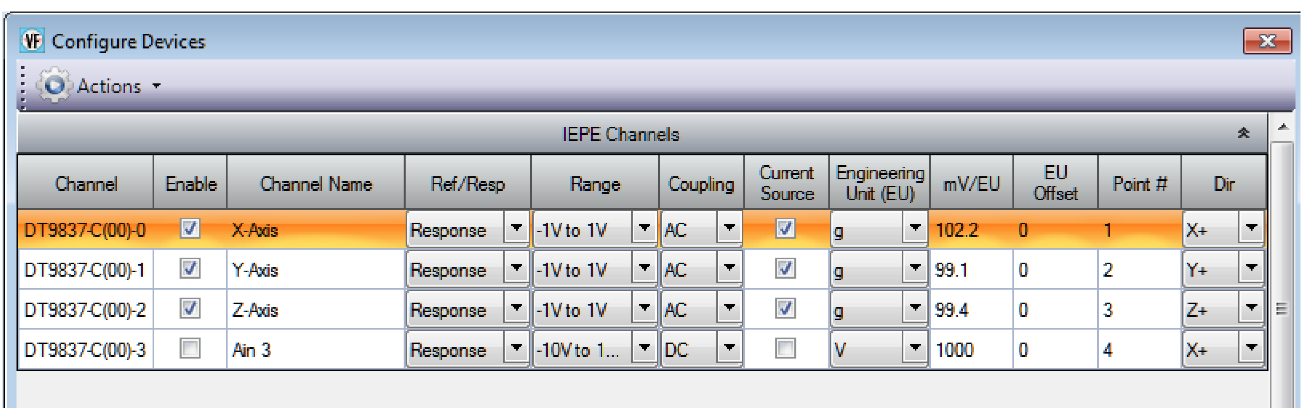
If you are using the FFT Analysis option or Advanced FFT Analysis option of QuickDAQ, you can use the following steps to perform an acceleration measurement.

This example uses a triaxial accelerometer connected to a DT9837C module to measure the vibration of a fan with an unbalanced blade. The FFT Analyzer interface of QuickDAQ is used.

Configure the Channels

Configure the channels as follows:

1. Ensure that the accelerometer is connected to your data acquisition device. For a triaxial accelerometer, each axis should be connected to a unique analog input channel.
   In this example, the x-axis connector is attached to analog input channel 0, the y-axis connector is attached to analog input channel 1, and the z-axis connector is attached to analog input channel 2 of a DT9837C module.
2. Configure each analog input channel by clicking the **Input Channel Configuration** toolbar button ( ) or by clicking the **Configuration** menu and clicking **Input Channel Configuration**.
3. Enable analog input channels 0, 1, and 2 at once by highlighting these analog input channels, clicking the **multi-channel Actions** button (Actions), and clicking **Enable all Selected Channels**. Alternatively, you can enable the channels by clicking the **Enable** checkbox for each of these channels.
   In this example, analog input channels 0, 1, and 2 are enabled.



4. Under the **Channel Name** column, enter the following names for the analog input channels:
   – For analog input channel 0, enter **X-Axis**.
   – For analog input channel 1, enter **Y-Axis**.
   – For analog input channel 2, enter **Z-Axis**.
5. Under the **Ref/Resp** column, select **Response** for the analog input channe1s.

6. Under the **Range** column, select the best effective input range (±10 V or ±1 V) for each signal. For example, if your accelerometer measures 100 mV/g and your signal of interest is less than 100 g, choose the ±1 V range; similarly, if your signal of interest is greater than 100 g, choose the ±10 V range.
   In this example, ***±1 V*** *is used.*
7. Under the **Coupling** column, select the coupling type (AC or DC) for your sensor. For most IEPE accelerometers, you will want to choose **AC** coupling to remove any DC offset from your measurement. Choose **DC** coupling only if the signal source has no offset voltage or if the DC content of the acquired signal is important.
   In this example, ***AC*** *is used.*
8. Under the **Current Source** column, select whether to enable or disable use of the 4 mA current source on the data acquisition device. Since most IEPE accelerometers require a current source of 4 mA and a compliance voltage of at least 18 V to drive their internal circuitry, in most cases, you will want to enable use of the current source for each analog input channel.
   In this example, the ***Current Source*** *checkbox is checked (enabled) for all three analog input channels.*

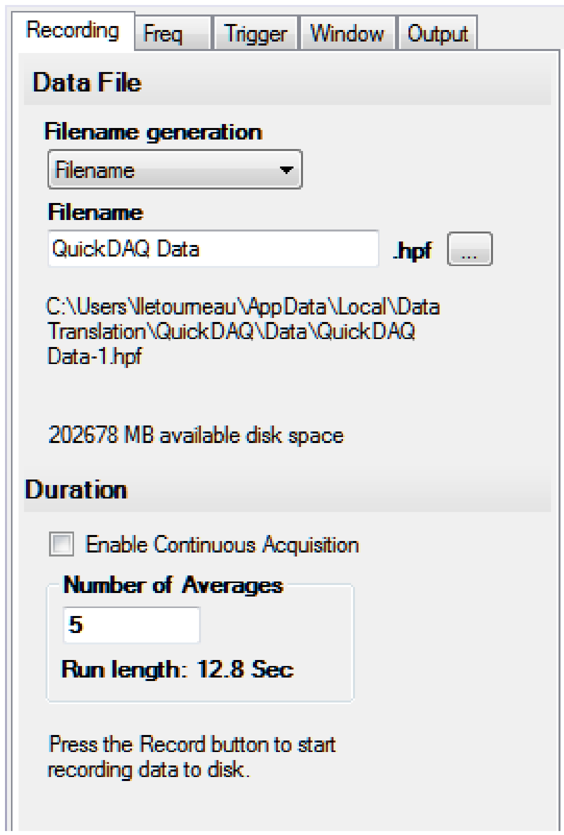
Note: If you enable the use of the internal 4 mA excitation current source, it is recommended that you choose AC coupling.

9. Under the **Engineering Units** column, select the engineering units for the accelerometer.
   In this example, ***g*** *is used.*
10. Using the calibration certificate that was supplied with your accelerometer, enter the calibrated sensitivity for the axis that is connected to the selected analog input channel in the **mV/EU** field. In this example, the following values are used:
    – Analog input channel 0 (X-Axis) = 102.2 mV/g
    – Analog input channel 1 (Y-Axis) = 99.1 mV/g
    – Analog input channel 2 (Z-Axis) = 99.4 mV/g
11. If an offset is specified for the accelerometer, enter the value in the **EU Offset** field.
    In this example, no offset (0) is specified.
12. If desired, enter a test point value for each channel. In this example, the following values are used:
    – Analog input channel 0 (X-Axis) = 1
    – Analog input channel 1 (Y-Axis) = 2
    – Analog input channel 2 (Z-Axis) = 3
13. If desired, enter a sensor direction for each channel. In this example, the following values are used:
    – Analog input channel 0 (X-Axis) = X+
    – Analog input channel 1 (Y-Axis) = Y+
    – Analog input channel 2 (Z-Axis) = Z+
14. Click **Close** to close the Configure Devices dialog box.

Configure the Recording Settings

For this example, configure the recording settings as follows:

1. Click the **Recording** tab of the Acquisition Config window.

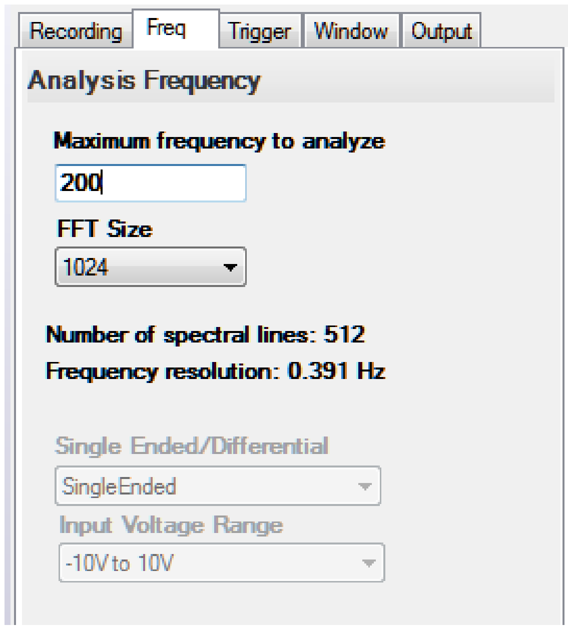


2. For **Filename generation**, use the **Filename** option.
3. For **Filename**, use the default name for the data file.
4. Leave the **Enable Continuous Acquisition** checkbox unchecked.
5. For **Number of Averages**, enter **5** as the number of records to acquired, process, and average before acquisition stops.
   The number of seconds for the total run and the amount of available disk space are shown.

Configure the Frequency Settings

Configure the frequency settings as follows:

1. Click the **Freq** tab of the Acquisition Config window.

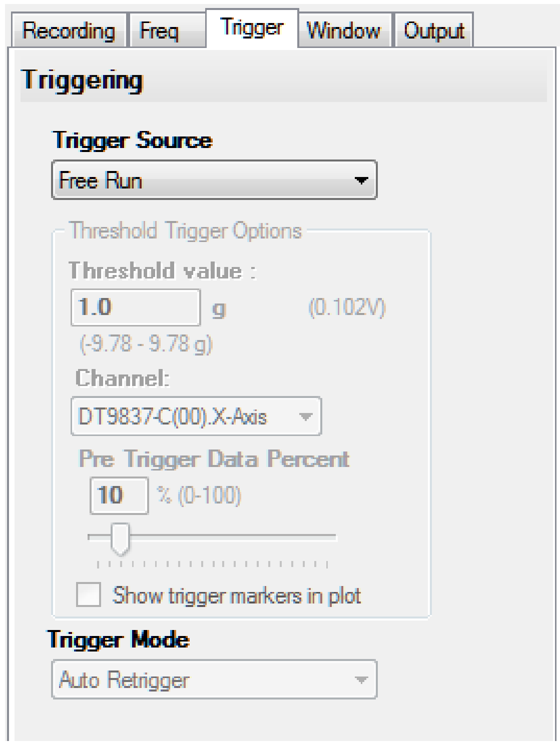


2. In the **Maximum frequency to analyze** text box, enter **200** as the bandwidth or span, in Hz, of the input signals that you want to analyze
   Internally, the sampling rate of the data acquisition device is set to twice the bandwidth, or 400 Hz in this example.
3. In the **FFT Size** text box, select **1024** from the drop-down combo box as the number of data points to use for the FFT calculation.
   The number of spectral lines and the frequency resolution are displayed.

Configure the Trigger Settings

For this example, configure the trigger settings as follows:

1. Click the **Trigger** tab of the Acquisition Config window.

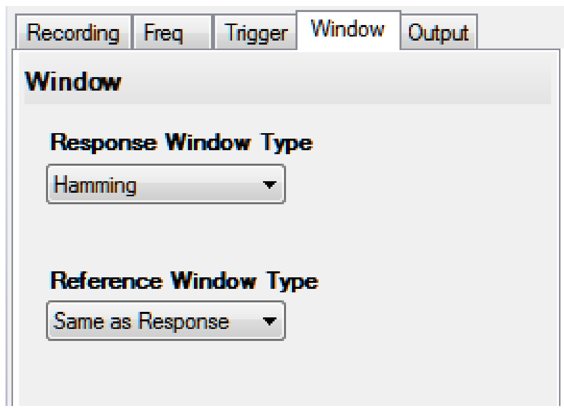


2. For the **Trigger Source**, select **Free Run** to ensure that the measurement starts as soon as the **Record** button is clicked.
   In this mode, the Trigger Mode is not used.

Configure the Windowing Function

For this example, configure the windowing function as follows:

1. Click the **Window** tab of the Acquisition and Config window.
   The Window tab for the Advanced FFT Analysis option of QuickDAQ is shown. Note that the FFT Analysis option of QuickDAQ does not use response and reference terms, and does not provide the reference window selections.



2. For the **Response Window Type** (Advanced FFT Analysis option) or **Window Type** (FFT Analysis option), select **Hamming**.
   This is a setting is applied to the FFTs for all enabled analog input channels.
3. If you are using the Advanced FFT Analysis option, leave the setting for the **Reference Window Type** as **Same As Response**.
   This example does not use reference channels.
4. If desired, hide the Acquisition Config window by clicking the Auto-Hide pin ( ) in the top, right corner of the window.

Add FFTs Channel to the Channel Plot Window

The following steps show how to use the FFT Analyzer interface of QuickDAQ to add an FFT channel for each enabled analog input channel to the Channel Plot window:

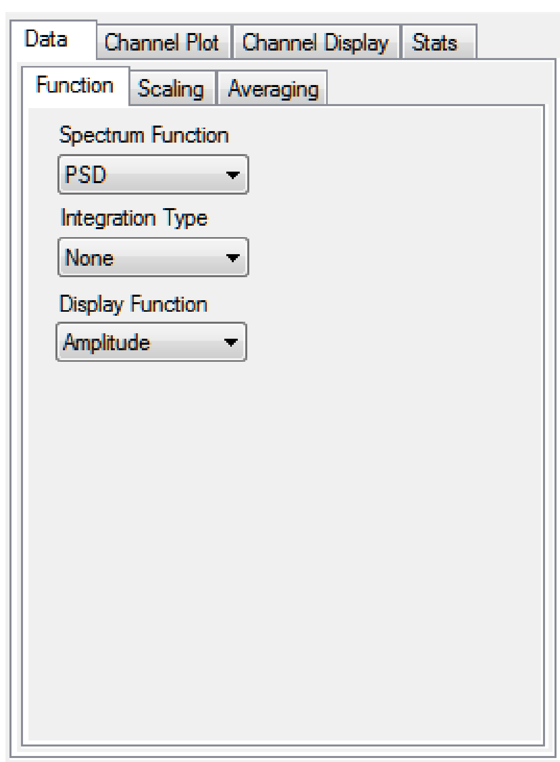
1. Click the **Add single channel FFT function** ( ) button, and select the channel named **DT9837-C(00).X-Axis**.
2. Click the **Add single channel FFT function** ( ) button, and select the channel named **DT9837-C(00s).Y-Axis**.
3. Click the **Add single channel FFT function** ( ) button, and select the channel named **DT9837-C(00).Z-Axis**.

By default, these channels are visible in the Channel Plot window.

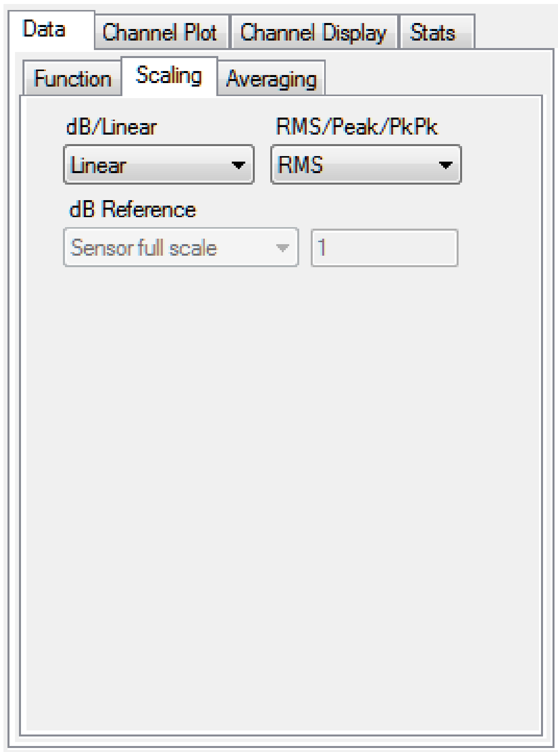
Configure FFT Parameters to Analyze Acceleration Data

The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used to generate the FFTs for each analog input channel:

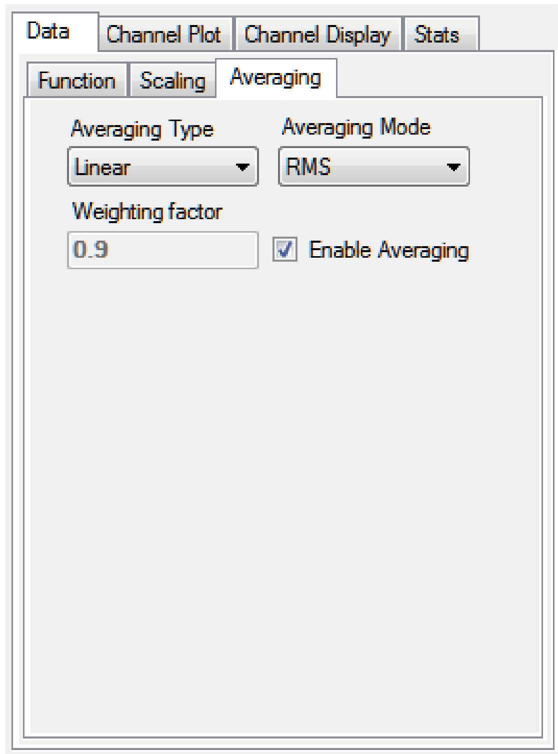
1. Under the **Channel Name** column in the Plot and Data Config window, hold down the **Ctrl** key and select the three FFT channels (**DT9837-C(00).X-Axis-FFT, DT9837-C(00).Y-Axis-FFT, and DT9837-C(00).Z-Axis-FFT**) to configure the settings the same way for all three channels.
2. Click the **Data** tab, and then click the **Function** tab.



3. For this example, configure the settings of this tab as follows:
   a. For **Spectrum Function**, select **PSD** (Power Spectral Density).
   b. For **Integration Type**, select **None** for acceleration data.
   c. For **Display Function**, select **Amplitude**.
4. Click the **Scaling** tab.



5. For this example, configure the settings of this tab as follows:
   a. For **dB/Linear**, select **Linear**.
   b. For **RMS/Peak/PkPk**, select **RMS**.
6. Click the **Averaging** tab.

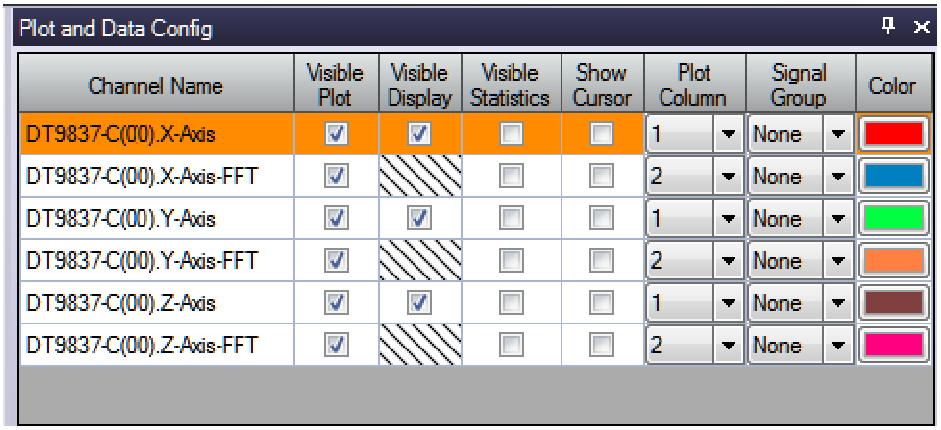


7. For this example, configure the settings of this tab as follows:
   - **a.** Check the **Enable Averaging** checkbox.
   - **b.** For **Averaging Type**, select **Linear**.
   - **c.** For **Averaging Mode**, select **RMS**.

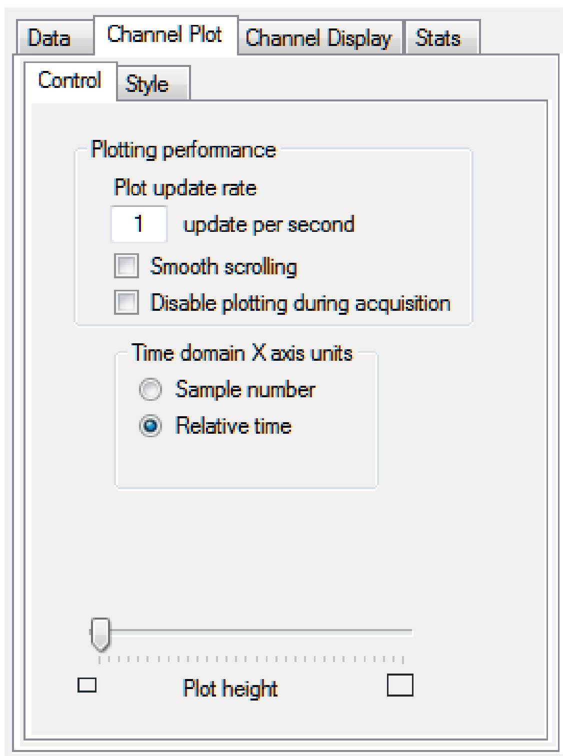
Configure the Appearance of the Channel Plot Window

Because, in this example, we want to look at data from three analog input channels, as well as their corresponding FFTs, in one view without scrolling, perform the following steps to configure the appearance of the Channel Plot window:

1. Under the **Plot Column** heading, assign the analog input channels to plot column 1, and assign the FFT channels to plot column 2, as follows:
   - **a.** Leave the analog input channels unchanged with the default plot column setting of 1.
   - **b.** Hold down the Ctrl key and select all the FFT channels.
   - **c.** Click the **Multi-channel Actions** button ().
   - **d.** Click **Select channels Plot Column**.
   - **e.** Select **2** as the column number to change the plot column for all FFT channels at once.



2. Assign a unique color to each trace, if desired.
3. Click the **Channel Plot - Control** tab.

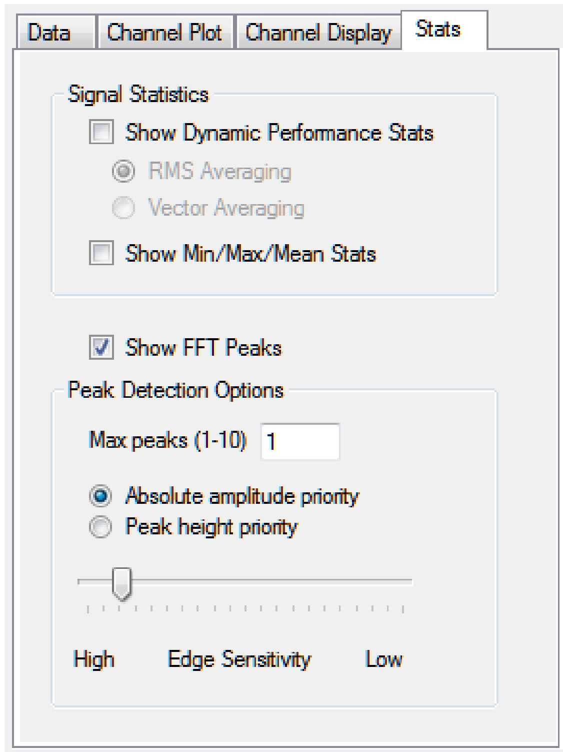


4. Slide the **Plot height** bar to the left until you can see all 6 plots on the screen at once.

Configure Statistics to Analyze Acceleration Data

The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure FFT statistics that are useful when analyzing acceleration data:

1. Click the **Stats** tab.



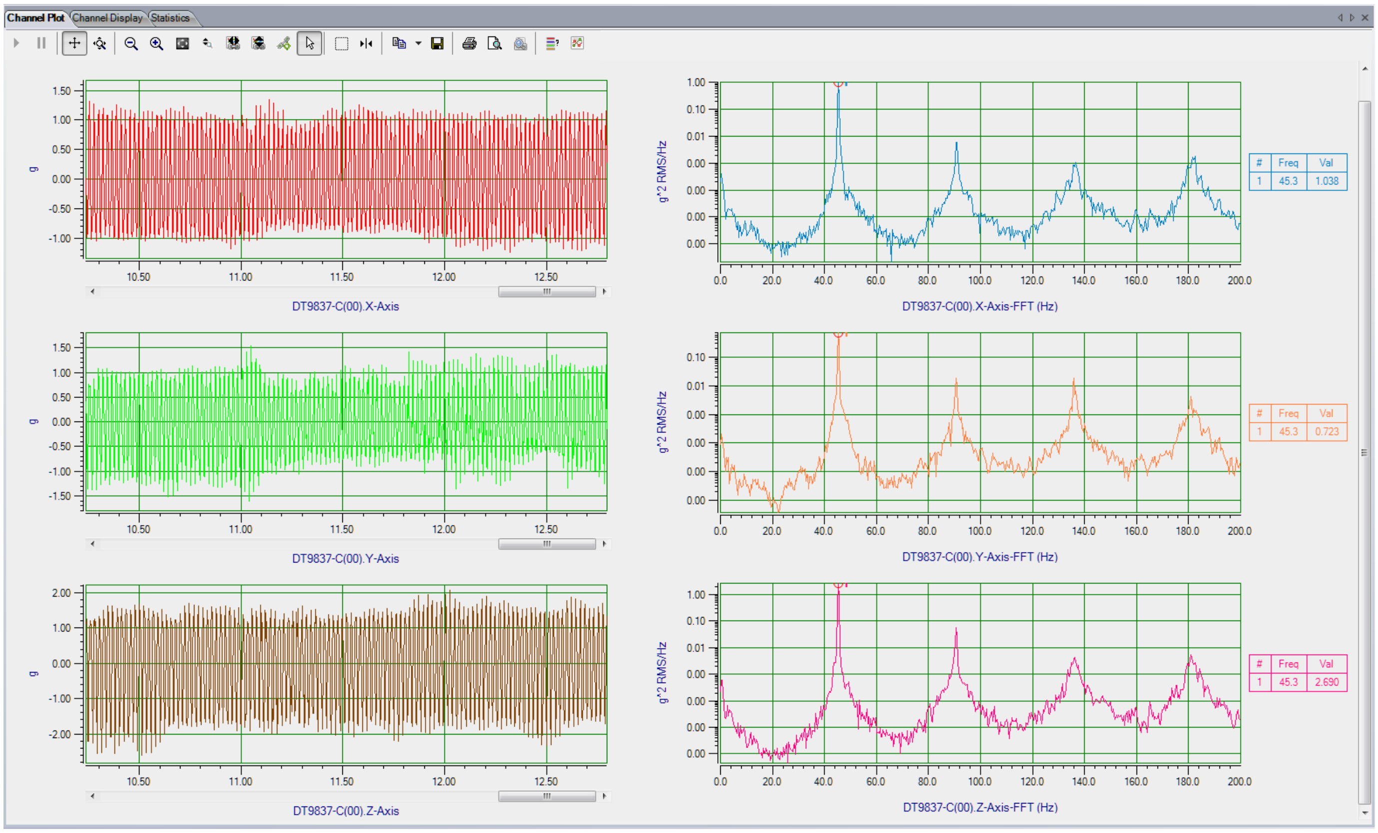
2. For this example, check the **Show FFT Peaks** checkbox to show a graphic similar to the following next to each FFT plot:

#	Freq	Val
1	45.3	1.038

3. For **Max peaks to detect**, select **1**.
4. For **Priority**, select **Absolute amplitude**.
5. For **Edge sensitivity**, slide the bar to the left for the most edge sensitivity (this will help ensure that the most peaks are detected).
6. If desired, hide the **Plot and Data Config** window by clicking the Auto-Hide pin ( ) in the top, right corner of the window.

Start the Acceleration Measurement

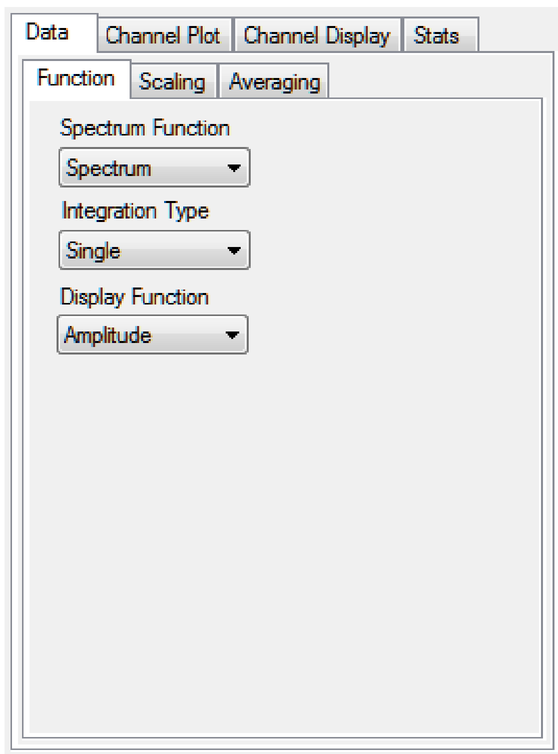
Once you have configured the channels and the parameters for the Channel Plot window, start acquisition and log data to disk by clicking the **Record** toolbar button ().
Results similar to the following are displayed in the Channel Plot window.



Perform Further Analysis of the Data

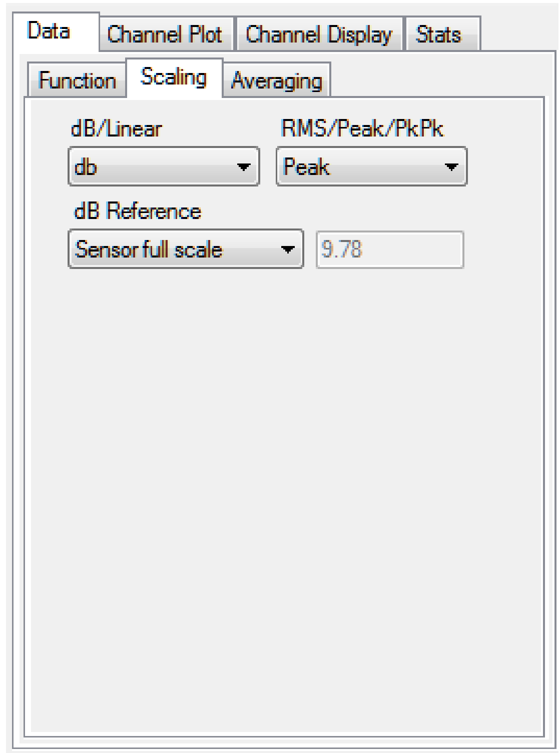
Once you have logged data to disk, you can modify the parameters of the Plot and Data Config window to perform further data analysis. An example follows:

1. If you hid the Plot and Data Config window previously, show the window by hovering over the **Plot and Data Config** tab with the mouse and then clicking the **Auto-Hide** pin () in the top, right corner of the window.
2. Ensure that the **Auto-reprocess on parameter change** checkbox is checked.
3. Click the **Stats** tab, and uncheck the **Show FFT Peaks** checkbox.
   Notice the change to the Channel Plot window takes effect immediately.
4. Under the **Channel Name** column, click the FFT channel to modify (for example, **DT9837-C(00).X-Axis-FFT**).
5. Click the **Data** tab, and then click the **Function** tab.



6. For this example, configure the settings of this tab as follows:
   a. For **Spectrum Function**, select **Spectrum.**
   b. For **Integration Type**, select **Single** for velocity data.
   c. For **Display Function**, leave **Amplitude** selected.

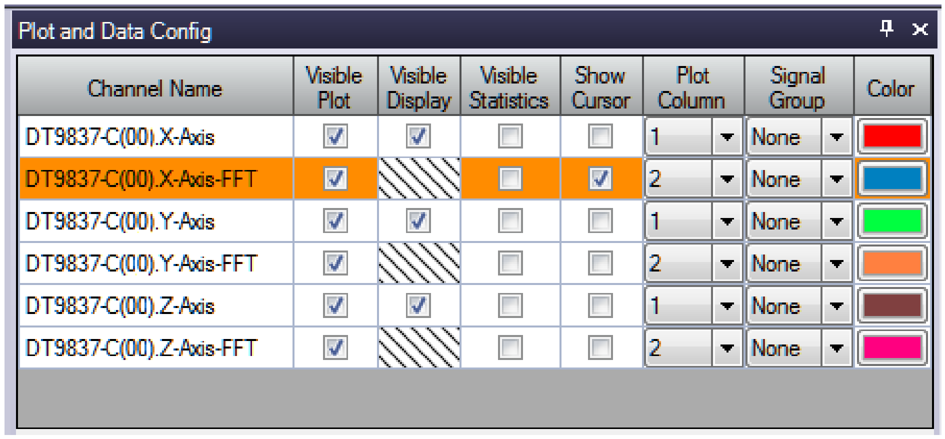
7. Click the **Scaling** tab.



8. For this example, configure the settings of this tab as follows:
   a. For **dB/Linear**, select **db**.
   b. For **RMS/Peak/PkPk**, select **Peak**.
   c. Leave the other settings unchanged.

9. Add a data-cursor to the plot, as follows:

   a. Under the **Show Cursor** column, click the checkbox next to the FFT channel for analog input channel 0 (in this example, **DT9837-C(00).X-Axis-FFT**).
   A data cursor appears on the FFT plot for that channel. (Note that you can remove the data cursor later, if desired, by unchecking this checkbox.)



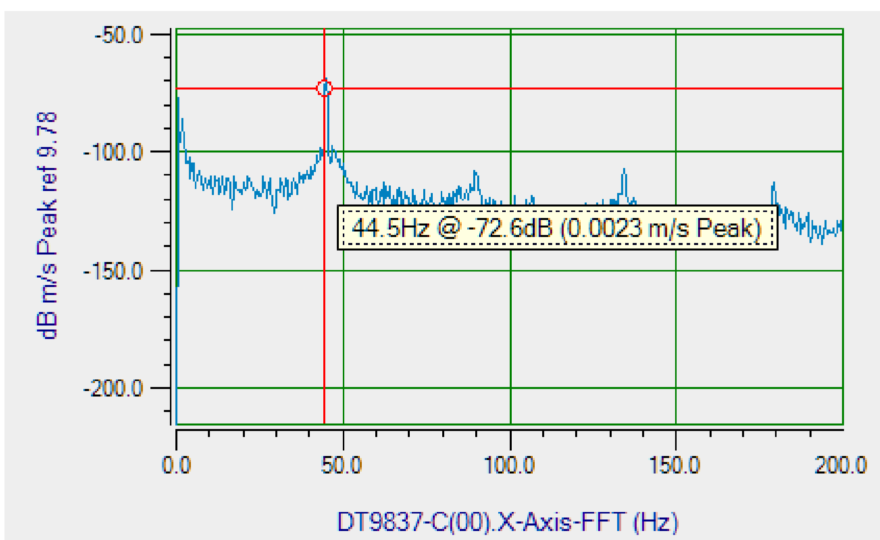
Plot and Data Config

Channel Name	Visible Plot	Visible Display	Visible Statistics	Show Cursor	Plot Column	Signal Group	Color
DT9837-C(00).X-Axis	☑	☑	☐	☐	1	None	
DT9837-C(00).X-Axis-FFT	☑		☐	☑	2	None	
DT9837-C(00).Y-Axis	☑	☑	☐	☐	1	None	
DT9837-C(00).Y-Axis-FFT	☑		☐	☐	2	None	
DT9837-C(00).Z-Axis	☑	☑	☐	☐	1	None	
DT9837-C(00).Z-Axis-FFT	☑		☐	☐	2	None	

   b. Click the data-cursor control ( ) at the top of the Channel Plot window.

   c. Right-click on the data cursor on the Channel Plot window, select **Color**, and change the color of the data cursor to red.

   d. Move the position of a data-cursor on the plot by holding down the left mouse button and dragging the mouse.
   The values of the selected points are shown; an example follows:



   Notice that, in this example, the velocity is 0.0023 m/s for the highlighted peak.

10. If desired, modify the other parameters of the Plot and Data Config window as you wish, and view the results.

Impact Testing Example

If you are using the Advanced FFT Analysis option of QuickDAQ, you can use the following steps to perform an impact test.

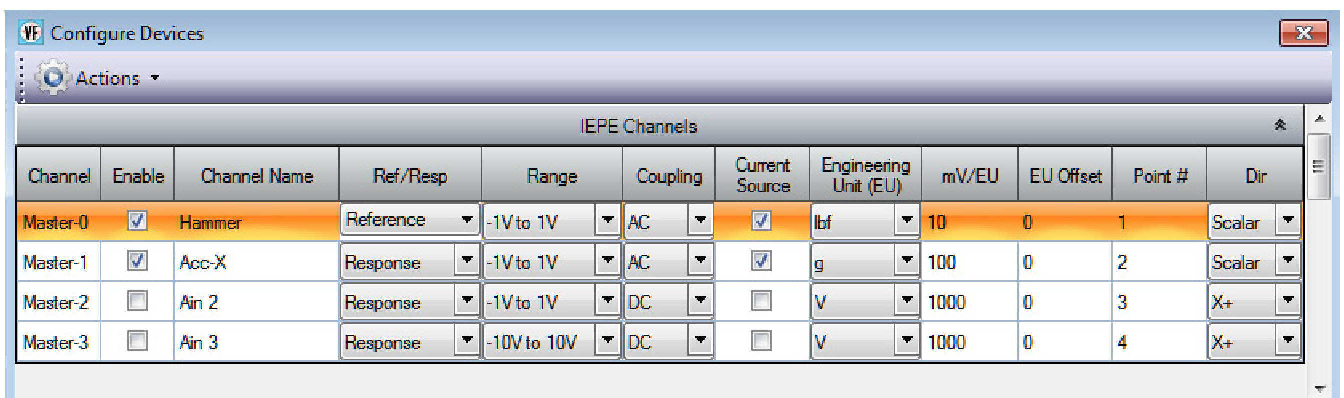
This example uses a hammer connected to analog input channel 0 of the DT9837B module and an accelerometer connected to analog input channel 1 of the DT9837B module to measure the impact of hitting a hammer against a table. The FFT Analyzer interface of QuickDAQ is used.

Configure the Channels

Configure the channels as follows:

1. Ensure that a hammer and an accelerometer are connected to your data acquisition device.
   In this example, connect the hammer to analog input channel 0 and connect the accelerometer to analog input channel 1 of a DT9837B module.
2. Configure each analog input channel by clicking the **Configuration** menu, and clicking **Input Channel Configuration**.

   Enable analog input channels 0 and 1 by clicking the checkbox under the **Enable** column:
   In this example, analog input channels 0 and 1 are enabled.



3. Under the **Channel Name** column, enter the following names for the analog input channels:
   – For analog input channel 0, enter **Hammer**.
   – For analog input channel 1, enter **Acc-X**.
4. Under the **Ref/Resp** column, select **Reference** for analog input channe1 0 (Hammer) and **Response** for analog input channel 1 (Acc-X).
5. Under the **Range** column, select the best effective input range (±10 V or ±1 V) for each signal. For example, if your hammer and accelerometer measure 100 mV/g and your signal of interest is less than 100 g, choose the ±1 V range; similarly, if your signal of interest is greater than 100 g, choose the ±10 V range.
   In this example, ***±1 V*** *is used.*

6. Under the **Coupling** column, select the coupling type (AC or DC) for your sensor. For most IEPE accelerometers, you will want to choose **AC** coupling to remove any DC offset from your measurement. Choose **DC** coupling only if the signal source has no offset voltage or if the DC content of the acquired signal is important.
   *In this example, **AC** is used.*

7. Under the **Current Source** column, select whether to enable or disable use of the 4 mA current source on the data acquisition device. Since most IEPE accelerometers require a current source of 4 mA and a compliance voltage of at least 18 V to drive their internal circuitry, in most cases, you will want to enable use of the current source for each analog input channel.
   *In this example, the **Current Source** checkbox is checked (enabled) for both analog input channels.*

Note: If you enable the use of the internal 4 mA excitation current source, it is recommended that you choose AC coupling.

---

8. Under the **Engineering Units** column, select the engineering units for the hammer and the accelerometer.
   *In this example, **lbf** is used for the hammer and **g** is used for the accelerometer.*

9. Using the calibration certificate that was supplied with your hammer and accelerometer, enter the calibrated sensitivity for the axis that is connected to the selected analog input channel in the **mV/EU** field. In this example, the following values are used:
   - Analog input channel 0 (Hammer) = 10 mV/lbf
   - Analog input channel 1 (Accelerometer) = 100 mV/g

10. If an offset is specified for the hammer and/or accelerometer, enter the value in the **EU Offset** field.
    In this example, no offset (0) is specified.

11. If desired, enter a test point value for each channel. In this example, the following values are used:
    - Analog input channel 0 (Hammer) = 1
    - Analog input channel 1 (Acc-X) = 2

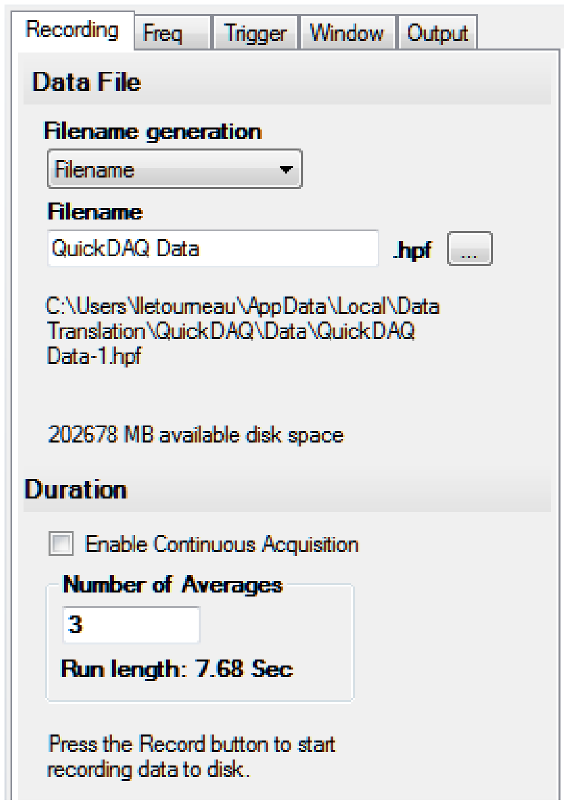
12. If desired, enter a sensor direction for each channel. In this example, the following values are used:
    - Analog input channel 0 (Hammer) = Scalar
    - Analog input channel 1 (Acc-X) = Scalar

13. Click **Close** to close the Channel Configuration dialog box.

Configure the Recording Settings

For this example, configure the recording settings as follows:

1. Click the **Recording** tab of the Acquisition Config window.

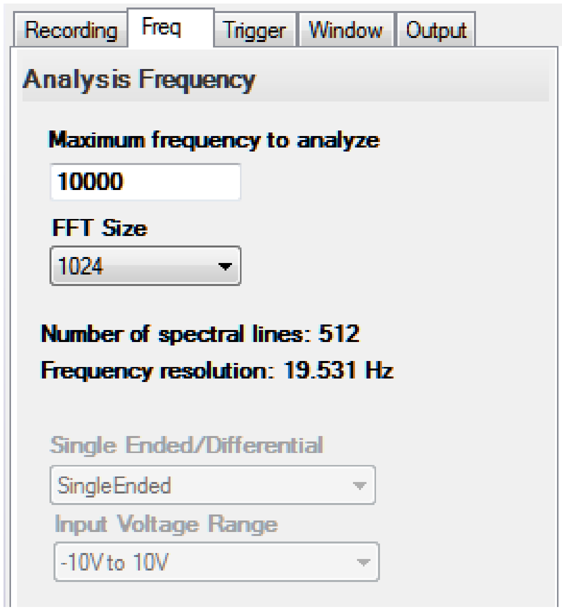


2. For **Filename generation**, use the **Filename** option.
3. For **Filename**, use the default name for the data file.
4. Leave the **Enable Continuous Acquisition** checkbox unchecked.
5. For **Number of Averages**, enter **3** as the number of records to acquire, process, and average before acquisition stops.
   The number of seconds for the entire run and the amount of available disk space are shown.

Configure the Frequency Settings

For this example, configure the frequency settings as follows:

1. Click the **Freq** tab of the Acquisition Config window.



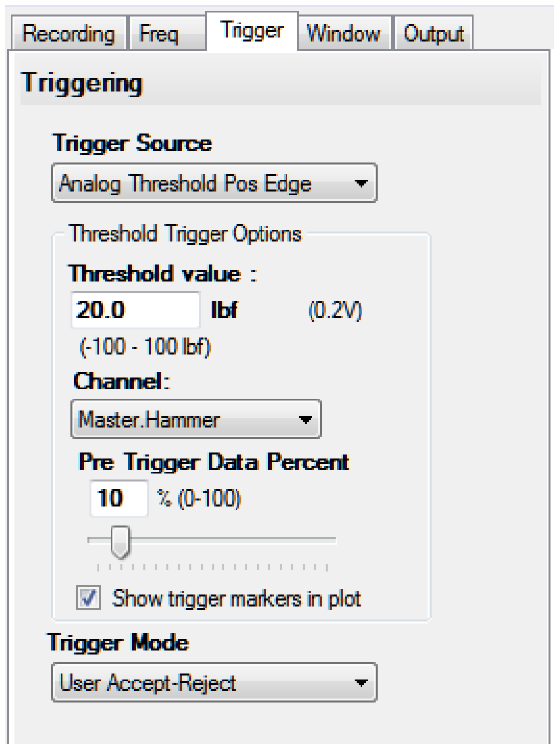
2. In the **Maximum frequency to analyze** text box, enter **10000** as the bandwidth or span, in Hz, of the input signals that you want to analyze
Internally, the sampling rate of the data acquisition device is set to twice the bandwidth, or 20000 Hz in this example.

3. In the **FFT Size** text box, select **1024** from the drop-down combo box as the number of data points to use for the FFT calculation.
The number of spectral lines and the frequency resolution are displayed.

Configure the Trigger Settings

For this example, configure the trigger settings as follows:

1. Click the **Trigger** tab of the Acquisition Config window.

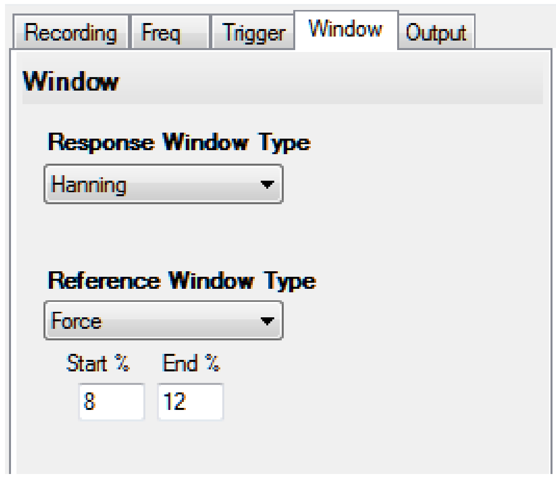


2. For the **Trigger Source**, select **Analog Threshold Pos Edge.**
3. For the **Threshold value**, enter **20.00 lbf**.
   This ensures that the measurement starts when the hammer hits the table and at least 20 lbf is detected.
4. Select Analog input channel 0 **(Hammer)** as the channel for the threshold trigger.
5. For the **Pre Trigger Data Percent**, enter 10%.
6. Select the **Show trigger markers in plot** checkbox.
7. For the **Trigger Mode**, select **User Accept-Reject**.

Configure the Windowing Function

For this example, configure the windowing function as follows:

1. Click the **Window** tab of the Acquisition and Config window.



2. For the **Response Window Type**, select **Hanning**.
   This is a global setting that is applied to the FFTs for all enabled analog input channels that are configured as response channels.
3. For the **Reference Window Type**, select **Force**, and enter **8** for **Start%** and **12** for **End%**.
4. If desired, hide the Acquisition Config window by clicking the Auto-Hide pin ( ) in the top, right corner of the window.

Add an FFT Channel to the Plot and Data Config Window

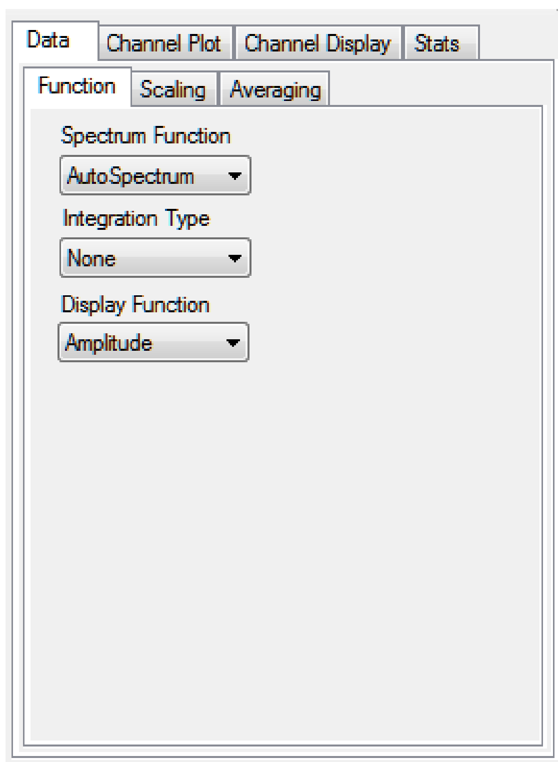
The following steps show how to use the FFT Analyzer interface of QuickDAQ to add an FFT channel to the Plot and Data Config window:

1. Click the **Add single channel FFT function** ( ) button to add an FFT channel to the Plot and Data Config window.
2. Select **Acc-X** as the FFT channel to add.
   The Acc-X-FFT channel is added to the Plot and Data Config window.

Configure the FFT Channel

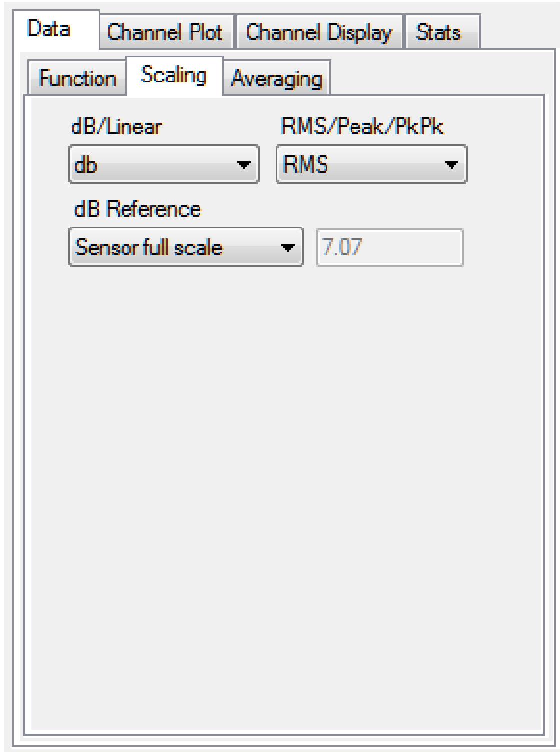
The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used for the FFT channel:

1. Select the **Acc-X-FFT** channel in the channel list.
2. Click the **Data** tab, and then click the **Function** tab.



3. For this example, configure the settings of this tab as follows:
   a. For **Spectrum Function**, select **AutoSpectrum.**
   b. For **IntegrationType**, select **None**.
   c. For **Display Function**, select **Amplitude**.

4. Click the **Scaling** tab, and configure the settings of this tab as follows:

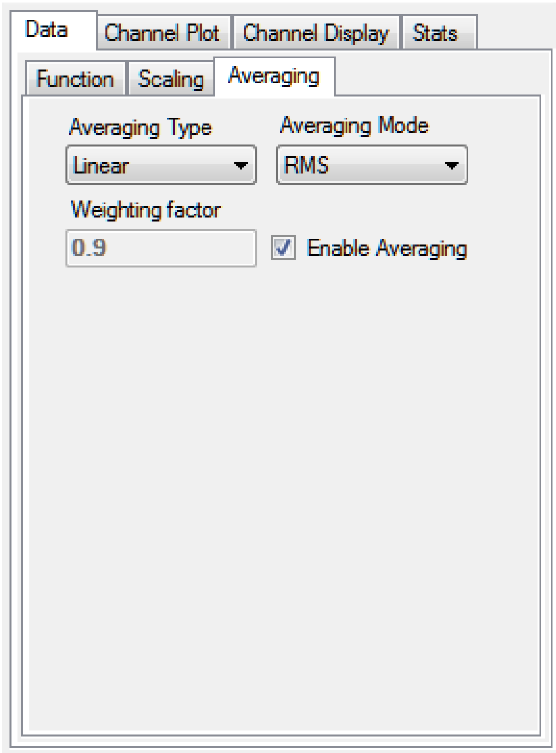


a. For **dB/Linear,** select **dB**.

b. For **RMS/Peak/PkPk**, select **RMS**.

c. For **dB Reference**, select **Sensor full scale**.

5. Click the **Averaging** tab, and configure the settings of this tab as follows:



a. Select the **Enable Averaging** checkbox.

b. For **Averaging Type**, select **Linear**.

c. For **Averaging Mode**, select **RMS**.

Add an FRF Channel to the Plot and Data Config Window

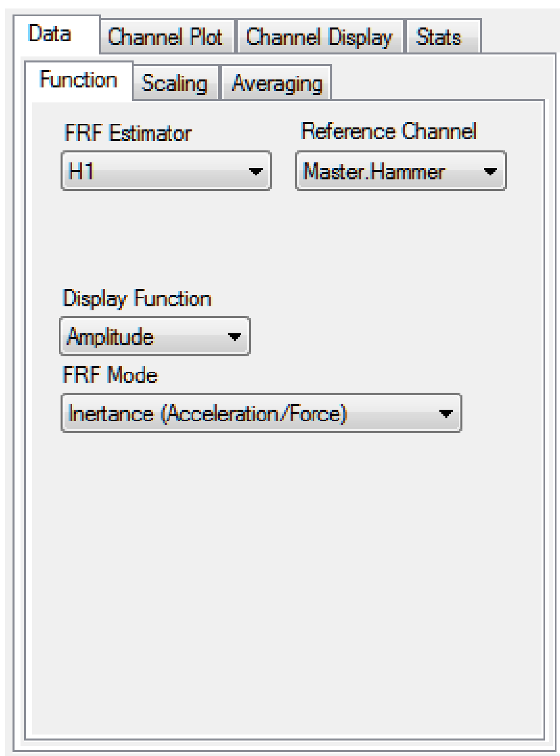
The following steps show how to use the FFT Analyzer interface of QuickDAQ to add an FRF channel to the Plot and Data Config window to calculate the frequency response of analog input channel 0 (Hammer) and analog input channel 1 (Acc-X).

1. Click the **Add 2 channel FFT function** () button, and select **FRF** to add an FRF channel to the Plot and Data Config window.
2. Select **Acc-X** as the response channel to add.
   The Acc-X-FRF channel is added to the Plot and Data Config window.

Configure the FRF Channel

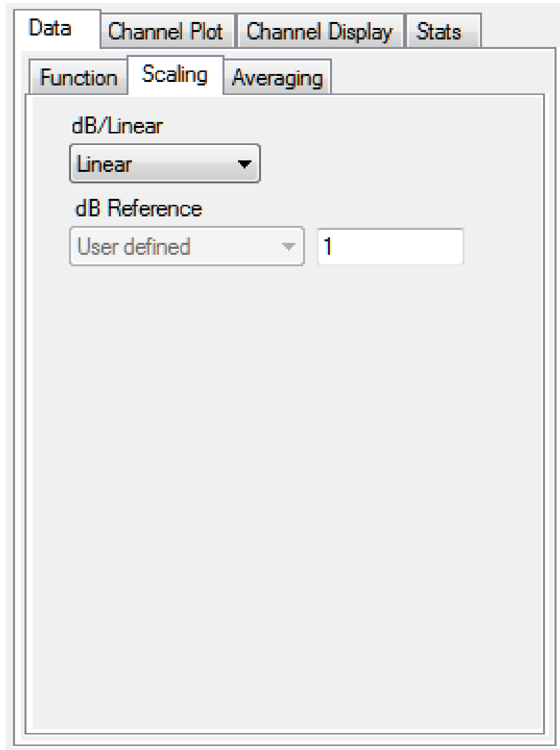
The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used to generate the FRF channels for the analog input channels:

1. Select the **Acc-X-FRF** channel in the channel list.
2. Click the **Data** tab, and then click the **Function** tab.

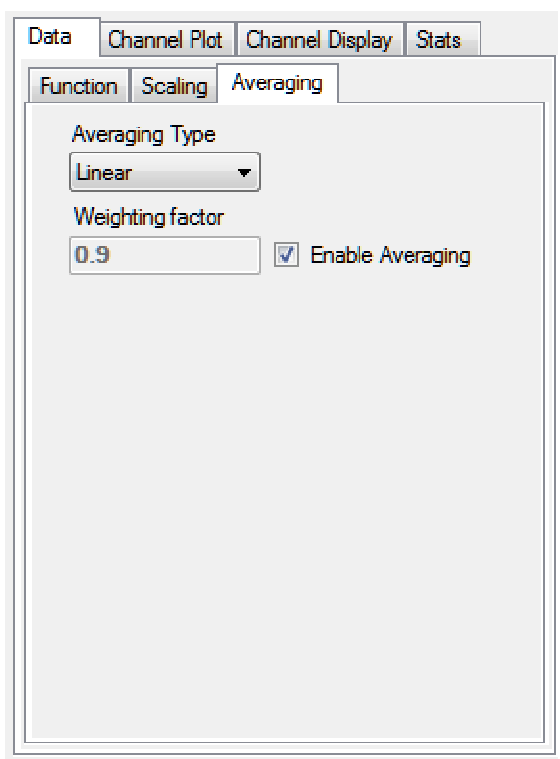


3. For the **Acc-X-FRF** channel, configure the settings of this tab as follows:
   a. For **FRF Estimator**, select **H1** (reduces noise on the response channel).
   b. For **Reference Channel**, select **Hammer** (analog input channel 0).
   c. For **Display Function**, select **Amplitude**.
   d. For **FRF Mode**, select **Inertance (Acceleration/Force)**.

4. Click the **Scaling** tab, and select **Linear** for **dB/Linear**:



5. Click the **Averaging** tab, select the **Enable Averaging** checkbox, and select **Linear** for **Averaging Type**:



Add a Coherence Channel to the Plot and Data Config Window

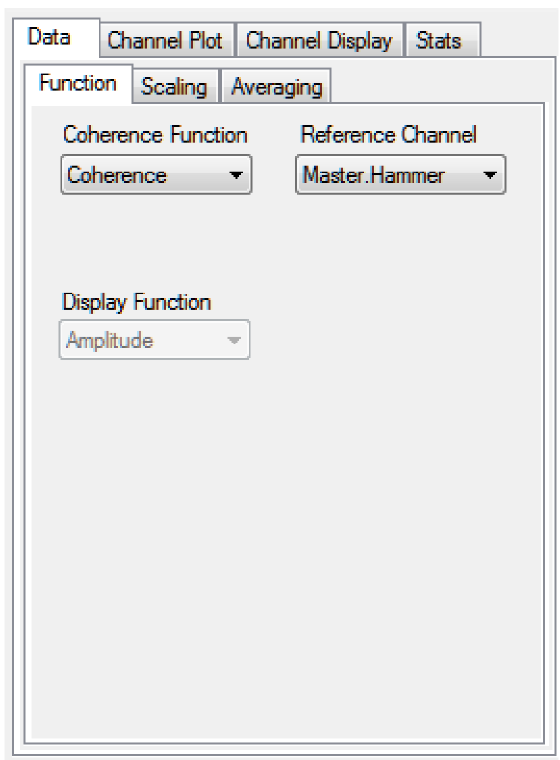
The following steps show how to use the FFT Analyzer interface of QuickDAQ to add a Coherence channel to the Plot and Data Config window:

1. Click the **Add 2 channel FFT function** () button, and select **Coherence** to add a Coherence channel to the Plot and Data Config window.
2. Select **Acc-X** as the response channel to add.
   The Acc-X-Coherence channel is added to the Plot and Data Config window.

Configure the Coherence Channel

The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used to generate the Coherence channel for the analog input channels:

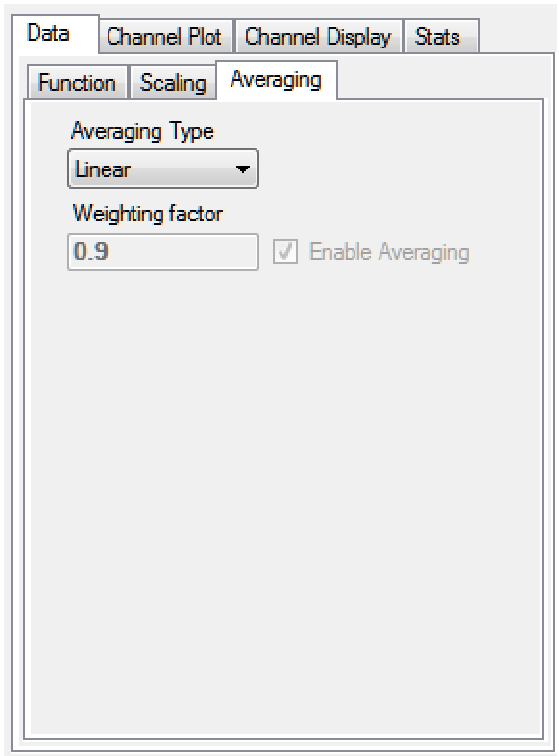
1. Select the **Acc-X-Coherence** channel in the channel list.
2. Click the **Data** tab, and then click the **Function** tab.



3. For this example, configure the settings of this tab as follows:
   a. For **Coherence Function**, select **Coherence**.
   b. For **Reference Channel**, select **Hammer** (analog input channel 0).

Note: Linear scaling is selected automatically for Coherence channels under the **Scaling** tab.

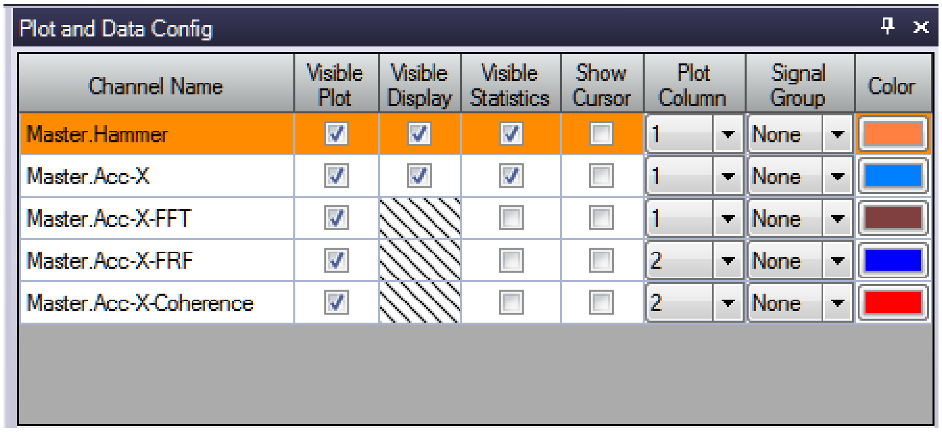
4. Click the **Averaging** tab, and select **Linear** for **Averaging Type**:



Configure the Appearance of the Channel Plot Window

Perform the following steps to configure the appearance of the Channel Plot window:

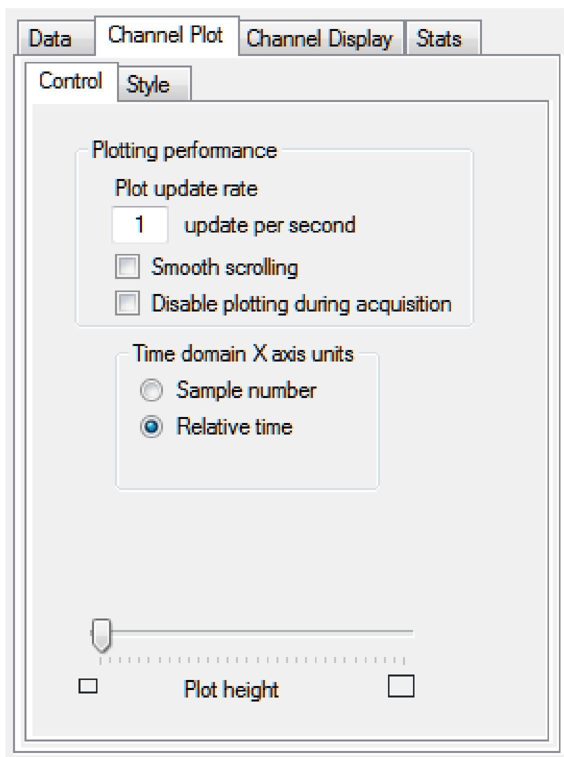
1. Select the **Visible Plot** checkbox for the Hammer, Acc-X, Acc-X-Coherence, Acc-X-FRF, and Acc-X-FFT channels.



2. Under **Plot Column**, select plot column **1** for the Hammer, Acc-X, and Acc-X-FFT channels, and select plot column **2** for the Acc-X-Coherence and Acc-X-FRF channels.

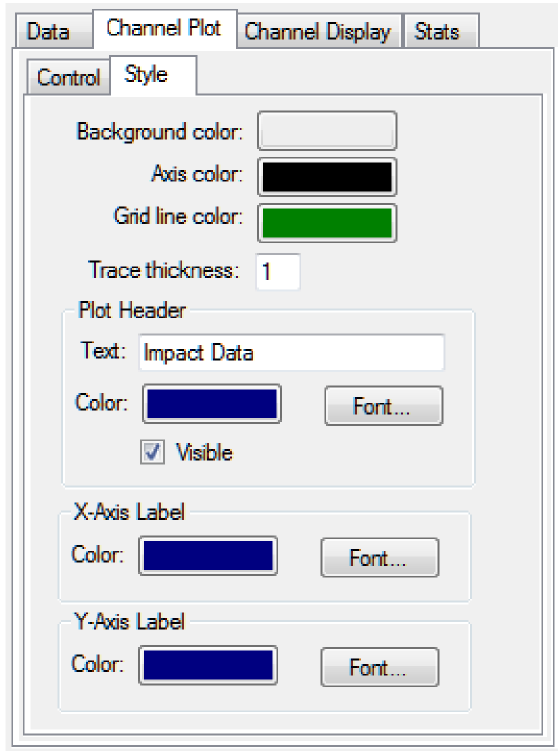
3. Assign a unique color to each trace for the visible channels.

4. Click the **Channel Plot - Control** tab.



5. Slide the **Plot height** bar to the left until you can see all 5 plots on the Channel Plot window at once.

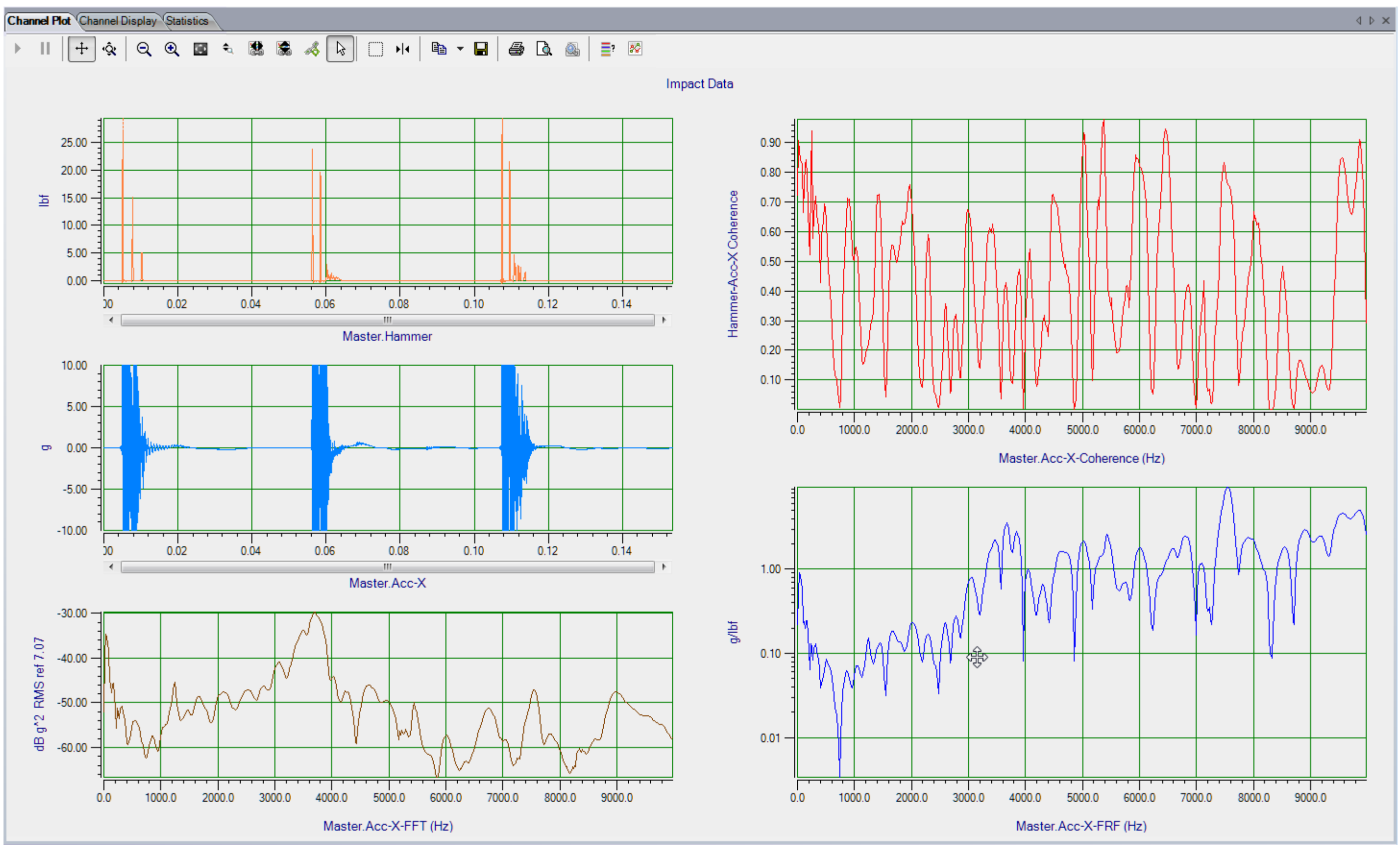
6. Click the **Channel Plot - Style** tab, and change the **Text** field to **Impact Data**.


Data
Channel Plot
Channel Display
Stats
Control
Style
Background color:
Axis color:
Grid line color:
Trace thickness:
1
Plot Header
Text:
Impact Data
Color:
Font...
Visible
X-Axis Label
Color:
Font...
Y-Axis Label
Color:
Font...

Start the Impact Test

Once you have configured the channels and the parameters for the Channel Plot window, do the following to conduct the impact test:

1. Click the **Record** toolbar button () or press the **F5** key, and strike the hammer on the table.
2. Click **Accept** if the data appears correct or **Reject** if the data reflects a bad hammer strike.
3. Strike the hammer on the table and repeat steps 2 and 3 until three acquisitions have been accepted.
   Results similar to the following are displayed in the Channel Plot window.



In this example, you can see by the trigger marker for the Hammer trace, which is in the same color as the Hammer trace color, that all the pre-trigger data (10% of the data) was acquired. The result of the hammer strike is measured by the Accelerometer (Acc-X). In addition, you can see the FFT, Coherence, and FRF of the accelerometer data.

Add Another FRF Channel to the Plot and Data Config Window

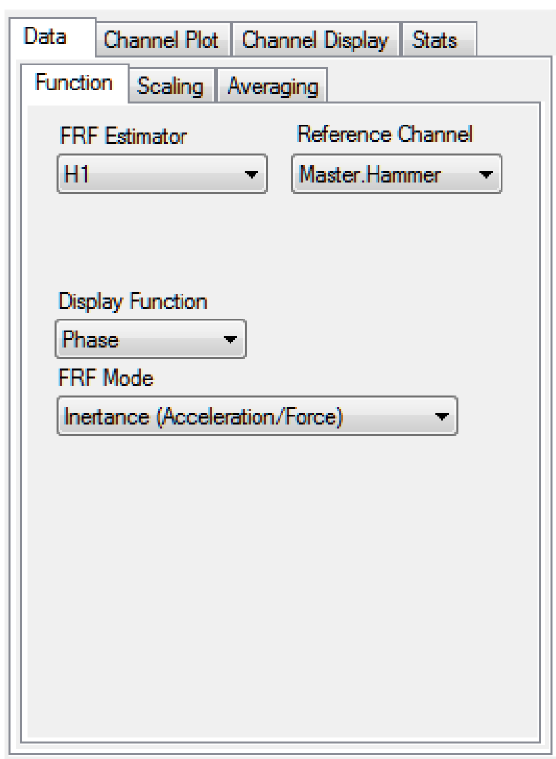
The following steps show how to use the FFT Analyzer interface of QuickDAQ to add another FRF channel to the Plot and Data Config window for calculating the frequency response and phase information.

1. Click the **Add 2 channel FFT function** ( ) button, and select **FRF** to add an FRF channel to the Plot and Data Config window.
2. Select **Acc-X** as the response channel to add.
   The Acc-X-FRF1 channel is added to the Plot and Data Config window.
3. Select this channel and enter the following text to rename it: **Acc-X-FRF-Phase**.

Configure the FRF Channel for Phase

The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used to generate the FRF channel:

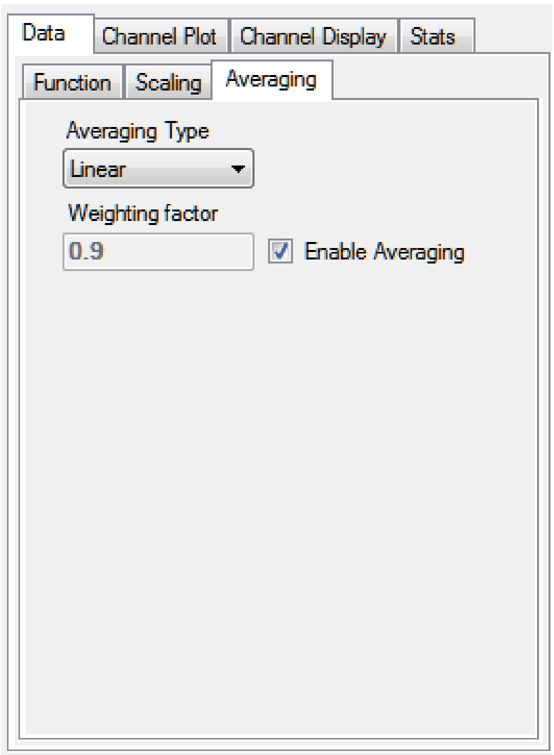
1. Select the **Acc-X-FRF-Phase** channel in the channel list.
2. Click the **Data** tab, and then click the **Function** tab.



3. For the **Acc-X-FRF-Phase** channel, configure the settings of this tab as follows:
   a. For **FRF Estimator**, select **H1** (reduces noise on the response channel).
   b. For **Reference Channel**, select **Hammer** (analog input channel 0).
   c. For **Display Function**, select **Phase**.
   d. For **FRF Mode**, select **Inertance (Acceleration/Force)**.

Note: Linear scaling is automatically selected for the **Scaling** tab.

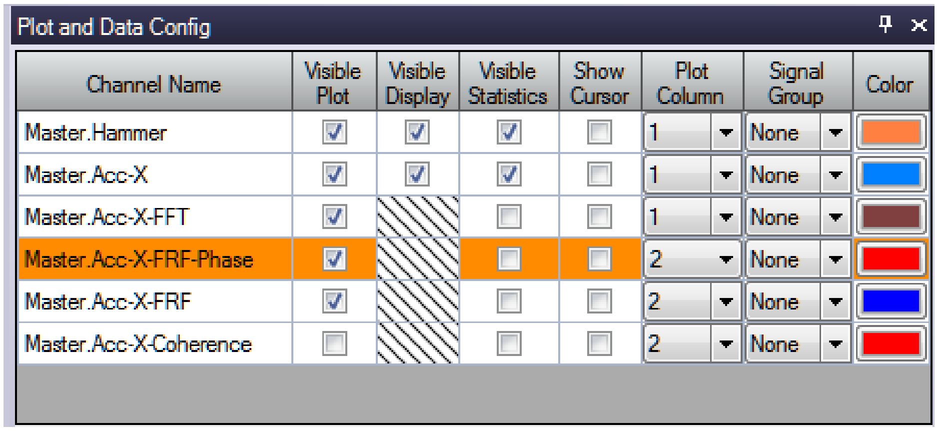
4. Click the **Averaging** tab, select the **Enable Averaging** checkbox, and select **Linear** for **Averaging Type**:



Configure the Appearance of the Channel Plot Window

Perform the following steps to configure the appearance of the Channel Plot window:

1. Select the **Visible Plot** checkbox for the Hammer, Acc-X, Acc-X-FRF-Phase, Acc-X-FRF, and Acc-X-FFT channels.


Plot and Data Config

Channel Name	Visible Plot	Visible Display	Visible Statistics	Show Cursor	Plot Column	Signal Group	Color
Master.Hammer	☑	☑	☑	☐	1	None	
Master.Acc-X	☑	☑	☑	☐	1	None	
Master.Acc-X-FFT	☑		☐	☐	1	None	
Master.Acc-X-FRF-Phase	☑		☐	☐	2	None	
Master.Acc-X-FRF	☑		☐	☐	2	None	
Master.Acc-X-Coherence	☐		☐	☐	2	None	

2. Under **Plot Column**, select plot column **1** for the Hammer, Acc-X, and Acc-X-FFT channels, and plot column **2** for the Acc-X-FRF-Phase and Acc-X-FRF channels.
3. Assign a unique color to each trace for the visible channels, if desired.

Restart the Impact Test

Once you have added and configured the Acc-X-Phase channel and configured the parameters for the Channel Plot window, do the following to restart the impact test:

1. Click the **Record** toolbar button (

) or press the **F5** key, and strike the hammer on the table.
2. Click **Accept** if the data appears correct or **Reject** if the data reflects a bad hammer strike.
3. Strike the hammer on the table and repeat steps 2 and 3 until three acquisitions have been accepted.
   Results similar to the following are displayed in the Channel Plot window.

In this example, you can see by the trigger marker for the Hammer trace, which is in the same color as the Hammer trace color, that all the pre-trigger data (10% of the data) was acquired. The result of the hammer strike is measured by the Accelerometer (Acc-X). In addition, you can see the FFT, FRF-Phase, and FRF (Amplitude) of the accelerometer data.

Strain Gage Example

If you are using the FFT Analysis or Advanced FFT Analysis option of QuickDAQ, you can use the following steps to perform a strain gage measurement.

This example uses a full-bridge strain gage (full-bridge bending configuration) connected to a DT9838 module to measure strain. In this example, TEDS is not supported for the strain gage. The FFT Analyzer interface is used.

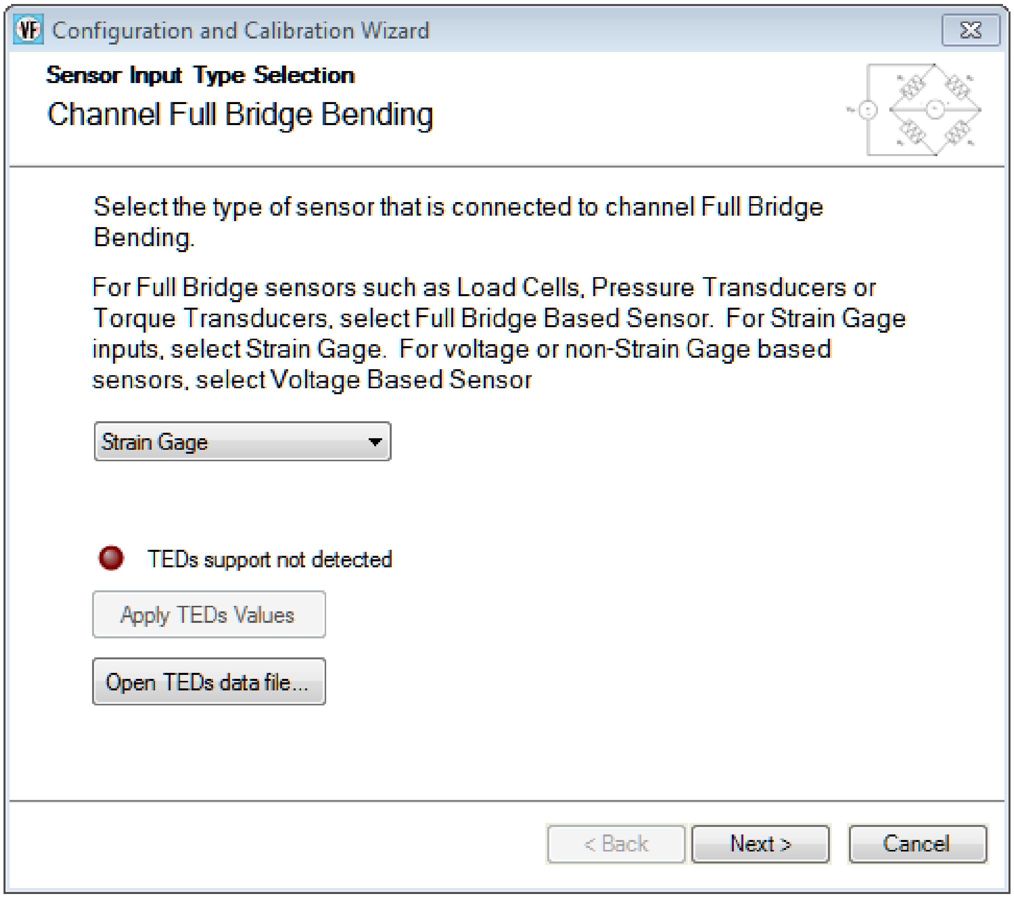
Configure the Channels

Configure the channels as follows:

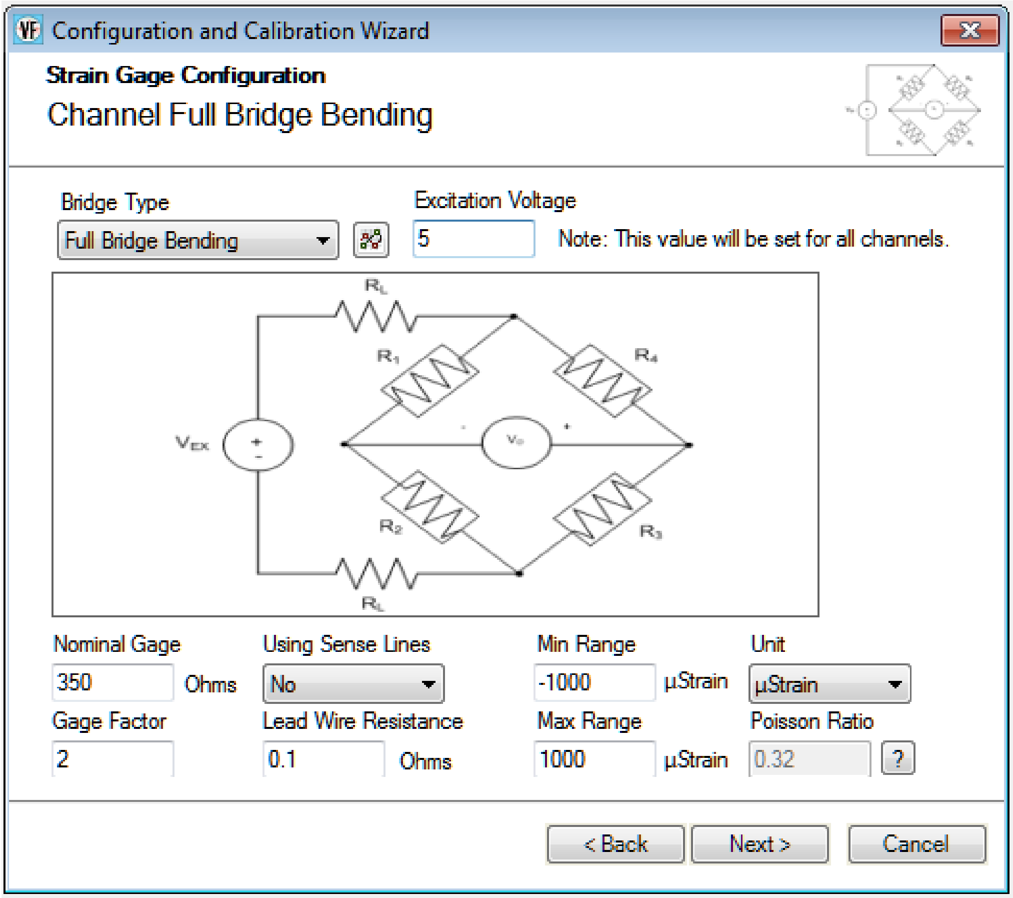
1. Ensure that the strain gage is connected to your data acquisition device.
   In this example, the strain gage is connected to analog input channel 0 of the DT9838. Refer to the wiring diagram for a full-bridge bending configuration in the DT9838 User's Manual for more information.
2. Configure each analog input channel by clicking the **Input Channel Configuration** toolbar button ( ) or by clicking the **Configuration** menu and clicking **Input Channel Configuration**.
3. Enable analog input channel 0 by clicking the **Enable** checkbox for analog input channel 0.

Channel	Enable	Channel Name	Ref/Resp	Enable Shunt Resistor	Configure and Calibrate	Point #
DT9838(00)-0	☑	Full Bridge Bending	Response	☐	Configure and Calibrate	1
DT9838(00)-1	☐	Ain 1	Response	☐	Configure and Calibrate	2
DT9838(00)-2	☐	Ain 2	Response	☐	Configure and Calibrate	3
DT9838(00)-3	☐	Ain 3	Response	☐	Configure and Calibrate	4

4. In the **Channel Name** column, enter **Full Bridge Bending** for the name of analog input channel 0.
5. In the **Ref/Resp** column, leave **Response** selected for this example.
6. In the **Enable Shunt Resistor** column, leave the checkbox unchecked; this disables the shunt resistor.
7. Click the **Configure and Calibrate** button.
   The following wizard appears:



8. Select **Strain Gage**, and click **Next**. (Note that TEDS is not supported for this sensor; therefore, you do not need to click the **Open TEDS data file...** button.)
   A screen similar to the following appears:



9. For the **Bridge Type** field, select the bridge configuration of your strain gage.
   In this example, the **Full Bridge Bending** *bridge configuration is used.*

10. For the **Excitation Voltage** field, enter the excitation voltage value for your strain gage.
    In this example, **5 V** *is used as the excitation voltage value.*

11. For the **Nominal Gage** field, enter the nominal gage resistance, in ohms, for your strain gage as determined by the manufacturer.
    In this example, **350** *is used as the nominal gage resistance.*

12. For the **Gage Factor** field, enter the gage factor for your strain gage as determined by the manufacturer.
    In this example, **2** *is used as the gage factor.*

13. For the **Using Sense Lines** field, select **Yes** if you are using remote sense lines in your wiring, or **No** if you are not using remote sensing in your wiring.
    In this example, remote sensing is not used.

14. For the **Lead Wire Resistance** field, enter the lead wire resistance, in ohms, for your strain gage.
    In this example, **0.1** *is used as the lead wire resistance.*

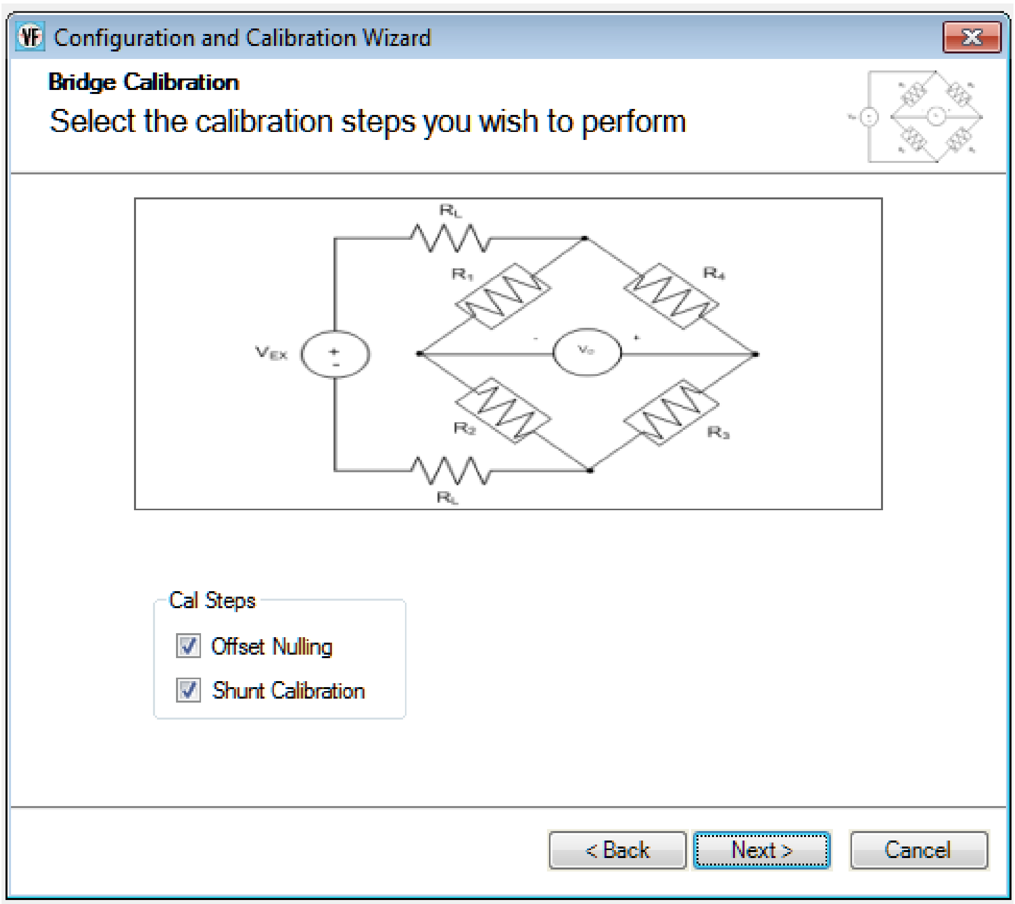
15. For the **Min Range** field, enter the minimum value of the range for your strain gage.
    In this example, **–1000** *is used as the minimum strain value.*

16. For the **Max Range** field, enter the maximum value of the range for your strain gage.
In this example, **1000** *is used as the maximum strain value.*

17. For the **Units** field, select the engineering units for the strain gage.
In this example, **µStrain** *is used.*

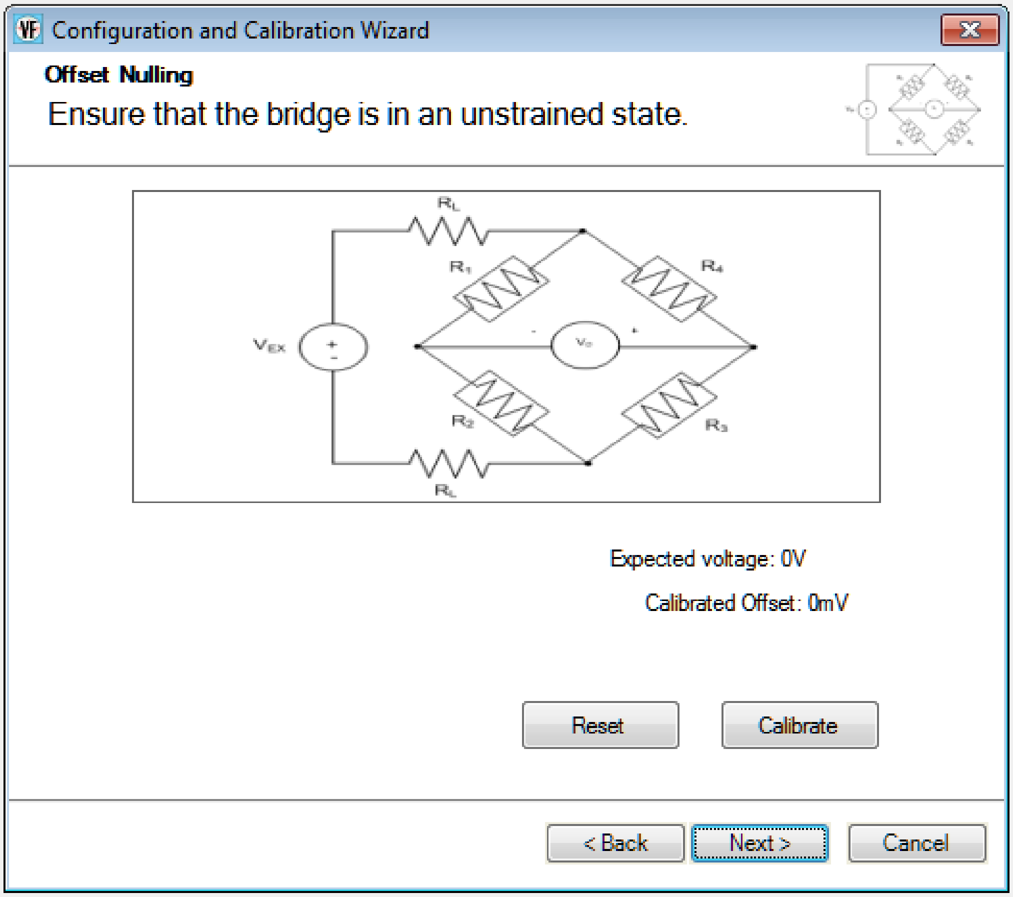
18. In the **Poisson Ratio** column, enter the Poisson ratio for your strain gage, if applicable.
In this example, the Poisson ratio is not used.

19. Click **Next**.
A screen similar to the following appears:

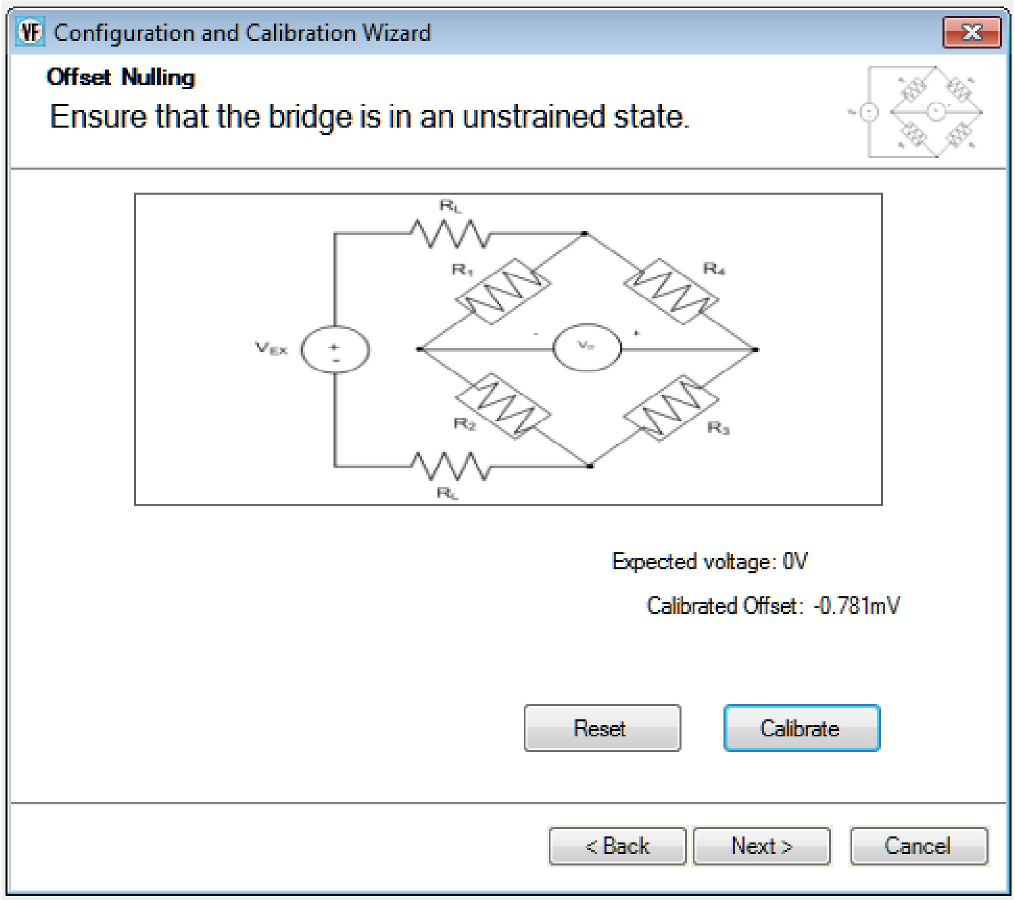


20. Select the calibration steps to perform.
In this example, **Offset Nulling** *and* **Shunt Calibration** *are selected.*

21. Click **Next**.
A screen similar to the following appears:

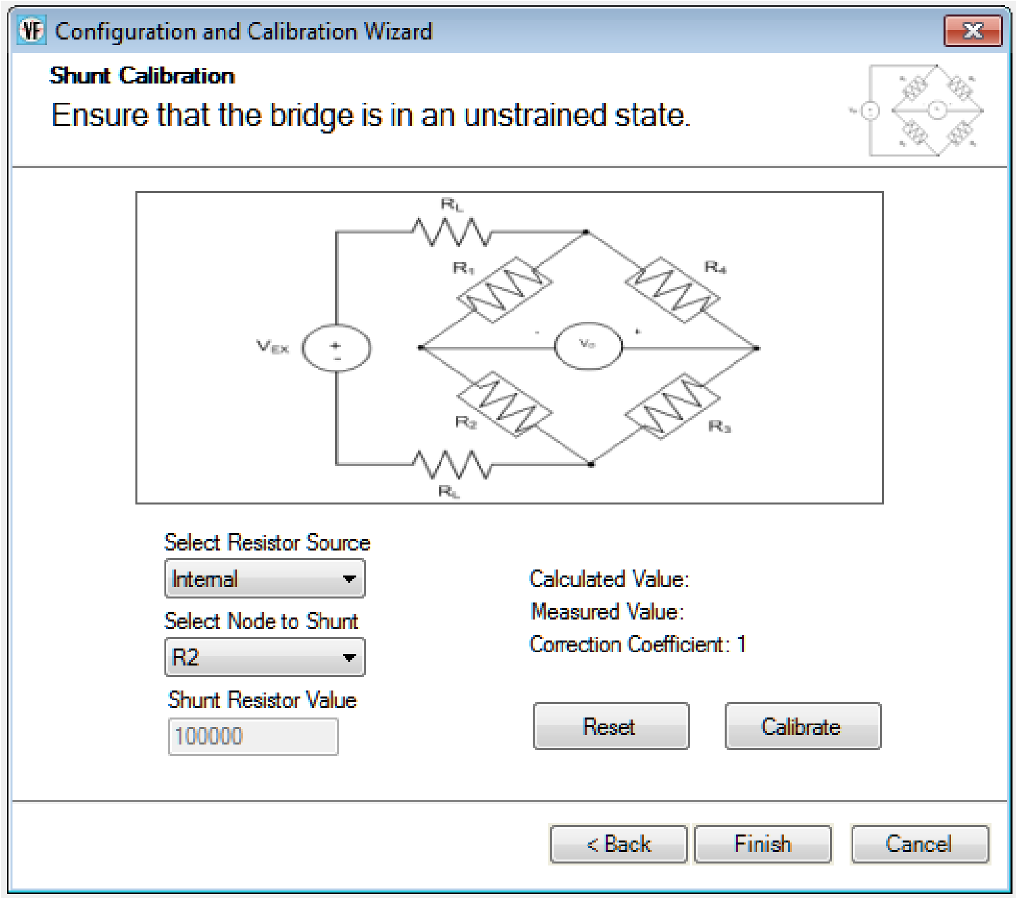


22. Ensure that the bride is in the unstrained state, and the click the **Calibrate** button to perform offset nulling procedure.
The expected voltage is shown along with the calibrated offset voltage.



23. Click **Next**.

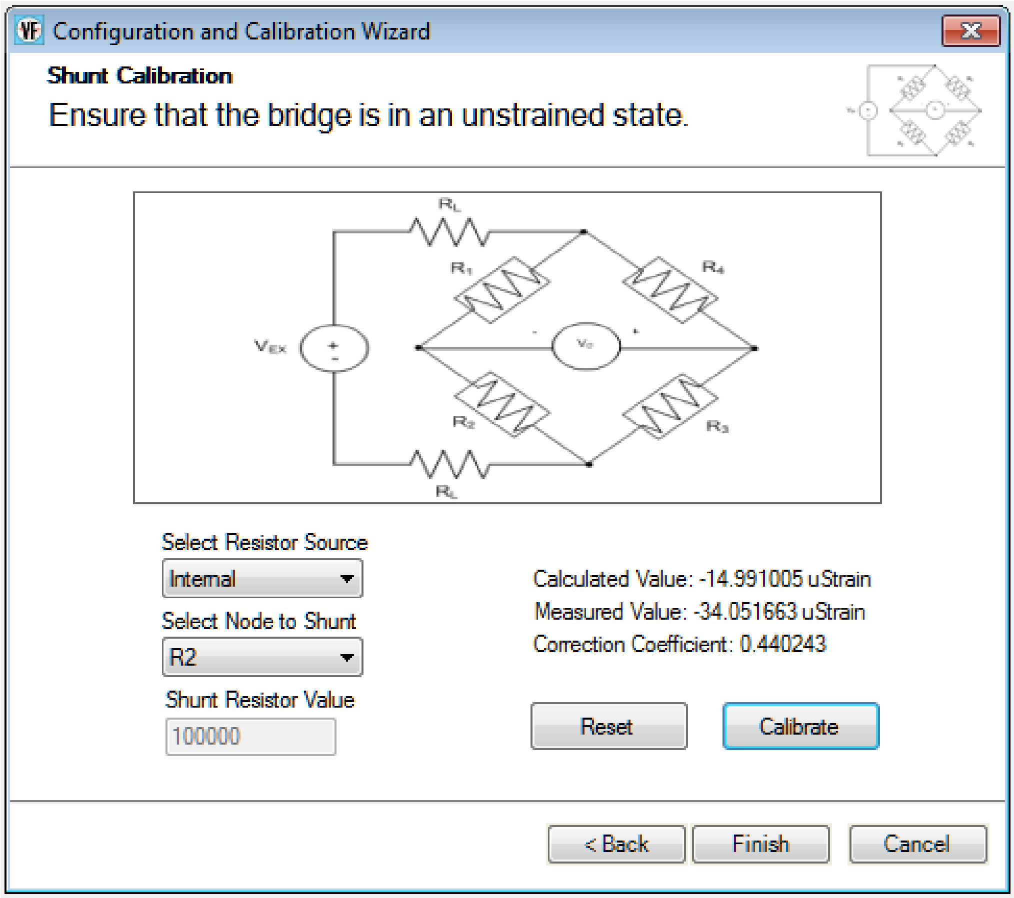
A screen similar to the following appears:



24. For the **Select Resistor Source** field, select **Internal** if you are using the internal shunt resistor provided on the DT9838 module to perform shunt calibration, or **External** if you are using your own external resistor to perform shunt calibration.
In this example, **Internal** *is used.*

25. For the **Select Node to Shunt** field, select the resistive node or element to which to apply the shunt resistor.
In this example, **R2** *is used.*

26. Click **Calibrate** to perform the shunt calibration procedure.
The calculated value is displayed along with the measured value and correction coefficient.



27. Click **Finish**.

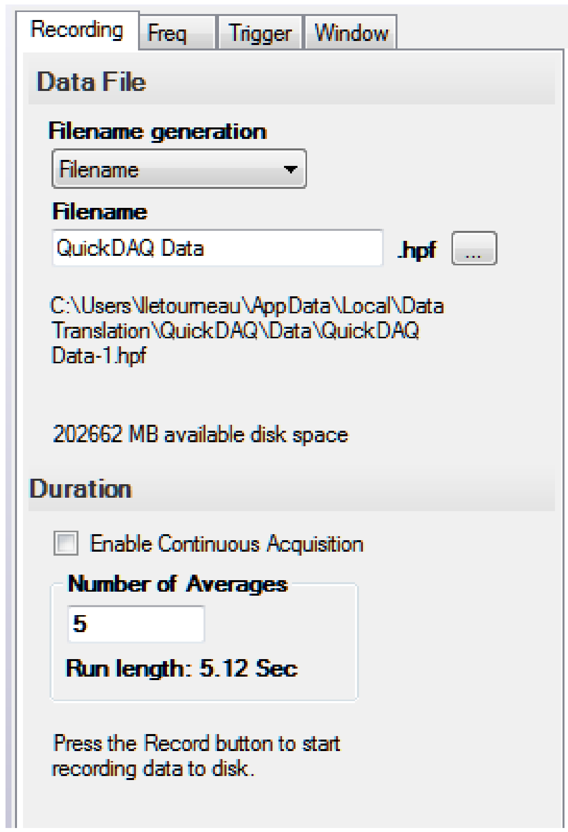
28. If desired, enter a test point number under the **Point #** column.
*In this example, **1** is used.*

29. Click **Close** to close the **Configure Devices** dialog box.

Configure the Recording Settings

For this example, configure the recording settings as follows:

1. Click the **Recording** tab of the Acquisition Config window.

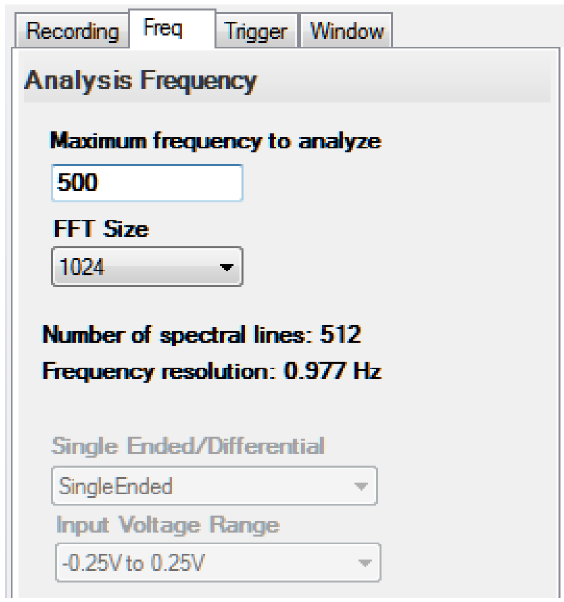


2. For **Filename generation**, use the **Filename** option.
3. For **Filename**, use the default name for the data file.
4. Leave the **Enable Continuous Acquisition** checkbox unchecked.
5. For **Number of Averages**, enter **5** as the number of records to acquired, process, and average before acquisition stops.
   The number of seconds for the total run and the amount of available disk space are shown.

Configure the Frequency Settings

For this example, configure the frequency settings as follows:

1. Click the **Freq** tab of the Acquisition Config window.



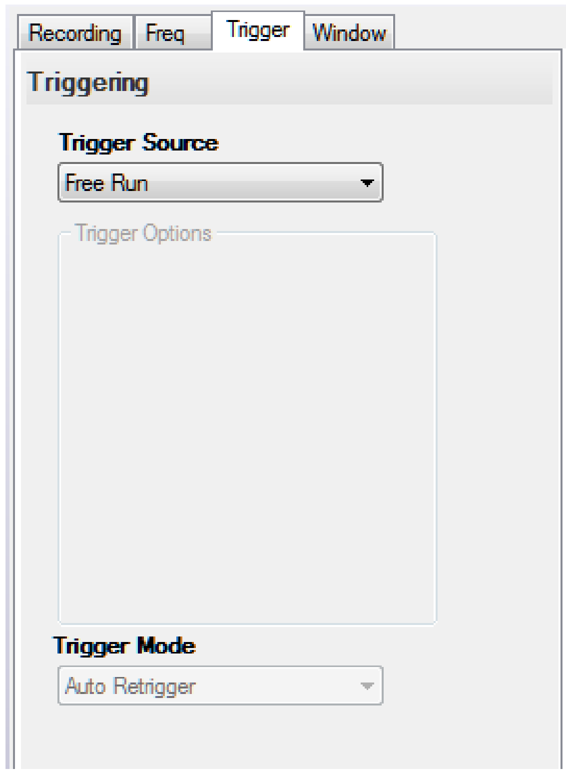
2. In the **Maximum frequency to analyze** text box, enter **500** as the bandwidth or span, in Hz, of the input signals that you want to analyze
   Internally, the sampling rate of the data acquisition device is set to twice the bandwidth, or 1000 Hz in this example.

3. In the **FFT Size** text box, select **1024** from the drop-down combo box as the number of data points to use for the FFT calculation.
   The number of spectral lines and the frequency resolution are displayed.

Configure the Trigger Settings

For this example, configure the trigger settings as follows:

1. Click the **Trigger** tab of the Acquisition Config window.

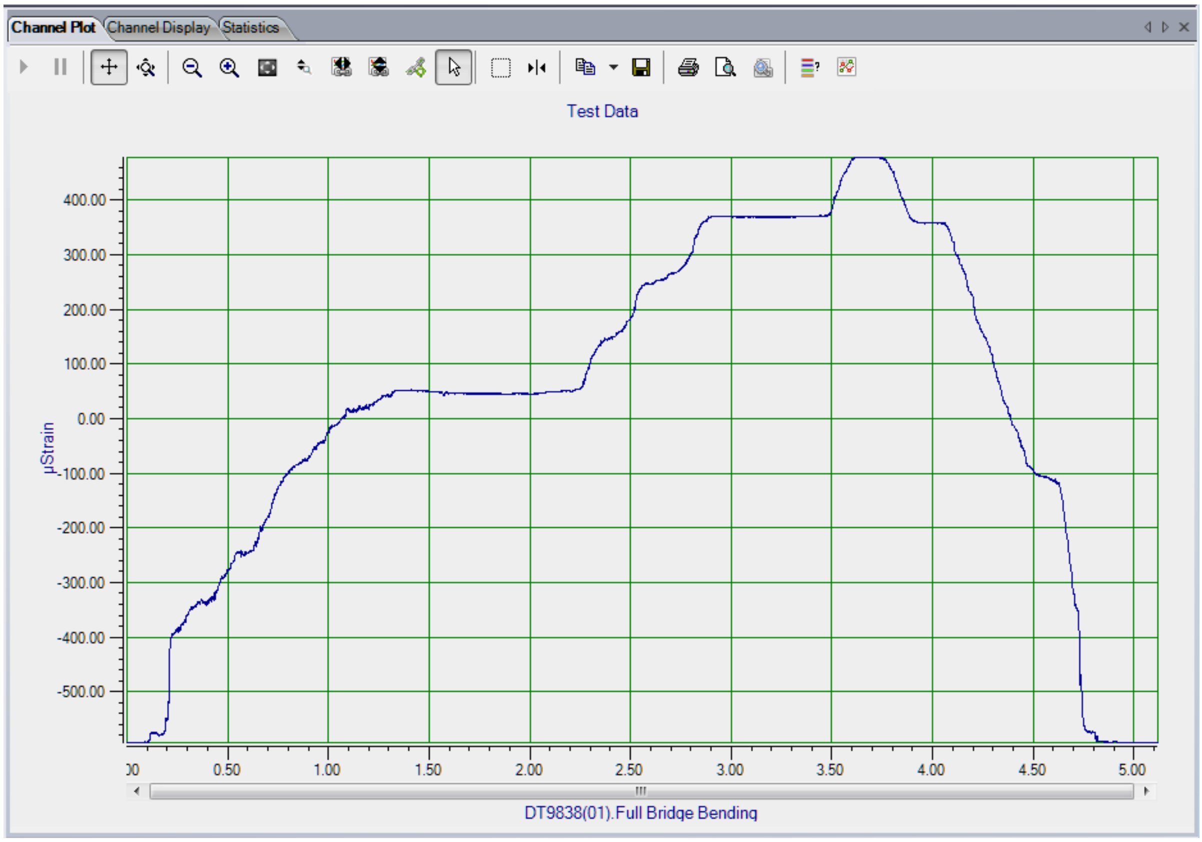


2. For the **Trigger Source**, select **Free Run** to ensure that the measurement starts as soon as the **Record** button is clicked.

Start the Strain Measurement

Once you have configured the channels, start acquisition and log data to disk by clicking the **Record** toolbar button ().

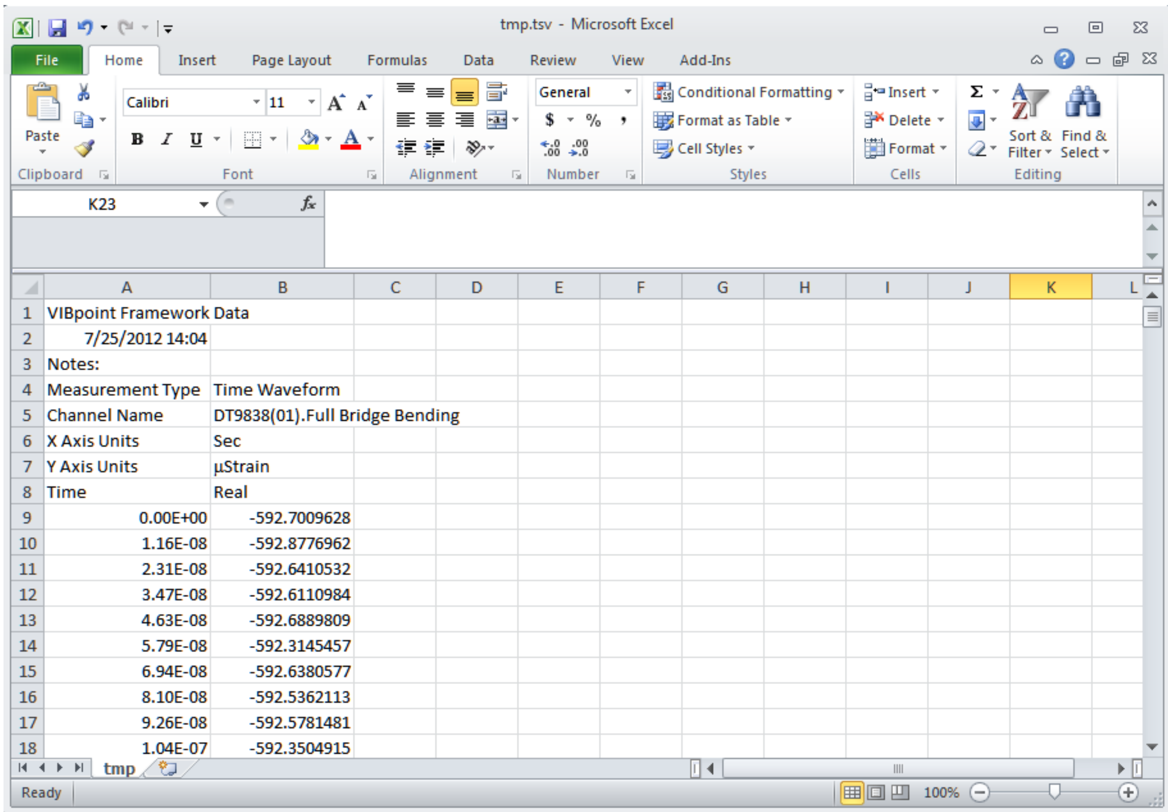
Results similar to the following are displayed in the Channel Plot window.



Export the Data to Excel

You can export the acquired data to Microsoft Excel for further analysis by performing the following steps:

1. Click the **Open Current Data in Excel** toolbar button .
2. Click **Time Series Data..**
   Microsoft Excel then opens and displays the data that was acquired, as shown in the following example.



Bridge-Based Sensor Example

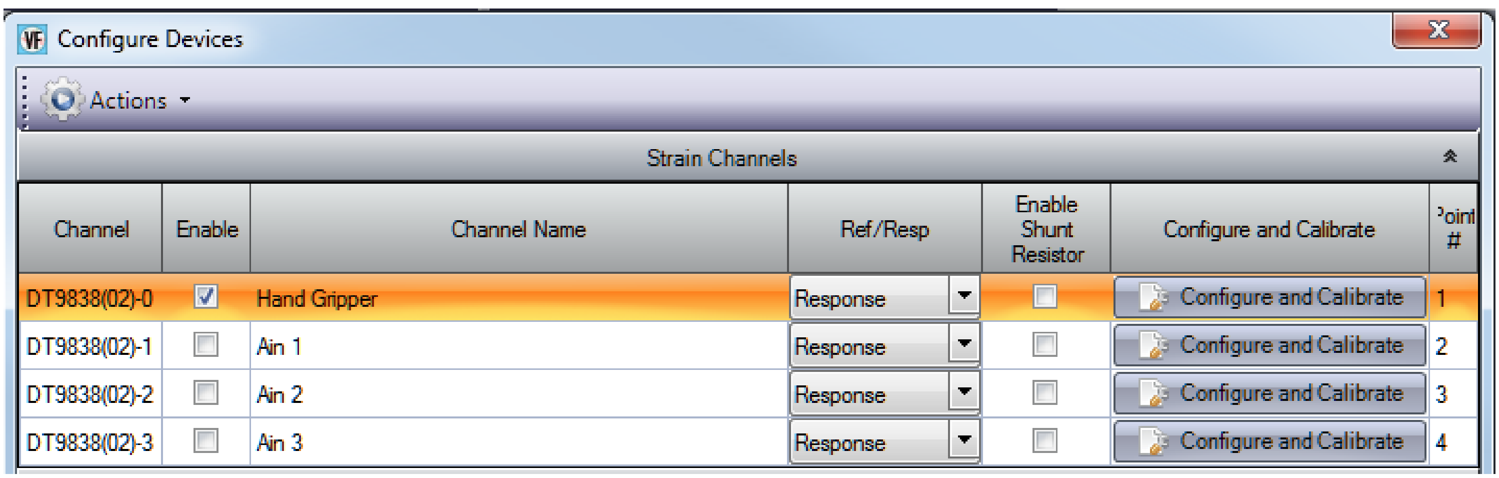
If you are using the FFT Analysis or Advanced FFT Analysis option of QuickDAQ, you can use the following steps to configure a bridge-based sensor to measure compression.

This example uses a load cell (a hand gripper) connected to a DT9838 module to measure compression, in pounds. In this example, the load cell supports hardware TEDS. The FFT Analyzer interface is used.

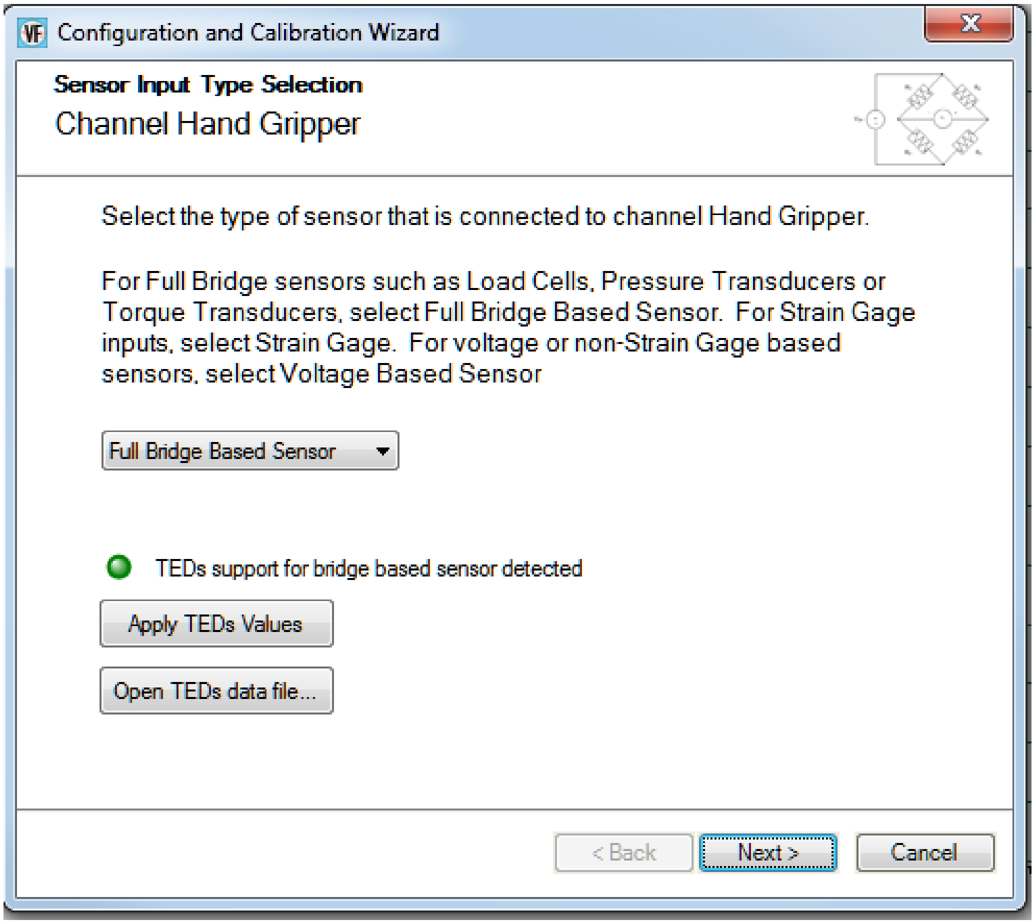
Configure the Channels

Configure the channels as follows:

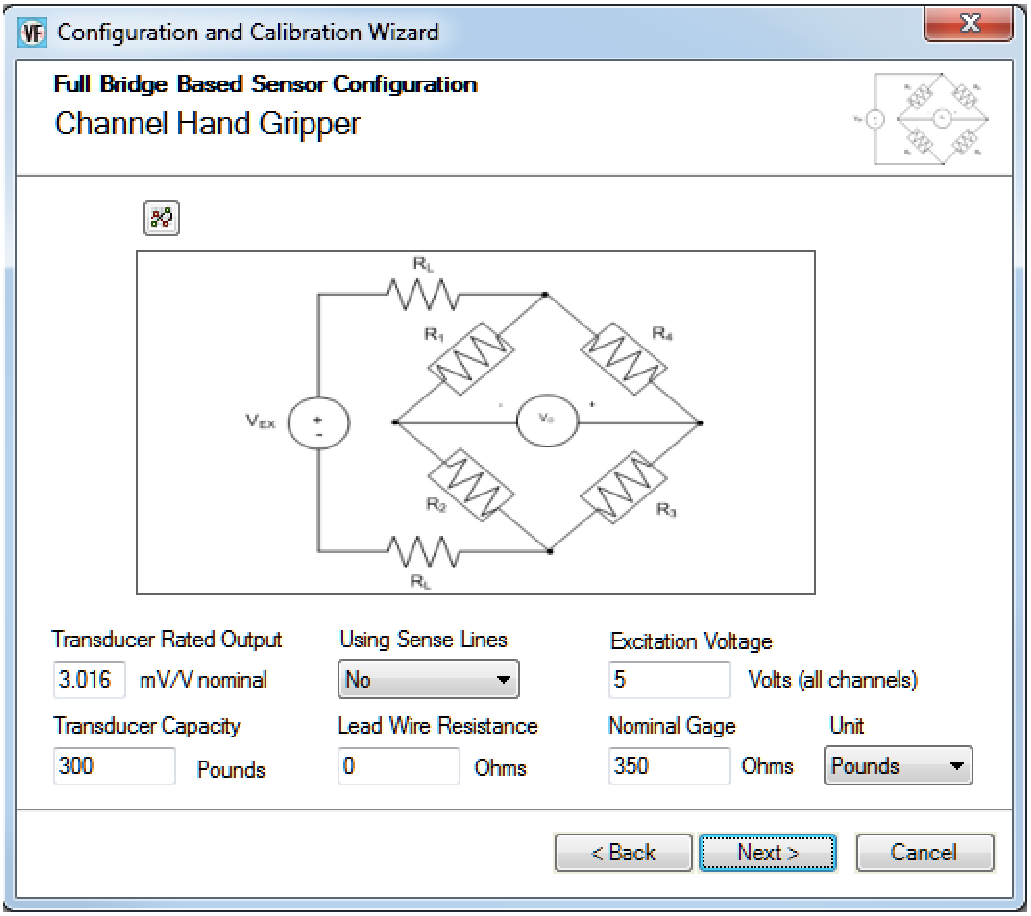
1. Ensure that the load cell is connected to your data acquisition device.
   In this example, the hand gripper load cell is connected to analog input channel 0 of the DT9838.
2. Configure each analog input channel by clicking the **Input Channel Configuration** toolbar button ( ) or by clicking the **Configuration** menu and clicking **Input Channel Configuration**.
3. Enable analog input channel 0 by clicking the **Enable** checkbox for analog input channel 0.



4. In the **Channel Name** column, enter **Hand Gripper** for the name of analog input channel 0.
5. In the **Ref/Resp** column, leave **Response** selected for this example.
6. In the **Enable Shunt Resistor** column, leave the checkbox unchecked; this disables the shunt resistor.
7. Click the **Configure and Calibrate** button.
   The following wizard appears:



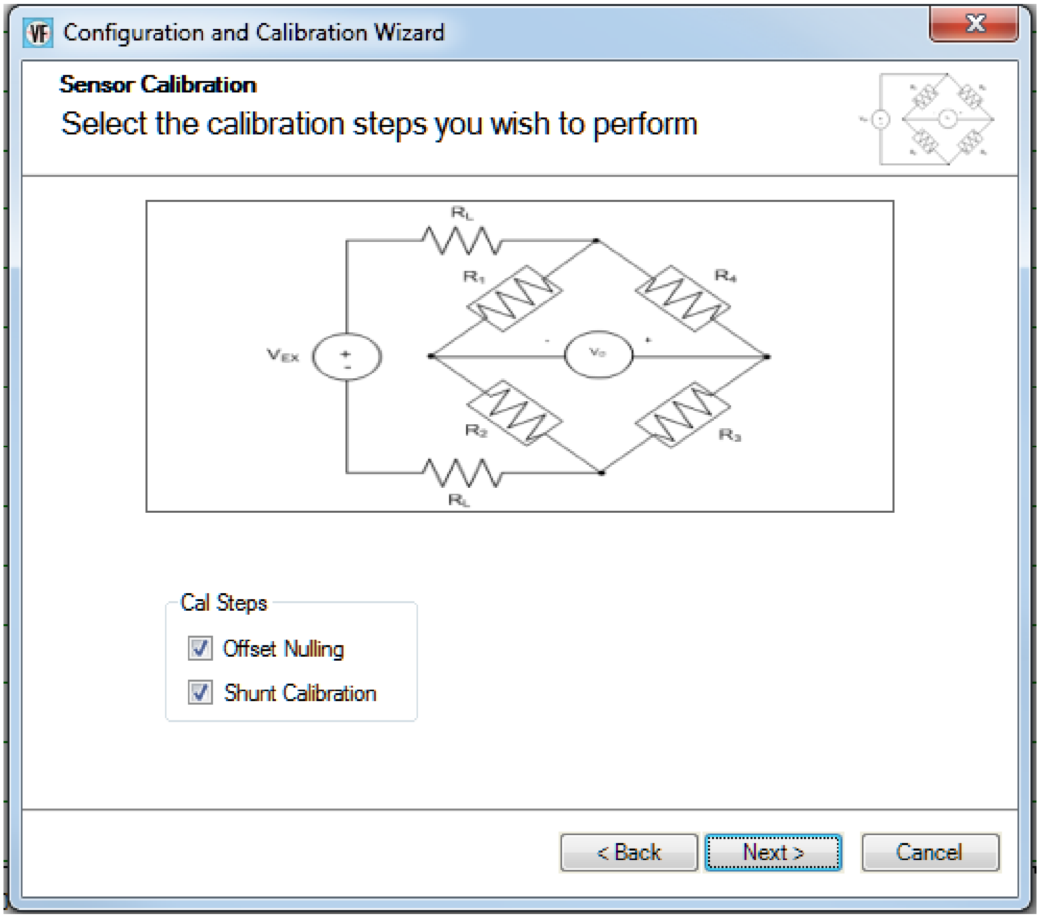
8. Select **Full Bridge Based Sensor**.
   Since TEDS is supported by the hardware, the LED turns green and the message "TEDs support for bridge based sensor detected" appears.
9. Click **Apply TEDs Values** to apply the TEDS values from the hardware to the channel configuration.
10. Click **Next**.
    A screen similar to the following appears:



The TEDS values are read from the hardware, applied to the channel configuration, and displayed in the fields on this screen.

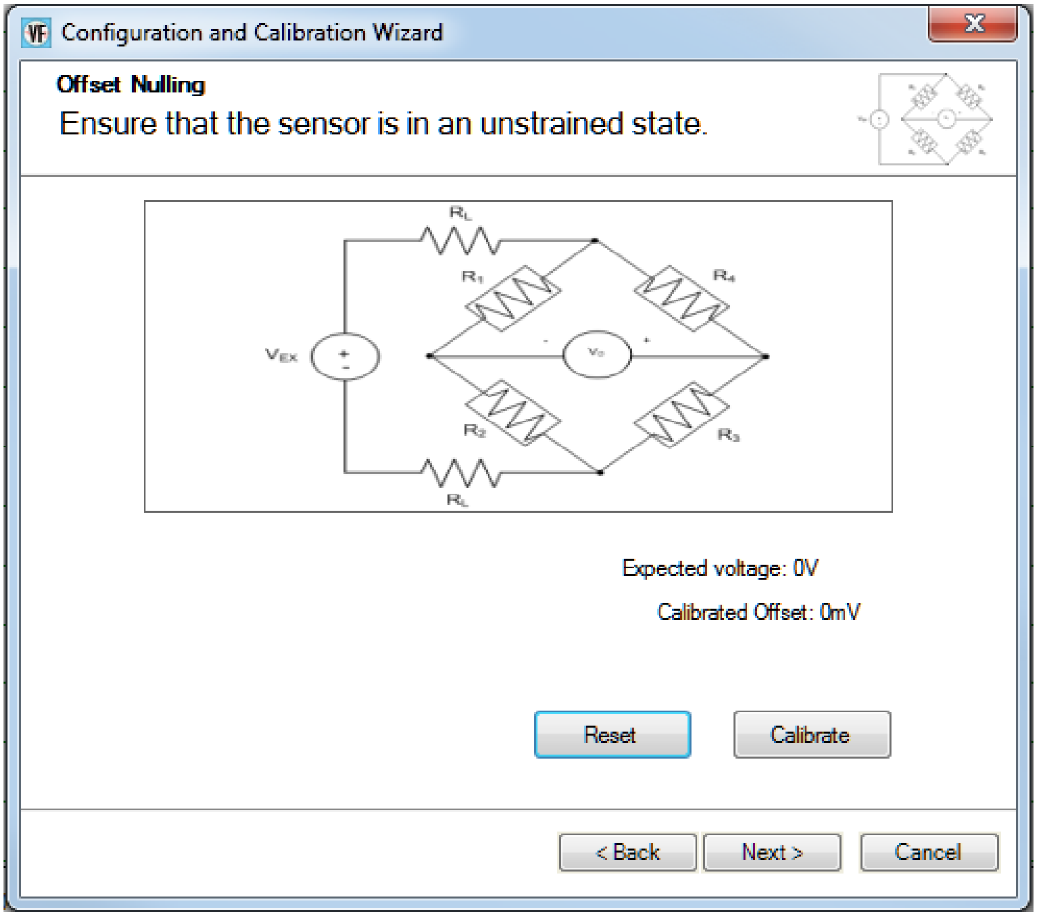
11. If desired, you can modify any of the parameters for the load cell.

12. Click **Next**.
A screen similar to the following appears:

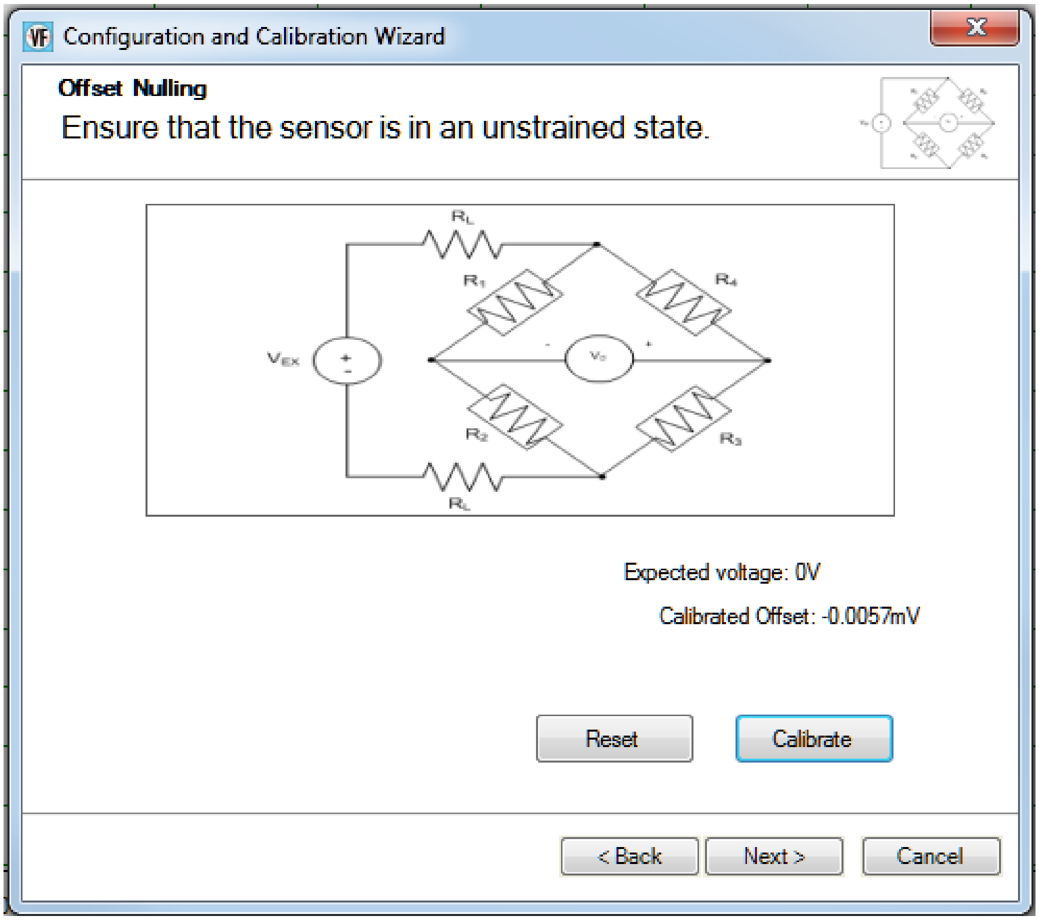


13. Select the calibration steps to perform.
In this example, **Offset Nulling** *and* **Shunt Calibration** *are selected.*

14. Click **Next**.
A screen similar to the following appears:

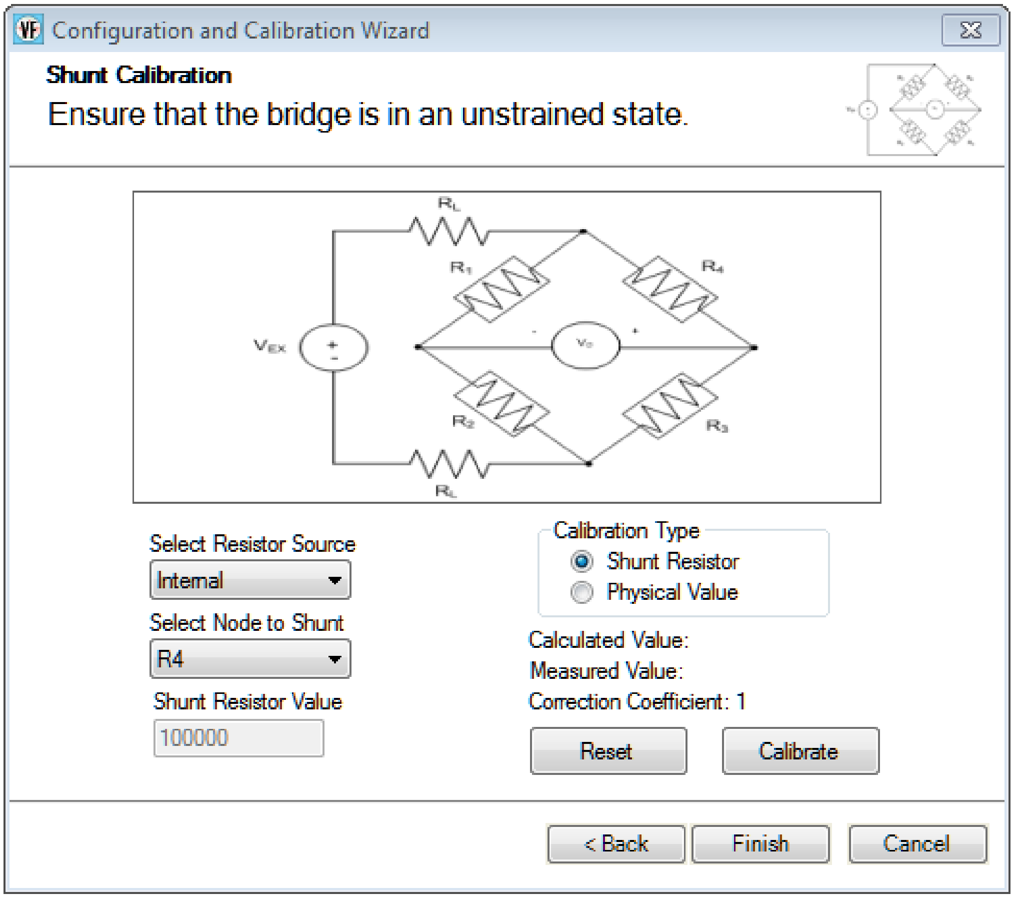


15. Ensure that the bridge is in the unstrained state, and the click the **Calibrate** button to perform offset nulling procedure.
The expected voltage is shown along with the calibrated offset voltage.



16. Click **Next**.

A screen similar to the following appears:

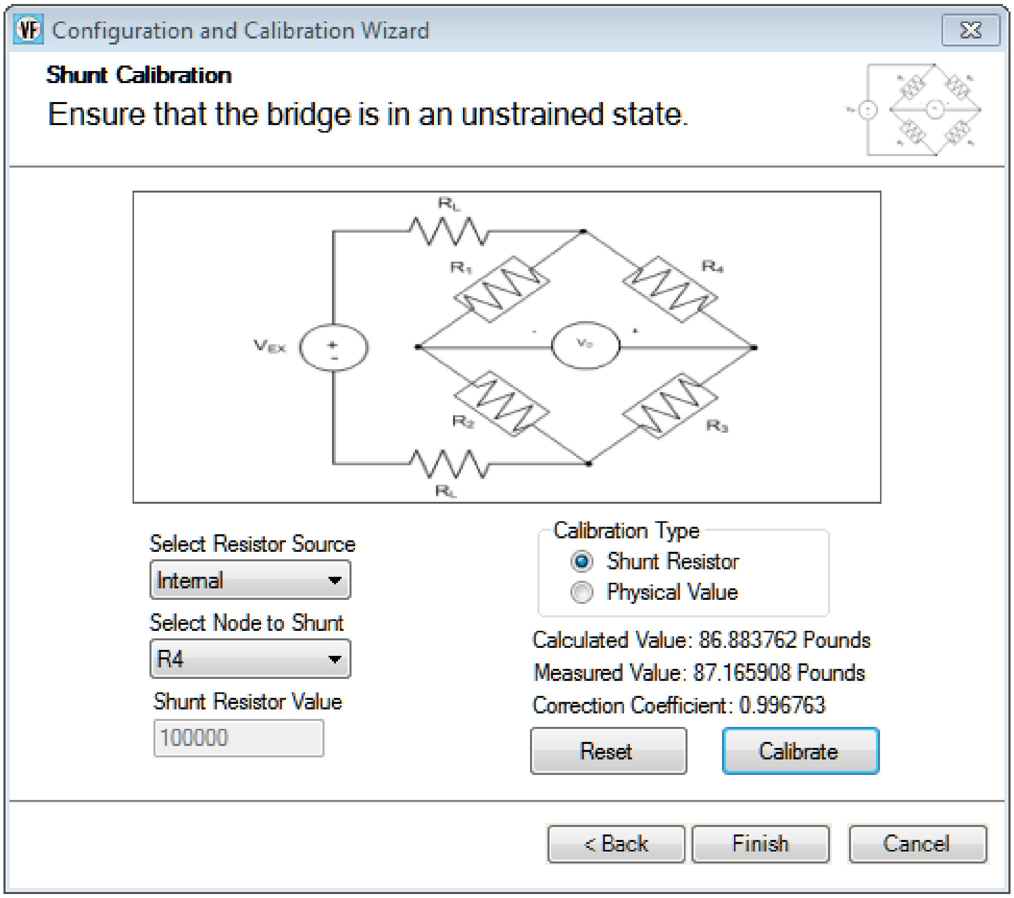


17. For the **Calibration Type**, select **Shunt Resistor** if you want to use a shunt resistor to perform the shunt calibration procedure or **Physical Value** if you want to enter a value to perform the shunt calibration procedure.
    In this example, **Shunt Resistor** *is used.*

18. For the **Select Resistor Source** field, select **Internal** if you are using the internal shunt resistor provided on the DT9838 module to perform shunt calibration, or **External** if you are using your own external resistor to perform shunt calibration.
    In this example, **Internal** *is used.*

19. For the **Select Node to Shunt** field, select the resistive node or element to which to apply the shunt resistor.
    In this example, **R4** *is used.*

20. Click **Calibrate** to perform the shunt calibration procedure.
    The calculated value is displayed along with the measured value and correction coefficient.



21. Click **Finish**.

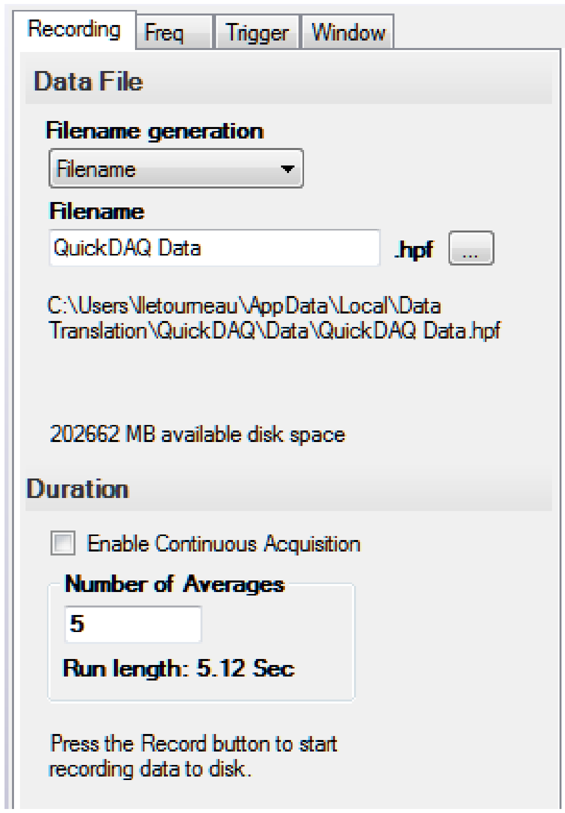
22. If desired, enter a test point number under the **Point #** column.
In this example, ***1*** *is used.*

23. Click **Close** to close the **Configure Devices** dialog box.

Configure the Recording Settings

For this example, configure the recording settings as follows:

1. Click the **Recording** tab of the Acquisition Config window.

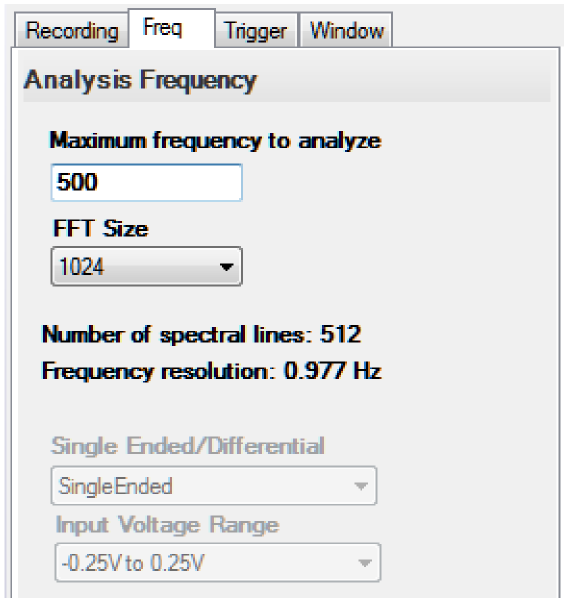


2. For **Filename generation**, select **Filename**.
3. For **Filename**, use the default name for the data file.
4. Leave the **Enable Continuous Acquisition** checkbox unchecked.
5. For **Number of Averages**, enter **5** as the number of records to acquired, process, and average before acquisition stops.
   The number of seconds for the total run and the amount of available disk space are shown.

Configure the Frequency Settings

For this example, configure the frequency settings as follows:

1. Click the **Freq** tab of the Acquisition Config window.



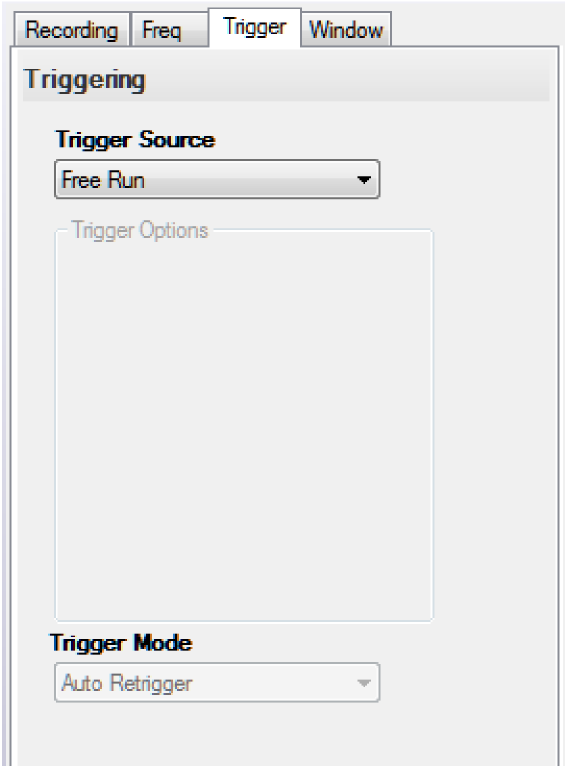
2. In the **Maximum frequency to analyze** text box, enter **500** as the bandwidth or span, in Hz, of the input signals that you want to analyze
Internally, the sampling rate of the data acquisition device is set to twice the bandwidth, or 1000 Hz in this example.

3. In the **FFT Size** text box, select **1024** from the drop-down combo box as the number of data points to use for the FFT calculation.
The number of spectral lines and the frequency resolution are displayed.

Configure the Trigger Settings

For this example, configure the trigger settings as follows:

1. Click the **Trigger** tab of the Acquisition Config window.

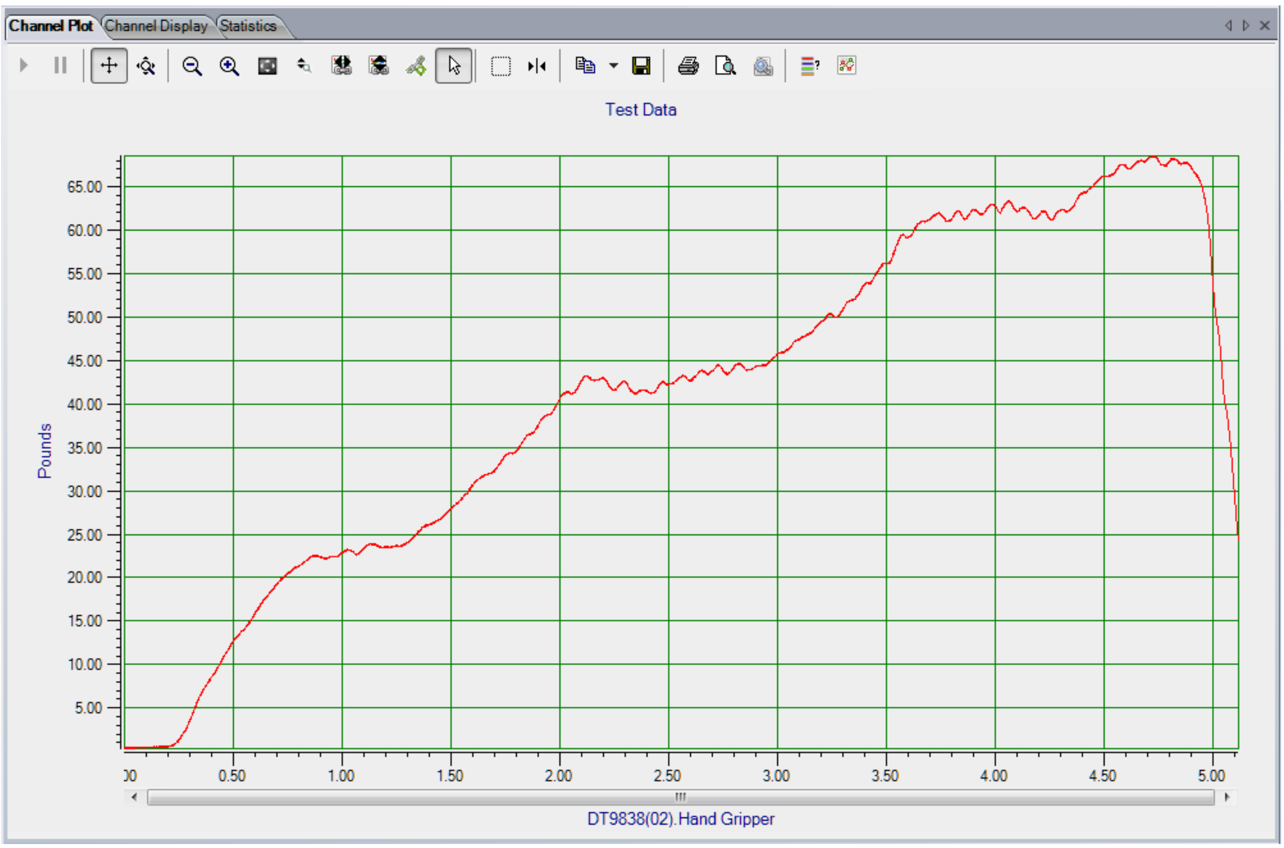


2. For the **Trigger Source**, select **Free Run** to ensure that the measurement starts as soon as the **Record** button is clicked.

Start the Measurement

Once you have configured the channels, start acquisition and log data to disk by clicking the **Record** toolbar button (Record).

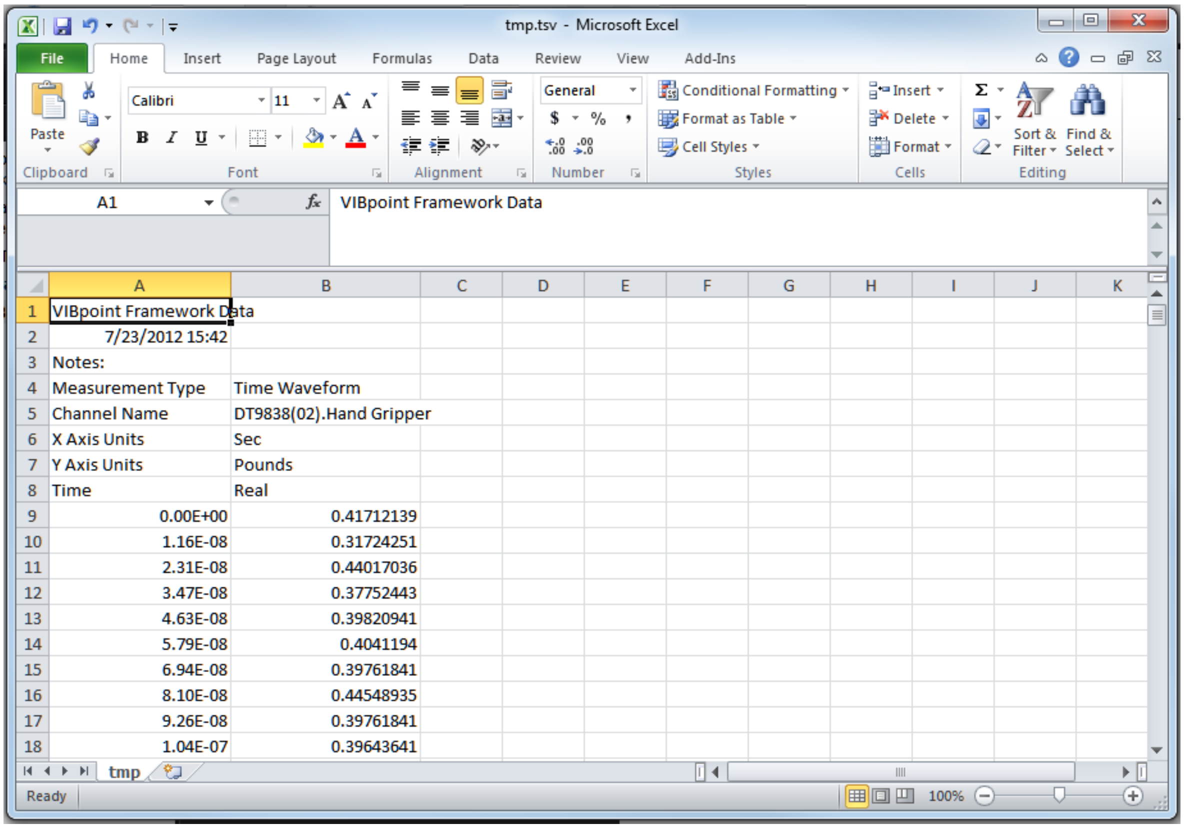
Results similar to the following are displayed in the Channel Plot window.



Export the Data to Excel

You can export the acquired data to Microsoft Excel for further analysis by performing the following steps:

1. Click the **Open Current Data in Excel** toolbar button  .
2. Click **Time Series Data..**
   Microsoft Excel then opens and displays the data that was acquired, as shown in the following example.



Free Run Swept Sine Example

If you are using the Advanced FFT Analysis option of QuickDAQ, you can use the following steps to output a swept sine wave from an analog output channel while simultaneously acquiring data on the analog input channels in Free Run trigger mode. Both the waveform output and the analog input operations are continuous.

This example uses a DT9837A and the following equipment:

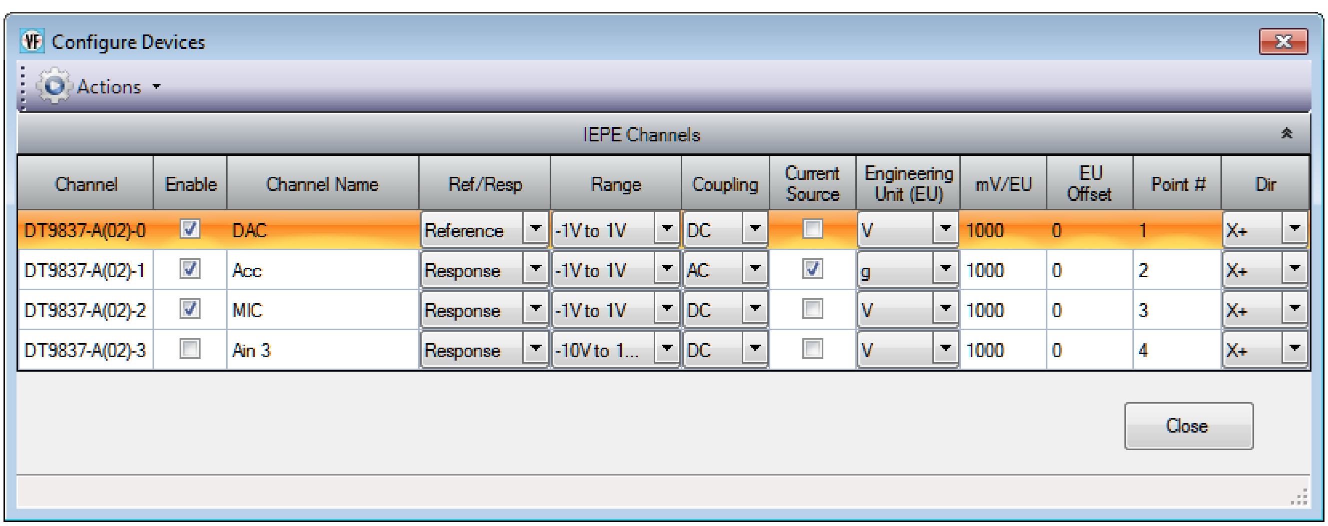
- The analog output channel of the DT9837A is connected to analog input channel 0 of the DT9837A and to an audio speaker.
- An accelerometer is attached to the audio speaker and connected to analog input channel 1 of the DT9837A.
- A microphone is positioned very close to the audio speaker and connected to analog input channel 2 of the DT9837A.

The FFT Analyzer interface is used.

Configure the Channels

Configure the channels as follows:

1. Ensure that the proper signals are attached to the analog input channels of your data acquisition device.
   In this example, the analog output signal is connected to analog input channel 0 and to the audio speaker, an accelerometer is attached to the audio speaker and connected to analog input channel 1, and the microphone is positioned close to the audio speaker and connected to analog input channel 2 of a DT9837A.
2. Configure each analog input channel by clicking the **Input Channel Configuration** toolbar button () or by clicking the **Configuration** menu and clicking **Input Channel Configuration**.
3. Enable analog input channel 0 by clicking the **Enable** checkbox for analog input channel 0.



4. Under the **Channel Name** column, enter the following names for the analog input channels:
   - For analog input channel 0, enter **DAC**.
   - For analog input channel 1, enter **Acc**.
   - For analog input channel 2, enter **MIC**.
5. In the **Ref/Resp** column, select **Reference** for analog input channel 0 (DAC), and select **Response** for analog input channels 1 (Acc) and 2 (MIC).
6. Under the **Range** column, select the best effective input range (±10 V or ±1 V) for each signal. For example, if your hammer and accelerometer measure 100 mV/g and your signal of interest is less than 100 g, choose the ±1 V range; similarly, if your signal of interest is greater than 100 g, choose the ±10 V range.
   In this example, ***±1 V*** *is used for analog input channels 0 (DAC), 1 (Acc), and 2 (MIC).*
7. Under the **Coupling** column, select the coupling type (AC or DC) for your sensor. For most IEPE accelerometers, you will want to choose **AC** coupling to remove any DC offset from your measurement. Choose **DC** coupling only if the signal source has no offset voltage or if the DC content of the acquired signal is important.
   In this example, ***AC*** *is used for analog input channel 1 (Acc), and DC is used for analog input channel 0 (DAC) and analog input channel 2 (MIC).*
8. Under the **Current Source** column, select whether to enable or disable use of the 4 mA current source on the data acquisition device. Since most IEPE accelerometers require a current source of 4 mA and a compliance voltage of at least 18 V to drive their internal circuitry, in most cases, you will want to enable use of the current source for each analog input channel.
   In this example, the ***Current Source*** *checkbox is checked (enabled) for analog input channel 1 (Acc) and is unchecked (disabled) for analog input channel 0 (DAC) and analog input channel 2 (MIC).*

Note: If you enable the use of the internal 4 mA excitation current source, it is recommended that you choose AC coupling.

---

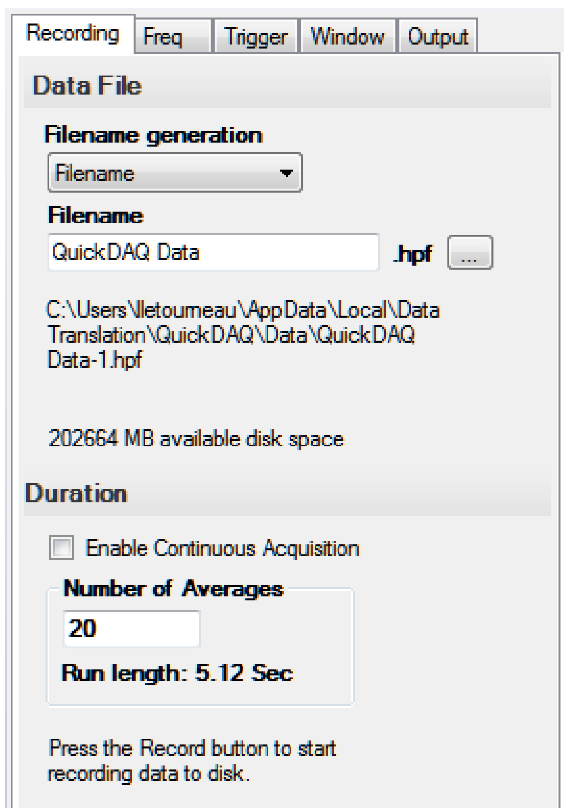
9. Under the **Engineering Units** column, select the engineering units for the analog input channels.
   In this example, ***V*** *is used for analog input channel 0 (DAC) and analog input channel 2 (MIC), and* ***g*** *is used for analog input channel 2 (Acc).*
10. Using the calibration certificate that was supplied with your accelerometer, enter the calibrated sensitivity for the axis that is connected to the selected analog input channel in the **mV/EU** field.
    In this example, 1000 mV/g is selected for analog input channel 1 (Acc).
11. If an offset is specified for the accelerometer, enter the value in the **EU Offset** field.
    In this example, no offset (0) is specified.

12. If desired, enter a test point value for each channel.
    In this example, the following values are used:
    - *Analog input channel 0 (DAC) = 1*
    - *Analog input channel 1 (Acc) = 2*
    - *Analog input channel 2 (MIC) = 3*
13. If desired, enter a sensor direction for each channel.
    In this example, the direction is not used (all analog input channels are X+).
14. Click **Close** to close the Channel Configuration dialog box.

Configure the Recording Settings

For this example, configure the recording settings as follows:

1. Click the **Recording** tab of the Acquisition Config window.

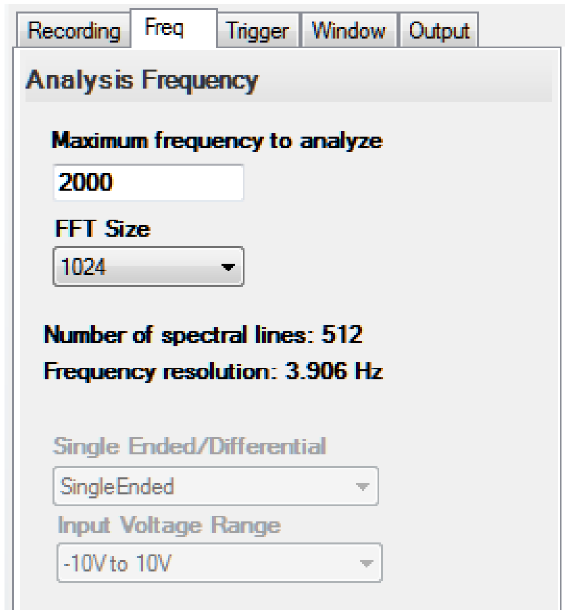


2. For **Filename generation**, select the **Filename** option.
3. For **Filename**, use the default name for the data file.
4. Leave the **Enable Continuous Acquisition** checkbox unchecked.
5. For **Number of Averages**, enter **20** as the number of records to acquire, process, and average before acquisition stops.
   The number of seconds for the entire run and the amount of available disk space are shown.

Configure the Frequency Settings

For this example, configure the frequency settings as follows:

1. Click the **Freq** tab of the Acquisition Config window.

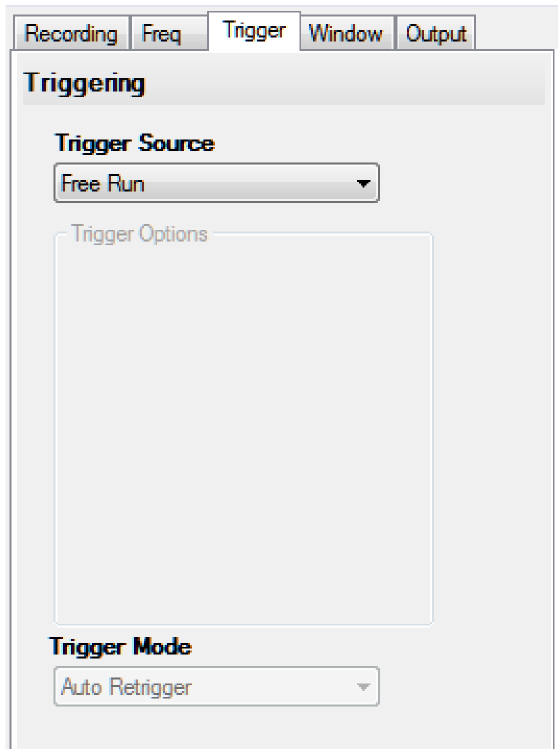


2. In the **Maximum frequency to analyze** text box, enter **2000** as the bandwidth or span, in Hz, of the input signals that you want to analyze
   Internally, the sampling rate of the data acquisition device is set to twice the bandwidth, or 4000 Hz in this example.
3. In the **FFT Size** text box, select **1024** from the drop-down combo box as the number of data points to use for the FFT calculation.
   The number of spectral lines and the frequency resolution are displayed.

Configure the Trigger Settings

For this example, configure the trigger settings as follows:

1. Click the **Trigger** tab of the Acquisition Config window.

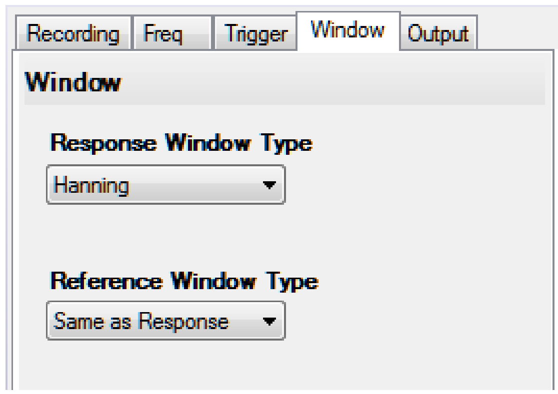


2. For the **Trigger Source**, select **Free Run**.
   *This trigger setting ensures that the module acquires contiguous, gap-free, records when you click the **Record** button or press the **F5** key.*

Configure the Windowing Function

For this example, configure the windowing function as follows:

1. Click the **Window** tab of the Acquisition and Config window.

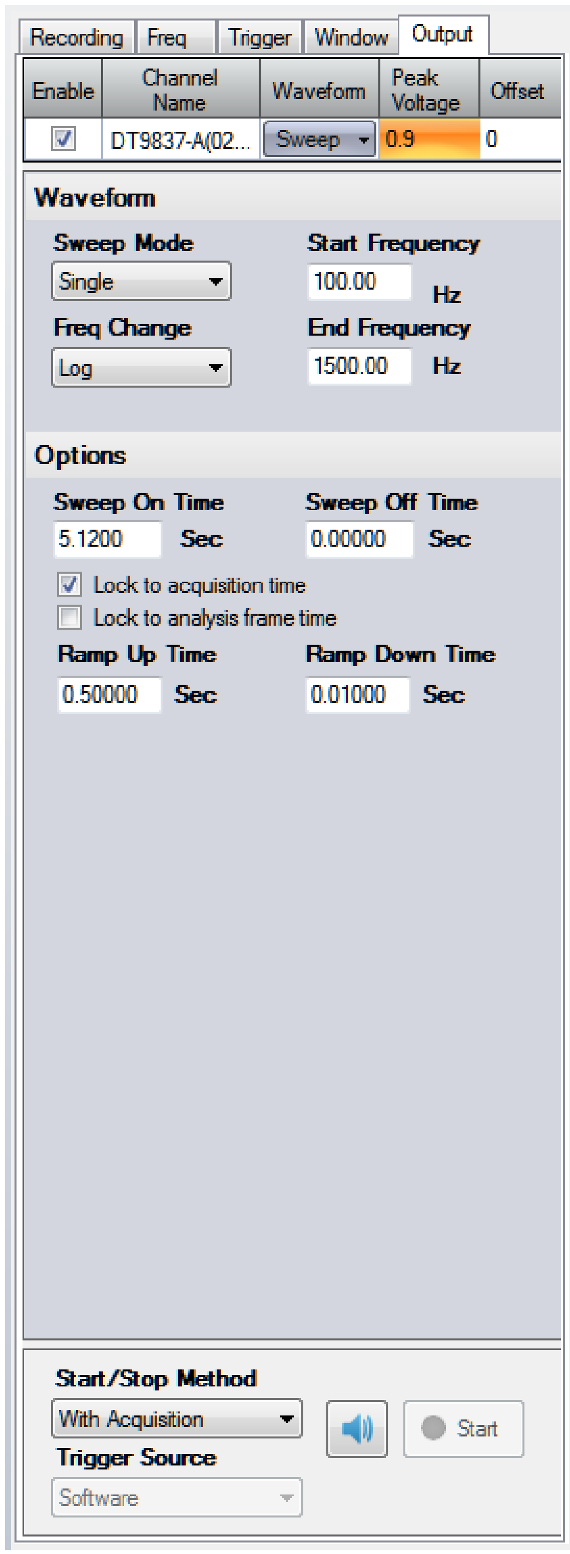


2. For the **Response Window Type**, select **Hanning**.
   This is a global setting that is applied to the FFTs for all enabled analog input channels that are configured as response channels.
3. For the **Reference Window Type**, select **Same as Response**.

Configure the Analog Output Settings

For this example, configure the analog output channel as follows:

1. Click the **Output** tab of the Acquisition and Config window.



2. Select the **Enable** checkbox to enable the analog output channel on the device.
3. For the **Waveform** type, select **Sweep** to output a swept sine wave.
4. For the **Peak Voltage**, enter **0.9** as the maximum peak voltage for this waveform.
   The maximum peak voltage of the output waveform will be ±0.9 V.
5. For the **Offset** value, enter 0.
6. For the **Sweep Mode**, select **Single** to output a single waveform.
7. For the **Freq Change**, select **Log** to output a waveform where the frequency increases at a fixed rate of octave change per time (Octave/s).
8. For the **Start Frequency**, enter 100 Hz; for the **End Frequency**, enter 1500 Hz.
   The output sine wave will start at 100 Hz and sweep to 1500 Hz.
9. Select the **Lock to acquisition time** checkbox.
   The waveform will be output in the time that it takes to do all the averages (an acquisition of all the records specified in the ***Recording*** *tab). Note that the* ***Sweep On Time*** *is set to the acquisition time, which in this case, is 5.12 s.*
10. For this example, leave the **Sweep Off Time** as **0 s**.
    In this example, the output signal will be on for 5.12 s and then stop.
11. For the **Ramp Up Time**, specify **0.5 s**; for the **Ramp Down Time**, specify **0.01 s**.
    The signal will go up to specified peak voltage in 0.5 s and will go down to 0 V in 0.01 s.
12. For the **Start/Stop Method**, select **With Acquisition**.
    The analog output operations will start when you start acquisition.
13. If desired, hide the Acquisition Config window by clicking the Auto-Hide pin ( ) in the top, right corner of the window.

Add FFT Channels to the Plot and Data Config Window

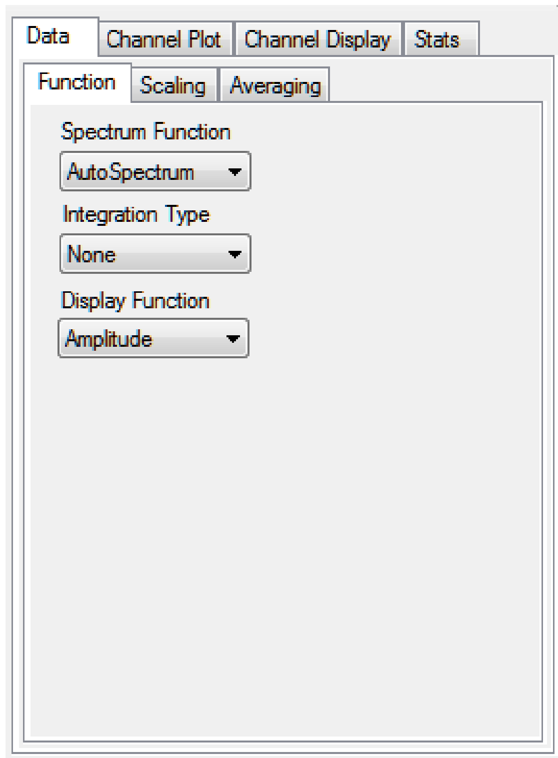
The following steps show how to use the FFT Analyzer interface of QuickDAQ to add three FFT channels to the Plot and Data Config window:

1. Click the **Add single channel FFT function** ( ) button and select **DT9837-A(02).DAC**.
   The DAC-FFT channel is added to the Plot and Data Config window.
2. Click the **Add single channel FFT function** ( ) button and select **DT9837-A(02).Acc**.
   The Acc-FFT channel is added to the Plot and Data Config window.
3. Click the **Add single channel FFT function** ( ) button and select **DT9837-A(02).MIC**.
   The MIC-FFT channel is added to the Plot and Data Config window.

Configure the FFT Channels

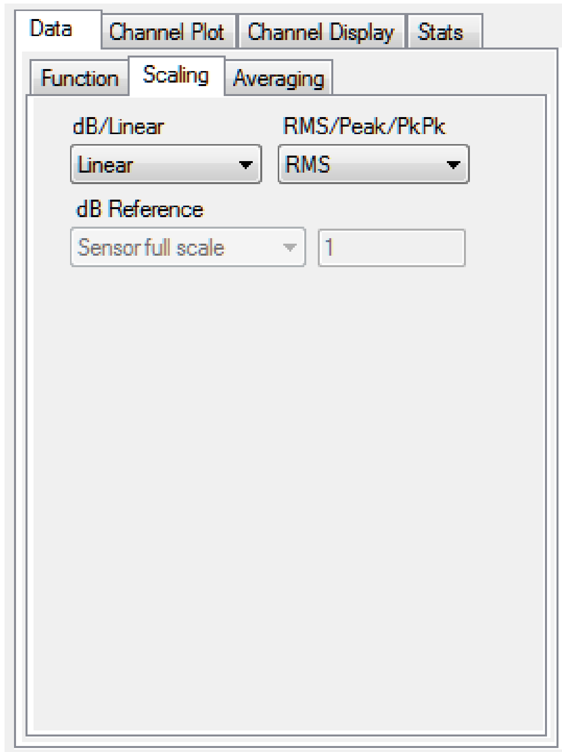
The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used for the FFT channel:

1. Under the **Channel Name** column in the Plot and Data Config window, hold down the **Ctrl** key and select the FFT channels (**DAC-FFT**, **Acc-FFT**, and **MIC-FFT** in this example) that you want to configure.
2. Click the **Data** tab, and then click the **Function** tab.

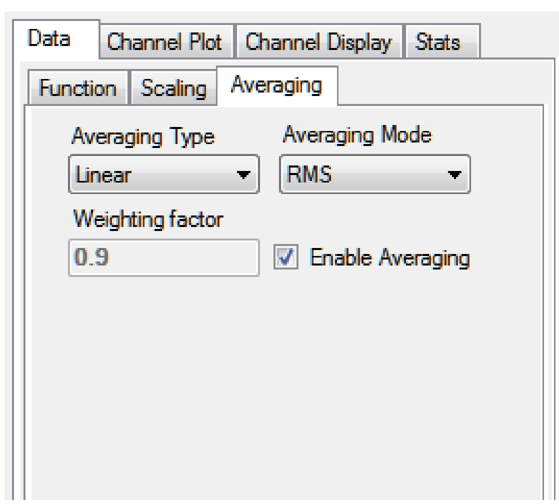


3. For this example, configure the settings of this tab as follows:
   a. For **Spectrum Function**, select **AutoSpectrum.**
   b. For **IntegrationType**, select **None**.
   c. For **Display Function**, select **Amplitude**.

4. Click the **Scaling** tab, and configure the settings of this tab as follows:
   a. For **dB/Linear,** select **Linear**.
   b. For **RMS/Peak/PkPk**, select **RMS**.



5. Click the **Averaging** tab, and configure the settings of this tab as follows:
   a. Select the **Enable Averaging** checkbox.
   b. For **Averaging Type**, select **Linear**.
   c. For **Averaging Mode**, select **RMS**.



Add FRF Channels to the Plot and Data Config Window

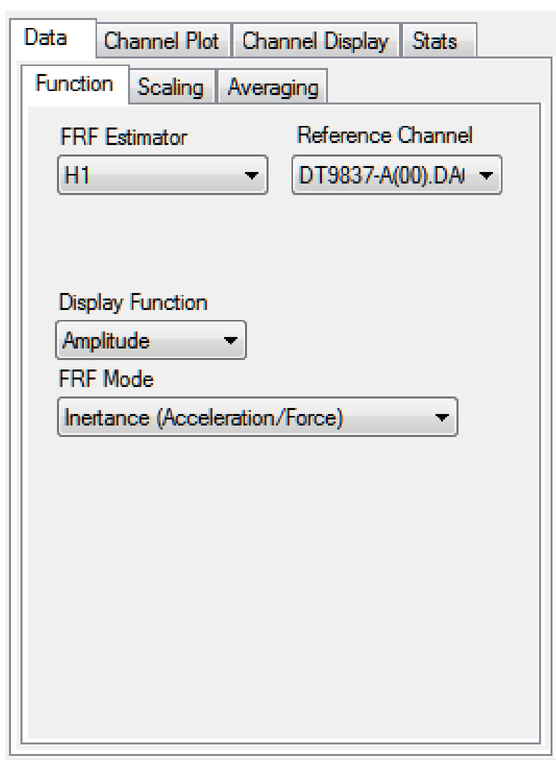
The following steps show how to use the FFT Analyzer interface of QuickDAQ to add two FRF channels to the Plot and Data Config window to calculate the frequency response of analog input channel 0 (DAC) and analog input channel 1 (Acc), and analog input channel 0 (DAC) and analog input channel 2 (MIC).

1. Click the **Add 2 channel FFT function** ( ) button, select **FRF**, and select **DT9837-A(02).Acc**.
   The Acc-FRF channel is added to the Plot and Data Config window.
2. Click the **Add 2 channel FFT function** ( ) button, select **FRF**, and select **DT9837-A(02).MIC**.
   The MIC-FRF channel is added to the Plot and Data Config window.

Configure the FRF Channels

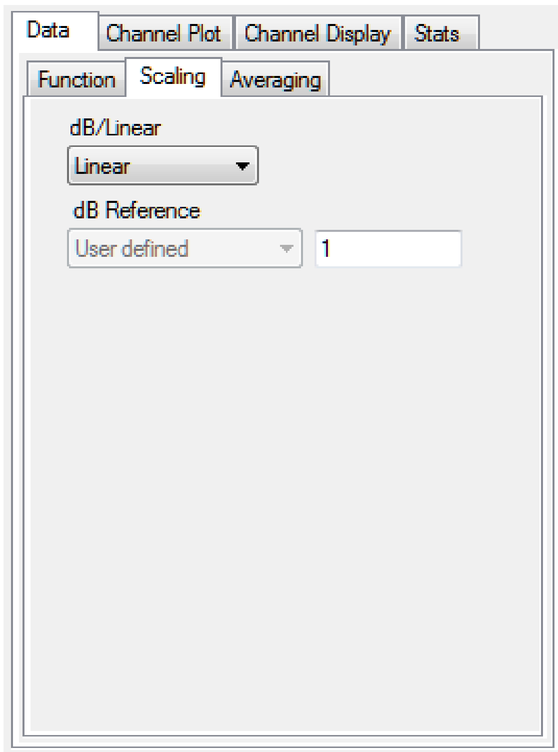
The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used to generate the FRF channels for the analog input channels:

1. Under the **Channel Name** column in the Plot and Data Config window, hold down the **Ctrl** key and select the FRF channels (**Acc-FRF** and **MIC-FRF** in this example) that you want to configure.
2. Click the **Data** tab, and then click the **Function** tab.

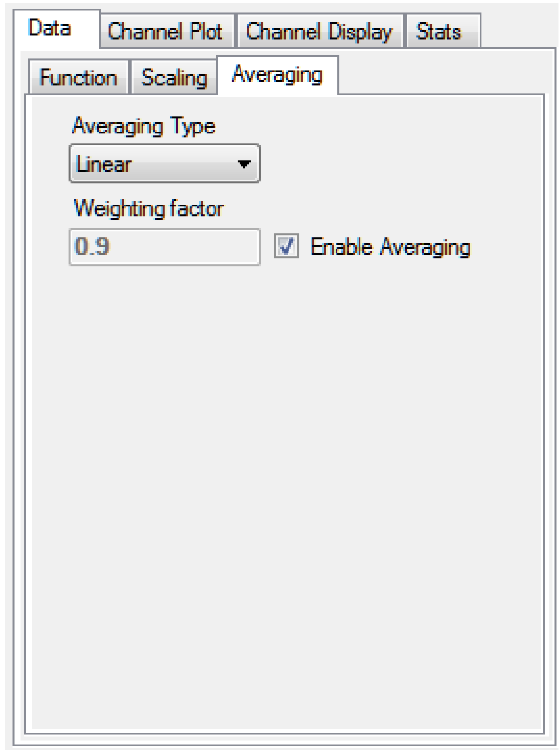


3. For the **Acc-FRF** channel, configure the settings of this tab as follows:
   a. For **FRF Estimator**, select **H1** (reduces noise on the response channel).
   b. For **Reference Channel**, select **DAC** (analog input channel 0).
   c. For **Display Function**, select **Amplitude**.
   d. For **FRF Mode**, select **Inertance (Acceleration/Force)**.

4. Click the **Scaling** tab, and select **Linear** for **dB/Linear**:



5. Click the **Averaging** tab, select the **Enable Averaging** checkbox, and select **Linear** for **Averaging Type**:



Add Coherence Channels to the Plot and Data Config Window

The following steps show how to use the FFT Analyzer interface of QuickDAQ to add two Coherence channels to the Plot and Data Config window:

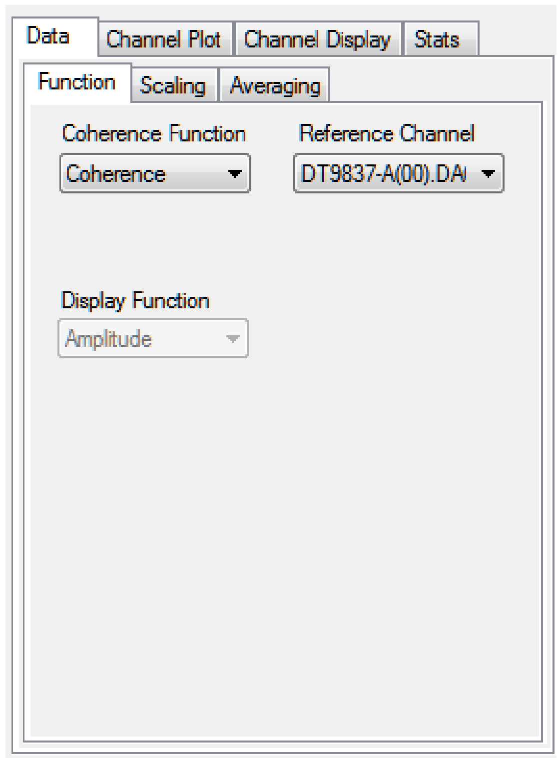
1. Click the **Add 2 channel FFT function** ( ) button, select **Coherence** and select **DT9837-A(02).Acc**.
   The Acc-Coherence channel is added to the Plot and Data Config window.

2. Click the **Add 2 channel FFT function** ( ) button, select **Coherence**, and select **DT9837-A(02).MIC**.
   The MIC-Coherence channel is added to the Plot and Data Config window.

Configure the Coherence Channel

The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used to generate the Coherence channel for the analog input channels:

1. Under the **Channel Name** column in the Plot and Data Config window, hold down the **Ctrl** key and select the Coherence channels (**Acc-Coherence** and **MIC-Coherence** in this example) that you want to configure.
2. Click the **Data** tab, and then click the **Function** tab.

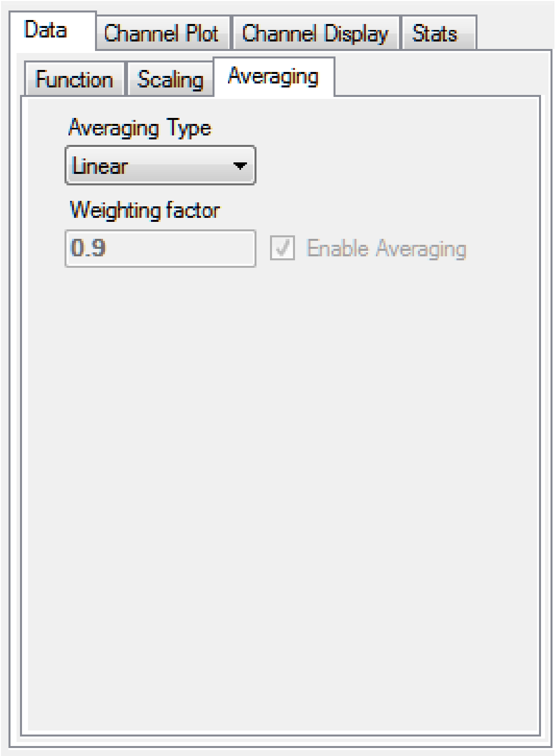


3. For this example, configure the settings of this tab as follows:
   a. For **Coherence Function**, select **Coherence**.
   b. For **Reference Channel**, select **DAC** (analog input channel 0).

Note: **Linear** scaling is selected automatically for Coherence channels under the **Scaling** tab.

---

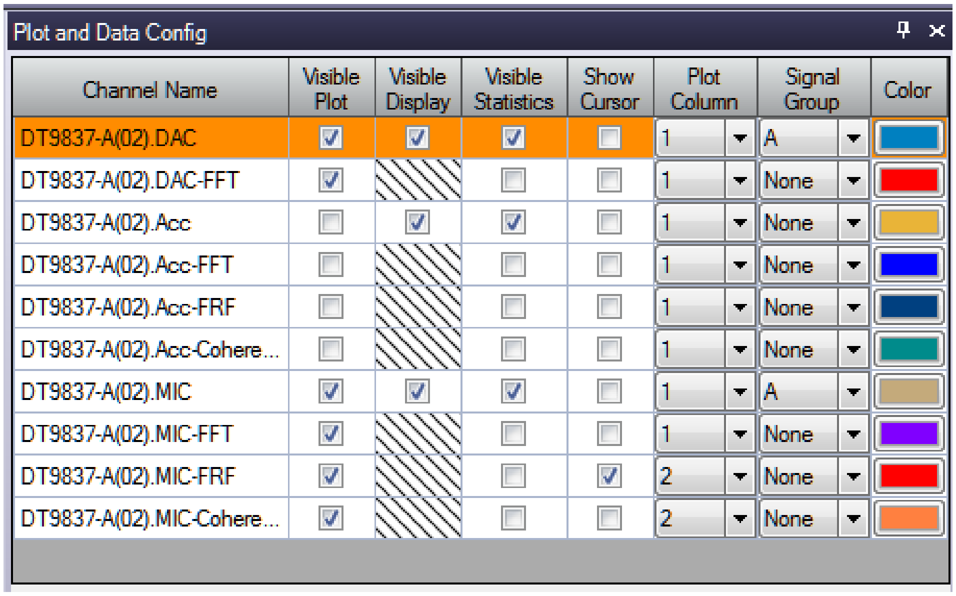
4. Click the **Averaging** tab, and select **Linear** for **Averaging Type**:



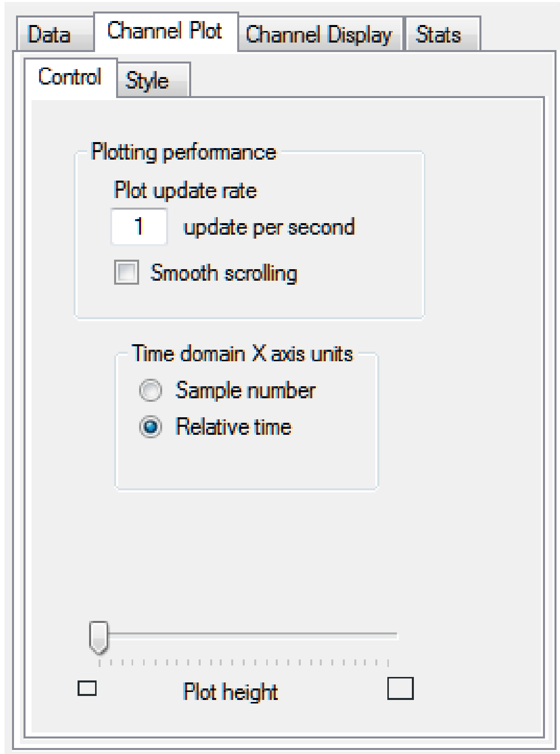
Configure the Appearance of the Channel Plot Window

Perform the following steps to configure the appearance of the Channel Plot window:

1. Select the **Visible Plot** checkbox for the DAC, DAC-FFT, MIC, MIC-Coherence, MIC-FRF, and MIC-FFT channels.



2. Select the **Show Cursor** checkbox for the MIC-FRF channel.
3. Under **Plot Column**, select plot column **1** for the DAC, DAC-FFT, MIC, and MIC-FFT channels, and select plot column **2** for the MIC-Coherence and MIC-FRF channels.
4. Under the **Signal Group** column, select group **A** for the DAC and MIC channels, and leave **None** as the signal group for the rest of the channels.
5. Assign a unique color to each trace for the visible channels.
6. Click the **Channel Plot - Control** tab.



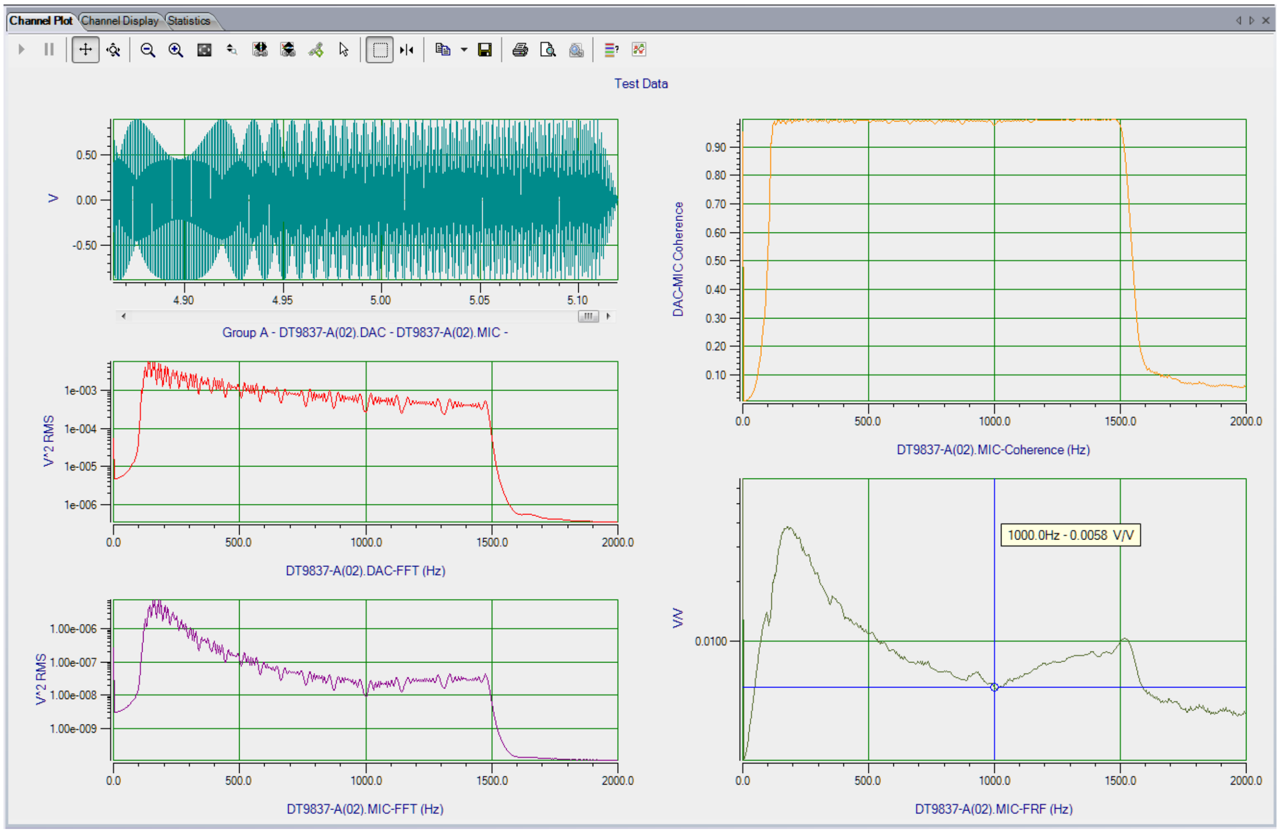
7. Slide the **Plot height** bar to the left until you can see all 5 plots on the Channel Plot window at once.

Start the Simultaneous Swept Sine and Analog Input Operation

Once you have configured the channels and the parameters for the Channel Plot window, click the **Record** toolbar button (

) or press the **F5** key.

You will hear the waveform as it is output from the audio speaker as it sweeps from 100 Hz to 1500 Hz. Results similar to the following are displayed in the Channel Plot window.



In column 1, you can see the signal that is output from the analog output channel (DAC) and read in as analog input channel 0, and the signal that is measured by the microphone (MIC); this is Group A of the first plot. You can also see the FFT of the analog output channel (DAC) and the FFT of the microphone (MIC).

In column 2, you can see the coherence of the microphone (MIC) and the FRF of the microphone (MIC). In this example, the y-axis of the FRF plot is shown in log10 format (right-click the y-axis, select **Y-Axis Setup...**, and select **Log10** for **Lin/Log**).

Note: If desired, you can show the data for the Acc, Acc-FFT, Acc-FRF, and Acc-Coherence channels by selecting the **Visible Plot** checkbox in the Channel Plot window.

Triggered Burst Random Example

If you are using the Advanced FFT Analysis option of QuickDAQ, you can use the following steps to output multiple burst random waveforms while simultaneously acquiring data on the analog input channels in Software trigger mode. The duration of each burst random signal corresponds to the duration of one analysis record. Between each burst, the analog input and analog output operations stop, wait the specified delay time, and then repeat the burst.

This example uses a DT9837A and the following equipment:

- The analog output channel of the DT9837A is connected to analog input channel 0 of the DT9837A and to an audio speaker.
- An accelerometer is attached to the audio speaker and connected to analog input channel 1 of the DT9837A.
- A microphone is positioned very close to the audio speaker and connected to analog input channel 2 of the DT9837A.

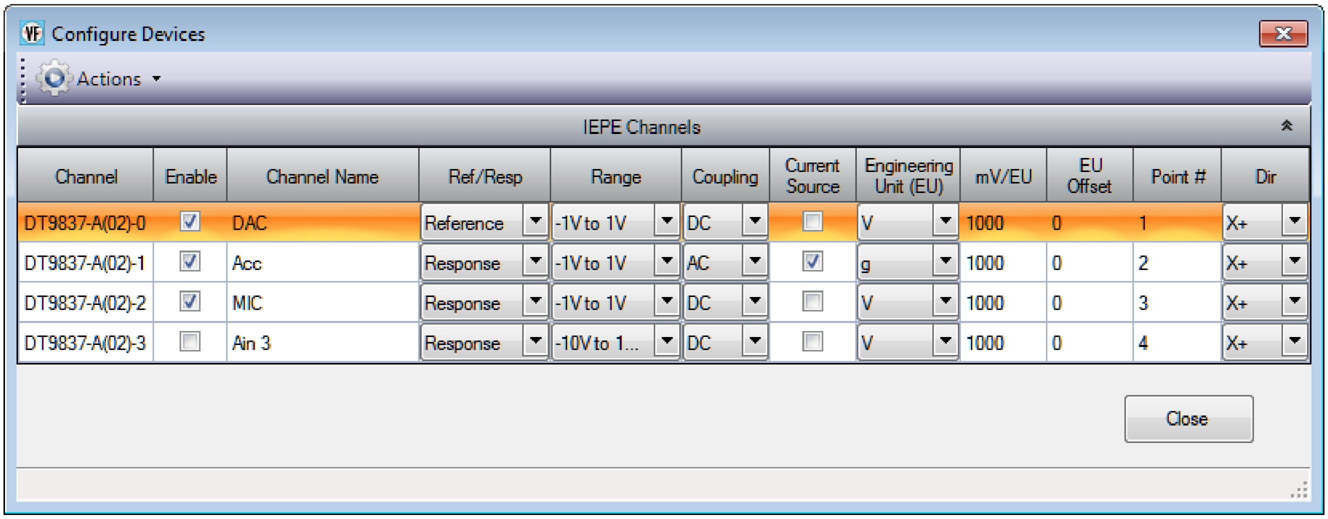
The FFT Analyzer interface is used.

Configure the Channels

Configure the channels as follows:

1. Ensure that the proper signals are attached to the analog input channels of your data acquisition device.
   In this example, the analog output signal is connected to analog input channel 0 and to the audio speaker, an accelerometer is attached to the audio speaker and connected to analog input channel 1, and the microphone is positioned close to the audio speaker and connected to analog input channel 2 of a DT9837A.
2. Configure each analog input channel by clicking the **Input Channel Configuration** toolbar button ( ) or by clicking the **Configuration** menu and clicking **Input Channel Configuration**.

3. Enable analog input channel 0 by clicking the **Enable** checkbox for analog input channel 0.



4. Under the **Channel Name** column, enter the following names for the analog input channels:
   – For analog input channel 0, enter **DAC**.
   – For analog input channel 1, enter **Acc**.
   – For analog input channel 2, enter **MIC**.
5. In the **Ref/Resp** column, select **Reference** for analog input channel 0 (DAC), and select **Response** for analog input channels 1 (Acc) and 2 (MIC).
6. Under the **Range** column, select the best effective input range (±10 V or ±1 V) for each signal. For example, if your hammer and accelerometer measure 100 mV/g and your signal of interest is less than 100 g, choose the ±1 V range; similarly, if your signal of interest is greater than 100 g, choose the ±10 V range.
   In this example, ***±1 V*** *is used for analog input channels 0 (DAC), 1 (Acc), and 2 (MIC).*
7. Under the **Coupling** column, select the coupling type (AC or DC) for your sensor. For most IEPE accelerometers, you will want to choose **AC** coupling to remove any DC offset from your measurement. Choose **DC** coupling only if the signal source has no offset voltage or if the DC content of the acquired signal is important.
   In this example, ***AC*** *is used for analog input channel 1 (Acc), and DC is used for analog input channel 0 (DAC) and analog input channel 2 (MIC).*
8. Under the **Current Source** column, select whether to enable or disable use of the 4 mA current source on the data acquisition device. Since most IEPE accelerometers require a current source of 4 mA and a compliance voltage of at least 18 V to drive their internal circuitry, in most cases, you will want to enable use of the current source for each analog input channel.
   In this example, the ***Current Source*** *checkbox is checked (enabled) for analog input channel 1 (Acc) and is unchecked (disabled) for analog input channel 0 (DAC) and analog input channel 2 (MIC).*

Note: If you enable the use of the internal 4 mA excitation current source, it is recommended that you choose AC coupling.

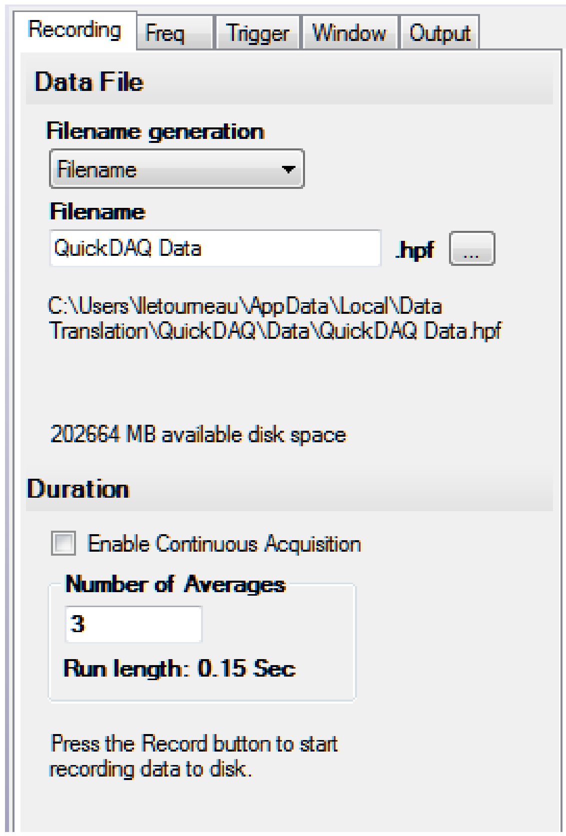
---

9. Under the **Engineering Units** column, select the engineering units for the analog input channels.
   In this example, ***V*** *is used for analog input channel 0 (DAC) and analog input channel 2 (MIC), and* ***g*** *is used for analog input channel 2 (Acc).*
10. Using the calibration certificate that was supplied with your accelerometer, enter the calibrated sensitivity for the axis that is connected to the selected analog input channel in the **mV/EU** field.
    In this example, 1000 mV/g is selected for analog input channel 1 (Acc).
11. If an offset is specified for the accelerometer, enter the value in the **EU Offset** field.
    In this example, no offset (0) is specified.
12. If desired, enter a test point value for each channel.
    In this example, the following values are used:
    - *Analog input channel 0 (DAC) = 1*
    - *Analog input channel 1 (Acc) = 2*
    - *Analog input channel 2 (MIC) = 3*
13. If desired, enter a sensor direction for each channel.
    In this example, the direction is not used (all analog input channels are X+).
14. Click **Close** to close the Channel Configuration dialog box.

Configure the Recording Settings

For this example, configure the recording settings as follows:

1. Click the **Recording** tab of the Acquisition Config window.

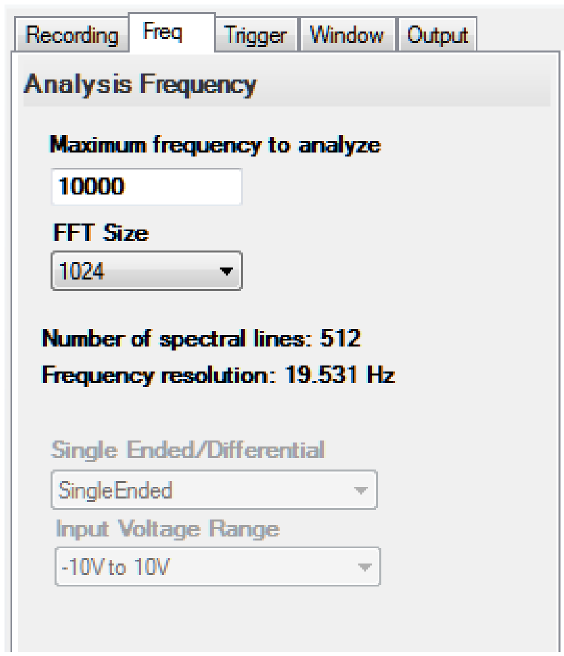


2. For **Filename generation**, select the **Filename** option.
3. For **Filename**, use the default name for the data file.
4. Leave the **Enable Continuous Acquisition** checkbox unchecked.
5. For **Number of Averages**, enter **3** as the number of records to acquire, process, and average before acquisition stops.
   The number of seconds for the entire run and the amount of available disk space are shown.

Configure the Frequency Settings

For this example, configure the frequency settings as follows:

1. Click the **Freq** tab of the Acquisition Config window.

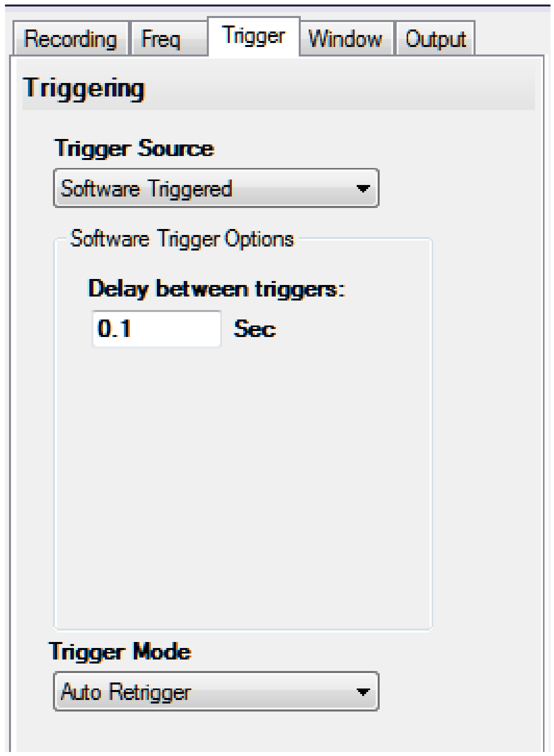


2. In the **Maximum frequency to analyze** text box, enter **10000** as the bandwidth or span, in Hz, of the input signals that you want to analyze
   Internally, the sampling rate of the data acquisition device is set to twice the bandwidth, or 20000 Hz in this example.
3. In the **FFT Size** text box, select **1024** from the drop-down combo box as the number of data points to use for the FFT calculation.
   The number of spectral lines and the frequency resolution are displayed.

Configure the Trigger Settings

For this example, configure the trigger settings as follows:

1. Click the **Trigger** tab of the Acquisition Config window.

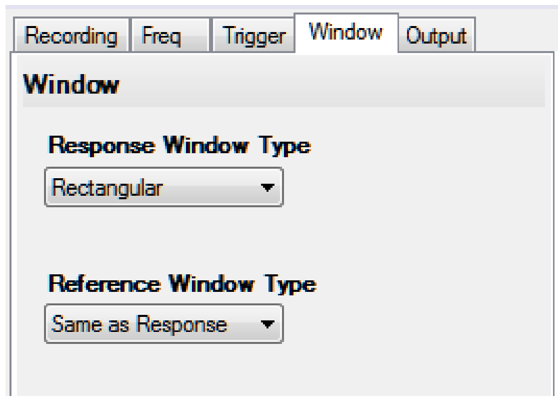


2. For the **Trigger Source**, select **Software Triggered**.
3. For the **Trigger Mode**, select **Auto Retrigger**.
4. For **Delay between triggers**, enter **0.1 s**.
   *With these trigger settings, once you click the **Record** button or press the **F5** key, the module acquires a record based on a internal software trigger, waits 0.1 second, and then acquires the next record based on the internal software trigger.*

Configure the Windowing Function

For this example, configure the windowing function as follows:

1. Click the **Window** tab of the Acquisition and Config window.

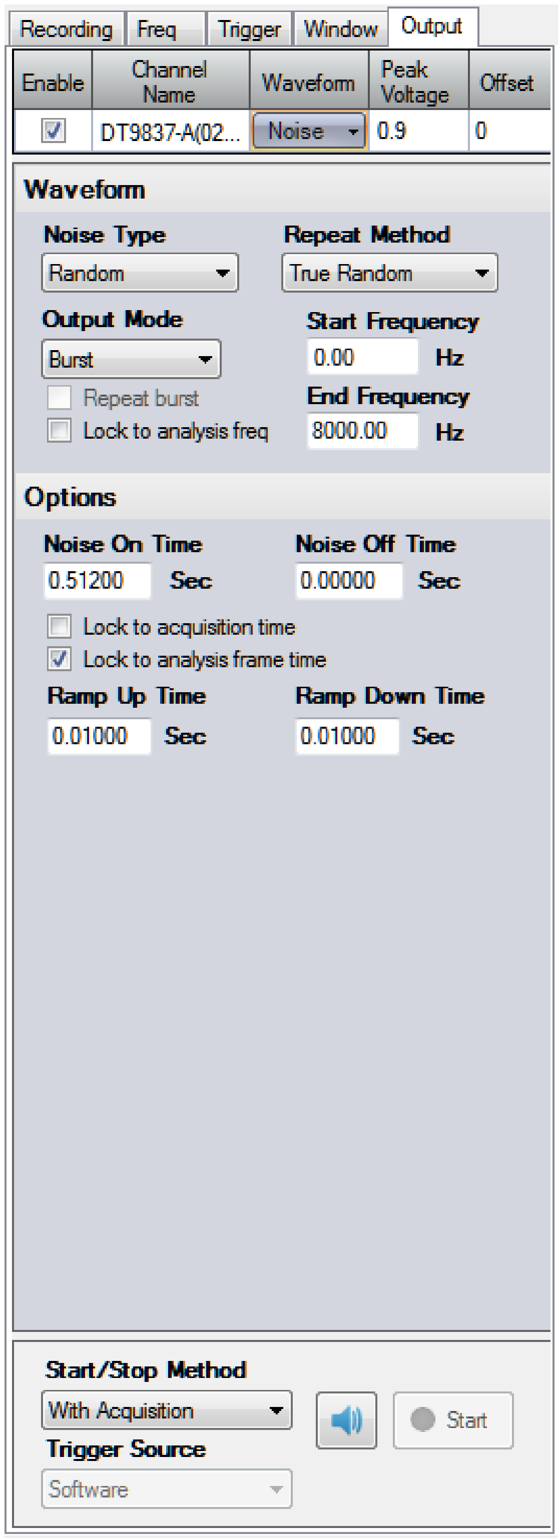


2. For the **Response Window Type**, select **Rectangular**.
   This is a global setting that is applied to the FFTs for all enabled analog input channels that are configured as response channels.
3. For the **Reference Window Type**, select **Same as Response**.

Configure the Analog Output Settings

For this example, configure the analog output channel as follows:

1. Click the **Output** tab of the Acquisition and Config window.



2. Select the **Enable** checkbox to enable the analog output channel on the device.
3. For the **Waveform** type, select **Noise** to output a noise waveform.

Note: To output a burst chirp signal, select **Sweep** instead of **Noise** to output a swept sine wave rather than a noise output signal.

---

4. For the **Peak Voltage**, enter **0.9** as the maximum peak voltage for this waveform.
   The maximum peak voltage of the output waveform will be ±0.9 V.
5. For the **Offset** value, enter 0.
6. For the **Noise Type**, select **Random** to output white noise that is created with a software generator function.
7. For the **Output Mode**, select **Burst** to output the waveform for a specified time.
8. For the **Start Frequency**, select **0 Hz**; this is the starting frequency component of the output waveform.
9. For the **End Frequency**, select **8000 Hz**; this is the ending frequency component of the output waveform.
10. For the **Repeat Method**, select **True Random** to generate a new random noise signal for each burst period.
11. Select the **Lock to analysis frame time** checkbox.
    *The waveform will be output in the time that it takes to do one average (an acquisition of one record). Note that the **Noise On Time** is set to the analysis frame time, which in this case, is 0.5120 s.*
12. For this example, leave the **Noise Off Time** as **0 s.**
    In this example, the output signal will be on for 0.5120 s and then stop.
13. For the **Ramp Up Time**, specify **0.01 s**; for the **Ramp Down Time**, specify **0.01 s**.
    The signal will go up to specified peak voltage in 0.01 s and will go down to 0 V in 0.01 s.
14. For the **Start/Stop Method**, select **With Acquisition**.
    The analog output operations will start when you start acquisition.
15. If desired, hide the Acquisition Config window by clicking the Auto-Hide pin ( ) in the top, right corner of the window.

Add FFT Channels to the Plot and Data Config Window

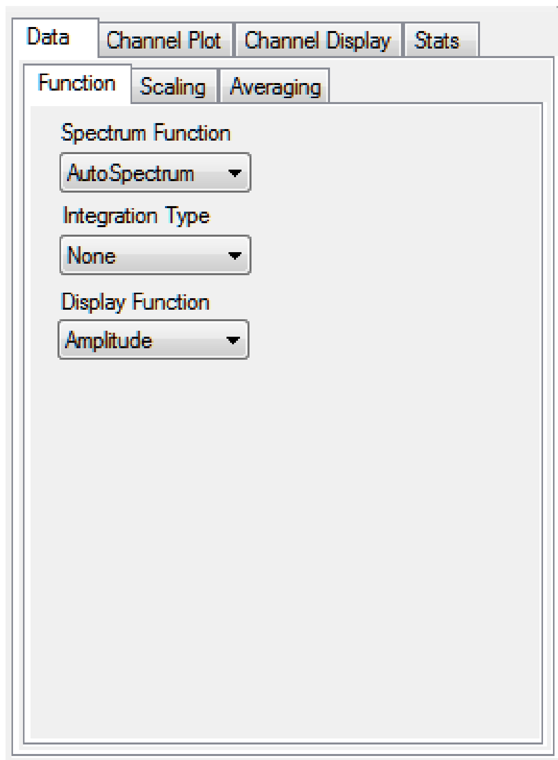
The following steps show how to use the FFT Analyzer interface of QuickDAQ to add three FFT channels to the Plot and Data Config window:

1. Click the **Add single channel FFT function** ( ) button and select **DT9837-A(02).DAC**.
   The DAC-FFT channel is added to the Plot and Data Config window.
2. Click the **Add single channel FFT function** ( ) button and select **DT9837-A(02).Acc**.
   The Acc-FFT channel is added to the Plot and Data Config window.
3. Click the **Add single channel FFT function** ( ) button and select **DT9837-A(02).MIC**.
   The MIC-FFT channel is added to the Plot and Data Config window.

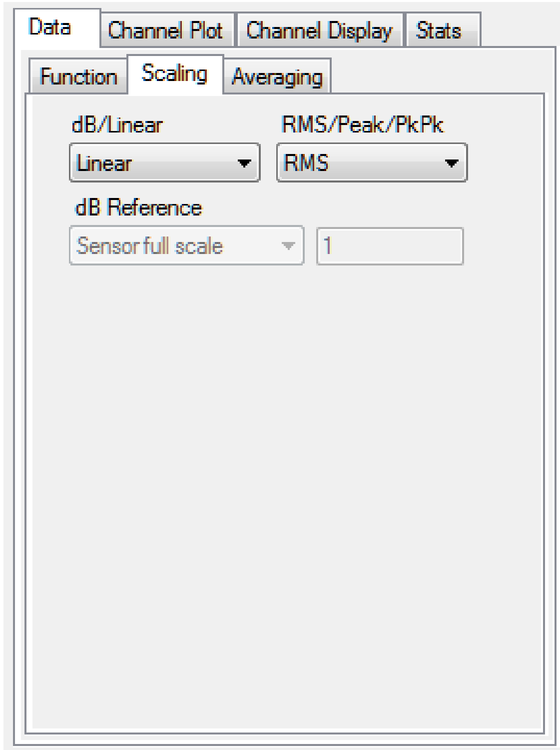
Configure the FFT Channels

The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used for the FFT channel:

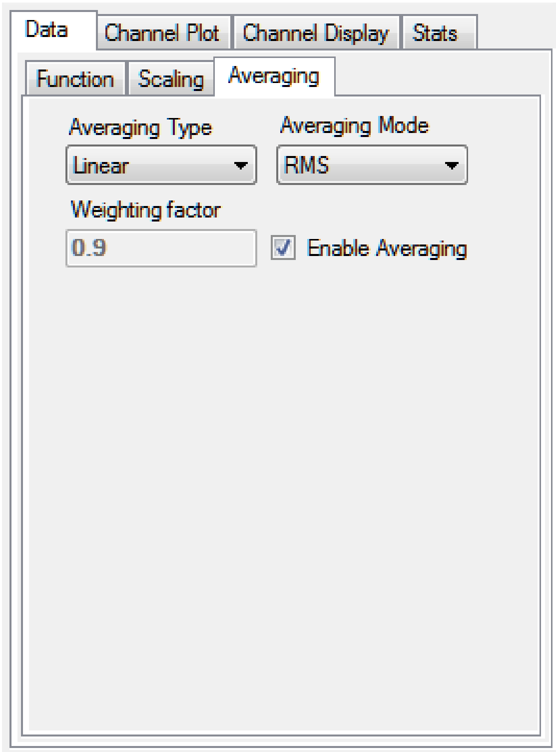
1. Under the **Channel Name** column in the Plot and Data Config window, hold down the **Ctrl** key and select the FFT channels (**DAC-FFT**, **Acc-FFT**, and **MIC-FFT** in this example) that you want to configure.
2. Click the **Data** tab, and then click the **Function** tab.



3. For this example, configure the settings of this tab as follows:
   a. For **Spectrum Function**, select **AutoSpectrum.**
   b. For **IntegrationType**, select **None**.
   c. For **Display Function**, select **Amplitude**.
4. Click the **Scaling** tab, and configure the settings of this tab as follows:
   a. For **dB/Linear,** select **Linear**.
   b. For **RMS/Peak/PkPk**, select **RMS**.



5. Click the **Averaging** tab, and configure the settings of this tab as follows:
   a. Select the **Enable Averaging** checkbox.
   b. For **Averaging Type**, select **Linear**.
   c. For **Averaging Mode**, select **RMS**.



Add FRF Channels to the Plot and Data Config Window

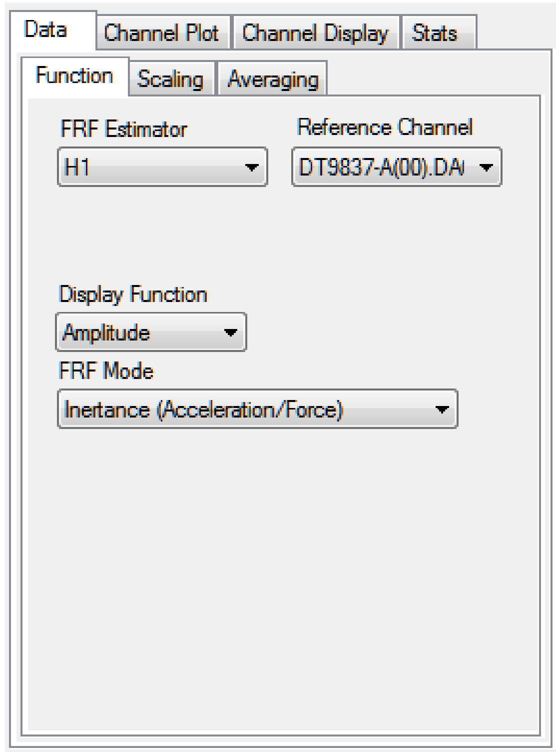
The following steps show how to use the FFT Analyzer interface of QuickDAQ to add two FRF channels to the Plot and Data Config window to calculate the frequency response of analog input channel 0 (DAC) and analog input channel 1 (Acc), and analog input channel 0 (DAC) and analog input channel 2 (MIC).

1. Click the **Add 2 channel FFT function** ( ) button, select **FRF**, and then select **DT9837-A(02).Acc**.
   The Acc-FRF channel is added to the Plot and Data Config window.
2. Click the **Add 2 channel FFT function** ( ) button, select **FRF**, and then select **DT9837-A(02).MIC**.
   The MIC-FRF channel is added to the Plot and Data Config window.

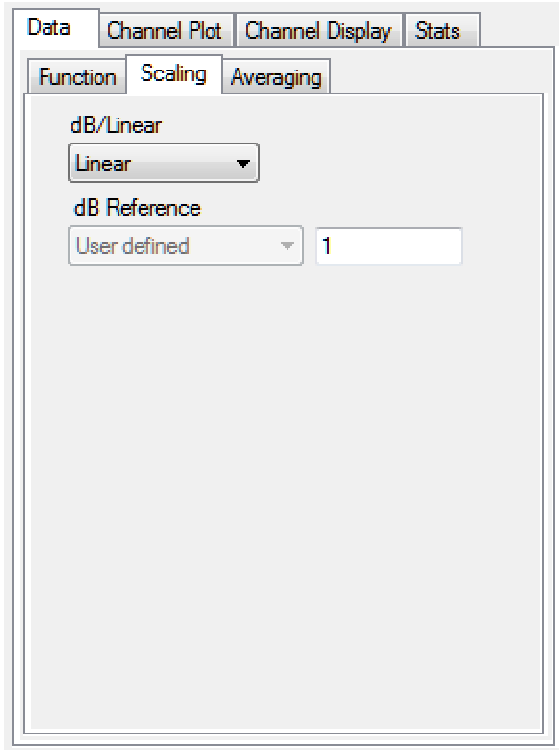
Configure the FRF Channels

The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used to generate the FRF channels for the analog input channels:

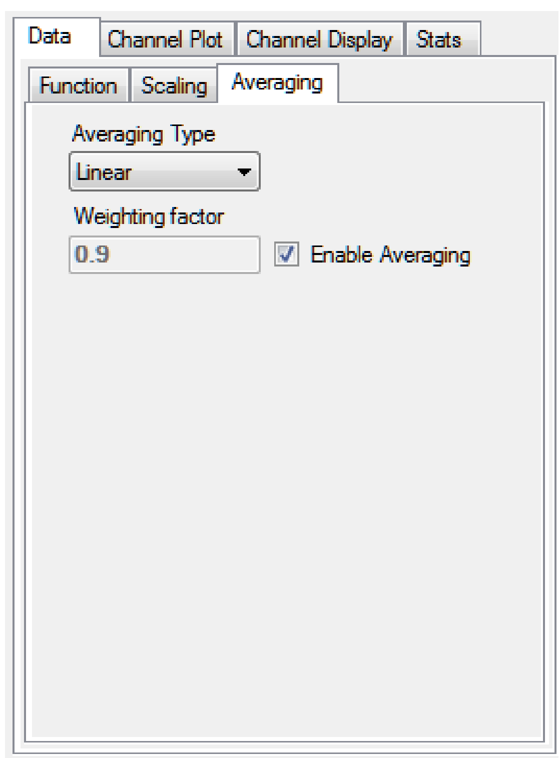
1. Under the **Channel Name** column in the Plot and Data Config window, hold down the **Ctrl** key and select the FRF channels (**Acc-FRF** and **MIC-FRF** in this example) that you want to configure.
2. Click the **Data** tab, and then click the **Function** tab.



3. For the **Acc-FRF** channel, configure the settings of this tab as follows:
   a. For **FRF Estimator**, select **H1** (reduces noise on the response channel).
   b. For **Reference Channel**, select **DAC** (analog input channel 0).
   c. For **Display Function**, select **Amplitude**.
   d. For **FRF Mode**, select **Inertance (Acceleration/Force)**.
4. Click the **Scaling** tab, and select **Linear** for **dB/Linear**:



5. Click the **Averaging** tab, select the **Enable Averaging** checkbox, and select **Linear** for **Averaging Type**:



Add Coherence Channels to the Plot and Data Config Window

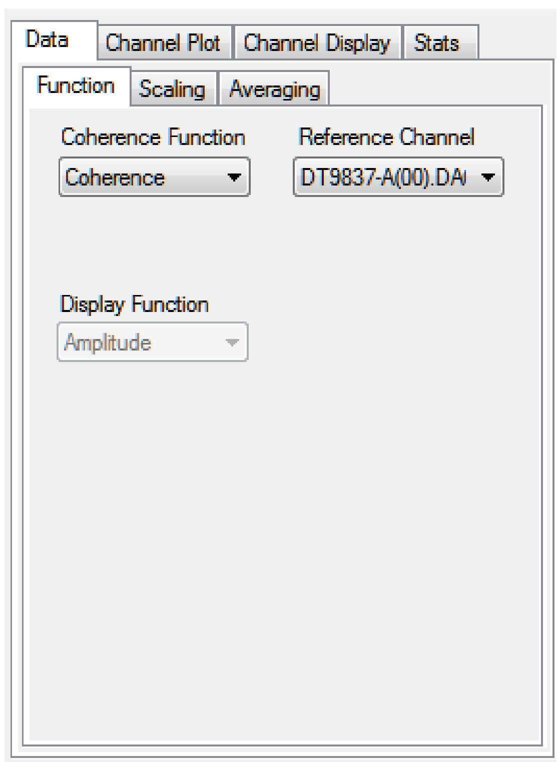
The following steps show how to use the FFT Analyzer interface of QuickDAQ to add Coherence channels to the Plot and Data Config window:

1. Click the **Add 2 channel FFT function** ( ) button, select **Coherence**, and select **DT9837-A(02).Acc**.
   The Acc-Coherence channel is added to the Plot and Data Config window.
2. Click the **Add 2 channel FFT function** ( ) button, select **Coherence**, and select **DT9837-A(02).MIC**.
   The MIC-Coherence channel is added to the Plot and Data Config window.

Configure the Coherence Channel

The following steps show how to use the FFT Analyzer interface of QuickDAQ to configure the parameters used to generate the Coherence channels for the analog input channels:

1. Under the **Channel Name** column in the Plot and Data Config window, hold down the **Ctrl** key and select the Coherence channels (**Acc-Coherence** and **MIC-Coherence** in this example) that you want to configure.
2. Click the **Data** tab, and then click the **Function** tab.

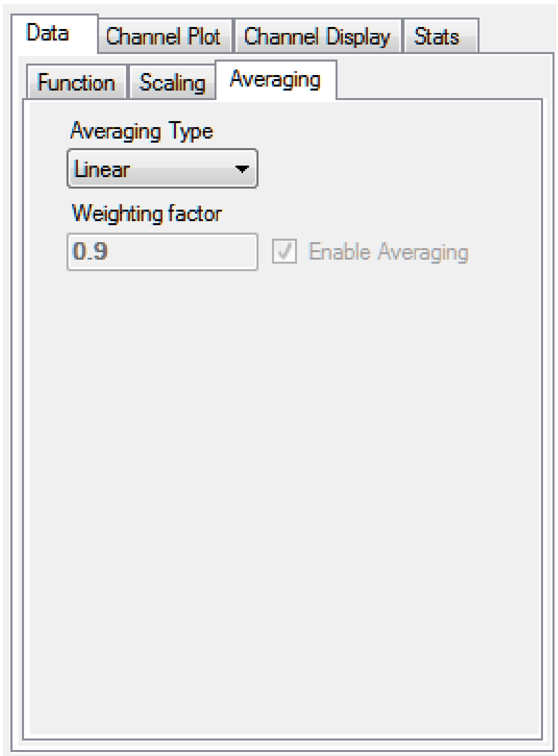


3. For this example, configure the settings of this tab as follows:
   a. For **Coherence Function**, select **Coherence**.
   b. For **Reference Channel**, select **DAC** (analog input channel 0).

Note: **Linear** scaling is selected automatically for Coherence channels under the **Scaling** tab.

---

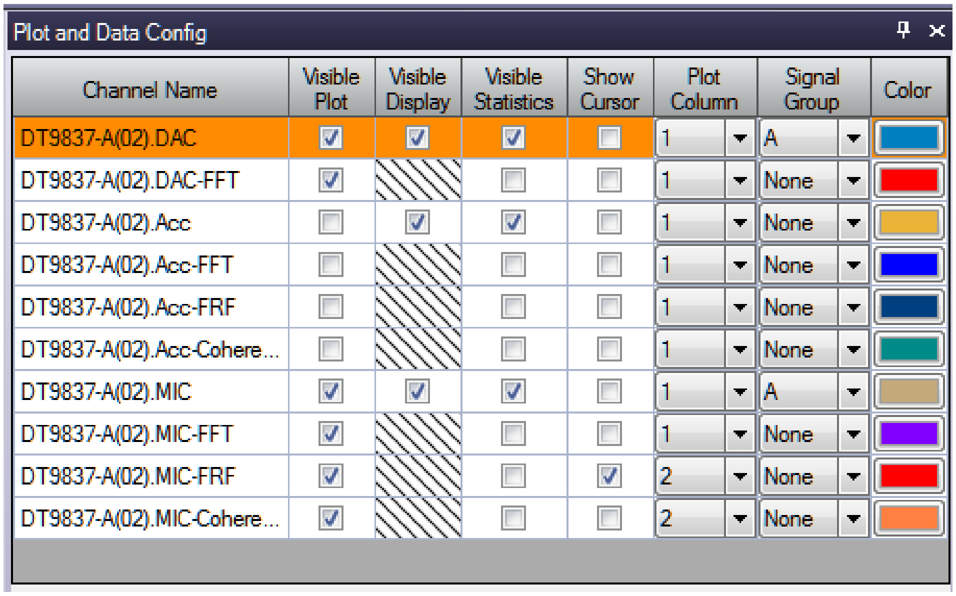
4. Click the **Averaging** tab, and select **Linear** for **Averaging Type**:



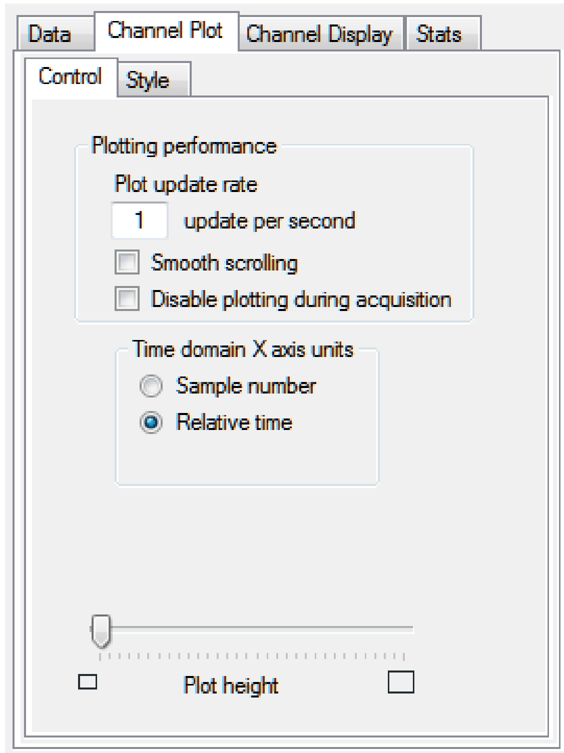
Configure the Appearance of the Channel Plot Window

Perform the following steps to configure the appearance of the Channel Plot window:

1. Select the **Visible Plot** checkbox for the DAC, DAC-FFT, MIC, MIC-Coherence, MIC-FRF, and MIC-FFT channels.



2. Select the **Show Cursor** checkbox for the MIC-FRF channel.
3. Under **Plot Column**, select plot column **1** for the DAC, DAC-FFT, MIC, and MIC-FFT channels, and select plot column **2** for the MIC-Coherence and MIC-FRF channels.
4. Under the **Signal Group** column, select group **A** for the DAC and MIC channels, and leave **None** as the signal group for the rest of the channels.
5. Assign a unique color to each trace for the visible channels, if desired.
6. Click the **Channel Plot - Control** tab.



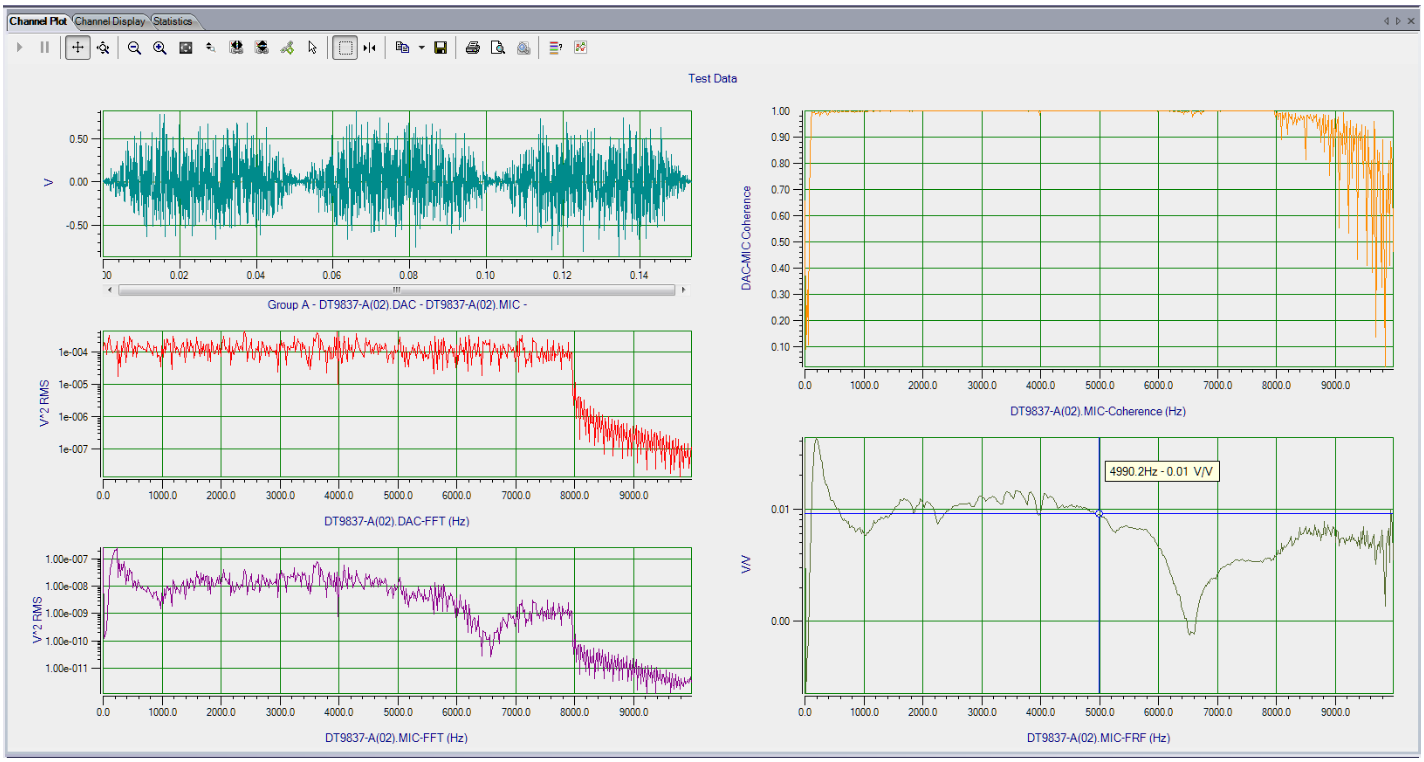
7. Slide the **Plot height** bar to the left until you can see all 5 plots on the Channel Plot window at once.

Start the Simultaneous Burst Random Output and Analog Input Operation

Once you have configured the channels and the parameters for the Channel Plot window, click the **Record** toolbar button (

) or press the **F5** key.

You will hear the waveform three times as it is output from the audio speaker. Results similar to the following are displayed in the Channel Plot window.



In column 1, you can see the signal that is output from the analog output channel (DAC) and read in as analog input channel 0, and the signal that is measured by the microphone (MIC); this is Group A of the first plot. You can also see the FFT of the analog output channel (DAC) and the FFT of the microphone (MIC).

In column 2, you can see the coherence of the microphone (MIC) and the FRF of the microphone (MIC). In this example, the y-axis of the FRF plot is shown in log10 format (right-click the y-axis, select **Y-Axis Setup...**, and select **Log10** for **Lin/Log**).

Note: If desired, you can show the data for the Acc, Acc-FFT, Acc-FRF, and Acc-Coherence channels by selecting the **Visible Plot** checkbox in the Channel Plot window.

Example of Monitoring an Analog Output Signal

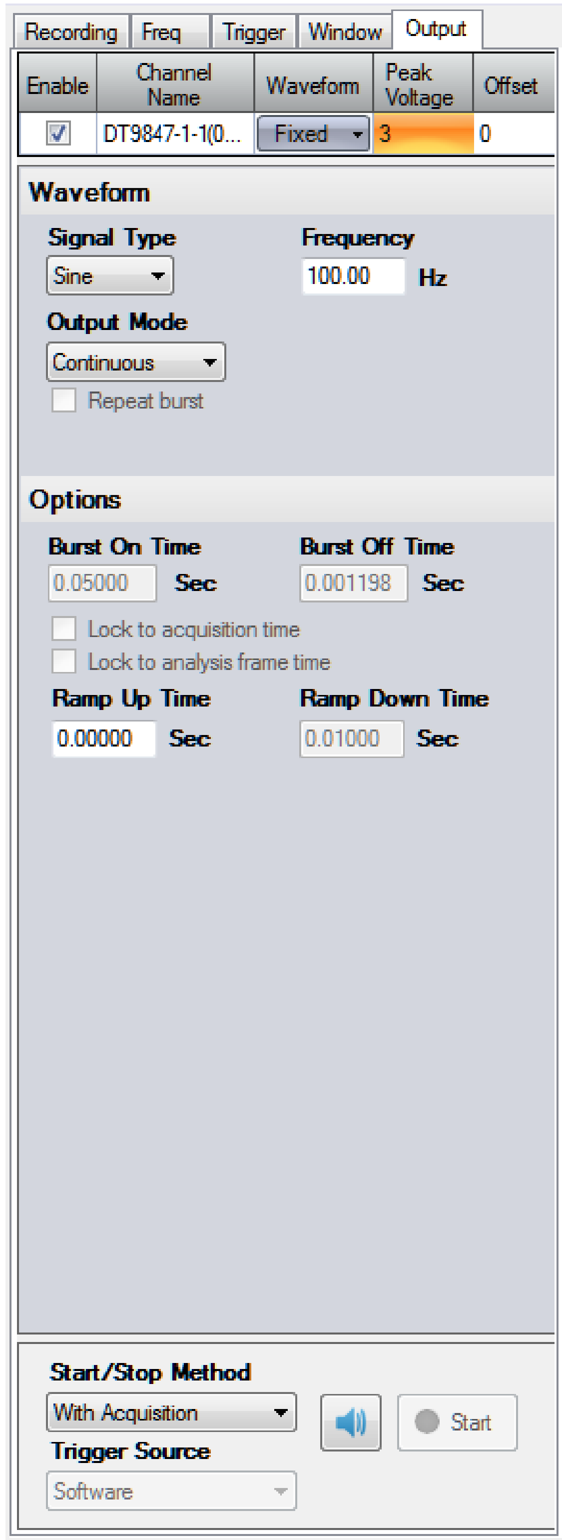
If you are using the FFT Analysis option or Advanced FFT Analysis option of QuickDAQ, you can use the following steps to monitor the output of an analog output signal on the device using an analog input channel.

In this example, a 3 V sine wave is output on analog output channel 0 of a DT9847-1-1 module. The analog output channel is connected to analog input channel 0 on the DT9847-1-1 module. The FFT Analyzer interface is used.

Configure the Analog Output Settings

For this example, configure the analog output channel as follows:

1. Click the **Output** tab of the Acquisition and Config window.



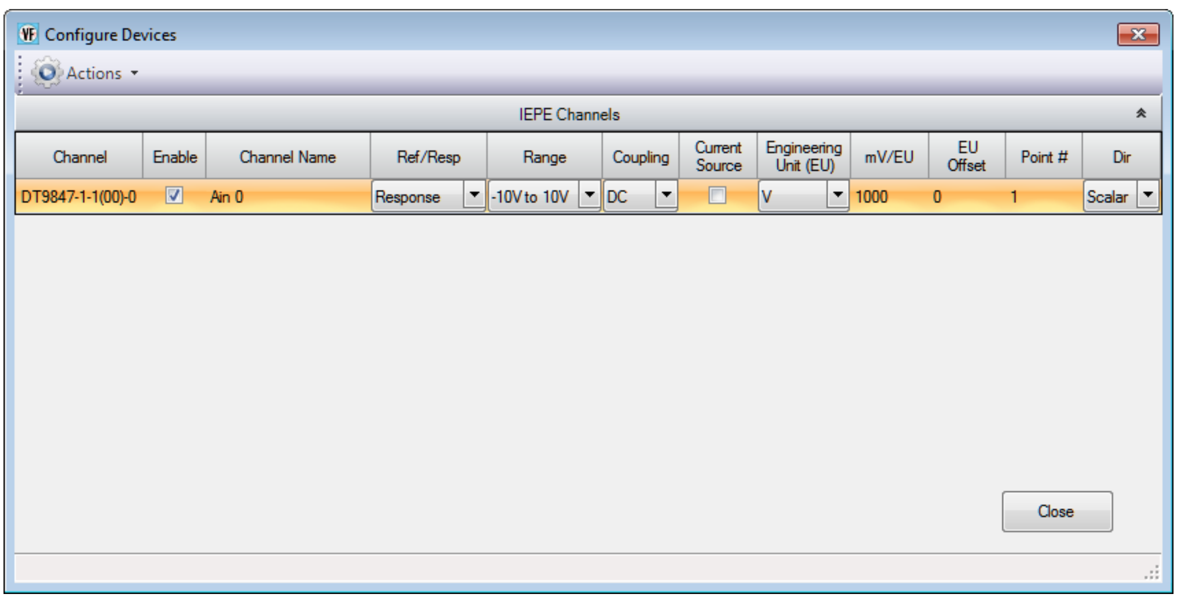
2. Select the **Enable** checkbox to enable the analog output channel on the module.
3. For the **Waveform** type, select **Fixed**.
4. For **Peak Voltage,** enter **3** to output a ±3 V signal.

5. For **Offset**, enter **0**.
6. For **Signal Type**, select **Sine** to output a sine wave.
7. For **Output Mode**, select **Continuous** to output a waveform that repeats continuously.
8. For **Frequency**, select **100 Hz**; this is the frequency of the output waveform.
9. For **Ramp Up Time**, enter **0**.
10. For the **Start/Stop Method**, select **With Acquisition**.
    The analog output operation will start when you start acquisition.
11. If desired, hide the Acquisition Config window by clicking the Auto-Hide pin ( ) in the top, right corner of the window.

Configure the Analog Input Channel

Configure the analog input channel as follows:

1. Ensure that the output from analog output channel 0 is connected to the input of analog input channel 0.
2. Configure the analog input channel by clicking the **Input Channel Configuration** toolbar button ( ) or by clicking the **Configuration** menu and clicking **Input Channel Configuration**.
3. Enable analog input channel 0 by clicking the **Enable** checkbox next to the channel.



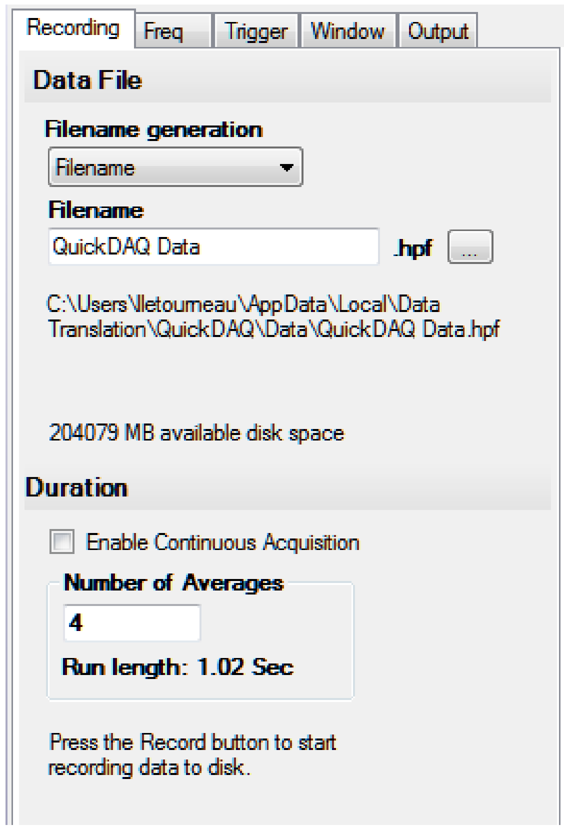
4. Under the **Channel Name** column, leave the default channel name as Ain 0.
5. Under the **Ref/Resp** column, select **Response** for the analog input channe1.
6. Under the **Range** column, select the input range for analog input channel.
   In this example, ***±10 V*** *is used.*

7. Under the **Coupling** column, select the coupling type (AC or DC) for your sensor.
   *Since this example is monitoring a voltage input, **DC** is used.*
8. Under the **Current Source** column, select whether to enable or disable use of the 4 mA current source on the data acquisition device.
   *Since this example is monitoring a voltage input, the **Current Source** checkbox is not checked (disabled) for the analog input channel.*
9. Under the **Engineering Units** column, select the engineering units for the input.
   *In this example, **V** is used.*
10. Enter the number of mV per engineering unit in the **mV/EU** field.
    *In this example, **1000** is used.*
11. If an offset is specified for the input, enter the value in the **EU Offset** field.
    *In this example, no offset (**0**) is specified.*
12. If desired, enter a test point value for the channel.
    *In this example, **0** is used.*
13. If desired, enter a sensor direction for each channel.
    *In this example, **Scalar** is used.*
14. Click **Close** to close the Configure Devices dialog box.

Configure the Recording Settings

For this example, configure the recording settings as follows:

1. Click the **Recording** tab of the Acquisition Config window.

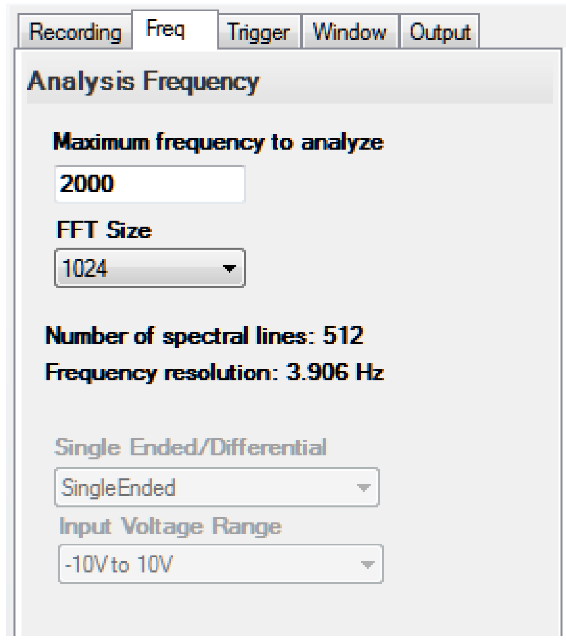


2. For **Filename generation**, use the **Filename** option.
3. For **Filename**, use the default name for the data file.
4. Leave the **Enable Continuous Acquisition** checkbox unchecked.
5. For **Number of Averages**, enter **4** as the number of records to acquired, process, and average before acquisition stops.
   The number of seconds for the total run and the amount of available disk space are shown.

Configure the Frequency Settings

For this example, configure the frequency settings as follows:

1. Click the **Freq** tab of the Acquisition Config window.

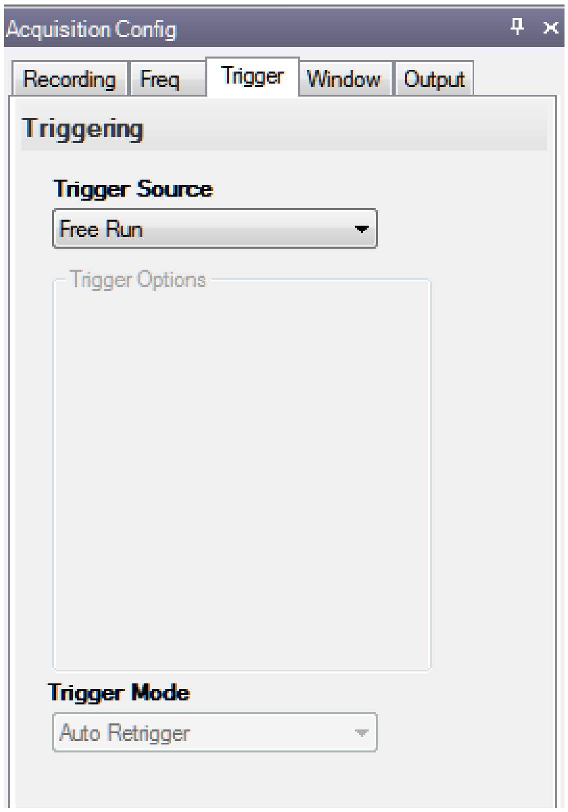


2. In the **Maximum frequency to analyze** text box, enter **2000** as the bandwidth or span, in Hz, of the input signals that you want to analyze
Internally, the sampling rate of the data acquisition device is set to twice the bandwidth, or 4000 Hz in this example.
3. In the **FFT Size** text box, select **1024** from the drop-down combo box as the number of data points to use for the FFT calculation.
The number of spectral lines and the frequency resolution are displayed.

Configure the Trigger Settings

For this example, configure the trigger settings as follows:

1. Click the **Trigger** tab of the Acquisition Config window.

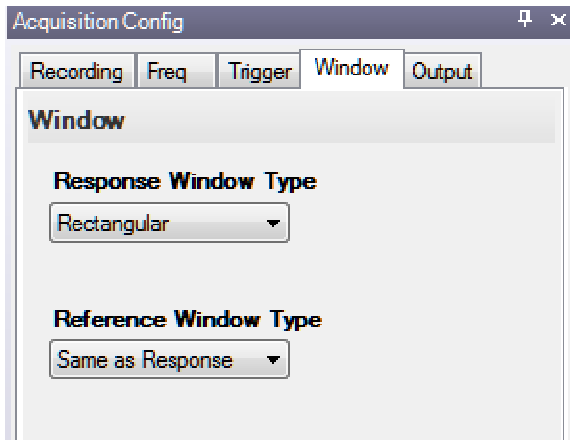


2. For the **Trigger Source**, select **Free Run** to ensure that the measurement starts as soon as the **Record** button is clicked.

Configure the Windowing Function

For this example, configure the windowing function as follows:

1. Click the **Window** tab of the Acquisition and Config window.
   The Window tab for the Advanced FFT Analysis option of QuickDAQ is shown. The FFT Analysis option does not use the response and reference window terms and does not support the reference window type.



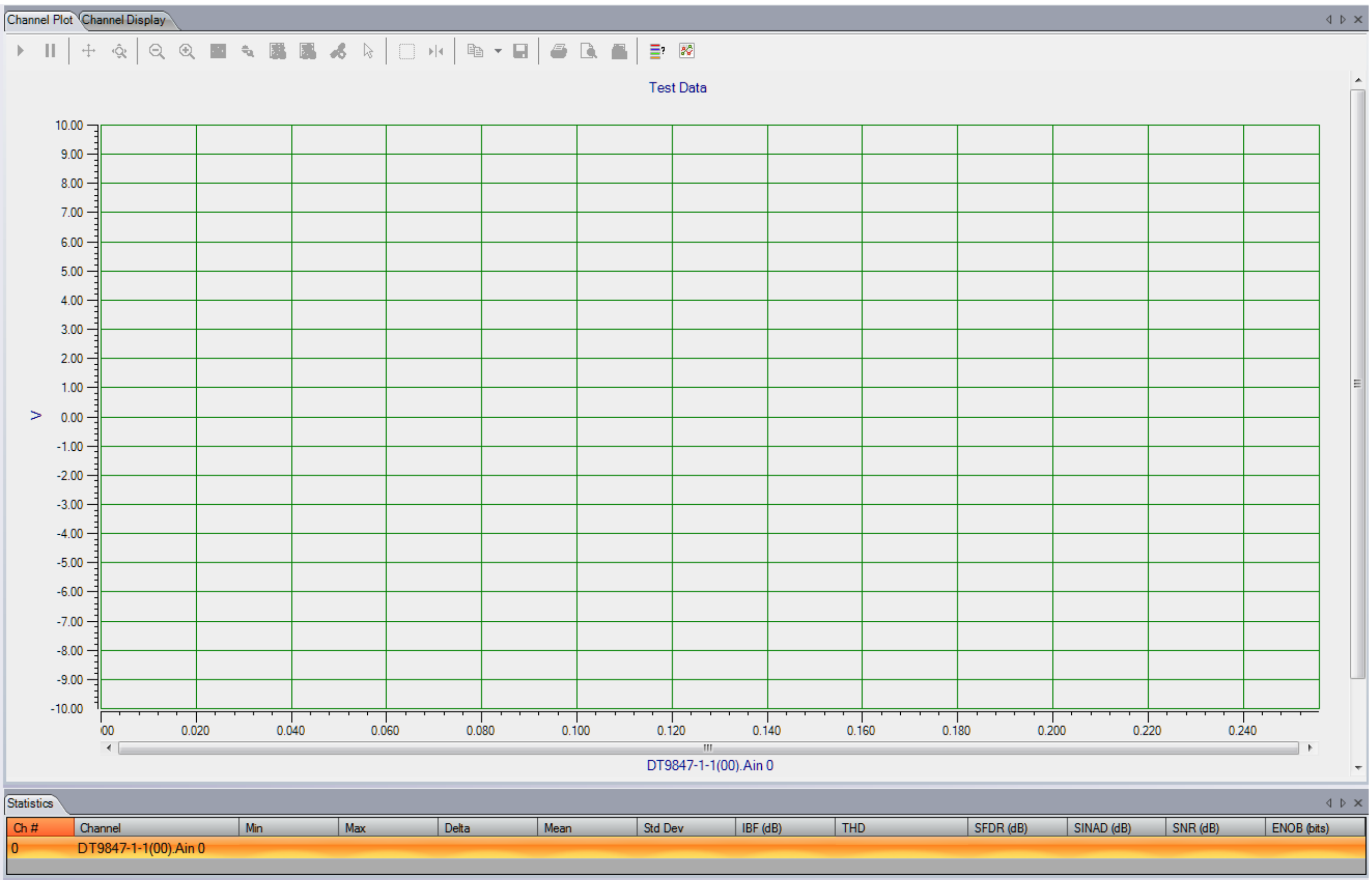
2. For the **Response Window Type** (Advanced FFT Analysis option) or **Window Type** (FFT Analysis option), select **Rectangular**.
   This is a global setting that is applied to the FFTs for all enabled analog input channels.
3. If you are using the Advanced FFT Analysis option, select **Same As Response** for the **Reference Window Type**.
   This example does not use reference channels.
4. If desired, hide the Acquisition Config window by clicking the Auto-Hide pin () in the top, right corner of the window.

Configure the Appearance of the Statistics Window

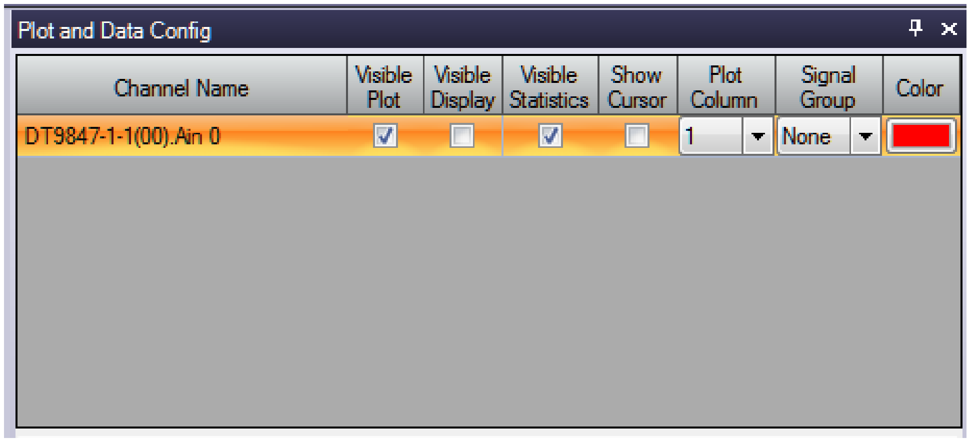
In this example, we want to look at the dynamic performance statistics of the analog output signal that is monitored by the analog input channel.

To configure the appearance of the statistics window, perform the following steps:

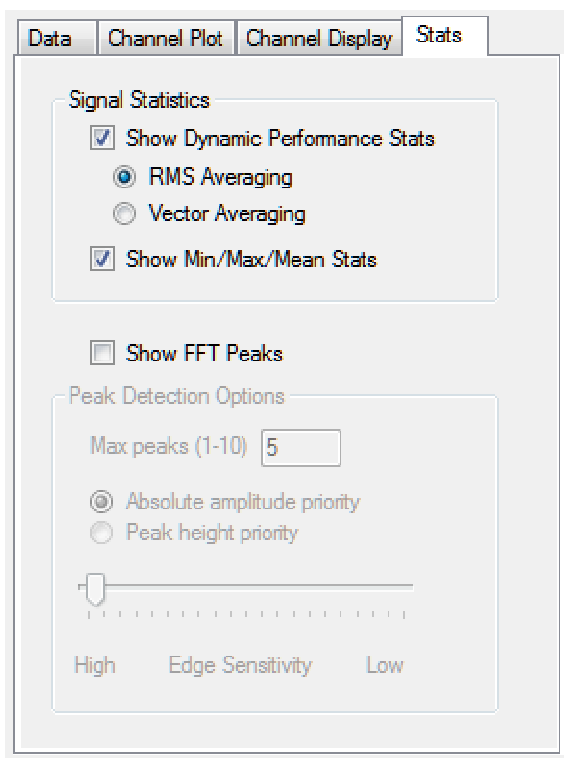
1. Click the **Statistics** tab and drag it to the lower section of the display area under the Channel Plot window, resizing it as desired:



2. In the Plot and Data Config window, check the **Visible Plot** and **Visible Statistics** checkboxes.



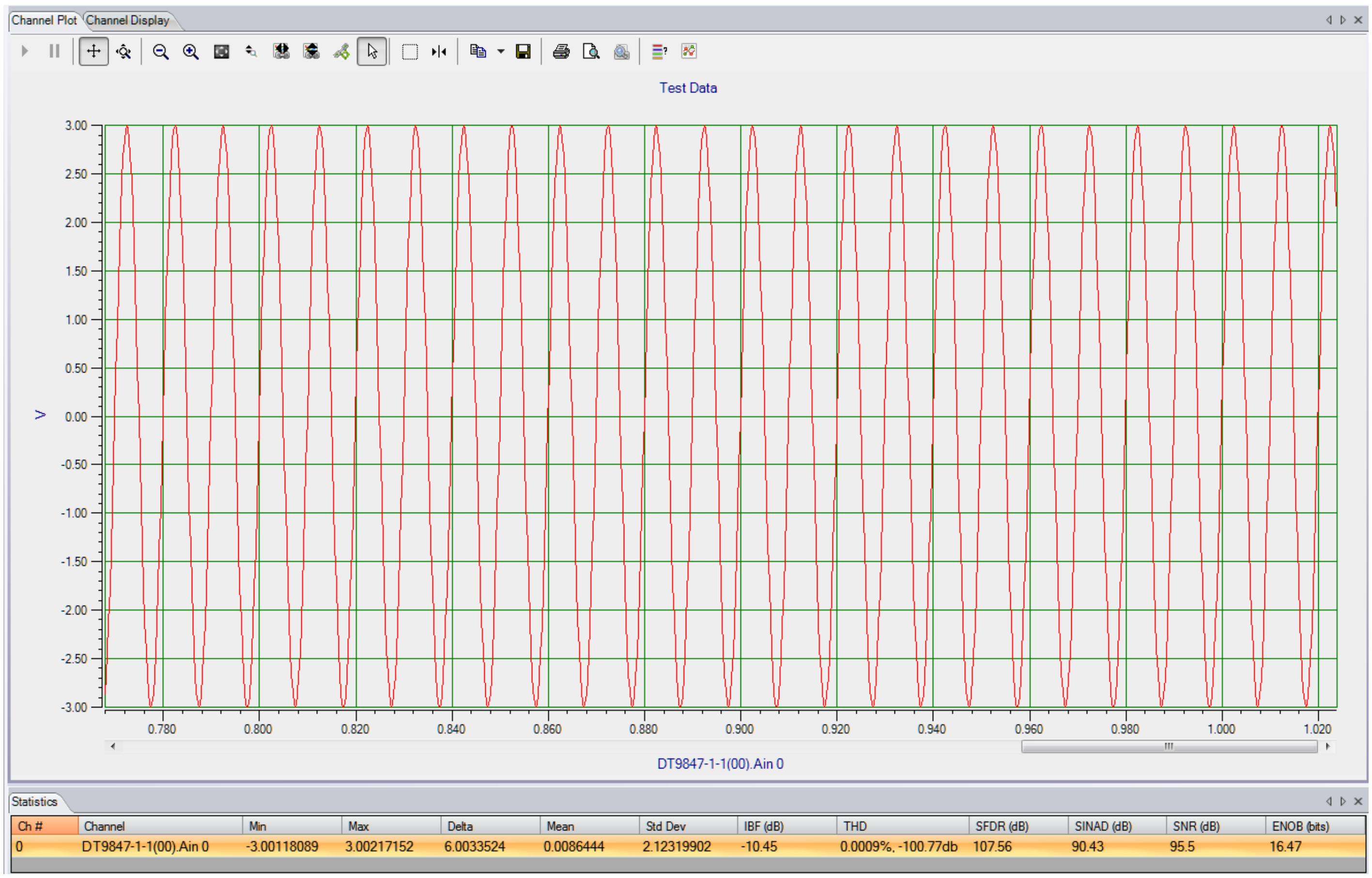
3. Click the Stats tab, and configure it as follows:
   a. Select the **Show Dynamic Performance Stats** checkbox.
   b. Select the **Show Min/Max/Mean Stats** checkbox.



Start the Operation

Once you have configured the channels and the application parameters, click the **Record** toolbar button () or press the **F5** key to start the operation.

Results similar to the following are displayed in the Channel Plot window, showing the output of analog output channel 0 as measured by analog input channel 0, and the statistics for the channel.



Example of Using a Digital Filter

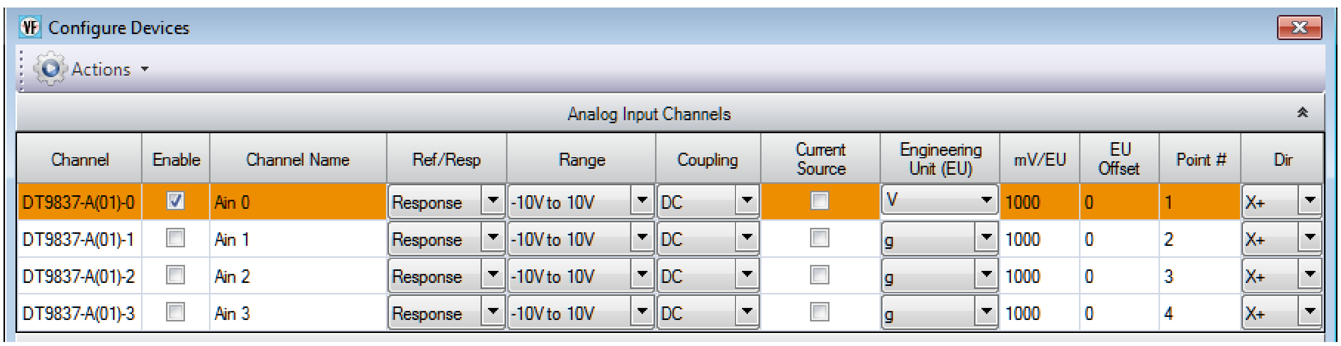
This example shows how to use a low-pass digital filter to eliminate unwanted high-frequency signals from your data.

In this example, a function generator, which is generating a 1 kHz output signal with an amplitude of approximately 6 V, is attached to analog input 0 of a DT9837A module.

Configure the Input Channel

To configure the input channel, perform the following steps:

1. Click the **Input Channel Configuration** toolbar button () or click the **Configuration** menu and click **Input Channel Configuration**.
2. Enable analog input channel 0 by clicking the **Enable** checkbox for this channel.

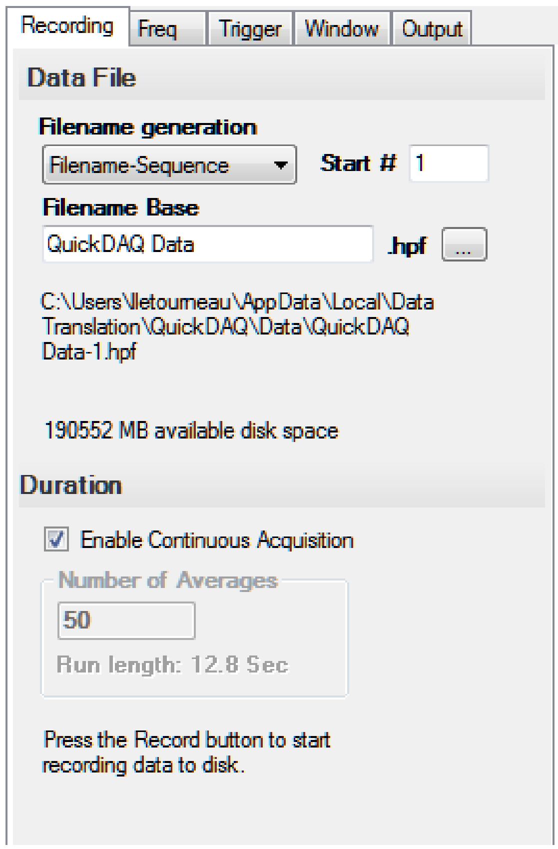


3. Under the **Range** column, select the an input range of ±10 V for the signal.
4. Under the **Engineering Unit (EU)** column, select **V** for voltage.
5. Leave the remainder of the settings at their default values.
6. Click **Close**.

Configure the Recording Settings

For this example, configure the recording settings as follows:

1. Click the **Recording** tab of the Acquisition Config window.

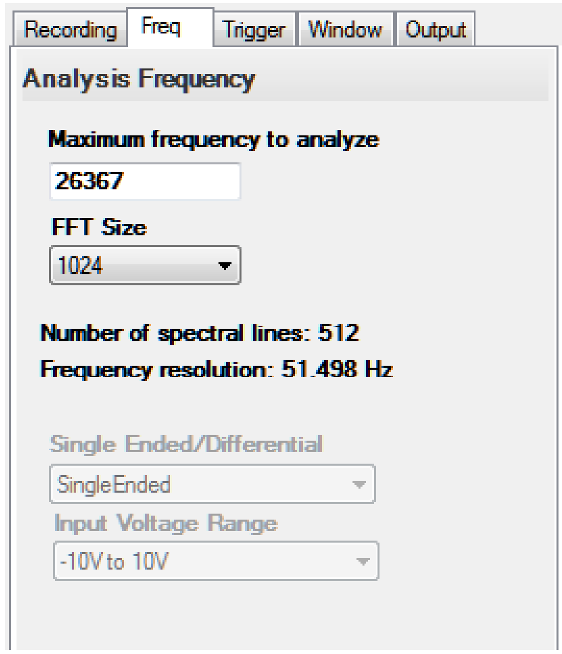


2. For **Filename generation**, use the **Filename-Sequence** option.
3. For **Filename**, use the default name for the data file.
4. Check the **Enable Continuous Acquisition** checkbox.

Configure the Frequency Settings

Configure the frequency settings as follows:

1. Click the **Freq** tab of the Acquisition Config window.

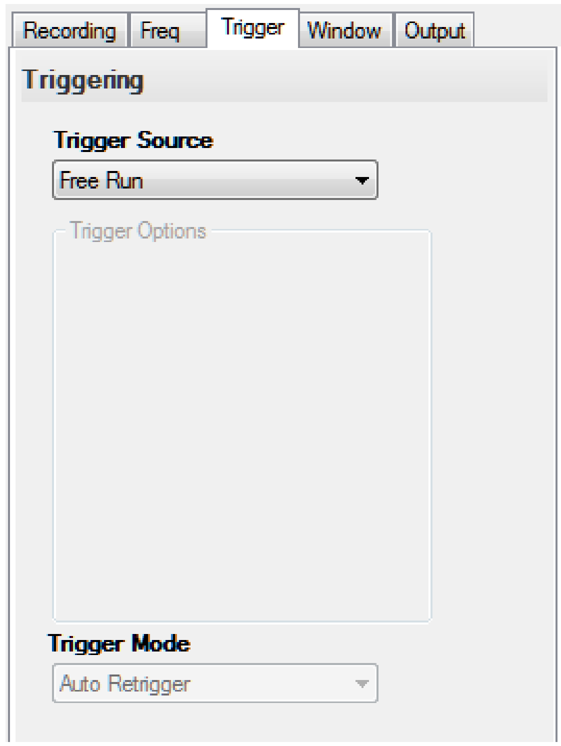


2. In the **Maximum frequency to analyze** text box, enter **26367** as the bandwidth or span, in Hz, of the input signals that you want to analyze
   Internally, the sampling rate of the data acquisition device is set to twice the bandwidth, or the maximum rate of the module (52734 Hz) in this example.
3. In the **FFT Size** text box, select **1024** from the drop-down combo box as the number of data points to use for the FFT calculation.
   The number of spectral lines and the frequency resolution are displayed.

Configure the Trigger Settings

For this example, configure the trigger settings as follows:

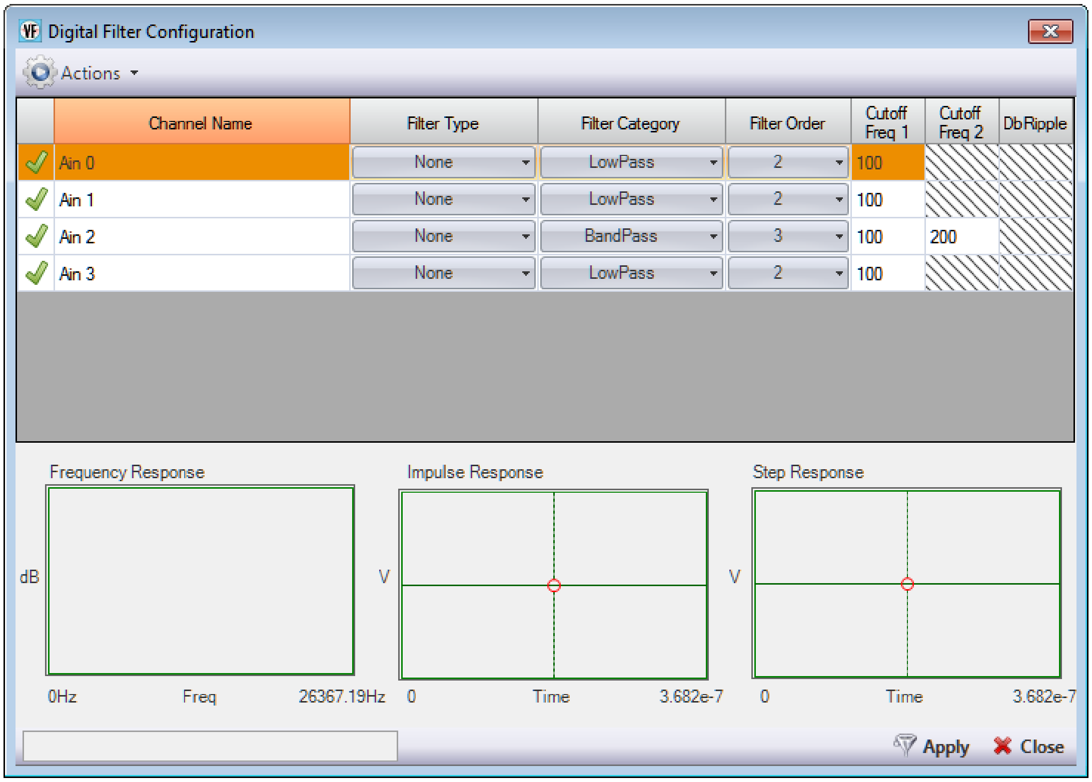
1. Click the **Trigger** tab of the Acquisition Config window.



2. For the **Trigger Source**, select **Free Run** to ensure that the measurement starts as soon as the **Record** button is clicked.
In this mode, the Trigger Mode is not used.

Open the Digital Filter Configuration Dialog

For this example, open the Digital Filter Configuration dialog by clicking the **Digital Filter Configuration** toolbar button ( ) or by clicking the **Configuration** menu and clicking **Digital Filter Configuration**.



For now, leave the default settings for analog input channel 0, so that no digital filter is applied.

Leave this dialog open.

Start the Acquisition

Once you have configured analog input channel 0 and the application parameters, click the **Record** toolbar button () or press the **F5** key to start the operation.

Results similar to the following are displayed in the Channel Plot window, showing the unfiltered 1 kHz signal that is generated by the function generator.

Let acquisition continue to run, and go to the next step.

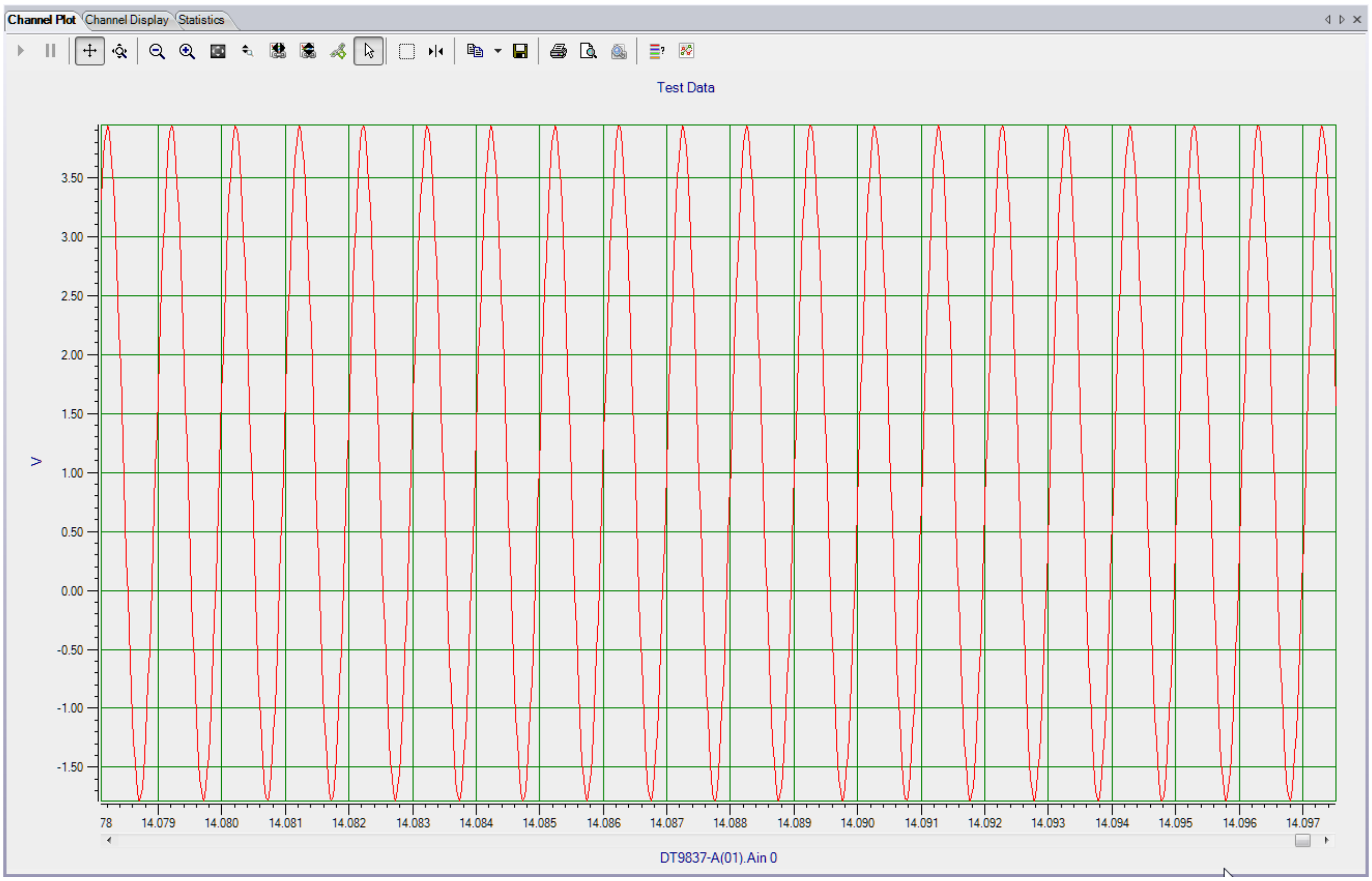
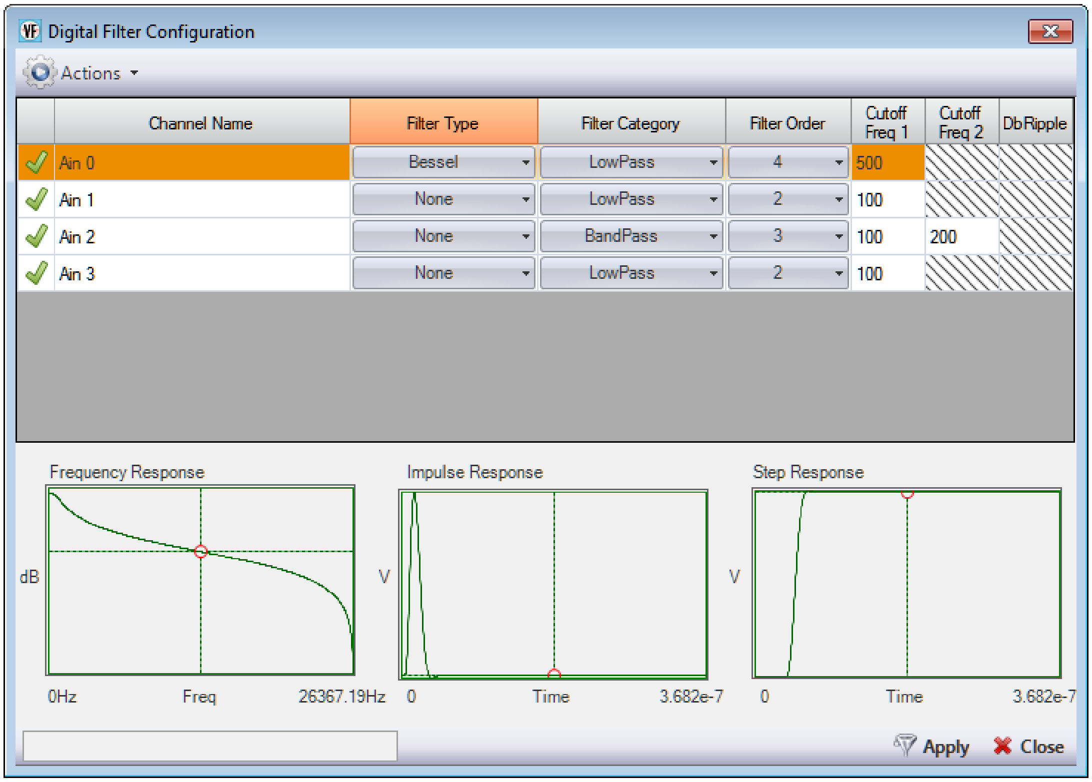


Figure 43: Unfiltered 1 kHz Signal

Configure the Digital Filter Configuration Dialog

While acquisition is running, change the settings in the Digital Filter Configuration dialog to configure a Bessel low-pass filter for analog input channel 0.



Configure the filter as follows:

1. Under the **Filter Type** column, select **Bessel**.
2. Under the **Filter Category** column, select **LowPass**.
3. Under the **Filter Order** column, select **4**.
4. Under the **Cutoff Freq 1** column, enter **500**. This is the frequency above which the signals will be attenuated.
5. Click **Apply**.

The attenuated signal is displayed. In this example, a low-pass filter is used so that all frequencies above 500 Hz are attenuated.

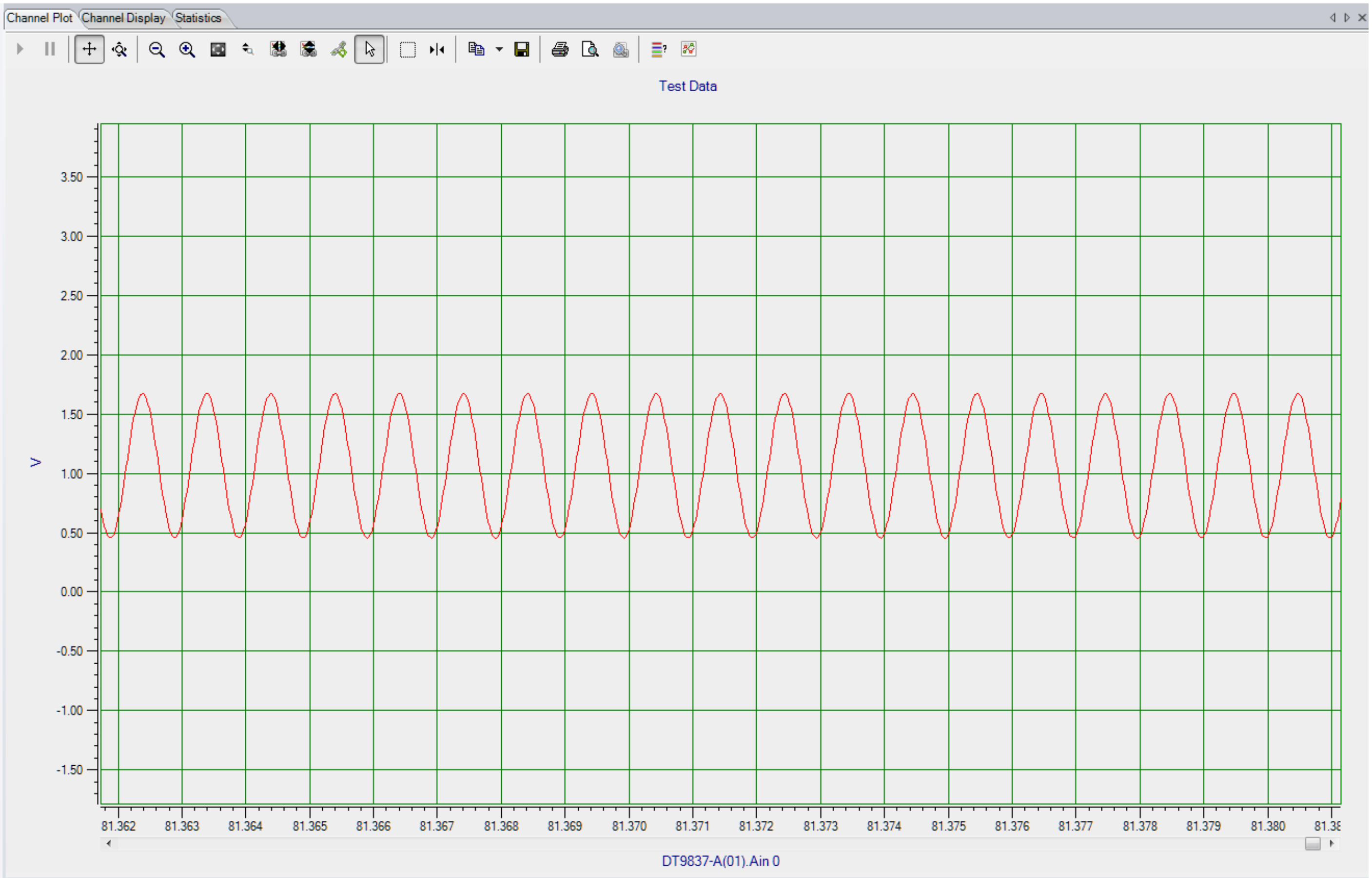


Figure 44: Attenuated 1 kHz Signal Using a Bessel Low-Pass Filter (500 Hz Cutoff Frequency)

9

Support

Troubleshooting

If you are performing an acquisition for a long duration, you may see that acquisition stops if your computer's power options are not set correctly. If this problem occurs, do the following to change the power options of your computer:

1. Go to the Control Panel application for your computer.
2. Select **Power Options**.
3. For the **Put the Computer to Sleep** option, select **Never.**

Should you experience other problems using QuickDAQ, do the following:

1. Read all the appropriate sections of this manual, including any "Read This First" information.
2. Check for a README file on the CD. If present, read this file for the latest installation and usage information.
3. Check that you have installed your data acquisition devices properly. For information, refer to the documentation supplied with your devices.
4. Check that you have installed the latest device drivers for your data acquisition devices. For information, refer to the documentation supplied with your devices.
5. Check that you have installed your software properly. For information, refer to page 26
6. Search the Data Translation web site (www.datatranslation.com) for information about QuickDAQ.
7. Search the Knowledgebase in the Support section of the Data Translation web site (at www.datatranslation.com) for an answer to your problem.

If you are still having problems, Data Translation's Technical Support Department is available to provide technical assistance to licensed users.

Technical Support

If you have any difficulty using QuickDAQ, Data Translation's Technical Support Department is available to provide technical assistance.

To request technical support, go to our web site at **http://www.datatranslation.com** and click on the **Support** link.

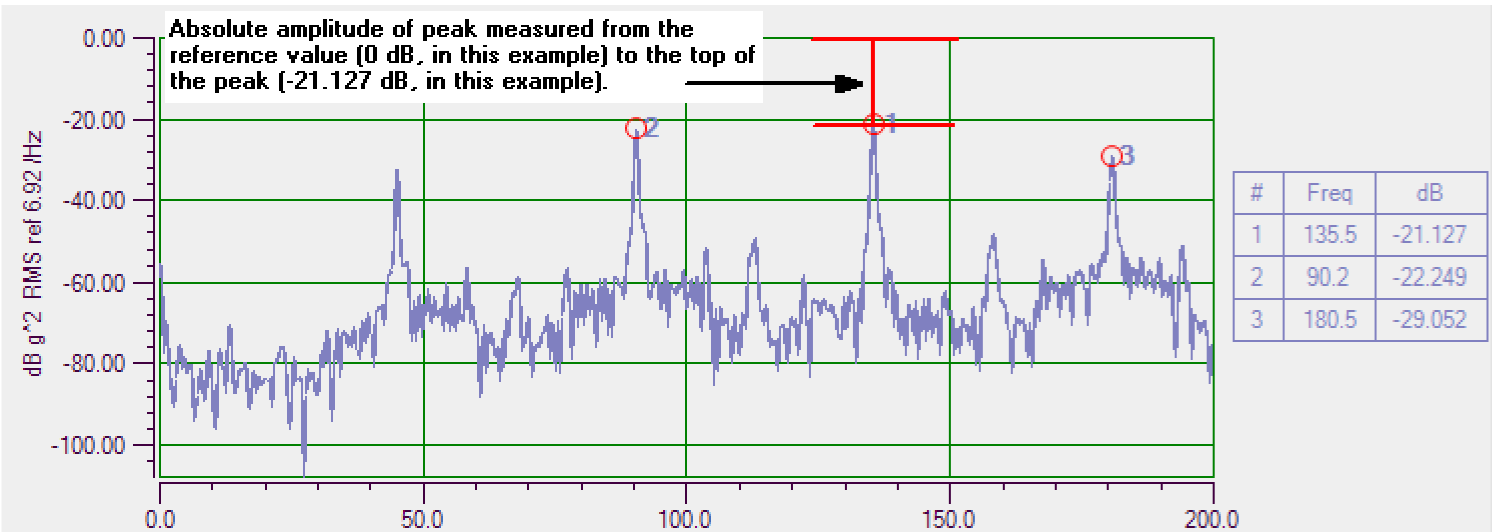
Note: Any unhandled exceptions that occur when running QuickDAQ are captured and displayed in a dialog box; these exceptions are also appended to a file called QuickDAQErrLog.log off the root directory (C:\QuickDAQErrLog.log). If you receive an unhandled exception, please email the QuickDAQErrLog.log file to the Data Translation Technical Support Department at **support@datx.com**.

When requesting technical support, be prepared to provide the following information:

- The version of QuickDAQ that you are using
- The serial number of your data acquisition device

Glossary

Absolute Amplitude of Peak
The largest peak measured from the reference value to the top of the peak. An example follows:



Accelerometer
A type of sensor that produces an output voltage that is proportional to acceleration.

Acquisition
Data from a specified list of analog input, tachometer, and/or counter channels is collected by a data acquisition device at a specified rate.

Amplitude Display Function
Plots the length of the frequency response.

Apparent Mass
An FRF mode that is calculated as the Force of the Reference Channel/Acceleration of the Response Channel. It is the recipriocal function of Inertance.

Auto Spectrum
A single-channel display function that plots the power of a periodic frequency component at a discrete frequency. For an accelerometer, for example, the units for **Auto Spectrum** are represented as g^2. Typically, **Auto Spectrum** is best for narrowband, deterministic signals.

Average
A series of measurements that are combined to create a result. The results from the last completed measurement are combined, point by point, with the results from the previous measurement. The combined results are retained and displayed as they are updated.

Bartlett Window
Triangular window.

Blackman Window
Similar to the Hanning and Hamming windows, this window has slightly wider central lobes and less sideband leakage.

Blackman Harris Window
Similar to Hanning window, this window has a slightly wider main lobe but much better dynamic range. It is optimized to provide the minimum side lobe level.

dB
Decibels. The decibel scale is a transformation of the linear scale into a logarithmic scale. A decibel is defined as follows:

dB = 20 * log10 (Voltage/RefVolt)

where *RefVolt* is the dB Reference value.

dB Reference
The reference value (*RefVolt*) that is used when computing values into the decibel scale:

dB = 20 *log10 (Voltage/RefVolt)

dB Reference is expressed in the selected engineering units, such a *V* for voltage.

Channel
In QuickDAQ, a channel is either a single input of a data acquisition device that is sampled periodically to generate data, the resultant FFT on the data from an input channel, or the result of an FRF, coherence, or cross spectrum function on the data from an input channel.

Chirp
A signal in which the frequency increases (sometimes called an up-chirp) or decreases (sometimes called a down-chirp) with time.

Coherence Function
A two-channel display function that evaluates the linearity between the input (reference) signal and the output (response) signal at each frequency, and generates a value between 0.0 and 1.0, where 0 is consistently not linear and 1.0 is consistently linear.

Compliance
An FRF mode that is calculated as the Displacement of the Response Channel/Force of the Reference Channel.

Cosine Taper
This window allows you to specify the percentage of the period to allow the signal to come through unaltered before smoothing the signal ends to zero.

Cross Spectrum Function
A two-channel display function of the frequency domain. It contains a building block that is used in FRF and other more advanced functions. It is the product of the complex Fourier spectrum of the response channel multiplied by the complex conjugate of the Fourier spectrum of the reference channel.

Dynamic Stiffness

An FRF mode that is calculated as the Force of the Reference Channel/Displacement of the Response Channel.

Double Integration

Double integration results in displacement (the engineering units change and are denoted with "~~").

ENOB

Effective Number of Bits, returned in bits, represents the quality of a digitized signal. The input signal is typically sinusoidal and should be near full-scale to properly characterize the A/D converter and/or system. Refer to page 244 for more information.

EU

Engineering Unit.

Exponential Averaging

With exponential averaging, new FFT data is weighted differently than older FFT data.

You specify the weighting factor to apply, which either increases or decreases the effect of the new FFT data on the average. This is a continuous averaging mode.

Exponential Window

Typically applied to the response channel in impact testing applications, this window is equal to 1.0 at the beginning of the record and decays exponentially to a smaller value at the end of the record. You can specify a value between 0 and 1 as the *Exp Factor.* This value indicates how much to decay the signal, allowing you to tailor the resultant FFT to the relevant data in the record.

FFT

Fast Fourier Transform (FFT). A technique for calculating the frequency components in a time-domain waveform based on digitized voltage measurements. The result is a display of amplitude versus frequency and phase versus frequency.

FFT Window Type

FFT windows are frequency weighting functions that are applied to the time domain data. An FFT window is typically applied to reduce spectral leakage errors that are associated with finite-duration time signals.

By multiplying each data frame by a suitable time-domain weighting window, an FFT window smooths the data, creating peaks in the middle frequencies and decreasing to zero at the edges.

The following table provides general information about each window type to help you choose which to use:

Table 3: FFT Window Types

Window	Description
Rectangular	No window. Recommended for use when your signal content contains random noise (white noise) or closely spaced sine waves.
Hanning	Bell-shaped cosine window typically used to analyze continuous signals. Recommended for use when your signal content contains sine waves, a combination of sine waves, vibration data consisting of narrowband random signals, or unknown content.
Hamming	Bell-shaped cosine window optimized to minimize the maximum (nearest) side lobe. Recommended for use when your signal content consists of closely spaced sine waves.
Bartlett	Triangular window.
Blackman	Similar to the Hanning and Hamming windows, this window has slightly wider central lobes and less sideband leakage.
Blackman Harris	Similar to Hanning window, this window has a slightly wider main lobe but much better dynamic range. It is optimized to provide the minimum side lobe level.
Flat Top	Optimized for measuring peak amplitudes of discrete spectral components that are separated by several spectral lines. Recommended for use when your signal content consists of sine waves and amplitude accuracy is important.
Exponential	Typically applied to the response channel in impact testing applications, this window is equal to 1.0 at the beginning of the record and decays exponentially to a smaller value at the end of the record. You can specify a value between 0 and 1 as the *Exp Factor.* This value indicates how much to decay the signal, allowing you to tailor the resultant FFT to the relevant data in the record.
Cosine Taper	This window allows you to specify the percentage of the period to allow the signal to come through unaltered before smoothing the signal ends to zero.
Force	Typically applied to the reference window in impact testing applications, allows you to specify the percentage of the signal period to which the rectangular window is applied. Data outside the selected percentage is forced to zero. You can enter the percentage of the period at which the rectangular window starts, as well as the percentage of the signal period at which the rectangular window stops. By applying this window to the output of a transducer, you can avoid collecting extraneous signals, such as forces from the retraction of a hammer, that are not important to the measurement.

Flat Top Window
Optimized for measuring peak amplitudes of discrete spectral components that are separated by several spectral lines. Recommended for use when your signal content consists of sine waves and amplitude accuracy is important.

Force Window
Typically applied to the reference window in impact testing applications, allows you to specify the percentage of the signal period to which the rectangular window is applied. Data outside the selected percentage is forced to zero.

You can enter the percentage of the period at which the rectangular window starts, as well as the percentage of the signal period at which the rectangular window stops.

By applying this window to the output of a transducer, you can avoid collecting extraneous signals, such as forces from the retraction of a hammer, that are not important to the measurement.

Frequency Domain

When plotted, amplitudes are graphed in relation to frequency. Also see *Time Domain*.

FRF (Frequency Response Function)

A two-channel function that represents the complex ratio between the output (response) signal and the input (reference) signal as a function of frequency. By choosing a FRF estimator, you can determine how the FRF is calculated.

FRF Estimator

The **FRF Estimator** determines how the frequency response is calculated. You can choose one of the following settings for **FRF Estimator**:

- **H1 –** Reduces noise on the response channel. It is calculated as follows:

  $H_1 = S_{xf}/S_{ff}$

  where,

  S_{xf} represents the average cross power spectrum between the response and the reference signals.

  S_{ff} represents the average auto power spectrum of the reference signal.

- **H2** – Reduces noise on the reference channel. It is calculated as follows:

  $H_2 = S_{xx}/S_{fx}$

  where,

  S_{xx} represents the average auto spectrum of the response signal.

  S_{fx} represents the average cross power spectrum between the response and the reference signals.

- **H3 –** Reduces noise on both the reference and response channels. H3 is computed as follows:

  $H_3 = (H_1 + H_2)/2$

g

Acceleration in relation to earth's gravity, where 1g = 980 cm/s^2 or 32.2 ft/s^2.

Gain

The factor by which to amplify a signal to allow greater resolution.

Hamming Window

Bell-shaped cosine window optimized to minimize the maximum (nearest) side lobe. Recommended for use when your signal content consists of closely spaced sine waves

Hanning Window

Bell-shaped cosine window typically used to analyze continuous signals. Recommended for use when your signal content contains sine waves, a combination of sine waves, vibration data consisting of narrowband random signals, or unknown content.

H1

This FRF Estimator mode reduces noise on the response channel. It is calculated as follows:

It is calculated as follows:

$$H_1 = S_{xf}/S_{ff}$$

where,

S_{xf} represents the average cross power spectrum between the response and the reference signals.

S_{ff} represents the average auto power spectrum of the reference signal.

H2

This FRF Estimator mode reduces noise on the reference channel. It is calculated as follows:

$$H_2 = S_{xx}/S_{fx}$$

where,

S_{xx} represents the average auto spectrum of the response signal.

S_{fx} represents the average cross power spectrum between the response and the reference signals.

H3

This FRF Estimator mode reduces noise on both the reference and response channels. H3 is computed as follows:

$$H_3 = (H_1 + H_2)/2$$

IBF

Input below full-scale, in dB, represents the level of the main signal component relative to the full-scale voltage. Refer to page 237 for more information.

Inertance

An FRF mode that is calculated as the Acceleration of the Response Channel/Force of the Reference Channel.

Impedance

An FRF mode that is calculated as the Force of the Reference Channel/Velocity of the Response Channel.

Integration

Integration is calculated by dividing each element of the function by (jw)^n, where *j* is the square root of -1; *w* is the product of 2 pi times the frequency of the block element; and *n* is an integer from +2 to -2.

If you specify an accelerometer as the sensor type, single integration results in velocity (the engineering units change and are denoted with "~"), and double integration results in displacement (the engineering units change and are denoted with "~~").

Linear

Data in the FFT is plotted in the engineering units of the specified signal, such as *V* for voltage or g for acceleration.

Linear Averaging

Data from each FFT is averaged with the data from the other FFTs; all data contributes equally to the average.

Magnitude

The FFT of a real signal is a complex number, having a real and an imaginary part. The magnitude is the square root of the sum of the squares of the real and imaginary parts.

Mobility

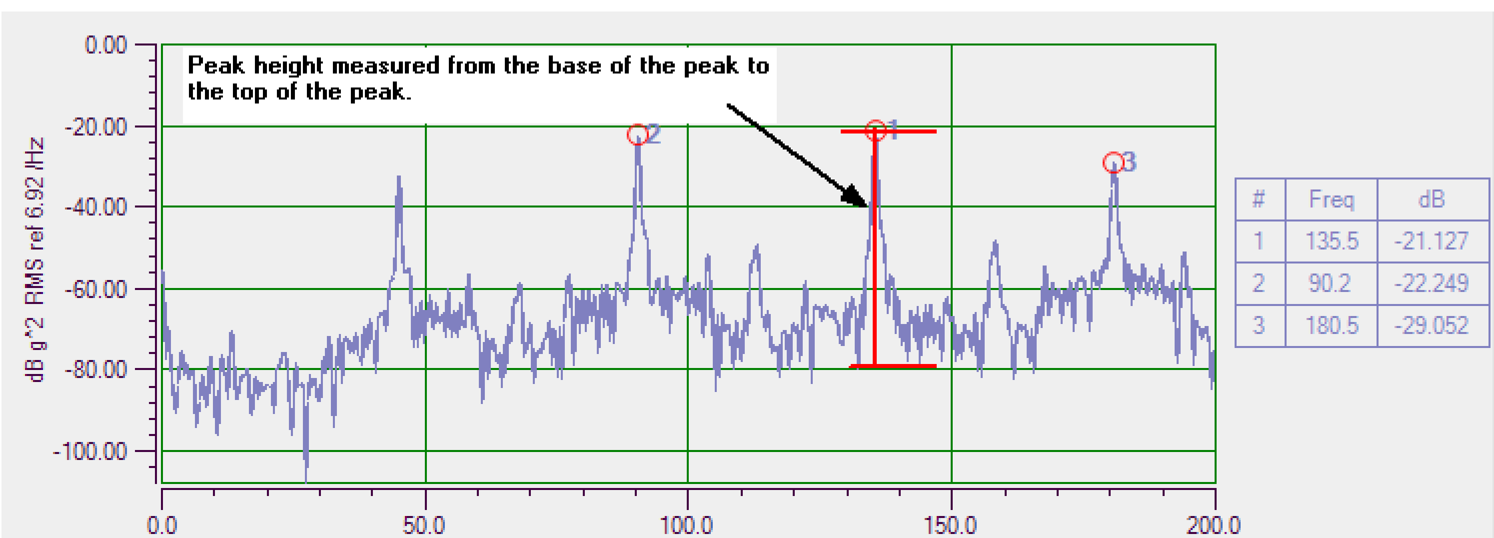
An FRF mode that is calculated as the Velocity of the Response Channel/Force of the Reference Channel.

Nyquist Display Function

This display function plots the resultant data in complex form, with real components on the x-axis and imaginary components on the y-axis. It used to assess the stability and performance of a closed loop system by analyzing the contour of the frequency response function on the complex plane.

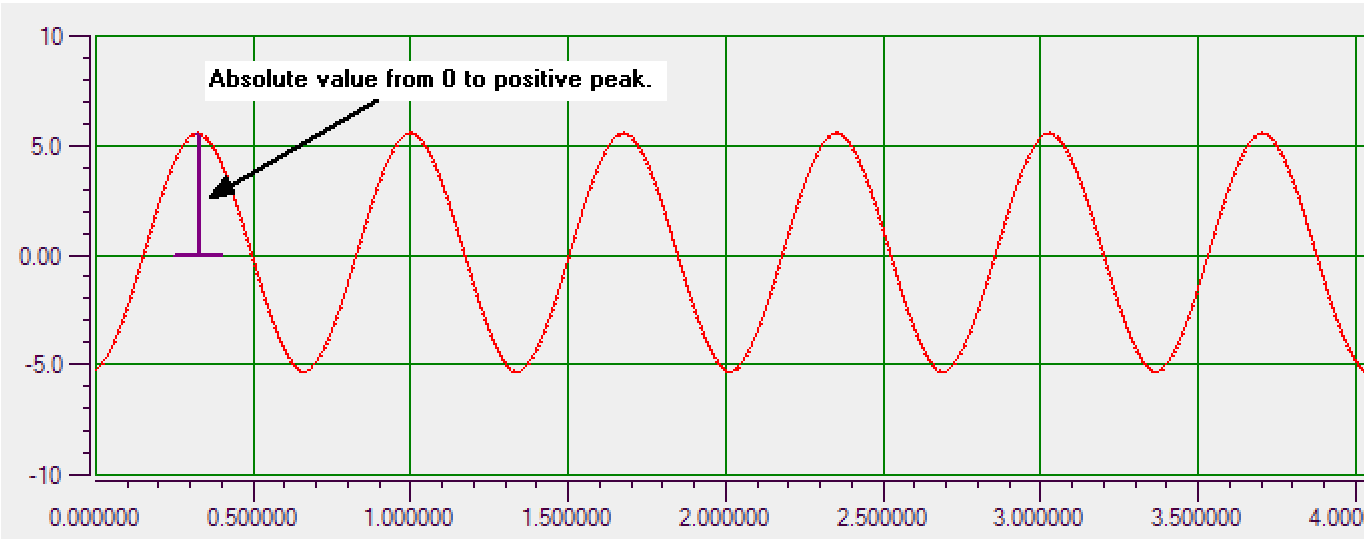
Peak Height

The largest peak measured from the bottom of the peak to the top of the peak:



Peak Scaling

The absolute value of the data from zero to the peak.



For example, the FFT plot for the above signal, appears as follows, where the peak height is just over 5 V:

Peak Hold Averaging

Each new FFT data point is compared with the current FFT data point, and the highest amplitude (peaks) from each FFT are retained. This is a continuous operation that combines the results of several measurements into a final FFT measurement.

Phase Display Function

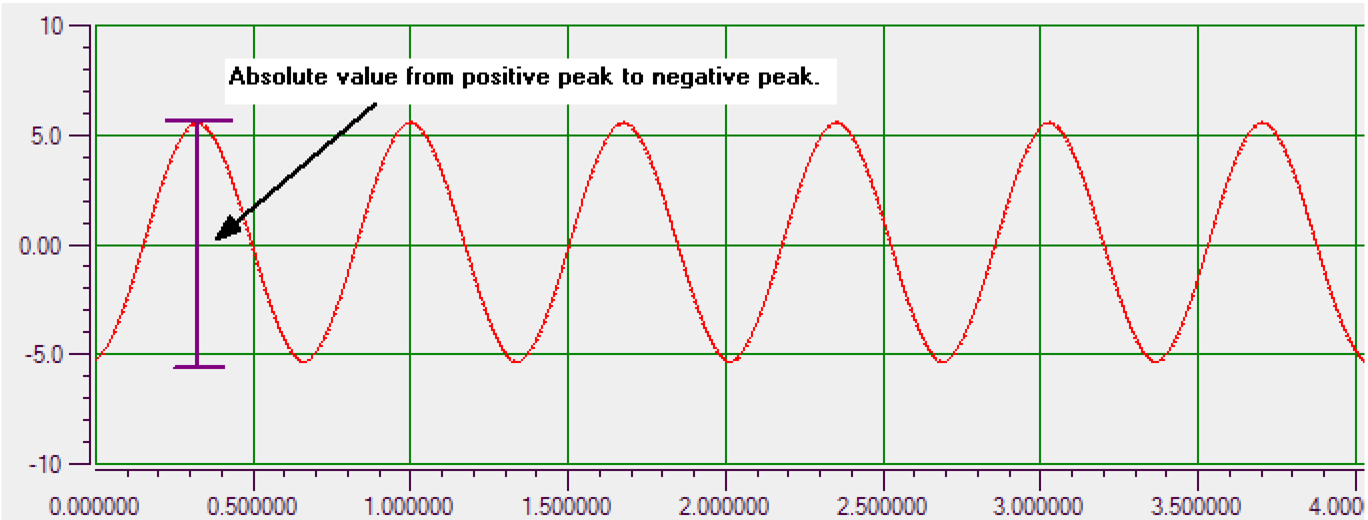
Plots the frequency response phase shift (angle).

Pink Noise

Pink noise is sometimes referred to as 1/f or flicker noise. In the frequency domain, the response is not flat. The power spectral density is inversely proportional to the frequency. Therefore, as the frequency increases, the power density decreases.

Pk-Pk Scaling

The absolute value of the data from the negative peak to the positive peak.



Power

The power in each frequency component represented by the FFT is calculated by squaring the magnitude of the frequency component.

Power Spectral Density (PSD)

The power of random vibration intensity as "mean-square acceleration per frequency unit." The spectrum is computed by squaring the magnitude of each frequency component in the FFT, and dividing this number by the change in frequency multiplied by the equivalent noise bandwidth of the windowing function (ENBW). For an accelerometer, for example, the units for PSD are represented as g^2/Hz. Typically, **PSD** is best for wideband, continuous signals.

Ramp Waveform

A non-sinusoidal waveform that ramps up and then drops sharply. This is sometimes called a sawtooth waveform.

Random Noise

White noise that is created with a software generator function. With white noise, the signal has a flat response in the frequency domain; it has an equal power in any band of a given bandwidth (power spectral density). In the time domain, the amplitudes of the signal form a flat histogram since all amplitude values are equally probable.

Random Normal Noise

White noise whose amplitude in the time domain has a Gaussian or normal distribution. The frequency content is the same as the Random noise type, having a flat response in the frequency domain.

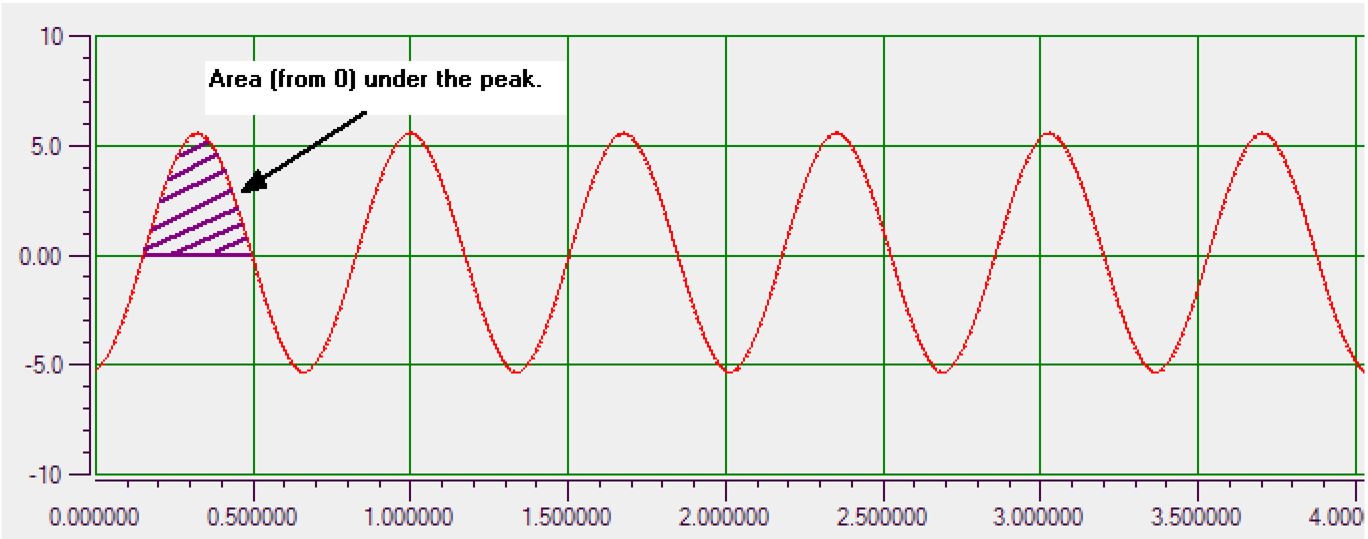
Record

A block of data whose length is equal to the size of the FFT.

Rectangular Window

No window. Recommended for use when your signal content contains random noise (white noise) or closely spaced sine waves.

RMS
Root Mean Square. The square root of the average of the square of the value of the function taken throughout one period.



SFDR
Spurious free dynamic range, returned in dB, represents the level of the full-scale range to the greatest noise or distortion component. Refer to page 240 for more information.

SINAD
Signal-to-noise and distortion ratio, returned in dB, represents the level of the main signal component relative to the total noise, V_{RMS}, including harmonics. Refer to page 241 for more information.

Sine Waveform
A sinusoidal, periodic waveform.

Single Integration
If you specify an accelerometer as the sensor type, single integration results in velocity (the engineering units change and are denoted with "~").

SNR
Signal-to-noise ratio, returned in dB, represents the level of the main signal component relative to the total noise, V_{RMS}. Refer to page 243 for more information.

Spectral Density

The spectral density describes how the variance of time-series data is distributed with frequency. It is computed as follows:

$$\frac{\text{Spectrum}}{\text{Change in Frequency x ENBW}}$$

where,

ENBW is the Equivalent Noise Bandwidth of the Windowing function.

For example, if you select **PSD (Power Spectral Density)** for the Spectrum Function setting when configuring the FFT parameters, the spectrum is computed by squaring the magnitude of each frequency component in the FFT, dividing this number by the change in frequency multiplied by the *ENBW*, and plotting the results. For example, if measuring acceleration with these settings, the units for power spectral density are represented as g^2/Hz.

Spectral Leakage

When the signal being analyzed is not periodic, energy from a signal at a specified frequency can leak into nearby spectral bins causing spectral amplitude inaccuracies; this is called spectral leakage.

Applying a windowing function controls, but doesn't completely eliminate, the error by multiplying each data frame by a suitable time-domain weighting window.

Spectrum

The frequency spectrum is a representation of a time-domain signal in the frequency domain. It is generated by performing an FFT on the signal.

For example, if you select **Spectrum** for the Spectrum Function setting when configuring the FFT parameters, the spectrum is computed by taking the magnitude of each frequency component in the FFT and plotting the results. For example, if measuring acceleration with these settings, the units are in g. Typically, **Spectrum** is best for narrowband, deterministic signals.

Square Waveform

A non-sinusoidal waveform that is shaped like a square, alternating between two levels.

Sweep Waveforms

A sine wave whose frequency changes over time. Swept sine waves provide the ability to test a device over a wide frequency range to determine if any resonances occur.

THD

Total harmonic distortion, returned in dB and %, represents the level of the harmonic distortion of the first five harmonics relative to the main signal component. Refer to page 238 for more information.

Time Domain

When plotted, amplitudes are graphed in relation to time. Also see *Frequency Domain*.

Triangle Waveform
A non-sinusoidal waveform that is shaped like a triangle.

Trigger
An event that starts data acquisition. QuickDAQ supports a number of trigger types, including a software trigger, external TTL trigger, and a hardware or software threshold trigger.

Index

A

D

E

F

G

H

I

L

M

N

O

P

Q

R

U

V

W

X

Y

Z