LabVIEW™ SignalExpress™

Getting Started with LabVIEW SignalExpress

January 2007
373873B-01

NATIONAL INSTRUMENTS™

Worldwide Technical Support and Product Information

ni.com

National Instruments Corporate Headquarters

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

Worldwide Offices

Australia 1800 300 800, Austria 43 0 662 45 79 90 0, Belgium 32 0 2 757 00 20, Brazil 55 11 3262 3599, Canada 800 433 3488, China 86 21 6555 7838, Czech Republic 420 224 235 774, Denmark 45 45 76 26 00, Finland 385 0 9 725 725 11, France 33 0 1 48 14 24 24, Germany 49 0 89 741 31 30, India 91 80 41190000, Israel 972 0 3 6393737, Italy 39 02 413091, Japan 81 3 5472 2970, Korea 82 02 3451 3400, Lebanon 961 0 1 33 28 28, Malaysia 1800 887710, Mexico 01 800 010 0793, Netherlands 31 0 348 433 466, New Zealand 0800 553 322, Norway 47 0 66 90 76 60, Poland 48 22 3390150, Portugal 351 210 311 210, Russia 7 495 783 68 51, Singapore 1800 226 5886, Slovenia 386 3 425 42 00, South Africa 27 0 11 805 8197, Spain 34 91 640 0085, Sweden 46 0 8 587 895 00, Switzerland 41 56 200 51 51, Taiwan 886 02 2377 2222, Thailand 662 278 6777, Turkey 90 212 279 3031, United Kingdom 44 0 1635 523545

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

Important Information

Warranty

The media on which you receive National Instruments software are warranted not to fail to execute programming instructions, due to defects in materials and workmanship, for a period of 90 days from date of shipment, as evidenced by receipts or other documentation. National Instruments will, at its option, repair or replace software media that do not execute programming instructions if National Instruments receives notice of such defects during the warranty period. National Instruments does not warrant that the operation of the software shall be uninterrupted or error free.

A Return Material Authorization (RMA) number must be obtained from the factory and clearly marked on the outside of the package before any equipment will be accepted for warranty work. National Instruments will pay the shipping costs of returning to the owner parts which are covered by warranty.

National Instruments believes that the information in this document is accurate. The document has been carefully reviewed for technical accuracy. In the event that technical or typographical errors exist, National Instruments reserves the right to make changes to subsequent editions of this document without prior notice to holders of this edition. The reader should consult National Instruments if errors are suspected. In no event shall National Instruments be liable for any damages arising out of or related to this document or the information contained in it.

EXCEPT AS SPECIFIED HEREIN, NATIONAL INSTRUMENTS MAKES NO WARRANTIES, EXPRESS OR IMPLIED, AND SPECIFICALLY DISCLAIMS ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. CUSTOMER'S RIGHT TO RECOVER DAMAGES CAUSED BY FAULT OR NEGLIGENCE ON THE PART OF NATIONAL INSTRUMENTS SHALL BE LIMITED TO THE AMOUNT THERETOFORE PAID BY THE CUSTOMER. NATIONAL INSTRUMENTS WILL NOT BE LIABLE FOR DAMAGES RESULTING FROM LOSS OF DATA, PROFITS, USE OF PRODUCTS, OR INCIDENTAL OR CONSEQUENTIAL DAMAGES, EVEN IF ADVISED OF THE POSSIBILITY THEREOF. This limitation of the liability of National Instruments will apply regardless of the form of action, whether in contract or tort, including negligence. Any action against National Instruments must be brought within one year after the cause of action accrues. National Instruments shall not be liable for any delay in performance due to causes beyond its reasonable control. The warranty provided herein does not cover damages, defects, malfunctions, or service failures caused by owner's failure to follow the National Instruments installation, operation, or maintenance instructions; owner's modification of the product; owner's abuse, misuse, or negligent acts; and power failure or surges, fire, flood, accident, actions of third parties, or other events outside reasonable control.

Copyright

Trademarks

National Instruments, NI, ni.com, and LabVIEW are trademarks of National Instruments Corporation. Refer to the *Terms of Use* section on `ni.com/legal` for more information about National Instruments trademarks.

Other product and company names mentioned herein are trademarks or trade names of their respective companies.

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, refer to the appropriate location: **Help»Patents** in your software, the `patents.txt` file on your CD, or `ni.com/patents`.

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

Contents

About This Manual

Chapter 1
Introduction to LabVIEW SignalExpress

Chapter 2
Installing LabVIEW SignalExpress

Chapter 3
Working with Projects

Chapter 4
Working with Signals

Chapter 5
Logging Data

Chapter 6
Performing Sweep Measurements

Chapter 7
Extending LabVIEW SignalExpress Projects with LabVIEW

Chapter 8
Where to Go from Here

Appendix A
Technical Support and Professional Services

About This Manual

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

This manual contains exercises that help you begin working with LabVIEW SignalExpress. These exercises teach you how to run projects, configure steps, work with signals, configure sweep measurements, log data, and extend LabVIEW 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 **File»Page Setup»Options** directs you to pull down the **File** menu, select the **Page Setup** item, and select **Options** from the last dialog box.

 — 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 input and output names, parameter names, 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 *LabVIEW SignalExpress Help*, available by selecting **Help» LabVIEW SignalExpress Help**, for more information as you read this manual.

1

Introduction to LabVIEW 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.

LabVIEW SignalExpress optimizes virtual instrumentation for design engineers by offering instant interactive measurements that require no programming. You can use LabVIEW SignalExpress interactively to acquire, generate, analyze, compare, import, and log signals. You can compare design data with measurement data in one step. LabVIEW 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 LabVIEW SignalExpress by importing a custom virtual instrument (VI) created in the LabVIEW Development System or by converting a LabVIEW SignalExpress project to a LabVIEW block diagram so you can continue development in LabVIEW. Refer to Chapter 7, *Extending LabVIEW SignalExpress Projects with LabVIEW*, for more information.

2

Installing LabVIEW SignalExpress

You can use the LabVIEW SignalExpress CD to install LabVIEW SignalExpress and complete the exercises in this manual.

LabVIEW SignalExpress LE

LabVIEW SignalExpress LE gives you a 30-day trial of the Full version of LabVIEW SignalExpress. After that period, you must activate your version of LabVIEW SignalExpress LE or purchase the Full version. If you activate your version of LabVIEW SignalExpress LE before your 30-day trial of the full version is complete, the full features are still available for the remainder of your 30-day trial. You can use LabVIEW SignalExpress LE to complete simple data acquisition and logging functions.

If you do not register your version of LabVIEW SignalExpress LE after 30 days, LabVIEW SignalExpress LE runs with the following limitations:

- You can no longer save projects, so you cannot convert projects into LabVIEW block diagrams.
- You can run projects for only 10 minutes per session.

Refer to the National Instruments Web site at `ni.com/signalexpress` to purchase the full version of LabVIEW SignalExpress.

Minimum System Requirements

LabVIEW SignalExpress requires a minimum of 256 MB of RAM and a Pentium III or later or Celeron 866 MHz or equivalent processor, but National Instruments recommends 512 MB of RAM and a Pentium 4/M or equivalent processor.

Installing LabVIEW SignalExpress

Complete the following steps to install LabVIEW SignalExpress on Windows Vista/XP/2000/NT.

Note If you want to convert LabVIEW SignalExpress projects to LabVIEW block diagrams, you must install LabVIEW 7.1 or later before you install LabVIEW SignalExpress.

1. Disable any automatic virus detection programs before you install. Some virus detection programs interfere with the installation program.
2. Log on as an administrator or as a user with administrator privileges.
3. Insert the LabVIEW SignalExpress Development or Evaluation CD and follow the instructions that appear on the screen.

   By default, LabVIEW SignalExpress installs in the `Program Files\National Instruments\SignalExpress` directory.
4. After installation, check the hard disk for viruses and enable any virus detection programs you disabled.

The example projects you use with this manual are located in `SignalExpress\Examples\Tutorial`. The solutions for each exercise are located in `SignalExpress\Examples\Tutorial\Solutions`.

If you use LabVIEW SignalExpress with National Instruments data acquisition or modular instrumentation hardware, you must install NI-DAQmx 8.5 or later, NI-SCOPE 2.7 or later, NI-FGEN 2.2 or later, or NI-DMM 2.4.2. LabVIEW SignalExpress includes a National Instruments Driver CD that includes these drivers.

3

Working with Projects

You can use LabVIEW 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. Most steps process input signals and produce output signals. You can configure the operation of a step by specifying values in a configuration view. A saved sequence of configured steps is a LabVIEW 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 LabVIEW SignalExpress.

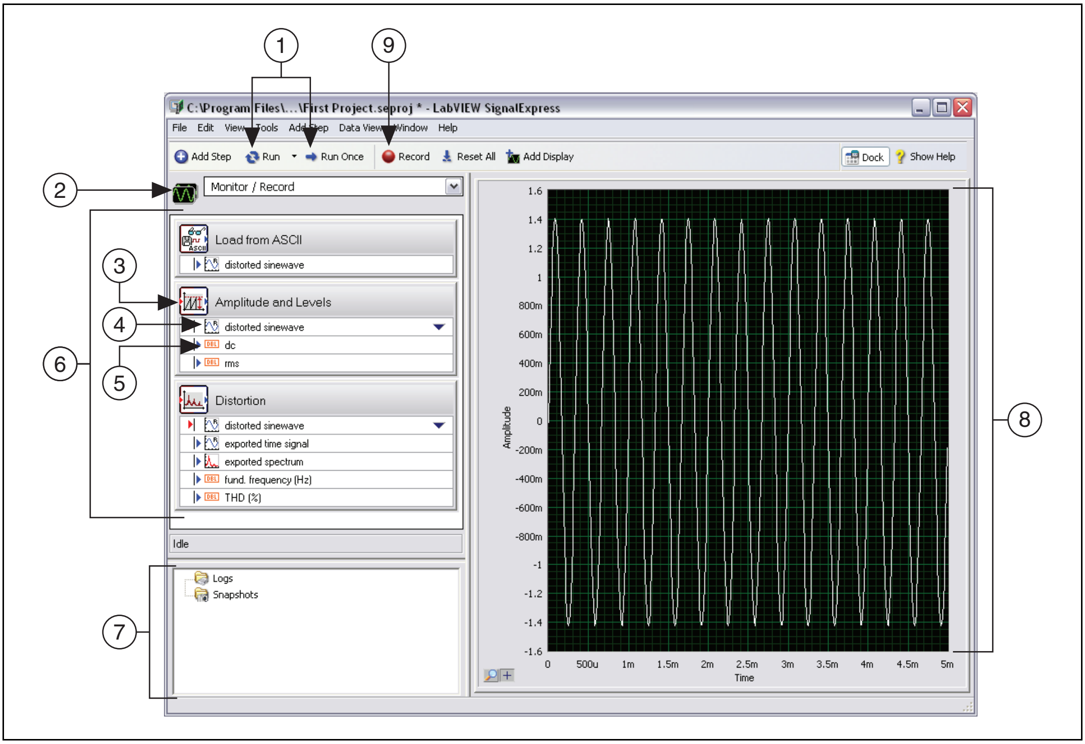
1. Launch LabVIEW SignalExpress.

   Notice that LabVIEW SignalExpress is split into three main views: the Project View on the left, the Data View in the middle, and the help on the right. Within the Data View, you should see the **Data View** tab, the **Logging Options** tab, and the **Project Documentation** tab.

2. If LabVIEW SignalExpress does not open in the factory-default configuration, select **View»Data View** to display the Data View.

3. Select **File»Open Project**, navigate to the `SignalExpress\Examples\Tutorial` directory, and double-click the `First Project.seproj` LabVIEW SignalExpress project.

4. Examine the window that appears, as shown in Figure 3-1 with the help hidden, to learn about different components of LabVIEW SignalExpress.



1	Execution control buttons	4	Input	7	Logged Data window
2	Work Area pull-down menu	5	Output	8	Data View
3	Step	6	Project View	9	Record button

Figure 3-1. First Project.seproj

The left pane is the Project View, which presents the order of operations, or steps, for the project. The right pane is the Data View, which displays the signal that the project generates and analyzes.

Running a Project and Displaying Signals

LabVIEW SignalExpress has two execution modes—Run and Run Once. When you click the **Run** button, LabVIEW SignalExpress executes all steps in the project continuously until you click the **Stop** button. The **Stop** button appears in place of the **Run** button as a project runs. While the steps in the project execute, the Data View updates continuously. While the project runs, you can change the measurement configurations and view the results immediately. If you modify the configuration of steps while a project runs, LabVIEW SignalExpress gives you direct, immediate feedback on the changes you make. When you click the **Run Once** button, LabVIEW SignalExpress executes all steps in the project one time.

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

1. Click the **Run** button, shown at left, to execute all 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, LabVIEW SignalExpress indicates inputs with red arrows and outputs with blue arrows.

   The graph in the Data View still contains the loaded signal, which is a time-domain signal. Graphs display time-domain, frequency-domain, or x-y signals.

2. Click the **exported spectrum** output signal of the Distortion step in the Project View and drag it to the Data View to display the signal.

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

3. Select **Help»LabVIEW SignalExpress Help**, click the **Search** tab, and enter `"signal types"` to refer to the *LabVIEW SignalExpress Help* for more information about signal types. The help provides information about using LabVIEW SignalExpress functionality such as projects, steps, and signals.

4. Click the **dc** output of the Amplitude and Levels step and drag it to the Data View to display the measurement.

   LabVIEW SignalExpress creates a table to display the scalar measurement from the **dc** output.

5. Drag the **rms** output to the table to display the scalar RMS measurement.

   LabVIEW SignalExpress creates a new row in the table to display the second measurement. The project should appear as shown in Figure 3-2.

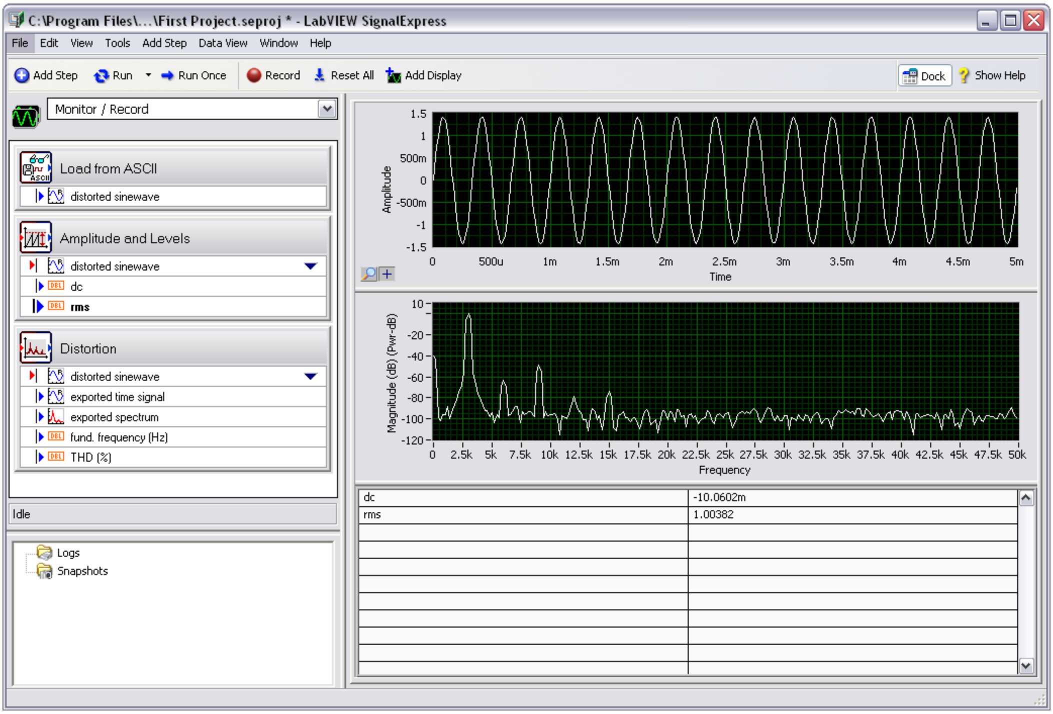


Figure 3-2. Outputs of First Project.seproj

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 in the **Step Setup** dialog box, or configuration view, for that step. While a project runs, you can modify the configuration of steps to view immediate feedback on the changes and adjust the 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. The Step Setup dialog box for the Distortion step appears, as shown in Figure 3-3.

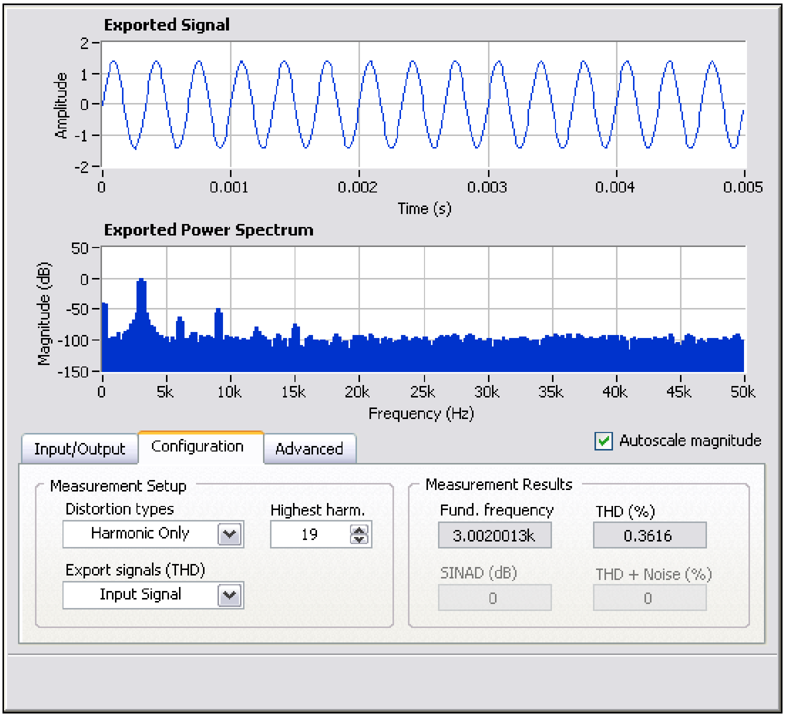


Figure 3-3. Distortion Step Setup Dialog Box

On the **Configuration** tab, the settings indicate that the Distortion step receives a time-domain waveform signal as an input, performs a power spectrum on the signal to convert it to the frequency domain, and computes the total harmonic distortion (THD) and fundamental frequency of the signal. The step generates 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, click the **Show Help** button, shown at left, to display complete reference information about the step. The upper half of the context help displays information about the step, and the lower half of the context help displays information about an object when you move the cursor over the object.

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

   The operation of the step 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. The graphed signals in the top half of the **Step Setup** dialog box update to reflect the change you made.

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

   The step changes to export only the spectrum of the harmonic signals from the input signal. Both the output signal of the Distortion step and the graph on the Data View update to reflect the change you made.

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

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

6. Click the **Input/Output** tab to display the list of possible inputs and outputs for this step, as shown in Figure 3-4.

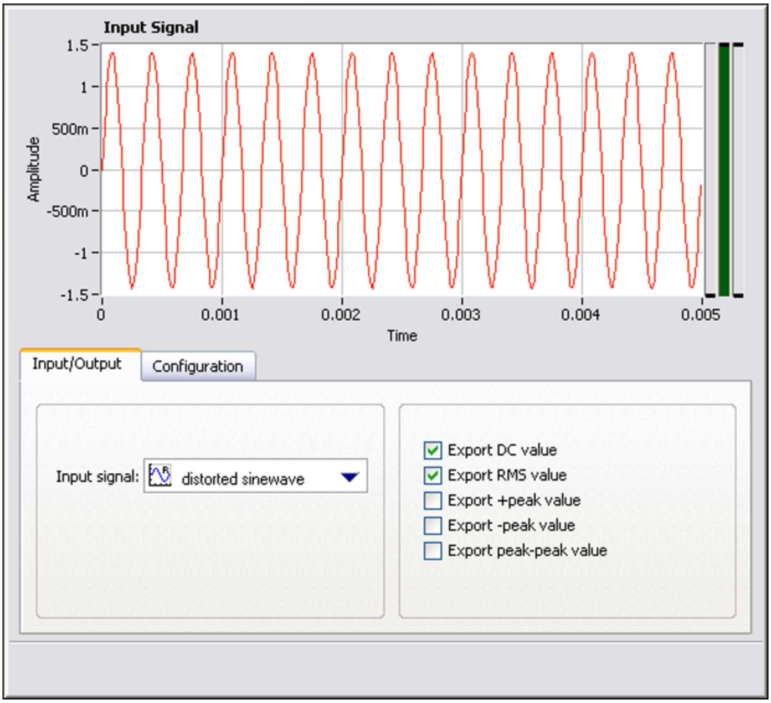


Figure 3-4. Amplitude and Levels Step Setup Dialog Box

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. Click the **Data View** tab to display the Data View.

9. Drag the three new outputs from the Project View to the table to display the scalar measurements.

10. Click the **Stop** button, shown at left, 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 click the **Abort** button 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 `SignalExpress\Examples\Tutorial` directory.
12. Select **File»Close Project** to close the project.

Moving and Deleting Steps

The steps in SignalExpress projects depend on input data, which means steps can operate only on signals exported from previous steps in the Project View. When you click the **Input signal** pull-down menu on the **Input** tab of the **Step Setup** dialog box of a step, the menu 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 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 shortcut menu. However, 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 shortcut 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** dialog box. Click the **Details** button to the right of the error description to display the full error description.

SignalExpress logs all errors and warnings in the Event Log while a project runs. To display the Event Log, select **View»Event Log** and click the **Event Log** tab in the Data View area. Refer to the *LabVIEW SignalExpress Help* for more information about errors and warnings by selecting **Help»LabVIEW SignalExpress Help**, clicking the **Search** tab, and entering `errors`.

4

Working with Signals

You can use LabVIEW SignalExpress to generate and analyze signals to evaluate designs without programming. This chapter teaches you how to work with signals in LabVIEW 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 **File»Open Project**, navigate to the `SignalExpress\Examples\Tutorial` directory, and double-click `Signals.seproj`. This project configures the Create 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 Signal step to the Data View.
3. Drag the **filtered step** output of the Filter step to the Data View.

   You can plot signals of the same signal type on the same graph. Both the **step signal** and the **filtered step** signals are time-domain signals, so they appear on the same graph. If you want to plot signals of different types, drag the signals to the same graph and LabVIEW SignalExpress creates a new graph.

4. Click the **Add Display** button, shown at left, on the toolbar to create a new graph.
5. Drag the **filtered step** output of the Filter step to the new graph.

6. Right-click the new graph and select **Visible Items»Cursors** from the shortcut menu to display two interactive cursors, as shown in Figure 4-1.

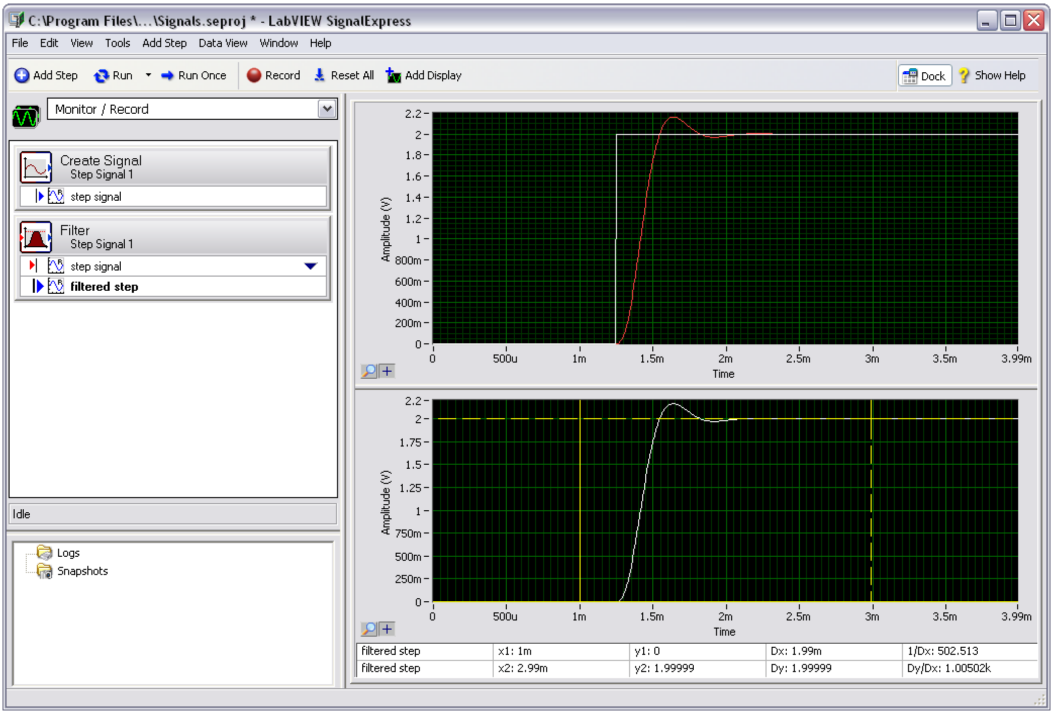


Figure 4-1. Signals.seproj

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

7. Select **File»Save Project As** and save the project as `My Signals.seproj` in the `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 EDA tools such as SPICE simulators.

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

1. Right-click the Project View and select **Load/Save Signals»Analog Signals»Load from ASCII** from the shortcut menu.

2. Click the browse button, shown at left, in the **Step Setup** dialog box, navigate to the `SignalExpress\Examples\Tutorial` directory, and double-click `Step Response.txt`. This step parses an ASCII file and displays the signals in the file.

   In the **File preview** section, column 1 shows the time stamp data, and column 2 shows the actual voltage values of the signal.
3. Click the **Import Signals** tab to display the available signals in the file.
4. Place a checkmark in the **Column 2** checkbox to import that signal, and remove the checkmark from the **Column 1** checkbox.

   The **Step Setup** dialog box displays a preview of the signal in the **Imported Signal** section.
5. Select **Column 1** from the **Input X values** pull-down menu to set the x-axis data of the waveform to the appropriate values.
6. Open the Data View.
7. Right-click the **Column 2** output in the Project View and select **Rename** from the shortcut menu.
8. Enter `step response` and press the <Enter> key to rename the output.
9. Drag the **step response** output of the Load from ASCII step to the lower graph in the Data View.

   The **filtered step** signal resembles the rising edge of the **step response** output, as shown in Figure 4-2.

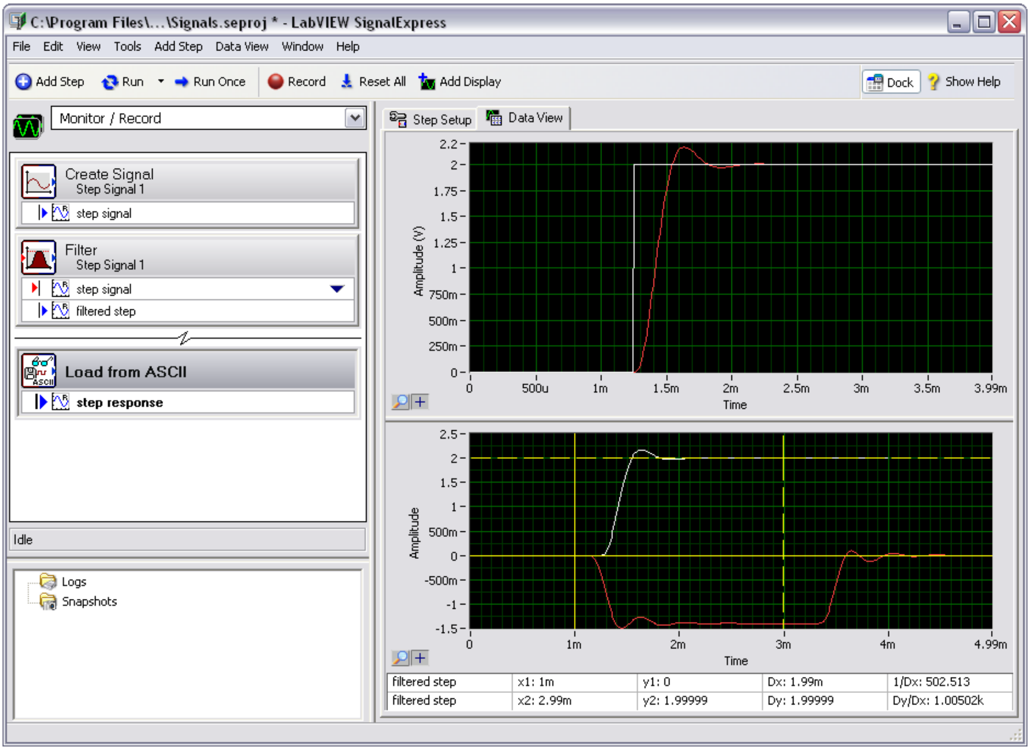


Figure 4-2. Signals of Signals.seproj

10. 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. Right-click the **step response** output and select **Send To» Processing»Analog Signals»Interactive Alignment** from the shortcut menu to pass the **step response** signal from the Load from ASCII step to the Interactive Alignment step.

The step selects the two most recent signals from the project to use as inputs and displays the signals on the graph of the **Step Setup** dialog box, as shown in Figure 4-3.

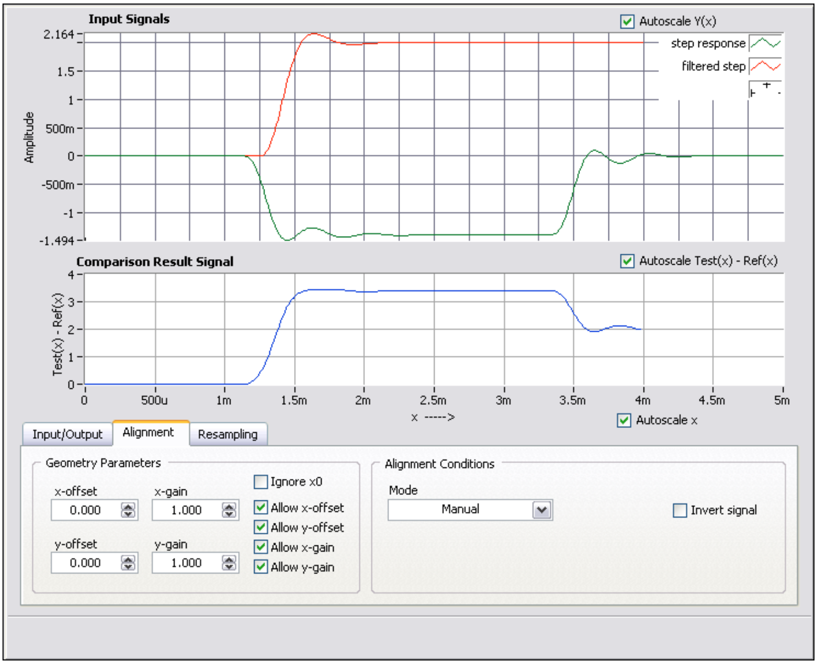


Figure 4-3. Interactive Alignment Step Setup Dialog Box

When you add a step to a project, LabVIEW 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. To change the input signals for a step, select different signals from the pull-down menu of compatible signals in the **Input/Output** tab of the **Step Setup** dialog box.

2. Click the red signal in the **Input Signals** graph and drag it to another point within the graph.

   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 to stretch the signal around that anchor point in the x and y directions.

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

4. Select **Auto - Step** from the **Mode** pull-down menu to align the signals. LabVIEW SignalExpress bases this alignment mode on built-in algorithms.

   The lower graph in the **Step Setup** dialog box displays the difference between the two signals.
5. On the **Input/Output** tab, place a checkmark in the **Export aligned signals** checkbox to add the signals to the outputs of the step.
6. Select the **Data View** tab to open the Data View.

7. Click the **Add Display** button, shown at left, to add a third graph.
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 LabVIEW 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, LabVIEW SignalExpress adds only their amplitudes. However, if you add two frequency-domain phase signals, LabVIEW SignalExpress adds the appropriate phase shift.

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

Exporting and Printing Signals

You can use LabVIEW 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 documentation feature to document the LabVIEW SignalExpress project.

Saving Signals to File

Complete the following steps to save a signal to a file.

1. Click the **Add Step** button and select **Load/Save Signals»Analog Signals»Save to ASCII/LVM**.
2. Click the **Signals** tab in the **Step Setup** dialog box and select **filtered step** from the **Input Data** pull-down menu.
3. On the **File Settings** tab, save the file to `SignalExpress\Examples\Tutorial\filtered signal.txt` in the Export file path control.
4. Select **Overwrite** from the **If file already exists** pull-down menu.
5. Select **Generic ASCII** 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.
6. Click the **Close** button to close the **Step Setup** dialog box.
7. Click the **Run** button 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.

Exporting Signals to Microsoft Excel

To export signal data to Microsoft Excel, launch Excel and drag the output signal of a step in LabVIEW SignalExpress to an Excel spreadsheet.

Printing Signals

To print an image of a graph, open the Data View and select **File»Print»Print Data View**.

Creating Reports in LabVIEW SignalExpress

Select **View»Project Documentation** to display the Project Documentation View. You can describe your project using text and drag and drop step outputs into your documentation. When you drag and drop a step output into the Project Documentation View, the value in the Project Documentation View automatically updates to match the current value of the step output.

To print your documentation, open the Project Documentation View and select **File»Print»Print Documentation**. To export your documentation to HTML, open the Project Documentation View and select **File»Export»Export Documentation to HTML**.

5

Logging Data

You can use LabVIEW SignalExpress to record and analyze measurements. You can record any time-domain, double, U32, or Boolean 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 LabVIEW 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 View 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 **File»Open Project**, navigate to the `SignalExpress\Examples\Tutorial` directory, and double-click `Logging.seproj`.

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

2. Click the **Record** button, shown at left, to open 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 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.

5. Click the **Stop** button, shown at left, to stop logging the signal.

   The logged data appears in the **Logged Data** window, as shown in Figure 5-1.

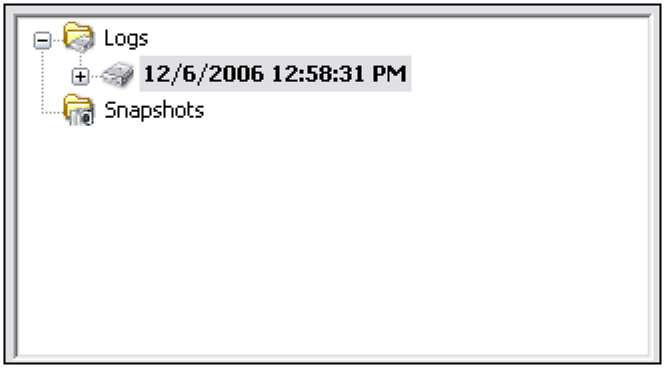


Figure 5-1. Logged Data Window

   By default, LabVIEW SignalExpress names the logged data according to the date and time you recorded the data. LabVIEW 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 LabVIEW 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** to save the project.

Viewing a Logged Signal

Complete the following steps to view the logged data.

1. If the Data View is not visible, select **View»Data View** to display the Data View.
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 into the Data View. The Data View displays the logged data and a preview graph, as shown in Figure 5-2. The signal displayed in the Data View may differ from the signal displayed in Figure 5-2, depending on how long you record the signal.

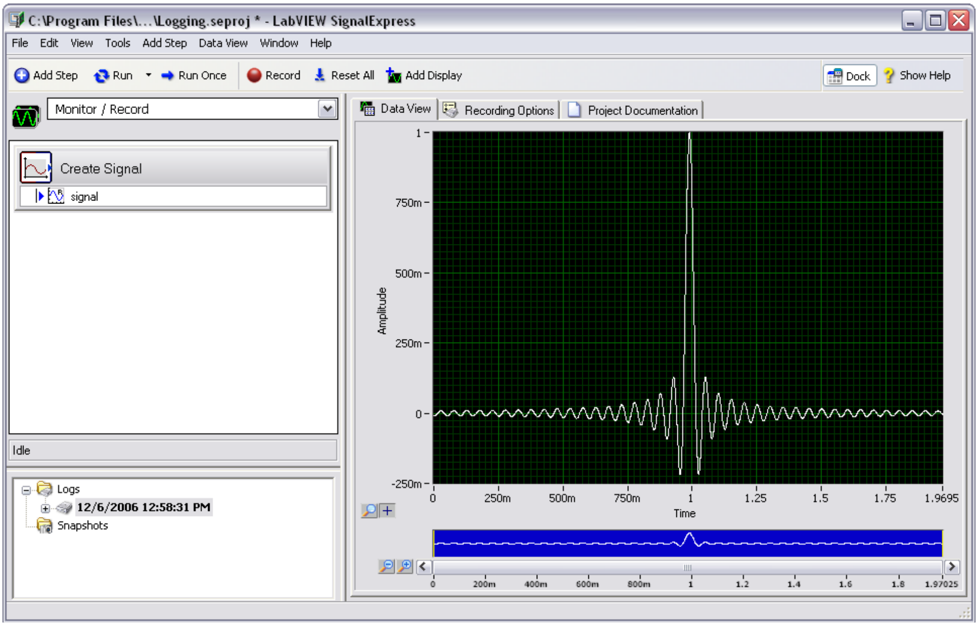


Figure 5-2. Logging.seproj

The preview graph provides a method for zooming and panning through data in the Data View. The preview graph appears by default when viewing logged data. When viewing live or non-logged data, right-click the Data View and select **Visible Items»Preview** to display the preview graph.

3. 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 preview graph. 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 the signals must meet before LabVIEW 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 View.

2. Select **Signal Selection** in the **Category** list in the Recording Options View.
3. Place a checkmark next to the signal in the **Record** column, as shown in Figure 5-3.

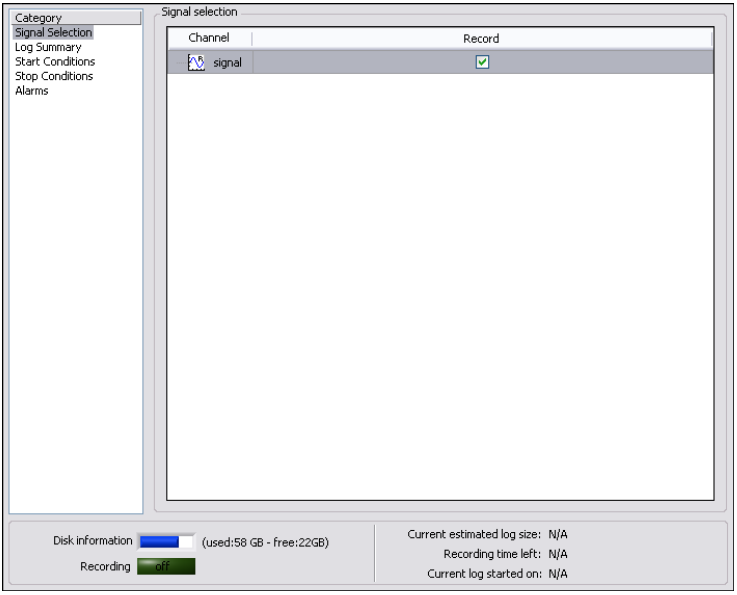
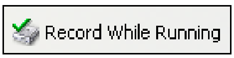


Figure 5-3. Signal Selection



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, LabVIEW SignalExpress records the selected signal when you click the **Run** or **Run Once** button.

4. Select **Start Conditions** in the **Category** list in the Recording Options View.
5. Click the **Add** button in the **Logging start conditions** page to customize a start condition for your logging task.
   a. Select the **Signal** option in the **Condition source** control to specify for LabVIEW SignalExpress to begin recording when the input signal meets the specified condition.
   b. Select **signal** in the **Signal** control.

c. Select **Rising slope** in the **Condition** control to begin recording the signal based on the value of the edge of the signal on a positive slope.

d. Enter `1` in the **Value** control to begin recording when the signal crosses 1 on a rising slope.

6. Select **Stop Conditions** in the **Category** list in the Recording Options View.

7. Click the **Add** button in the **Logging stop conditions** page to customize a stop condition for your logging task.

a. Select the **Duration** option in the **Condition source** control.

b. Enter `5` in the **Duration** control to record the signal for 5 seconds after the signal meets the start condition.

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

The **Recording** indicator on the bottom of the Recording Options View will turn **on** when the signal meets the start condition and logging is in progress. The **Disk information** indicator displays the available hard disk space on the computer for the log.

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. Navigate to the **Work Area** pull-down menu on the top left of the LabVIEW SignalExpress window, as shown in Figure 5-4. Click the down arrow and select **Playback** to switch to the Playback work area.

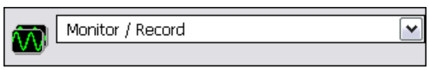


Figure 5-4. Work Area pull-down menu

Use work areas to perform multiple LabVIEW 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 your project, LabVIEW 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. You must be in the Playback work area to use logged data as an input for an analysis step. For example, you can take a measurement and log the signal in the Monitor/Record work area, and then run the logged data through an analysis step in the Playback work area.

2. Click the **Add Step** button, shown at left, and select the Filter step from **Processing»Analog Signals»Filter**. LabVIEW SignalExpress automatically selects the first signal you logged as the input to the Filter step.
3. Navigate to the Data View. Drag the **filtered data** output of the Filter step into the Data View to view the resulting signal.

4. Click the **Run** button, shown at left. The Data View displays the resulting filtered signal and LabVIEW SignalExpress plays back the entire log.

Advanced Playback

You can configure advanced data playback options by using the Playback Options View. The Playback Options View 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 open the Playback Options View, as shown in Figure 5-5. The Playback Options View allows you to preview the logged data, as well as specify a subset of the logged data to play back or send to analysis steps.

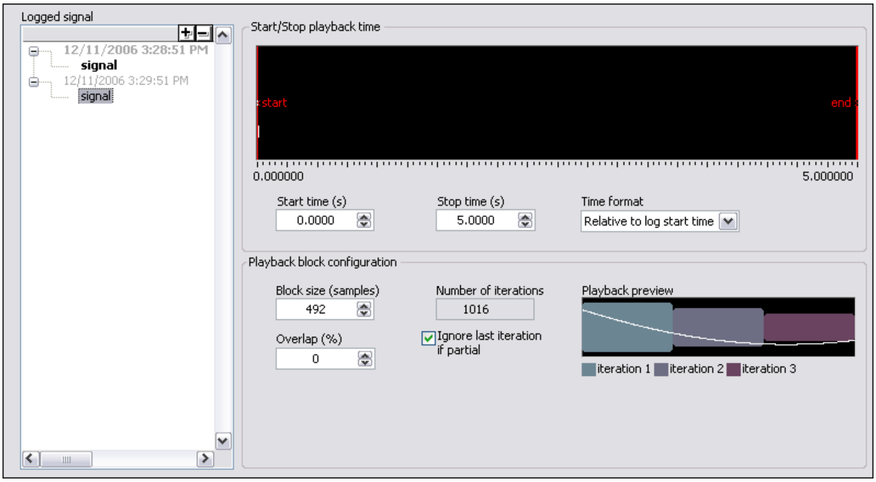


Figure 5-5. Playback Options View

2. Select **signal** from the second log you created in the **Logged signal** listbox.
3. Enter `1` in the **Start time** control 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** control to play back or analyze a subset of the logged signal ending four seconds after the start of the log.
5. Switch to the Data View and drag the filtered data output of the Filter step into the Data View.

6. Click the **Run** button, shown at left. LabVIEW SignalExpress filters the subset of the signal you specified in the Playback Options View and displays the resulting filtered signal in the Data View.
7. Click the **Stop** button to stop running the project.

Refer to the *LabVIEW SignalExpress Help* for more information about logging data, such as specifying alarm conditions and playback options.

6

Performing Sweep Measurements

You can use LabVIEW SignalExpress to automate measurements to characterize and validate designs by creating sweep operations. 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 LabVIEW 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 LabVIEW 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 **File»Open Project**, navigate to the `SignalExpress\Examples\Tutorial` directory, and double-click `Sweep.seproj`.

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

   The project generates a sine wave stimulus signal using the Create 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 analyses, or multifunction I/O (MIO) device.

3. Click the **Add Step** button, shown at left, and select **Execution Control»Sweep**.
4. Click the **Add** button in the **Step Setup** dialog box to display the list of sweep parameters from each step in the project, as shown in Figure 6-1.

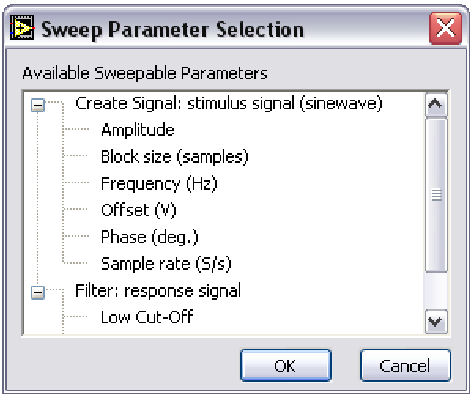


Figure 6-1. Sweep Parameter Selection Dialog Box

5. Select the **Frequency** parameter under **Create Signal** and click the **OK** button.

   The Sweep step encloses the Create Signal step, which provides the signal to sweep.
6. On the **Sweep Configuration** tab, select **Exponential** from the **Type** pull-down menu.
7. Enter `1k` in the **Start Frequency (Hz)** field, and enter `40k` in the **Stop Frequency (Hz)** field.
8. Enter `150` in the **Number of points** field.

   The **Sweep Configuration** tab should appear as shown in Figure 6-2.

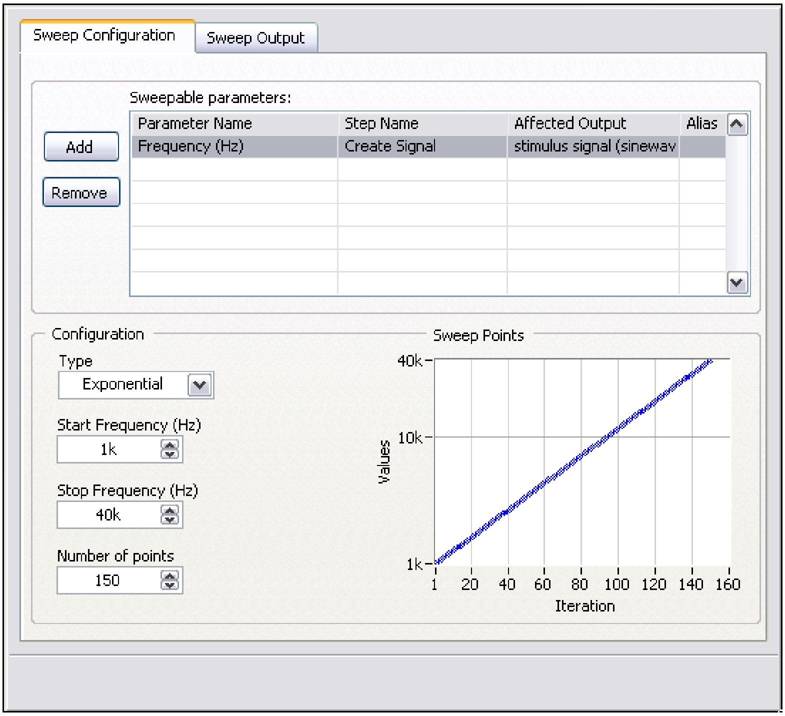


Figure 6-2. Sweep Step Setup Dialog Box

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

9. Click the **Sweep Output** tab.
10. Click the **Add** button to display the list of sweep outputs from each step in the project.
11. Select the **response amplitude in dB** output under **Formula** and click the **OK** button to plot this measurement against the swept **Frequency** parameter.

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

Plotting Sweep Results

Complete the following steps to run the sweep measurement.

1. Click the **Run Once** button, shown at left, to execute the sweep measurement.

   The **stimulus signal** output on the graph 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 to display the output of the sweep.

   LabVIEW SignalExpress creates a new graph. The data from a sweep operation is an x-y array that requires a separate graph, as shown in Figure 6-3.

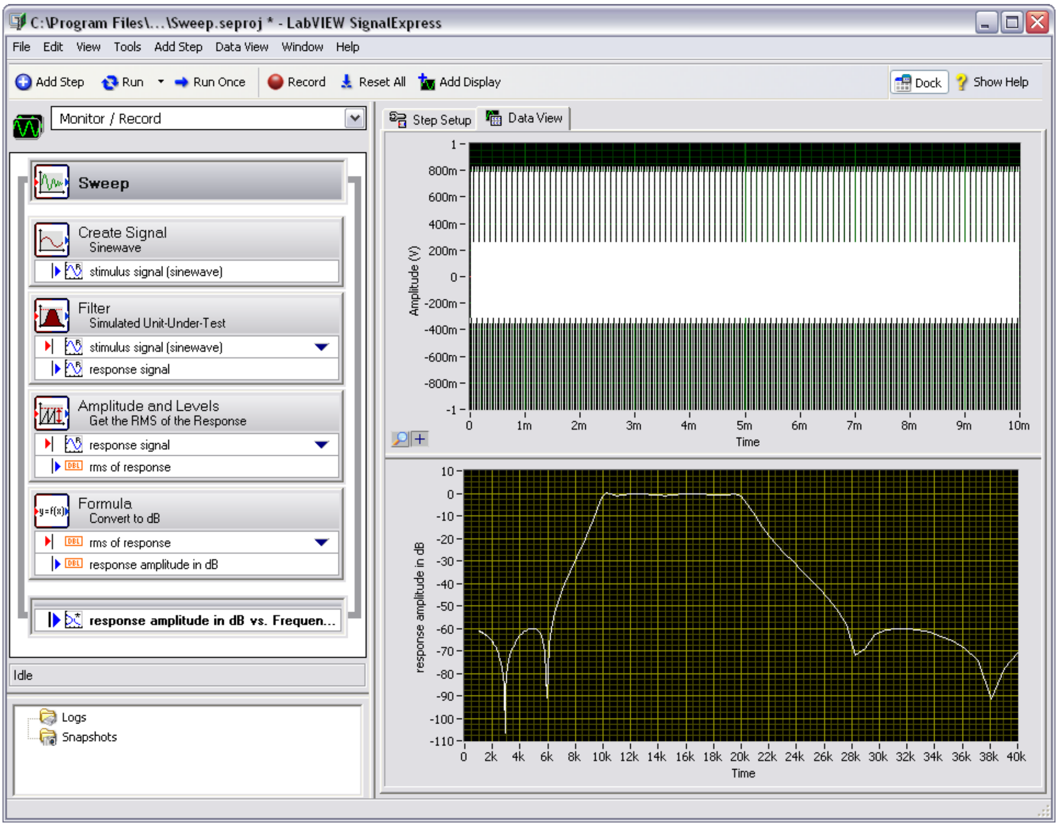


Figure 6-3. Sweep.seproj

3. Click the **Run Once** button again to execute the sweep.

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

4. Double-click the Filter step to display the filter specifications in the **Step Setup** dialog box.

   The frequency response of the filter in the **Filter Magnitude Response** graph matches the graph in the Data View.

5. Select **File»Save Project As** and save the project as `My Sweep.seproj` in the `SignalExpress\Examples\Tutorial` directory.

You can use the Sweep step to sweep multiple parameters simultaneously by adding additional parameters on the **Sweep Configuration** tab of the **Sweep Step Setup** dialog box. 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. You can use 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, and 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 Dimension** from the shortcut menu to add another sweep loop.

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

1. Select **File»Open Project**, navigate to the `SignalExpress\Examples\Tutorial` directory, and double-click `Nested Sweep.seproj`.

2. Click the **Run** button, shown at left, 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, and then appears on the lower graph to display all the sweeps at each cutoff frequency setting, as shown in Figure 6-4.

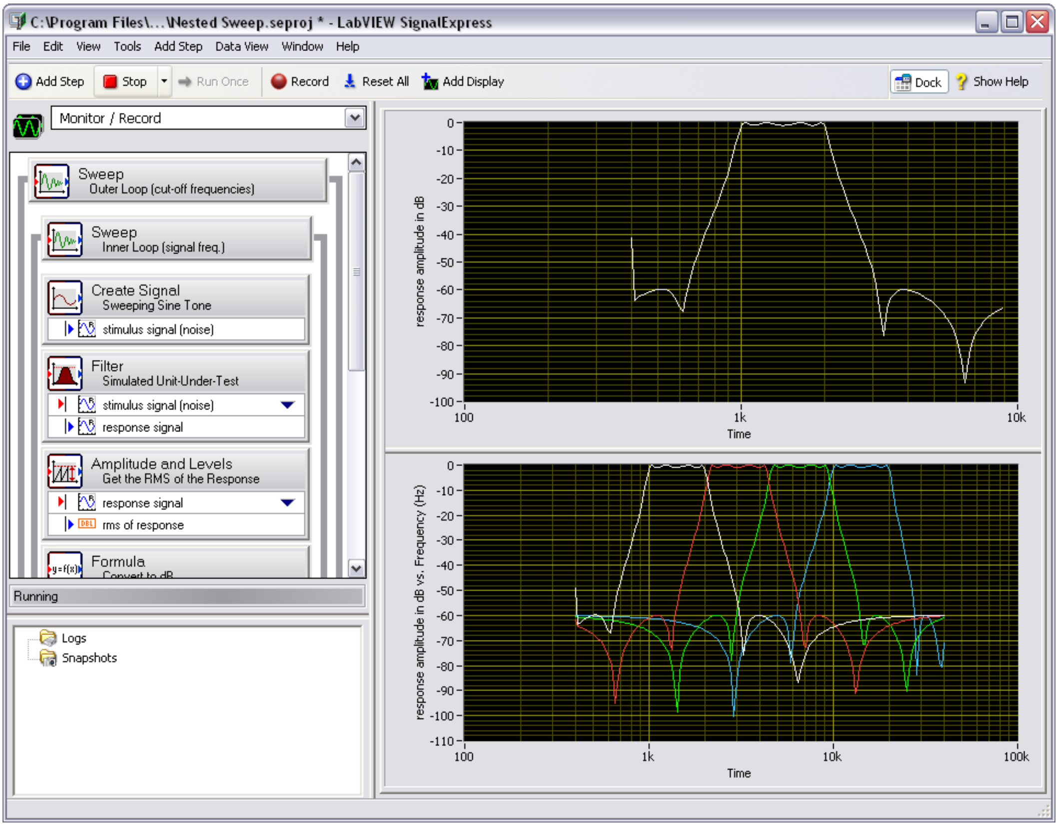


Figure 6-4. Nested Sweep.seproj

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

7

Extending LabVIEW SignalExpress Projects with LabVIEW

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

- Build a VI in LabVIEW and import this VI into LabVIEW SignalExpress to provide custom step functionality and expand the number of steps available in LabVIEW SignalExpress.
- Convert a LabVIEW SignalExpress project to a LabVIEW block diagram to continue development in LabVIEW.

You must have LabVIEW 7.1 or later to complete the exercises in this chapter.

Importing LabVIEW VIs into LabVIEW SignalExpress as Steps

Use the Run LabVIEW VI step in LabVIEW SignalExpress to call custom LabVIEW VIs. You can call a LabVIEW VI from LabVIEW SignalExpress to do the following:

- Control GPIB instruments
- Control National Instruments hardware that LabVIEW 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 **File»Open Project**, navigate to the `SignalExpress\Examples\Tutorial` directory, and double-click `User Step.seproj`.

   This project uses the Create Signal step to generate a signal.

2. Click the **Add Step** button and select **Run LabVIEW VI»Run LabVIEW 8.2 VI**.

   The VI you run in this exercise was saved in LabVIEW 8.2. You must use the version of the Run LabVIEW VI step that matches the version of LabVIEW you saved your VI in.

3. Click the browse button on the **Step Setup** dialog box and select `Limiter-LV82.vi` in the `LabVIEW SignalExpress\Examples\Tutorial` directory. The Limiter-LV82 VI accepts a time-domain waveform as an input, clips the signal above and below values that you specify in the **Step Setup** dialog box, and returns the clipped waveform as an output signal.

   When you import a LabVIEW VI, LabVIEW SignalExpress maps the inputs of the VI as parameters and the outputs of the VI as output signals in LabVIEW 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** dialog box 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** dialog box appears as shown in Figure 7-1.

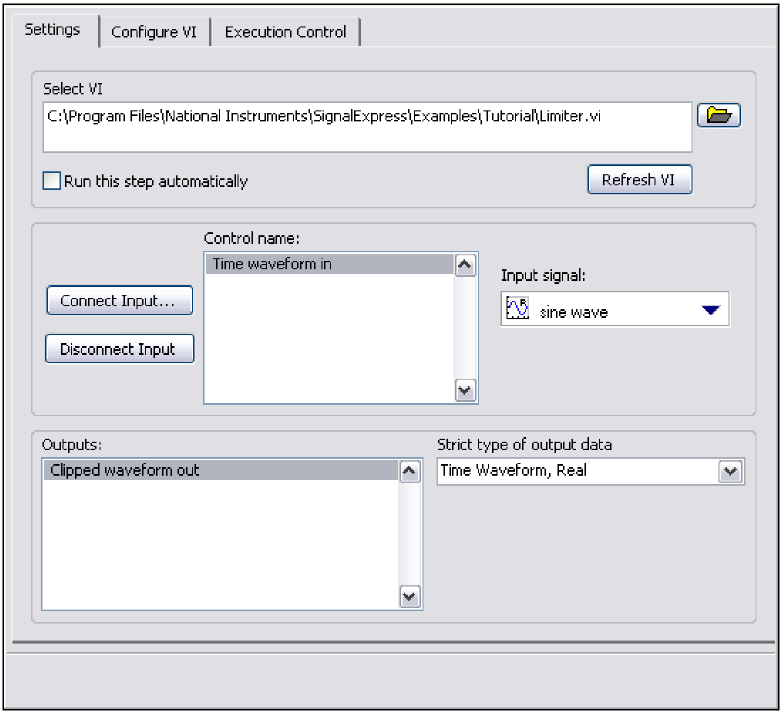


Figure 7-1. Limiter VI Step Setup Dialog Box

4. Click the **Run** button, shown at left, to run the project.
5. Drag the **Clipped waveform out** output signal from the Limiter.vi step to the Data View.
6. Double-click the Limiter.vi step to display the **Step Setup** dialog box.
7. Select the **Configure VI** tab 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. Click the **Apply** button to apply the changes.
10. Click the **Close** button to close the VI and click the **Close** button to close the **Step Setup** dialog box.

    The **Clipped waveform out** signal changes on the Data View to reflect the changes you made.

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

Converting LabVIEW SignalExpress Projects to LabVIEW Block Diagrams

LabVIEW SignalExpress can convert LabVIEW SignalExpress projects into LabVIEW block diagrams.

Note To convert an LabVIEW SignalExpress project to a LabVIEW VI, you must have the LabVIEW 7.1 Full Development System or greater installed.

Complete the following steps to convert a LabVIEW SignalExpress project to a LabVIEW block diagram.

1. Select **File»Open Project**, navigate to the `SignalExpress\Examples\Tutorial\Solutions` 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.

   The resulting LabVIEW block diagram reflects the execution of the project in LabVIEW SignalExpress. The LabVIEW block diagram consists of LabVIEW Express VIs wired together. Each Express VI correlates to a step in the LabVIEW SignalExpress project. You can double-click an Express VI to display the same **Step Setup** dialog box as in LabVIEW SignalExpress. You also can right-click an Express VI and select **Open Front Panel** from the shortcut 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 LabVIEW SignalExpress project with logging, LabVIEW SignalExpress generates a LabVIEW block diagram with one Express VI. You cannot convert the generated Express VI into a subVI.

8

Where to Go from Here

Refer to the following resources for more information about LabVIEW SignalExpress.

LabVIEW SignalExpress Sample Projects

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

Using Hardware with LabVIEW SignalExpress

This manual does not describe how to control hardware from LabVIEW SignalExpress because LabVIEW 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 *LabVIEW SignalExpress Help* by selecting **Help»LabVIEW SignalExpress Help** for more information about using hardware with LabVIEW SignalExpress.

Web Resources

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

A

Technical Support and Professional Services

Visit the following sections of the National Instruments Web site at `ni.com` for technical support and professional services:

- **Support**—Online technical support resources at `ni.com/support` include the following:
  - **Self-Help Resources**—For immediate answers and solutions, visit the award-winning National Instruments Web site 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.
  - **Free Technical Support**—All registered users receive free Basic Service, which includes access to hundreds of Application Engineers worldwide in the NI Developer Exchange at `ni.com/exchange`. National Instruments Application Engineers make sure every question receives an answer.
- **Training and Certification**—Visit `ni.com/training` for self-paced training, eLearning virtual classrooms, interactive CDs, and Certification program information. You also can 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`.

If you searched `ni.com` and could not find the answers you need, contact your local office or NI corporate headquarters. Phone numbers for our worldwide offices are listed at the front of this manual. You also can visit the Worldwide Offices section of `ni.com/niglobal` to access the branch office Web sites, which provide up-to-date contact information, support phone numbers, email addresses, and current events.