SignalExpress™

Getting Started with SignalExpress

NATIONAL INSTRUMENTS™

Worldwide Technical Support and Product Information

`ni.com`

Worldwide Offices

Visit `ni.com/niglobal` to access the branch office websites, which provide up-to-date contact information, support phone numbers, email addresses, and current events.

National Instruments Corporate Headquarters

11500 North Mopac Expressway   Austin, Texas 78759-3504   USA   Tel: 512 683 0100

For further support information, refer to the *Technical Support and Professional Services* appendix. To comment on National Instruments documentation, refer to the National Instruments website at `ni.com/info` and enter the Info Code `feedback`.

Important Information

Members of the National Instruments Alliance Partner Program are business entities independent from National Instruments and have no agency, partnership, or joint-venture relationship with National Instruments.

Patents

For patents covering National Instruments products/technology, refer to the appropriate location: **Help»Patents** in your software, the `patents.txt` file on your media, or the *National Instruments Patent Notice* at ni.com/patents.

Export Compliance Information

Refer to the *Export Compliance Information* at `ni.com/legal/export-compliance` for the National Instruments global trade compliance policy and how to obtain relevant HTS codes, ECCNs, and other import/export data.

WARNING REGARDING USE OF NATIONAL INSTRUMENTS PRODUCTS

(1) NATIONAL INSTRUMENTS PRODUCTS ARE NOT DESIGNED WITH COMPONENTS AND TESTING FOR A LEVEL OF RELIABILITY SUITABLE FOR USE IN OR IN CONNECTION WITH SURGICAL IMPLANTS OR AS CRITICAL COMPONENTS IN ANY LIFE SUPPORT SYSTEMS WHOSE FAILURE TO PERFORM CAN REASONABLY BE EXPECTED TO CAUSE SIGNIFICANT INJURY TO A HUMAN.

(2) IN ANY APPLICATION, INCLUDING THE ABOVE, RELIABILITY OF OPERATION OF THE SOFTWARE PRODUCTS CAN BE IMPAIRED BY ADVERSE FACTORS, INCLUDING BUT NOT LIMITED TO FLUCTUATIONS IN ELECTRICAL POWER SUPPLY, COMPUTER HARDWARE MALFUNCTIONS, COMPUTER OPERATING SYSTEM SOFTWARE FITNESS, FITNESS OF COMPILERS AND DEVELOPMENT SOFTWARE USED TO DEVELOP AN APPLICATION, INSTALLATION ERRORS, SOFTWARE AND HARDWARE COMPATIBILITY PROBLEMS, MALFUNCTIONS OR FAILURES OF ELECTRONIC MONITORING OR CONTROL DEVICES, TRANSIENT FAILURES OF ELECTRONIC SYSTEMS (HARDWARE AND/OR SOFTWARE), UNANTICIPATED USES OR MISUSES, OR ERRORS ON THE PART OF THE USER OR APPLICATIONS DESIGNER (ADVERSE FACTORS SUCH AS THESE ARE HEREAFTER COLLECTIVELY TERMED "SYSTEM FAILURES"). ANY APPLICATION WHERE A SYSTEM FAILURE WOULD CREATE A RISK OF HARM TO PROPERTY OR PERSONS (INCLUDING THE RISK OF BODILY INJURY AND DEATH) SHOULD NOT BE RELIANT SOLELY UPON ONE FORM OF ELECTRONIC SYSTEM DUE TO THE RISK OF SYSTEM FAILURE. TO AVOID DAMAGE, INJURY, OR DEATH, THE USER OR APPLICATION DESIGNER MUST TAKE REASONABLY PRUDENT STEPS TO PROTECT AGAINST SYSTEM FAILURES, INCLUDING BUT NOT LIMITED TO BACK-UP OR SHUT DOWN MECHANISMS. BECAUSE EACH END-USER SYSTEM IS CUSTOMIZED AND DIFFERS FROM NATIONAL INSTRUMENTS' TESTING PLATFORMS AND BECAUSE A USER OR APPLICATION DESIGNER MAY USE NATIONAL INSTRUMENTS PRODUCTS IN COMBINATION WITH OTHER PRODUCTS IN A MANNER NOT EVALUATED OR CONTEMPLATED BY NATIONAL INSTRUMENTS, THE USER OR APPLICATION DESIGNER IS ULTIMATELY RESPONSIBLE FOR VERIFYING AND VALIDATING THE SUITABILITY OF NATIONAL INSTRUMENTS PRODUCTS WHENEVER NATIONAL INSTRUMENTS PRODUCTS ARE INCORPORATED IN A SYSTEM OR APPLICATION, INCLUDING, WITHOUT LIMITATION, THE APPROPRIATE DESIGN, PROCESS AND SAFETY LEVEL OF SUCH SYSTEM OR APPLICATION.

Contents

Chapter 4
Logging Data

Chapter 5
Performing Sweep Measurements

Chapter 6
Extending SignalExpress Projects with LabVIEW

Chapter 7
Where to Go from Here

Appendix A
Technical Support and Professional Services

About This Manual

Use this manual to familiarize yourself with SignalExpress interactive measurements and the basic SignalExpress features that you use to acquire and analyze signals.

This manual contains exercises that help you begin working with SignalExpress. These exercises teach you how to run projects, configure steps, work with signals, configure sweep measurements, log data, and extend SignalExpress with LabVIEW graphical programming.

Conventions

The following conventions appear in this manual:

»	The **»** symbol leads you through nested menu items and dialog box options to a final action. The sequence **Options»Settings»General** directs you to pull down the **Options** menu, select the **Settings** item, and select **General** from the last dialog box.
	This icon denotes a tip, which alerts you to advisory information.
	This icon denotes a note, which alerts you to important information.
bold	Bold text denotes items that you must select or click in the software, such as menu items and dialog box options. Bold text also denotes parameter names, input and output names, views, dialog boxes, sections of dialog boxes, and menu names.
italic	Italic text denotes variables, emphasis, a cross-reference, or an introduction to a key concept. Italic text also denotes text that is a placeholder for a word or value that you must supply.
`monospace`	Text in this font denotes text or characters that you should enter from the keyboard. This font is also used for the proper names of disk drives, paths, directories, programs, subprograms, subroutines, device names, functions, operations, variables, filenames, and extensions.

Related Documentation

Refer to the *SignalExpress Help*, available by selecting **Help» SignalExpress Help**, for more information as you read this manual.

1

Getting Started with SignalExpress

National Instruments provides innovative solutions for scientists and engineers to build automated measurement systems based on industry-standard computers and platforms. National Instruments develops robust, industry-leading programming environments for automating measurement systems, such as LabVIEW for graphical development, LabWindows™/CVI™ for ANSI C programming, and Measurement Studio for Microsoft Visual Studio programming. You can use these programming tools with National Instruments measurement hardware and interfaces to traditional instruments to build custom, advanced virtual instrumentation systems.

SignalExpress optimizes virtual instrumentation for design engineers by offering instant interactive measurements that require no programming. You can use SignalExpress interactively to acquire, generate, analyze, compare, import, and log signals. You can compare design data with measurement data in one step. SignalExpress extends the ease of use and performance of virtual instrumentation to those who must acquire or analyze signals without programming applications. You also can extend the functionality of SignalExpress by importing a custom virtual instrument (VI) created in the LabVIEW Development System or by converting a SignalExpress project to a LabVIEW block diagram so you can continue development in LabVIEW. Refer to Chapter 6, *Extending SignalExpress Projects with LabVIEW*, for more information about advanced functionality in SignalExpress.

This chapter provides information about SignalExpress version availability and available licensing options.

SignalExpress Version Availability

SignalExpress is available in both a full and limited (LE) version. Refer to the following table for a list of features available in each version.

Table 1-1. Differences in SignalExpress Limited Edition (LE) and Full Edition

	LE	Full
Instrument Support		
Over 300 common standalone instruments	Acquire/Generate Signals steps installed by NI Device Drivers IVI DMM Acquire IVI FGEN Arbitrary Waveform IVI FGEN Standard Function IVI Power Supply IVI Scope Acquire	Acquire/Generate Signals steps installed by NI Device Drivers IVI DMM Acquire IVI FGEN Arbitrary Waveform IVI FGEN Standard Function IVI Power Supply IVI Scope Acquire
Visualization and Documentation		
Customizable graphing	Data View	Data View
Interactive cursors	Cursors	Cursors
Drag and drop data into Microsoft Excel, Word, and WordPad	Drag and drop	Drag and drop
Print and export graphs	—	Project Documentation tab
Operator mode with limited user-editing	—	Operator mode
Signal Creation		
Analog signal creation	Create Analog Signal	Create Analog Signal
Digital signal creation	Create Digital Signal	Create Digital Signal
Signal Processing		
Software filters	—	Filter
Scalar and waveform math	—	Arithmetic Formula Scaling and Conversion Time Averaging Window

Table 1-1. Differences in SignalExpress Limited Edition (LE) and Full Edition (Continued)

	LE	Full
Signal Processing (Continued)		
Analog and digital conversion	—	Convert Analog to Digital Convert Digital to Analog
Interactive signal comparisons	—	Interactive Alignment
Load simulation data from PSPICE, Multisim, and other SPICE packages	—	Load from SPICE
Time and Frequency Measurements		
Amplitude and level	—	Amplitude and Levels
Timing and transition	—	Timing and Transition
Power spectrum	—	Power Spectrum
Frequency response	—	Frequency Response
Distortion measurements	—	Distortion
Tone extraction	—	Tone Extraction
Data Logging		
Save signals to file	Load/Save Signals steps installed by NI Device Drivers Export to text and Microsoft Excel files	Load/Save Signals steps installed by NI Device Drivers Export to text and Microsoft Excel files Save to ASCII/LVM Export TDMS log files to Microsoft Excel
Data logging (Creating TDMS Files)	Record button[*] Playback	Record button Playback Record While Running button Recording Options tab Unlimited logs per project
Logging alarms and events	—	Alarms Events
Logging with start or stop conditions	—	Start conditions Stop conditions

Table 1-1. Differences in SignalExpress Limited Edition (LE) and Full Edition (Continued)

	LE	Full
Measurement Automation		
Parameter sweeping	—	Sweep
Limit testing	—	Limit Test
Software triggering	—	Trigger
Sequencing	—	Sequence
Remote Data Access		
Read/Write shared variables	—	Read Shared Variables Write to shared variables
LabVIEW Interaction		
LabVIEW code generation	Generate a LabVIEW VI from a project	Generate a LabVIEW VI from a project
LabVIEW Express VIs	Acquire/Generate Signals Express VIs installed by NI Device Drivers Create Signals Express VIs IVI DMM Acquire IVI FGEN Arbitrary Waveform IVI FGEN Standard Waveform IVI Power Supply IVI Scope Acquire	Acquire/Generate Signals Express VIs installed by NI Device Drivers Create Signals Express VIs IVI DMM Acquire IVI FGEN Arbitrary Waveform IVI FGEN Standard Waveform IVI Power Supply IVI Scope Acquire Load/Save Signals Express VIs Processing Express VIs Analysis Express VIs
Run LabVIEW VIs in SignalExpress	—	Run LabVIEW VI steps
[*] Recording is limited to one log per project		

SignalExpress Licensing Options

This section assists you in understanding the licensing policies for SignalExpress. This document does not replace the *National Instruments Software License Agreement*. Use this document only as a reference.

Evaluation Mode

The evaluation mode of SignalExpress gives you access to the SignalExpress Full Edition and the features in Table 1-1 for 7 days. After that period, you must either register the free, unlicensed LE version or activate the SignalExpress Full Edition. If you do not register or activate SignalExpress, you cannot launch the product after the 7 day evaluation period expires.

Note   If you install SignalExpress on a machine that has LabVIEW 8.0 or later installed, the trial of the SignalExpress Full Edition begins the first time you launch either LabVIEW or SignalExpress.

SignalExpress Full Edition

The full edition of SignalExpress provides you with the full functionality shown in Table 1-1. You can activate the full edition of SignalExpress using the National Instruments License Manager, the SignalExpress installer, or the National Instruments website at ni.com/activate.

Refer to the National Instruments website at ni.com/signalexpress to purchase SignalExpress Full Edition.

SignalExpress LE

You can register the free limited edition, SignalExpress LE, at any time during or after the 7 day evaluation period. After the 7 day evaluation period expires, registered versions of SignalExpress LE allow you to evaluate licensed features with the following restrictions:

- Each time you drop a step that is licensed, a dialog box prompts you to activate the software.

- You cannot save a project.

- Projects close after 10 minutes.

Activating SignalExpress LE

Complete the following steps to activate SignalExpress LE from the application.

1. Launch SignalExpress.

2. In the **National Instruments License Dialog** window that appears, click the **Register Limited Edition** button.

3. Follow the prompts to activate SignalExpress LE.

✏️   **Note**   You also can activate SignalExpress LE from the NI License Manager using the serial number listed in the *SignalExpress Readme*.

2

Working with Projects

You can use SignalExpress to define measurement procedures by adding and configuring steps in an interactive measurement environment. A step is a configurable function that acquires, generates, analyzes, loads, or stores signals. The **Add Step** menu and the **Add Step** palette show the steps available in SignalExpress.

Most steps process input signals and produce output signals. You can configure the operation of a step by specifying values on the **Step Setup** tab for the step. A saved sequence of configured steps is a SignalExpress project.

This chapter teaches you how to load and run existing projects and how to configure steps in these projects.

Opening a Project

Complete the following steps to load a sample project in SignalExpress.

1. Launch SignalExpress. Select **Empty SignalExpress Project** from the **Getting Started** window.

   Notice that SignalExpress is split into views that display various types of information. The primary view appears in the middle of the application window and contains tabs. If SignalExpress opens in the default configuration, the **Data View** tab, the **Recording Options** tab, and the **Project Documentation** tab appear in the primary view.

   The primary view is surrounded by supplementary views. In the default configuration, the **Project View** appears on the left, and the context help appears on the right.

Note If SignalExpress detects a supported hardware device, the **Channel View** appears across the bottom of the SignalExpress application window.

2. If SignalExpress does not open in the default configuration, select **View»Layout»Reset Project to NI Default Layout** to reset the application to the default configuration. You can use the **View** menu to display tabs and views or reset the layout at any time.

3. Select **Help»Open Example** to open the SignalExpress\ Examples directory. Navigate to the Tutorial directory and double-click First Project.seproj to open the example project.

4. Examine the window that appears, as shown in Figure 2-1, to learn about different components of SignalExpress.

1	Run button	6	Step
2	Record button	7	Input
3	Primary view with Data View tab visible	8	Output
4	Work Area pull-down menu	9	Logged Data window
5	Project View	10	Context Help
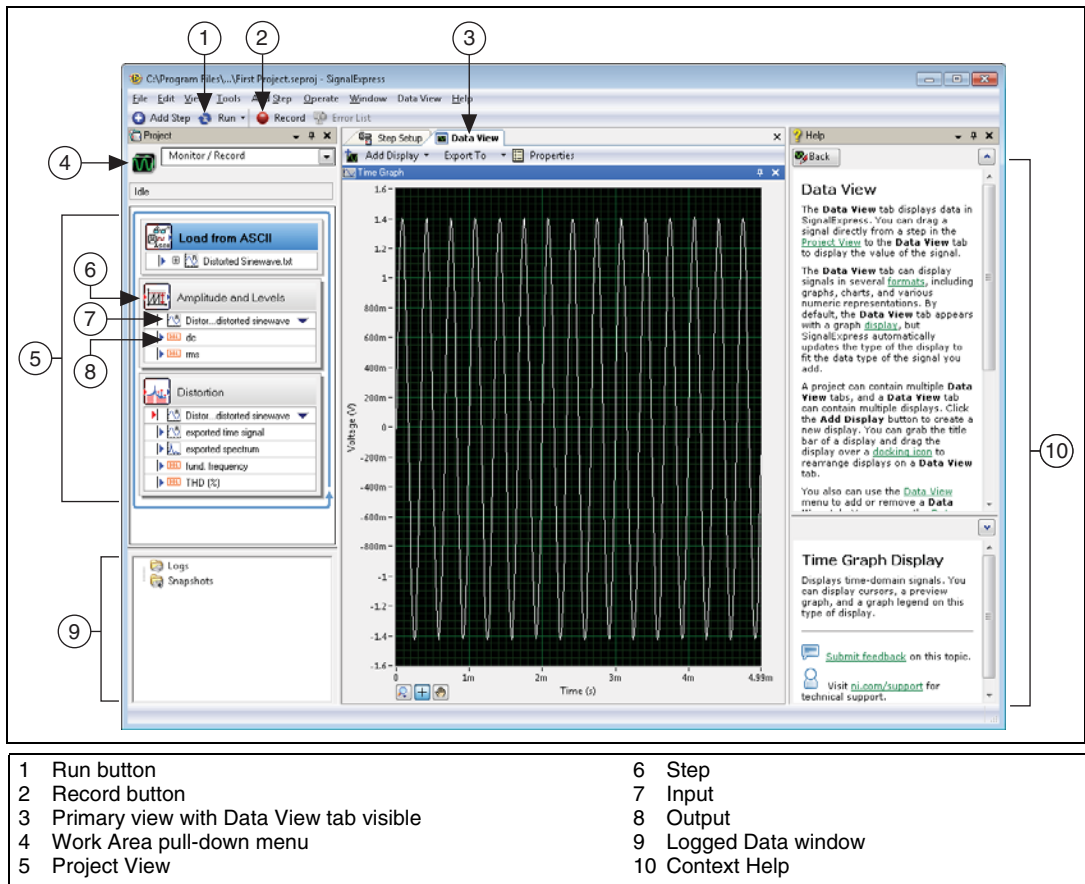
Figure 2-1. First Project.seproj

Running a Project and Displaying Signals

SignalExpress has multiple execution or run modes. You can run projects once, continuously, for a number of iterations you specify, or for an amount of time you specify. When you run a project once, SignalExpress executes all the steps that appear in the **Project View** one time. When you run a project continuously, SignalExpress executes all the steps in the project continuously. Click the down arrow on the **Run** button, shown at left, and select **Configure Run** from the pull-down menu to configure the run mode for a project.

Displays on the **Data View** tab update continuously while a project runs. When a project is running, you can change measurement configuration settings by changing settings on the **Step Setup** tab and view the results immediately. Projects you run continuously run until you click the **Stop** button, shown at left. The **Stop** button appears in place of the **Run** button as a project runs.

Complete the following steps to run the sample project and display signals.

1. Click the **Run** button. If the **Run Information** dialog box appears, click the **Run** button in the dialog box to execute all the steps in the project continuously.

   The project loads a signal from a text file and performs two operations on the signal—an amplitude and levels measurement and a distortion measurement. The Amplitude and Levels step and the Distortion step perform these measurements, respectively. When you run a project, steps analyze input signals and generate new output signals as a result of the analysis. In this project, the Load from ASCII step loads a distorted sine wave, the Amplitude and Levels step and the Distortion step analyze the sine wave, and both steps return new outputs. In the **Project View**, SignalExpress indicates inputs with red arrows and outputs with blue arrows.

   The graph display on the **Data View** tab still contains the loaded signal, which is a time-domain signal. Graphs display time-domain, frequency-domain, or XY signals.
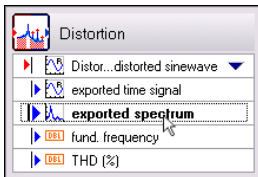
2. Drag the **exported spectrum** output signal of the Distortion step, shown at left, from the **Project View** to the **Data View** tab to display the signal.

   SignalExpress creates a new graph display on the **Data View** tab. SignalExpress does not display the **exported spectrum** signal on the same display as the time-domain signal because the **exported spectrum** signal is a frequency-domain signal. SignalExpress automatically recognizes different types of signals and renders them in the appropriate displays.

💡 **Tip**   Refer to the *SignalExpress Help* for more information about signal types by selecting **Help»SignalExpress Help**, clicking the **Index** tab, and entering signal types. The help provides information about using SignalExpress functionality such as projects, steps, and signals.

3. Drag the **dc** output of the Amplitude and Levels step to the **Data View** tab.

    SignalExpress creates a Chart display with both a graph and a **Legend** table to display the scalar measurement from the **dc** output. The **Legend** table displays the value of each output and the color SignalExpress uses to plot the output on the adjoining graph.

4. Drag the **rms** output of the Amplitude and Levels step to the Chart display to display the scalar RMS measurement.

    SignalExpress creates a new row in the table to display the second measurement. The project appears as shown in Figure 2-2.
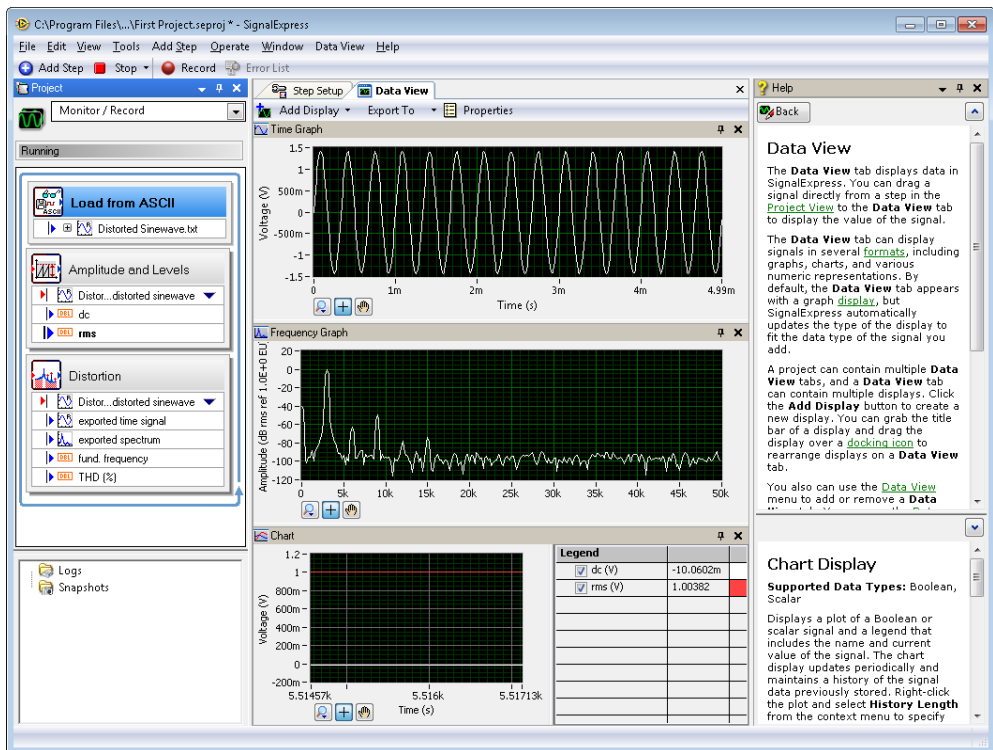
Figure 2-2. Outputs of First Project.seproj

Configuring a Step

A step is a configurable function that acquires, generates, analyzes, loads, or stores signals. Steps process input signals and produce output signals. You can configure the operation of a step in SignalExpress by specifying values on the **Step Setup** tab for the step. While a project runs, you can modify the configuration of steps and see immediate feedback on the **Data View** tab, and you can adjust measurements until you achieve the results you need.

Complete the following steps to configure the Distortion step and the Amplitude and Levels step.

1. Double-click the Distortion step in the **Project View**. SignalExpress displays the **Step Setup** tab for the Distortion step, as shown in Figure 2-3.
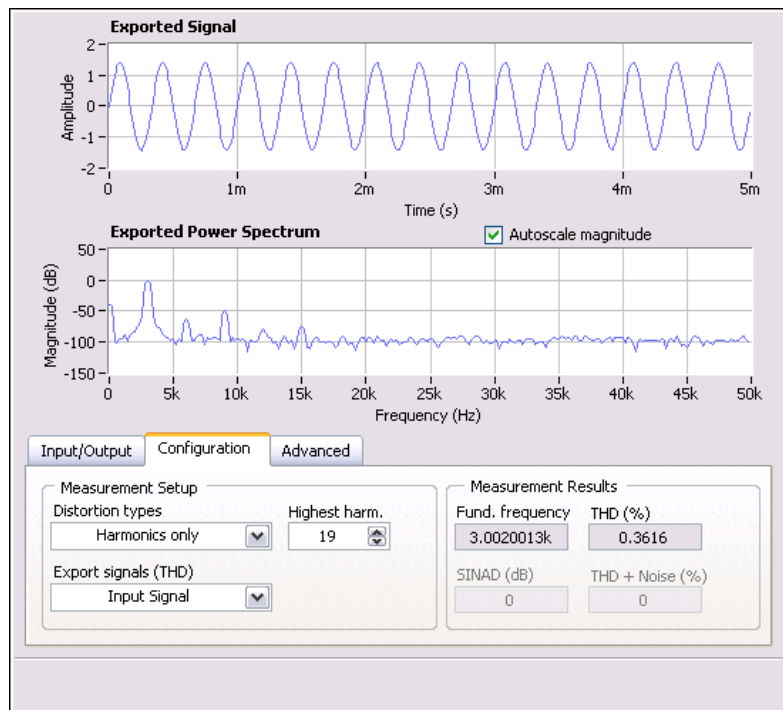
Figure 2-3. Distortion Step Setup Tab

On the **Configuration** page of the **Step Setup** tab, the **Export signals (THD)** field indicates that the Distortion step exports the input signal, and the **Exported Signal** graph displays a preview of the signal. The **Exported Power Spectrum** graph indicates that the step performs a power spectrum on the input signal to convert it to the frequency domain, and the **Measurement Results** field displays the fundamental frequency and total harmonic distortion (THD) of the signal. The step generates the exported signal and three measurements as outputs—the spectrum, the THD, and the fundamental frequency of the original time-domain waveform input.

2. If the context help does not appear on the right side of the screen, select **Help»Context Help** to display complete reference information about the step.

   The upper section of the context help displays information about the step, and the lower section of the context help displays information about specific parameters of the step when you move the cursor over a parameter. Move the cursor over the **THD (%)** parameter to display information about the parameter.

3. On the **Configuration** page, select **Fundamental Tone** from the **Export signals (THD)** pull-down menu.

   The **Exported Power Spectrum** graph changes from displaying the frequency-domain spectrum of the entire input signal to displaying only the frequency spectrum of the fundamental tone of the input signal. Both the output signal of the Distortion step and the graph display of the **exported spectrum** output on the **Data View** tab update to reflect the change you made.

4. Select **Harmonics only** from the **Export signals (THD)** pull-down menu.

   The **Exported Power Spectrum** graph on the **Step Setup** tab and the graph of the output on the **Data View** tab both change to display only the spectrum of the harmonic signals from the input signal.

5. Click the Amplitude and Levels step in the **Project View**.

   The **Step Setup** tab changes from displaying the configuration of the Distortion step to displaying the configuration of the Amplitude and Levels step.

6. Select the **Input/Output** page to display the list of possible inputs and outputs for this step, as shown in Figure 2-4.
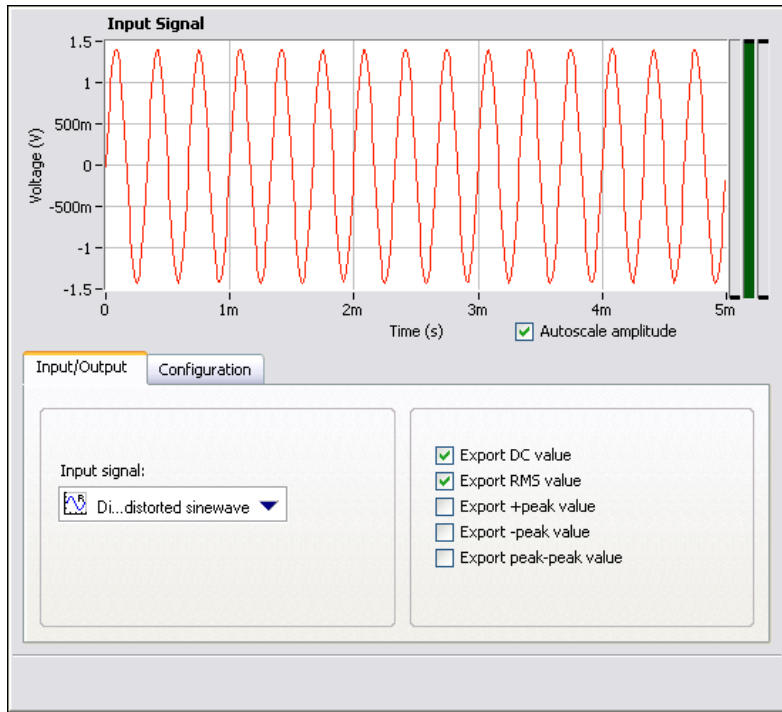
Figure 2-4. Amplitude and Levels Step Setup Tab

7. Place checkmarks in the **Export +peak value**, **Export -peak value**, and **Export peak-peak value** checkboxes to configure the Amplitude and Levels step to return three additional measurements.

    Three additional outputs appear in the **Project View**.

8. Switch to the **Data View** tab.

9. Drag the three new outputs from the **Project View** to the Chart display of scalar measurements. The new outputs appear on the graph and in the **Legend** table.

Note If you have been running the project for a while, you might need to stop and restart the project to see all the scalar signals on the graph. SignalExpress accumulates points of data for signals that you add to a graph, and new signals you add do not have as many data points to display on the graph.

10. Click the **Stop** button to stop the project.

    When you click the **Stop** button, the project stops running after completing the current cycle of operations, or the current iteration. Click the down arrow on the **Stop** button and select **Abort** from the pull-down menu to completely stop the project without finishing the current iteration.

11. Select **File»Save Project As** and save the project as `My First Project.seproj` in the `C:\Program Files\National Instruments\SignalExpress\Examples\Tutorial` directory.

12. Select **File»Close Project** to close the project.

Ordering, Moving, and Deleting Steps

Most steps in SignalExpress projects require input data. Steps can operate only on signals exported from previous steps in the **Project View**, so the order in which steps appear in the **Project View** can effect project functionality.

For steps that have an **Input** or **Input/Output** page on the **Step Setup** tab, the **Input signal** pull-down menu on that page displays only compatible signals exported from a previous step. When the output of a step becomes the input of another step, the steps become dependent on each other, and the two steps execute sequentially at the same rate. The first step generates an output signal that the second step must receive as an input before the second step can execute.

You can move a step within a project by dragging it up or down in the **Project View**. You can delete a step by right-clicking it in the **Project View** and selecting **Delete** from the context menu. When you move or delete a step, the status of signals in the project changes. For example, if you delete a step that generates output signals, the operation of the project breaks if any of the deleted output signals are inputs for other steps, and an error indicator appears in the **Project View**. You also can cut, copy, and paste steps within a project by pressing the <Ctrl-X>, <Ctrl-C>, and <Ctrl-V> keys, respectively, or by right-clicking a step in the **Project View** and selecting **Cut**, **Copy**, **Paste Before Selected Step**, or **Paste After Selected Step** from the context menu.

Handling Errors and Warnings

If an error occurs while a project runs, an error indicator, shown at left, appears in the **Project View** on the step that encountered the error. Double-click the step with the error to display an error description across the bottom of the **Step Setup** tab. Click the **Details** button to the right of the error description to display the full error description.

SignalExpress logs all errors and warnings on the **Event Log** tab while a project runs. To display the **Event Log** tab, select **View»Event Log**. Refer to the *SignalExpress Help* for more information about errors and warnings by selecting **Help»SignalExpress Help**, clicking the **Index** tab, and entering errors.

3

Working with Signals

You can use SignalExpress to generate and analyze signals without programming. This chapter teaches you how to work with signals in SignalExpress, including how to plot signals on graphs, import signals from a file, interactively align and compare two signals, and save signals to a file.

Graphing Signals

Complete the following steps to plot signals in a sample project and examine the signals visually using cursors.

1. Select **Help»Open Example**, to open the SignalExpress\Examples directory. Navigate to the Tutorial directory and double-click Signals.seproj. This project configures the Create Analog Signal step to create a square wave signal and the Filter step to perform a lowpass Butterworth filter.

2. Drag the **step signal** output of the Create Analog Signal step to the **Data View** tab.

3. Drag the **filtered step** output of the Filter step to the **Data View** tab.

   Both the **step signal** and the **filtered step** signals are time-domain signals, so they appear on the same graph display. If you try to plot signals of different types by dragging the signals to the same display, SignalExpress creates a new display.
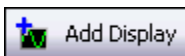
4. Click the **Add Display** button, shown at left, to create a new display.

Note You can remove individual displays by clicking the **X** at the top right corner of the display.

5. Drag the **filtered step** output of the Filter step to the new display.

6. Right-click the new display and select **Visible Items»Cursors** from the context menu to display two interactive cursors, as shown in Figure 3-1.
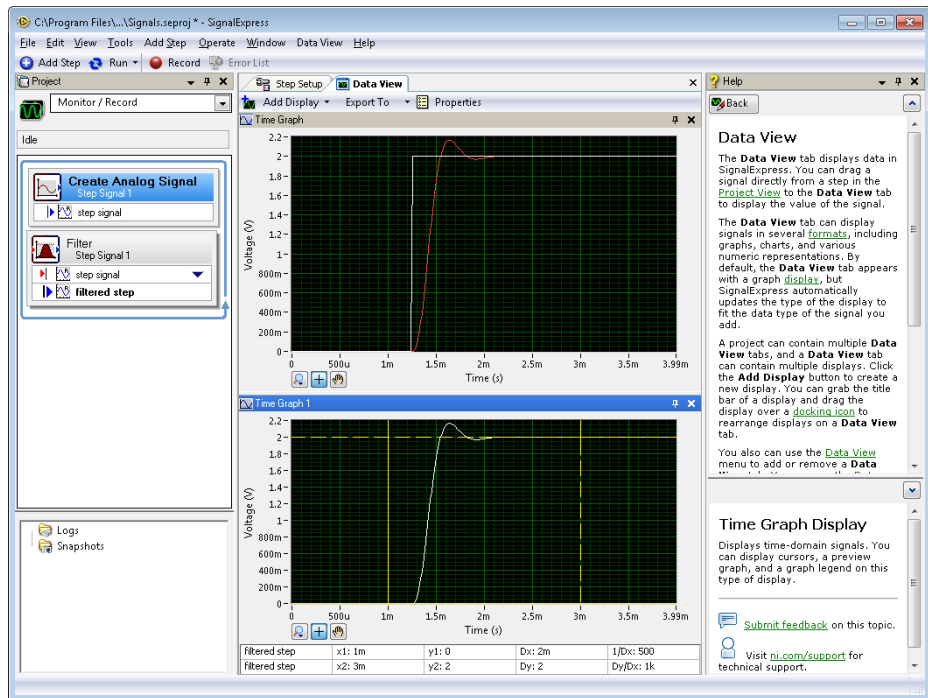
Figure 3-1. Signals.seproj

As you drag the cursors, SignalExpress displays the x- and y-values of the cursors in the cursor table at the bottom of the **Data View** tab.

7. Select **File»Save Project As** and save the project as My Signals.seproj in the C:\Program Files\National Instruments\SignalExpress\Examples\Tutorial directory.

Importing a Signal from a File

You can import signals from standard file formats such as ASCII comma- or tab-delimited files and LabVIEW measurement data files (`.lvm`). You also can import signals from simulated results of electronic design automation tools such as SPICE simulators.

Complete the following steps to import a signal from a file.

1. Right-click in the **Project View** and select **Load/Save Signals»Analog Signals»Load from ASCII** from the context menu to add the Load from ASCII step to the **Project View**. The **Step Setup** tab for the Load from ASCII step appears. This step parses an ASCII file and displays the signals in the file.

Note   You can add steps to a project from the **Add Step** menu, the **Add Step** palette, or the context menu that appears when you right-click in the **Project View**.

2. Confirm that **. (dot)** is selected in the **Decimal separator** pull-down menu to read the values in the example file correctly.

3. On the **Step Setup** tab, click the browse button, shown at left, navigate to the `C:\Program Files\National Instruments\SignalExpress\Examples\Tutorial` directory, and double-click `Step Response.txt`.

    In the **File preview** section, column 1 shows the time stamp data, and column 2 shows the actual voltage values of the signal.

4. Switch to the **Import Signals** page of the **Step Setup** tab to display the available signals in the file.

5. Confirm that a checkmark appears in the **Column 2** checkbox to import that signal.

    The **Step Setup** tab displays a preview of the signal on the **Imported Signal** graph.

6. Confirm that **Column 1 (Column 1)** appears in the **Input X values** pull-down menu to set the x-axis data of the waveform to the appropriate values.

7. Switch to the **Data View** tab.

8. In the **Project View**, expand the **Step Response.txt** output of the Load from ASCII step.

9. Right-click the **Column 2** output and select **Rename** from the context menu.

10. Enter `step response` and press the <Enter> key to rename the output.

11. Drag the new **step response** output of the Load from ASCII step to the lower display on the **Data View** tab.

    The lower graph display appears to show almost no signals. This is because the Create Analog Signal step produces signals with a current timestamp while the **step response** output has a zero timestamp. Right-click the lower graph display and select **Time Stamp»Ignore** to see both the **filtered step** signal and the **step response** output.

    The **filtered step** signal resembles the rising edge of the **step response** output, as shown in Figure 3-2.
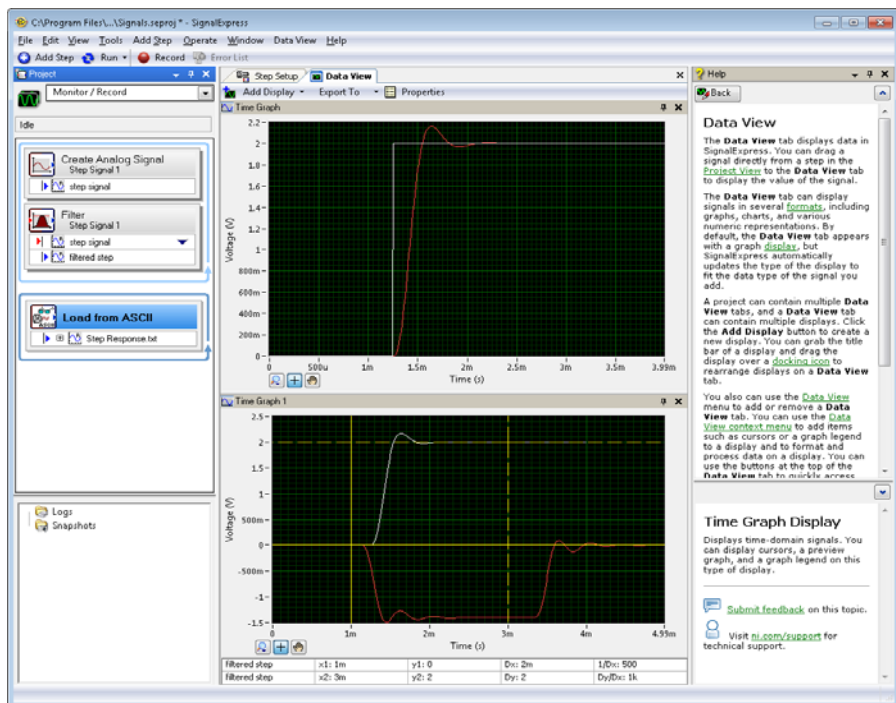
Figure 3-2. Signals of Signals.seproj

12. Select **File»Save Project** to save the project.

Aligning and Comparing Signals

Although the **filtered step** signal and the **step response** output both show an overshoot in the rising edge, assessing the similarity between the two is difficult because the signals come from different sources and vary in amplitude and timing. However, you can use the Interactive Alignment step to align and compare two signals, so you can choose which type of information you want to export from the operation to use in the project.

Complete the following steps to align two signals in the `My Signals.seproj` project.

1. Expand the **Step Response.txt** output, right-click the **step response** output and select **Send To»Processing»Analog Signals»Interactive Alignment** from the context menu to pass the **step response** signal from the Load from ASCII step to the Interactive Alignment step. Place a checkmark in the **Ignore x0** checkbox on the **Alignment** tab to ignore timestamp mismatches.

    The step selects the two most recent signals from the project to use as inputs and displays the signals on the graphs on the **Step Setup** tab, as shown in Figure 3-3.
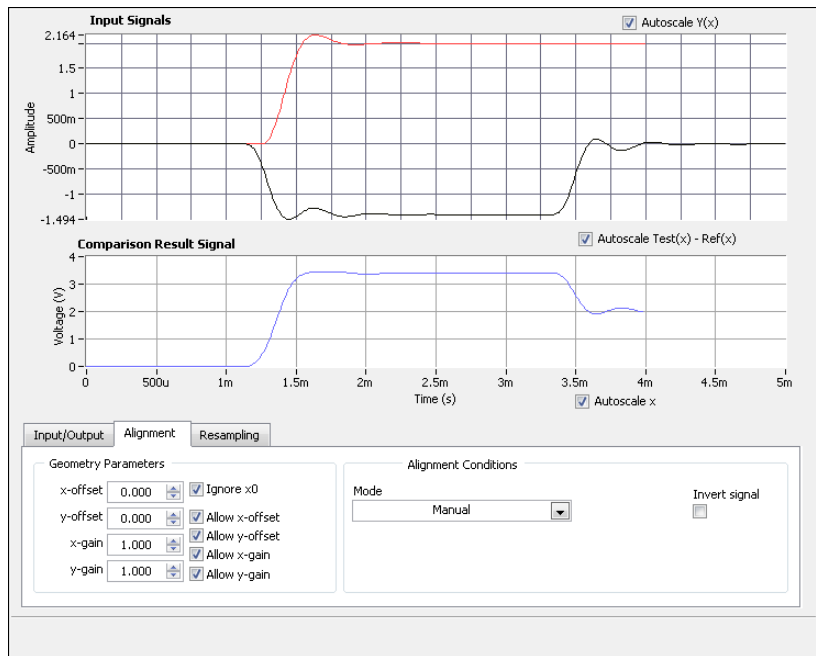
Figure 3-3. Interactive Alignment Step Setup Tab

When you add a step to a project, SignalExpress selects input signals based on the signal types the step can accept. For example, the Interactive Alignment step can operate only on time-domain waveform signals. Therefore, the step selects as inputs the last two time-domain signals created in the project.

Note   To change the input signals for a step, select different signals from the pull-down menu of compatible signals on the **Input** or **Input/Output** page of the **Step Setup** tab, or click the down arrow that appears next to the step input name on the step in the **Project View**.

2. Click the red signal in the **Input Signals** graph and drag it to another point within the graph. The **Comparison Result Signal** graph updates to show the new calculated difference between the signals.

    You can drag, expand, and contract signals on the graph.

3. Try to align the rising edges of the two signals by dragging a signal within the graph. Click a signal to set an anchor point and press and hold the <Alt> key while dragging the signal to stretch the signal around that anchor point along the x- and y-axes.

    On the **Alignment** page of the **Step Setup** tab, the step computes and displays the x- and y-gain and offset values you need to achieve the alignment specifications as you drag the signals.

4. Select **Auto - Step** from the **Mode** pull-down menu to align the signals. SignalExpress computes the alignment using built-in algorithms.

    The **Comparison Result Signal** graph on the **Step Setup** tab displays the difference between the two signals.

5. On the **Input/Output** page, place a checkmark in the **Export aligned signals** checkbox to add the signals to the outputs of the step.

6. Switch to the **Data View** tab.

7. Click the **Add Display** button to add a third display.

8. Drag the **aligned reference** and **aligned test** outputs of the Interactive Alignment step to the new graph to view the aligned signals.

9. Select **File»Save Project** to save the project.

Signal Types in SignalExpress

Some steps, such as the Arithmetic step, can operate on multiple signal types. For example, you can use the Arithmetic step to operate on time-domain or frequency-domain waveforms. The Arithmetic step changes behavior based on the type of input signals you select for the step. For example, if you add two time-domain signals, SignalExpress adds only their amplitudes. However, if you add two frequency-domain phase signals, SignalExpress adds the appropriate phase shift.

Refer to the *SignalExpress Help* for more information about signal types by selecting **Help»SignalExpress Help**, clicking the **Index** tab, and entering `signal types`.

Exporting and Printing Signals

You can use SignalExpress to document signals or continue analysis in another software application. This section teaches you how to export signals, including sending signals to an ASCII file, sending data to Microsoft Excel, printing signals, and using the built-in project documentation feature to document the SignalExpress project.

Saving Signals to File

Complete the following steps to save a signal from `My Signals.seproj` to a file.
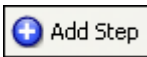
1. Click the **Add Step** button, shown at left, to display the **Add Step** palette.

2. Select **Load/Save Signals»Analog Signals»Save to ASCII/LVM** to add the Save to ASCII/LVM step to the **Project View**. The **Step Setup** tab for the Save to ASCII/LVM step appears.

3. Click the **Signals** page on the **Step Setup** tab and select **filtered step** from the **Input Data** pull-down menu.

4. On the **File Settings** page, specify where to save the text file you are creating by clicking the **Browse** button next to the **Export file path** field and navigating to the `C:\Program Files\National Instruments\SignalExpress\Examples\Tutorial` directory. Enter `filtered signal.txt` as the filename.

5. Select **Overwrite** from the **If file already exists** pull-down menu.

6. Select **Generic ASCII (.txt)** from the **Export file type** pull-down menu.

You can use a Load/Save Signals step to save data to a file every time the project runs.

7. Click the down arrow on the **Run** button and select **Run Once** to run the project and save the resulting signal to the specified ASCII file.

8. Select **File»Save Project** to save the project.

9. Select **File»Close Project** to close the project.

10. Browse to the `filtered signal.txt` and open the file in a text editor to see the values of the signal. In SignalExpress, you can use the Load from ASCII step to import the signal back into a project.

Exporting Signals to Microsoft Excel

To export signal data to Microsoft Excel, launch Excel and drag the output signal of a step in SignalExpress to an Excel spreadsheet. You also can right-click a display on the **Data View** tab and select **Export To» Microsoft Excel** to export all the data on the display.

Creating Reports in SignalExpress

Select **View»Project Documentation** to display the **Project Documentation** tab. You can use this tab to describe a project using text and images such as graphs. You can drag a step output from the **Project View** to the **Project Documentation** tab to display a graph of the output signal. If the project is running, the graph on the **Project Documentation** tab automatically updates to match the current value of the step output.

To print the documentation, display the **Project Documentation** tab and select **File»Print»Print Documentation** or click the **Print Documentation** button. To export the documentation to HTML, display the **Project Documentation** tab and select **File»Export»Export Documentation to HTML**.

4

Logging Data

You can use SignalExpress to record and analyze measurements. You can record any time-domain, double, 32-bit unsigned integer, Boolean, or digital step output. You also can analyze and process logged data by playing it through analysis steps.

This chapter teaches you how to record data using the integrated data logging features in SignalExpress. You learn how to record a specified signal, play back that signal, and analyze the signal using analysis steps. You also learn how to use the **Recording Options** tab to log signals based on specified start or stop conditions.

Recording a Signal

You can use the **Record** button to configure a data logging process.

Complete the following steps to specify a signal to record and to record the signal.

1. Select **Help»Open Example**, navigate to the `Tutorial` directory, and double-click `Logging.seproj`.

   This project uses the Create Analog Signal step to generate a signal based on a formula.

   ⬤ Record

2. Click the **Record** button, shown at left, to display the **Logging Signals Selection** dialog box.

   The **Logging Signals Selection** dialog box displays the signals in the project available for recording. You can select one signal or multiple signals to record. You also can specify a name and description for the log.

3. Place a checkmark in the **signal** checkbox to record the formula signal generated in the Create Analog Signal step.

4. Click the **OK** button to close the **Logging Signals Selection** dialog box and begin recording the signal. The logging operation continues until you click the **Stop** button.

Stop ▼

5. Click the **Stop** button, shown at left, to stop logging the signal. If you have not logged a signal before, the **First Log Complete** dialog box appears. Click the **OK** button to close the dialog box.

   The logged data appears in the **Logged Data** window at the bottom of the **Project View**, as shown in Figure 4-1.
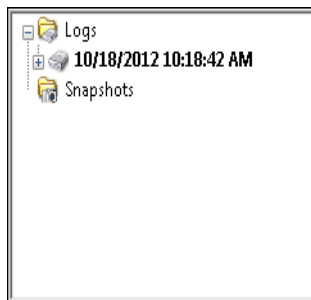
Figure 4-1. Logged Data Window

   By default, SignalExpress names the logged data according to the date and time you recorded the data. SignalExpress saves logged data in the `.tdms` file format in the directory you specify in the **Options** dialog box.

6. Select **Tools»Options** and select the **Logging** option to specify the directory for SignalExpress to save the logged data and to customize various preferences for logged data.

7. Click the **OK** button to close the **Options** dialog box.

8. Select **File»Save Project As** and save the project as `My Logging.seproj` in the `C:\Program Files\National Instruments\SignalExpress\Examples\Tutorial` directory.

Viewing a Logged Signal

Complete the following steps to view the logged data.

1. If the **Data View** tab is not visible, select **View»Data View** to display the **Data View** tab.

2. The **Logged Data** window displays a list of all logged data in the current project. Select the data log you just recorded from the **Logged Data** window and drag it to the **Data View** tab. The **Data View** tab displays the logged data and a Preview Graph, as shown in Figure 4-2 with the context help closed. The signal displayed on the **Data View**

tab might differ from the signal displayed in Figure 4-2, depending on how long you recorded the signal.

| 1   Logged Data | 2   Preview Graph |
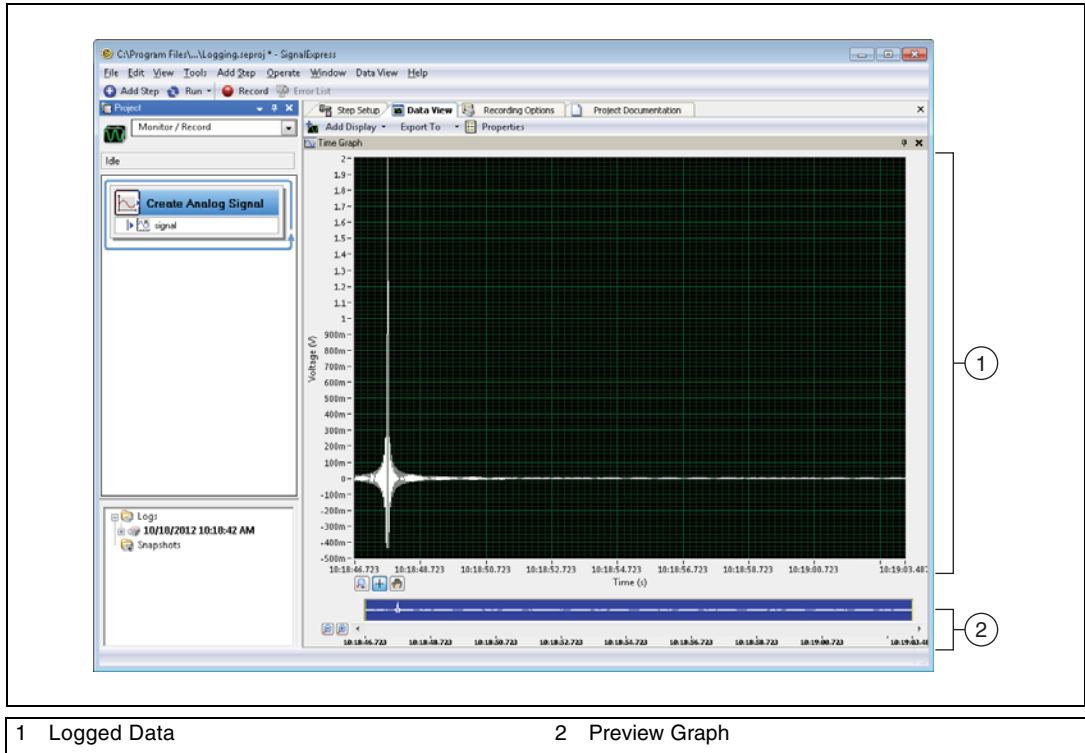
Figure 4-2. Logging.seproj

The Preview Graph provides a method for zooming and panning through data on the **Data View** tab. The Preview Graph appears by default when you view logged data. When viewing live or non-logged data, right-click a display on the **Data View** tab and select **Visible Items»Preview** from the context menu to display the Preview Graph.

Click the **Zoom In** button next to the Preview Graph to zoom in on the logged signal. The cursors on the Preview Graph show the subset of data currently displayed on the **Data View** tab. Use the scroll bar beneath the Preview Graph to scroll through the data. Click and drag the cursors on the Preview Graph to increase or decrease the subset of data you are viewing.

Logging Signals with Predefined Start and Stop Conditions

You can configure start and stop conditions that signals must meet before SignalExpress records or stops recording the signals. Complete the following steps to log data based on start and stop conditions.

1. If the **Recording Options** tab is not visible, select **View»Recording Options** to open the **Recording Options** tab.

2. Select **Signal Selection** from the **Category** list on the **Recording Options** tab.

3. Place a checkmark next to the signal in the **Record** column, as shown in Figure 4-3.

Figure 4-3. Signal Selection
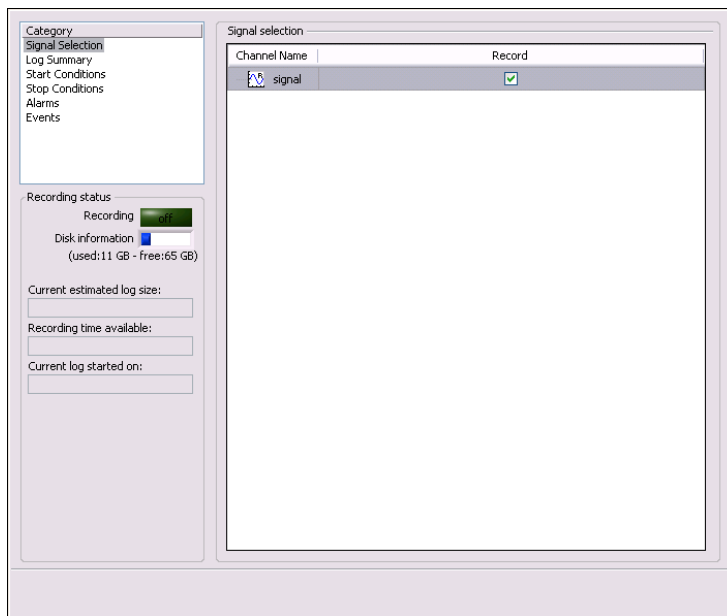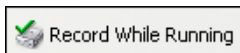
The **Record** button changes to the **Record While Running** button, shown at left. Ensure the **Record While Running** button is pressed. When the **Record While Running** button is pressed, SignalExpress records the selected signal when you click the **Run** button.

4. Select **Start Conditions** from the **Category** list on the **Recording Options** tab.

5. Click the **Add** button under the **Start condition list** to customize a start condition for the logging task.

    a. In the **Condition type** pull-down menu, verify that **Signal trigger** is selected to specify that SignalExpress begins recording when the input signal meets the specified condition.

    b. In the **Signal** pull-down menu, verify that **signal** is selected.

    c. In the **Trigger type** pull-down menu, verify that **Rising slope** is selected to specify to begin recording the signal based on the value of the edge of the signal on a positive slope.

    d. Enter 1 in the **Trigger value** field to begin recording when the signal crosses 1 on a rising slope.

6. Select **Stop Conditions** from the **Category** list on the **Recording Options** tab.

7. Click the **Add** button under the **Stop condition list** to customize a stop condition for the logging task.

    a. In the **Condition type** pull-down menu, verify that **Duration** is selected to specify that SignalExpress stops recording after a specified amount of time passes.

    b. In the **Duration (s)** control, verify that 5 appears to specify to stop recording 5 seconds after the signal begins.

8. Click the **Run** button, shown at left. SignalExpress begins recording the signal when the signal crosses level 1 on a rising slope and continues recording the signal for 5 seconds.

    The following indicators in the **Recording status** section of the **Recording Options** tab update while the project runs:

    •   **Recording** displays **ON** when the signal meets the start condition and logging is in progress.

    •   **Disk information** displays the available hard disk space on the computer for the log.

    •   **Current estimated log size** displays the size of the log file on disk.

    •   **Recording time available** displays the amount of time you can continue recording the log before running out of disk space.

    •   **Current log started on** displays the start time of the current log.

    The **Start Conditions**, **Stop Conditions**, **Alarms**, and **Events** pages of the **Recording Options** tab also include indicators that display the status of start and stop conditions, alarms, and events that you configure.

Analyzing Logged Signals

After you log a signal, you can play back the logged data or run the logged signal through analysis steps, just as you can with live data. Complete the following steps to analyze a logged signal.

1. Locate the **Work Area** pull-down menu above the **Project View**, as shown in Figure 4-4. Click the down arrow and select **Playback** to switch to the Playback work area.
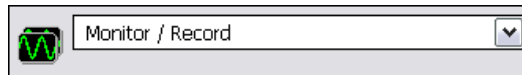
Figure 4-4. Work Area pull-down menu

Use work areas to perform multiple SignalExpress operations from within the same project. You can acquire data, process signals, record data, and perform measurements on logged data without opening a new project. When you save a project, SignalExpress saves every work area within the project in the same project file.

The default work area, Monitor/Record, allows you to take measurements, analyze live data, and log data. The Playback work area allows you to use logged data that you record in the Monitor/Record work area as an input for an analysis step.
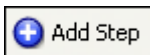
2. Click the **Add Step** button, shown at left, and select the Filter step from **Processing»Analog Signals»Filter**. SignalExpress automatically selects the first signal you logged as the input to the Filter step.

3. Switch to the **Data View** tab, and drag the **filtered data** output of the Filter step to the **Data View** tab to view the resulting signal.

4. Click the **Run** button. The **Data View** tab displays the resulting filtered signal and SignalExpress plays back the entire log.

Advanced Playback

You can use the **Playback Options** tab to configure advanced data playback options. The **Playback Options** tab displays a preview of the logged data and allows you to select a subset of that data to play back or run through analysis steps.

1. Select **View»Playback Options** to display the **Playback Options** tab, as shown in Figure 4-5.
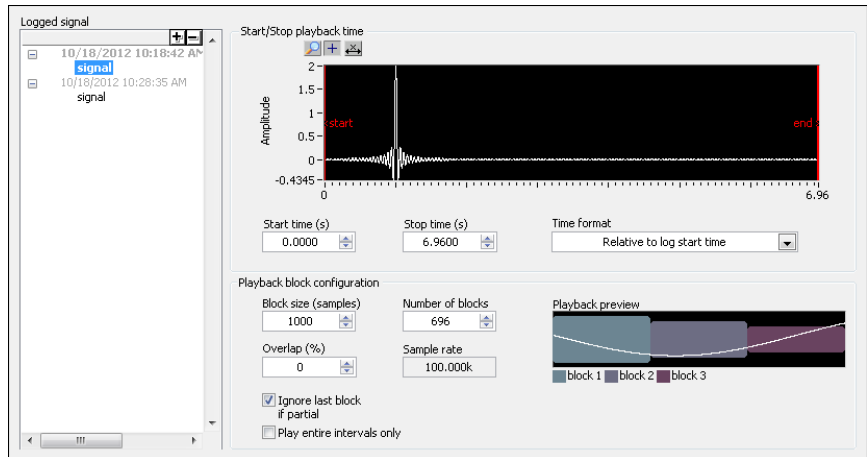
Figure 4-5. Playback Options Tab

2. In the **Logged signal** listbox, select **signal** from the second log you created.

3. Enter 1 in the **Start time (s)** field to play back or analyze a subset of the logged signal beginning 1 second after the start of the log.

4. Enter 4 in the **Stop time (s)** field to play back or analyze a subset of the logged signal ending four seconds after the start of the log. If the log is not at least 4 seconds long, enter an appropriate value in the **Stop time (s)** field.

5. In the **Logged signal** listbox, right-click the **signal** from the second log and select **Activate** from the context menu to make that signal the active log.

✎ **Note**   You also can right-click a log in the **Logged Data** window and select **Make Active Log** from the context menu to make that log the active log.

6. Switch to the **Data View** tab. The graph display on the **Data View** tab automatically updated to display the signal from the second log when you made that signal the active signal.

7. Click the **Run** button, shown at left. SignalExpress filters the subset of the signal you specified on the **Playback Options** tab and displays the resulting filtered signal on the **Data View** tab.

8. Select **File»Close Project** to close the project. The **Save Project Changes?** dialog box appears. Select **Discard project changes and delete new logs on disk** option. You can also select **Save project and logs** option and **Discard project changes and keep logs on disk** option.

Refer to the *SignalExpress Help*, available by selecting **Help» SignalExpress Help**, for more information about logging data, such as specifying alarm conditions, events, and playback options.

5

Performing Sweep Measurements

You can use SignalExpress to automate measurements to characterize and validate designs by creating sweep operations. A design can be anything you create and want to characterize and validate with SignalExpress. You can use the sweep measurements to gather data from designs over a range of conditions to document the performance of the designs. For example, you can use sweep operations to vary the frequency of a stimulus signal or vary the level of a supply voltage while taking measurements to characterize designs.

This chapter teaches you how to set up sweep operations using the Sweep step in SignalExpress. You learn how to characterize the performance of a filter by sweeping through a range of frequency values and measuring the output of the filter. You also learn how to display sweep results and perform multidimensional sweeps for more complex measurements.

Defining Sweep Ranges and Outputs

You can use the Sweep step in SignalExpress to define automated measurements for complex, repeatable sweep operations.

Complete the following steps to define a frequency range in a sample project to sweep through a filter.

1. Select **Help»Open Example**, navigate to the Tutorial directory, and double-click Sweep.seproj.
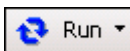
2. Click the **Run** button, shown at left, to run the project continuously.

   The project generates a sine wave stimulus signal using the Create Analog Signal step, passes it through a bandpass elliptic filter using the Filter step, measures the RMS level of the filter output using the Amplitude and Levels step, and converts the level to decibels (dB) using the Formula step. The Filter step acts as a simulated unit under test, so the project uses no hardware. However, you also can sweep physical signals generated from a National Instruments arbitrary waveform generator, function generator, dynamic signal analyzer, or multifunction I/O (MIO) device.

3. Click the **Stop** button to stop the project.
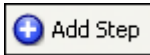
 Add Step

4. Click the **Add Step** button, shown at left, and select **Execution Control»Sweep** to add the Sweep step to the **Project View**.

5. On the **Step Setup** tab, click the **Add** button to display the list of sweepable parameters from each step in the project, as shown in Figure 5-1.
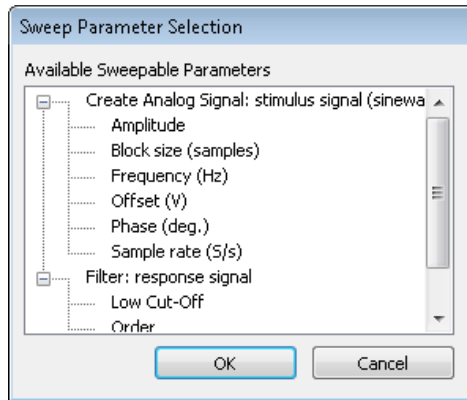
Figure 5-1. Sweep Parameter Selection Dialog Box

6. Select the **Frequency (Hz)** parameter under **Create Analog Signal** and click the **OK** button.

   The Sweep step encloses the Create Analog Signal step, which provides the signal to sweep.

7. On the **Sweep Configuration** page of the **Step Setup** tab, select **Exponential** from the **Type** pull-down menu.

8. Enter 1k in the **Start: Frequency (Hz)** field, and enter 40k in the **Stop: Frequency (Hz)** field.

9. Enter 150 in the **Number of points** field.

    The **Sweep Configuration** page appears as shown in Figure 5-2.
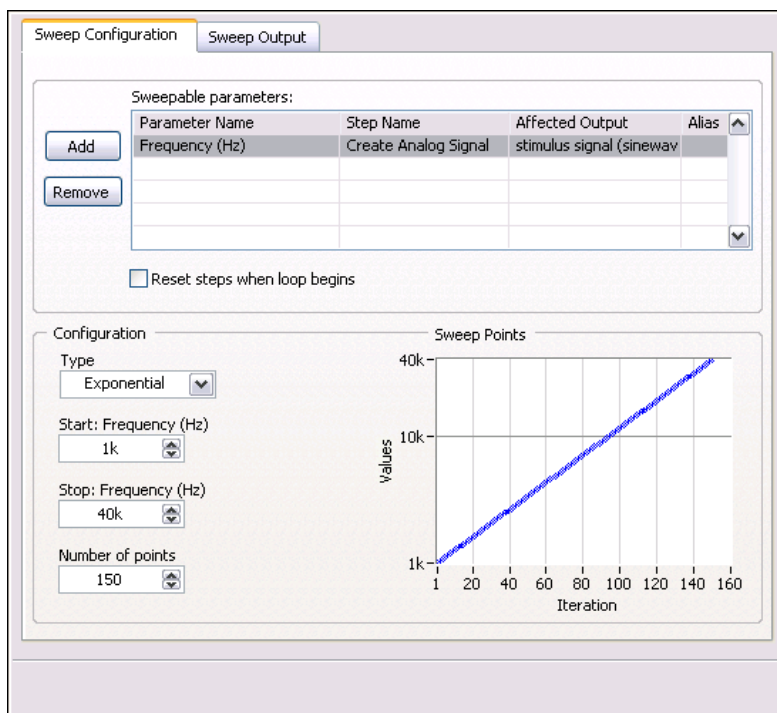
Figure 5-2. Sweep Step Setup Tab

You use the Sweep step to specify a range of values to iterate through the **Frequency (Hz)** parameter of the Create Analog Signal step. The Create Analog Signal step uses the defined frequency range to generate a sine wave at each of these frequencies. You can use the Sweep step to iterate through any sweepable parameter value of any sweepable step in a project.

10. Switch to the **Sweep Output** page.

11. Click the **Add** button to display the list of sweepable outputs from each step in the project.

12. Select the **response amplitude in dB** output under **Formula** and click the **OK** button to plot this measurement against the swept **Frequency (Hz)** parameter.

    The Sweep step creates a loop around all the steps in the **Project View** to include all the steps in the sweep operation.

Running Sweep Measurements

Complete the following steps to run the sweep measurement.

1. Switch to the **Data View** tab, click the down arrow on the **Run** button, and select **Run Once** to execute the sweep measurement.

   The white **stimulus signal** output on the graph display iterates through the specified range of frequencies.

2. Drag the **response amplitude in dB vs. Frequency** signal from the bottom of the Sweep loop to the **Data View** tab to display the output of the sweep.

   SignalExpress creates a new graph display. The data from a sweep operation is an XY array that requires a separate display, as shown in Figure 5-3.
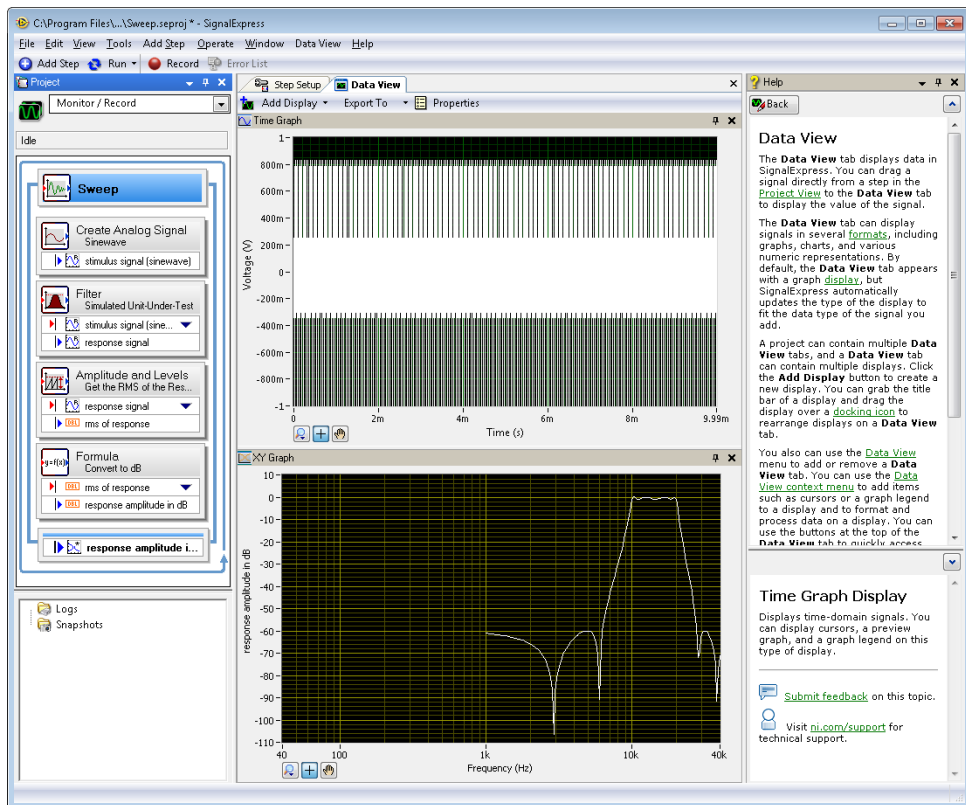

Figure 5-3. Sweep.seproj

3. Select the **Run Once** option again to execute the sweep.

    The frequency response of the Filter step plots on the new graph display while the project runs. The graph shows the transfer function of the filter, or the amplitude output expressed in decibels versus the frequency.

Note By default, SignalExpress does not clear displays on the **Data View** tab between iterations of a sweep. Because the frequency response of the Filter step is the same for each iteration, the graph that displays the signal does not appear to update when you run the project. You can use the **Data** page of the **Options** dialog box to specify whether SignalExpress clears displays between iterations of a sweep. Select **Tools»Options** to display the **Options** dialog box.

4. Double-click the Filter step to display the filter specifications on the **Step Setup** tab.

    The frequency response of the filter in the **Filter Magnitude Response (dB)** graph matches the graph on the **Data View** tab.

5. Select **File»Save Project As** and save the project as My Sweep.seproj in the C:\Progam Files\National Instruments\SignalExpress\Examples\Tutorial directory.

You can use the Sweep step to sweep multiple parameters simultaneously by adding additional parameters on the **Sweep Configuration** page of the Sweep **Step Setup** tab. Sweeping two or more parameters simultaneously is called a parallel sweep. For example, if you want to vary the amplitude of a stimulus signal, you can run a parallel sweep to maximize the precision of the acquisition by varying the input range of a digitizer or MIO device as you vary the signal level. As the signal level increases, you can increase the input range of the measurement device to ensure you use the entire resolution for the measurement.

Running Multidimensional Sweeps

Use multidimensional, or nested, sweeps to iterate through one range while you vary another range. For example, if you want to sweep through frequencies of a stimulus signal at different amplitudes, run a nested sweep. You can set the amplitude to level 1 and sweep through frequencies, then set the amplitude to level 2 and sweep through frequencies, and so on. You can build a nested sweep by right-clicking a Sweep step in a project and selecting **Add nested sweep** from the context menu to add another sweep loop.

Complete the following steps to run a sample nested sweep project.

1. Select **Help»Open Example**, navigate to the `Tutorial` directory, and double-click `Nested Sweep.seproj`.

2. Click the **Run** button to run the project.

   Each iteration of the inner sweep loop sweeps the frequency of the stimulus signal. The outer sweep loop varies the low and high cutoff frequencies of the Filter step. Each iteration appears in real time on the upper graph display, and then appears on the lower display to show all the sweeps at each cutoff frequency setting, as shown in Figure 5-4.
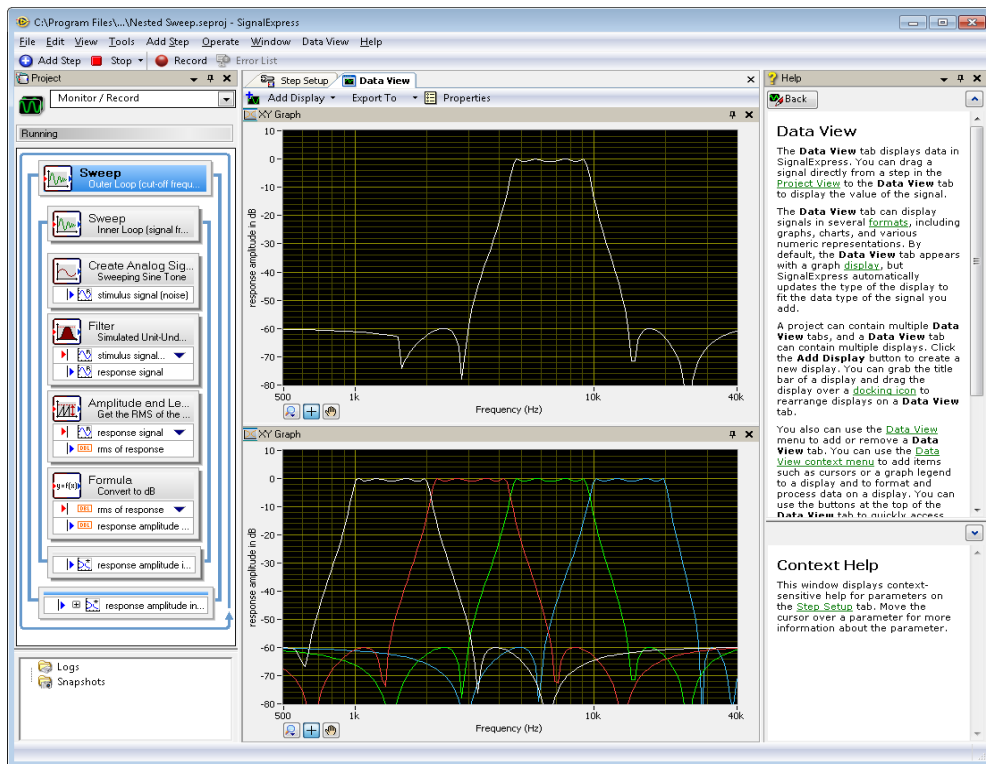

Figure 5-4. Nested Sweep.seproj

3. Click the **Stop** button, shown at left, to stop the project.

6

Extending SignalExpress Projects with LabVIEW

You can use SignalExpress to define automated measurements by using built-in steps for acquiring, generating, analyzing, or logging signals. You can extend the functionality of SignalExpress projects using LabVIEW in the following ways:

- Build a VI in LabVIEW and import the VI into SignalExpress to provide custom step functionality and expand the number of steps available in SignalExpress.

- Convert a SignalExpress project to a LabVIEW VI to continue development in LabVIEW.

Importing LabVIEW VIs into SignalExpress as Steps

Use the Run LabVIEW VI steps in SignalExpress to call custom LabVIEW VIs.

Note You do not need to install the LabVIEW Development System to use a Run LabVIEW VI step. The Run LabVIEW VI steps do not support VIs developed in 64-bit versions of LabVIEW.

You can call a LabVIEW VI from SignalExpress to do the following:

- Control GPIB instruments

- Control National Instruments hardware that SignalExpress does not support

- Read or write data to more file formats

- Display operator instructions in a pop-up dialog box

- Define a measurement algorithm

Complete the following steps to import a VI from LabVIEW with the Run LabVIEW VI step.

1. Select **Help»Open Example**, navigate to the `Tutorial` directory, and double-click `User Step.seproj`.

    This project uses the Create Analog Signal step to generate a signal.
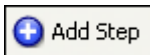
2. Click the **Add Step** button, shown at left, and select **Run LabVIEW VI»Run LabVIEW 2011 VI**.

Note   The VI you run in this exercise was saved in LabVIEW 2011. You must use the version of the Run LabVIEW VI step that matches the version of LabVIEW in which you saved the VI.

3. On the **Step Setup** tab, click the browse button in the **Select VI** section and select `Limiter-LV2011.vi` in the `C:\Program Files\ National Instruments\SignalExpress\Examples\ Tutorial` directory. The Limiter-LV2011 VI accepts a time-domain waveform as an input, clips the signal above and below values that you specify on the **Step Setup** tab, and returns the clipped waveform as an output signal.

    When you import a LabVIEW VI, SignalExpress maps the inputs of the VI as parameters and the outputs of the VI as output signals in SignalExpress.

    You can define whether the inputs for VIs become input signals or parameters. An input signal appears in the **Project View** as an input to a step, which means you can pass signals as inputs to a VI. A parameter is a value you can configure on the **Step Setup** tab of a step. You also can sweep parameters dynamically using the Sweep step. In this project, the VI has an input signal, **Time waveform in**, and scalar parameters, **Upper limit** and **Lower limit**.

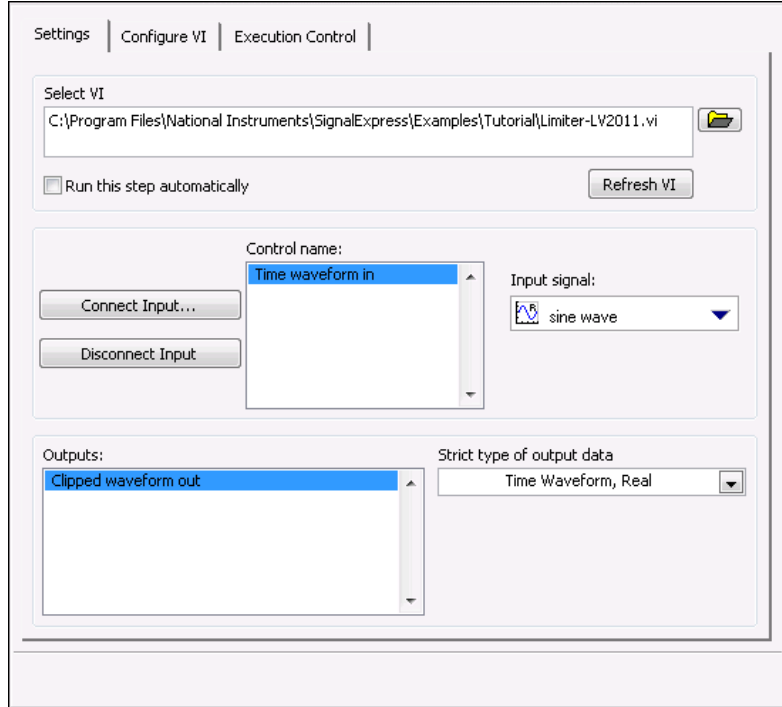Verify that the **Step Setup** tab appears as shown in Figure 6-1.

Figure 6-1. Limiter VI Step Setup Tab

4. Click the **Run** button, shown at left, to run the project.

5. Switch to the **Data View** tab and drag the **Clipped waveform out** output signal from the Run LabVIEW 2011 VI step to the **Data View** tab.

6. Double-click the Run LabVIEW 2011 VI step to display the **Step Setup** tab.

7. Select the **Configure VI** page to display the VI.

8. Enter new values in the **Upper limit** and **Lower limit** fields. For example, enter 100 in the **Upper limit** field.

9. Switch to the **Data View** tab. The **Clipped waveform out** signal changes to reflect the changes you made.

10. Click the **Stop** button, shown at left, to stop the project.

11. Select **File»Save As** and save the project as My User Step.seproj.

12. Select **File»Close Project** to close the project.

Refer to the *SignalExpress Help* for more information about using
LabVIEW VIs in SignalExpress and building VIs that work well in
SignalExpress.

Converting SignalExpress Projects to LabVIEW VIs

SignalExpress can convert SignalExpress projects into LabVIEW VIs.

Note   To convert a SignalExpress project to a LabVIEW VI, you must have the LabVIEW
8.2 Full Development System or later installed.

Complete the following steps to convert a SignalExpress project to a
LabVIEW VI.

1. Select **Help»Open Example**, navigate to the `Tutorial\Solutions`
    directory and double-click `My First Project.seproj`.

2. Select **Tools»Generate Code»LabVIEW Diagram**.

3. Specify a filename and location for the new LabVIEW VI and click the
    OK button. SignalExpress generates the new VI and opens the VI in
    the last version of LabVIEW you opened on the machine.

    The resulting LabVIEW VI reflects the execution of the project in
    SignalExpress. The block diagram consists of LabVIEW Express VIs
    wired together. Each Express VI correlates to a step in the
    SignalExpress project. You can double-click an Express VI to display
    a configuration dialog box that is identical to the **Step Setup** tab in
    SignalExpress. You also can right-click an Express VI and select **Open
    Front Panel** from the context menu to convert the Express VI into a
    LabVIEW subVI. You can view the block diagram to see how the
    LabVIEW block diagram executes and modify the functionality of the
    VI. When you convert an Express VI into a subVI, you cannot convert
    the subVI back into an Express VI.

Note   When you convert a SignalExpress project with logging, SignalExpress generates a
LabVIEW block diagram with one Express VI. You cannot convert the generated Express
VI into a subVI.

7

Where to Go from Here

Refer to the following resources for more information about SignalExpress.

SignalExpress Sample Projects

SignalExpress provides a variety of sample projects that demonstrate more capabilities of SignalExpress. These projects are located in the `C:\Program Files\National Instruments\SignalExpress\Examples` directory. Review these examples to learn more about the features of SignalExpress or to start with a project that closely resembles your needs.

Using Hardware with SignalExpress

SignalExpress supports a variety of National Instruments hardware for acquiring and generating signals. You can generate or acquire and log analog signals in SignalExpress using National Instruments MIO devices, dynamic signal acquisition devices, high-speed digitizers, or arbitrary waveform generator and function generator devices. You also can synchronize multiple devices in a system by sharing clocks and trigger signals between devices. Refer to the *SignalExpress Help* by selecting **Help»SignalExpress Help** for more information about using hardware with SignalExpress.

Web Resources

Refer to the National Instruments website at ni.com/signalexpress for resources such as example projects, technical documents, and LabVIEW VIs written for use in SignalExpress.

A

Technical Support and Professional Services

Log in to your National Instruments `ni.com` User Profile to get personalized access to your services. Visit the following sections of `ni.com` for technical support and professional services:

- **Support**—Technical support at `ni.com/support` includes the following resources:

  - **Self-Help Technical Resources**—For answers and solutions, visit `ni.com/support` for software drivers and updates, a searchable KnowledgeBase, product manuals, step-by-step troubleshooting wizards, thousands of example programs, tutorials, application notes, instrument drivers, and so on. Registered users also receive access to the NI Discussion Forums at `ni.com/forums`. NI Applications Engineers make sure every question submitted online receives an answer.

  - **Standard Service Program Membership**—This program entitles members to direct access to NI Applications Engineers via phone and email for one-to-one technical support as well as exclusive access to eLearning training modules at `ni.com/elearning`. All customers automatically receive a one-year membership in the Standard Service Program (SSP) with the purchase of most software products and bundles including NI Developer Suite. NI also offers flexible extended contract options that guarantee your SSP benefits are available without interruption for as long as you need them. Visit `ni.com/ssp` for more information.

  For information about other technical support options in your area, visit `ni.com/services`, or contact your local office at `ni.com/contact`.

- **Training and Certification**—Visit `ni.com/training` for training and certification program information. You can also register for instructor-led, hands-on courses at locations around the world.

- **System Integration**—If you have time constraints, limited in-house technical resources, or other project challenges, National Instruments Alliance Partner members can help. To learn more, call your local NI office or visit `ni.com/alliance`.

You also can visit the Worldwide Offices section of `ni.com/niglobal` to access the branch office websites, which provide up-to-date contact information, support phone numbers, email addresses, and current events.