LabVIEW™ SignalExpress™

Getting Started with LabVIEW SignalExpress

Tektronix® Edition

July 2008
374297D-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 43 662 457990-0, Belgium 32 (0) 2 757 0020, Brazil 55 11 3262 3599, Canada 800 433 3488, China 86 21 5050 9800, Czech Republic 420 224 235 774, Denmark 45 45 76 26 00, Finland 358 (0) 9 725 72511, France 01 57 66 24 24, Germany 49 89 7413130, India 91 80 41190000, Israel 972 3 6393737, Italy 39 02 41309277, Japan 0120-527196, 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 6851, 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 2005151, 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.

Tektronix® and Tek are registered trademarks of Tektronix, Inc. 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 media, 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 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.

Conventions

The following conventions are used 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 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.
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, sections of code, programming examples, and syntax examples. 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.

Contents

1

Introduction to LabVIEW SignalExpress Tektronix Edition

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.

Tektronix and National Instruments have partnered to help increase your productivity with the most seamless oscilloscope-to-computer connectivity solution in the industry.

LabVIEW SignalExpress Tektronix Edition for the Design Lab

LabVIEW SignalExpress Tektronix Edition is fully interactive measurement acquisition, analysis, and documentation software. LabVIEW SignalExpress Tektronix Edition also includes USB plug-and-play automatic configuration and connectivity. Using LabVIEW SignalExpress Tektronix Edition, you can take advantage of the following:

- Ability to configure and connect Tektronix instruments with the first, true USB plug-and-play connectivity solution in the industry.
- An intuitive drag-and-drop user interface that does not require any programming.
- Ability to capture, transfer, and save measurement settings, waveforms, and screen images.

- Ability to import simulated results and compare them to actual measured results live on-screen.
- More than 200 measurement, processing, analysis, and reporting operations that perform in real time.

LabVIEW SignalExpress Tektronix Edition also comes with built-in support for the following:

- Tektronix DPO2000, DPO3000, DPO4000, MSO2000, MSO4000, TDS1000, TDS2000, and TDS3000 series oscilloscopes
- Tektronix AFG3000 series arbitrary/function generators
- Over 400 other common standalone/modular instruments

Using LabVIEW SignalExpress Tektronix Edition

You can use LabVIEW SignalExpress Tektronix Edition to control instruments and define measurement procedures by adding and configuring functions called steps to the environment. A step acquires, generates, analyzes, loads, or stores signals. The **Add Step** menu and the **Add Step** palette show the steps available in LabVIEW SignalExpress Tektronix Edition.

LabVIEW SignalExpress Tektronix Edition includes Tektronix-specific steps you can use to control Tektronix DPO2000, DPO3000, DPO4000, MSO2000, MSO4000, TDS1000, TDS2000, and TDS3000 series oscilloscopes, and Tektronix AFG3000 series arbitrary/function generators.

You configure the operation of a step by specifying values on the **Step Setup** tab for the step. You also use this tab to control instruments using Tektronix-specific steps. Refer to Chapter 3, *Connecting and Controlling Instruments*, for more information about using LabVIEW SignalExpress Tektronix Edition to control instruments.

A saved sequence of configured steps is a LabVIEW SignalExpress Tektronix Edition project. Refer to Chapter 4, *Working with Projects*, for more information about LabVIEW SignalExpress Tektronix Edition projects.

Extending LabVIEW SignalExpress Tektronix Edition Projects with LabVIEW

You can use National Instruments LabVIEW to extend the functionality of LabVIEW SignalExpress Tektronix Edition. You can build virtual instruments (VIs) with LabVIEW and import them into LabVIEW SignalExpress Tektronix Edition to expand the built-in measurement options. You also can convert a LabVIEW SignalExpress Tektronix Edition project to a LabVIEW block diagram so you can continue development in the LabVIEW environment.

Refer to Chapter 8, *Extending LabVIEW SignalExpress Tektronix Edition Projects with LabVIEW*, for more information about using LabVIEW to extend the capabilities of LabVIEW SignalExpress Tektronix Edition.

2

Getting Started with LabVIEW SignalExpress Tektronix Edition

This chapter provides information about LabVIEW SignalExpress Tektronix Edition system requirements, installation instructions, version availability, and available licensing options.

Installing LabVIEW SignalExpress Tektronix Edition

Before you begin the exercises in this manual, you must install LabVIEW SignalExpress Tektronix Edition.

Minimum System Requirements

To run LabVIEW SignalExpress Tektronix Edition, National Instruments recommends that your system meet the following requirements:

- 512 MB of memory
- Pentium 4 processor or equivalent (Pentium III or Celeron 600 MHz minimum)

Installation Instructions

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

1. Exit all programs before you run the LabVIEW SignalExpress Tektronix Edition installer. Applications that run in the background, such as virus scanning utilities, might cause the installer to take longer than average to complete.
2. Log on as an administrator or as a user with administrator privileges.
3. Insert the LabVIEW SignalExpress Tektronix Edition media and follow the instructions that appear on the screen.

Note Unless you specify another location during installation, the LabVIEW SignalExpress Tektronix Edition installation program copies files to the `<Program Files>\National Instruments\SignalExpressTekEd` 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 the `<Program Files>\National Instruments\SignalExpress\Examples\Tutorial` directory. The solutions for each exercise are located in the `<Program Files>\National Instruments\SignalExpress\Examples\Tutorial\Solutions` directory.

LabVIEW SignalExpress Tektronix Edition also includes Tektronix-specific examples, which are located in the `<Program Files>\National Instruments\SignalExpressTekEd\Examples\Tutorial` directory.

LabVIEW SignalExpress Tektronix Edition Version Availability

LabVIEW SignalExpress Tektronix Edition 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 2-1. Differences in LabVIEW SignalExpress Tektronix Edition Full and LE Versions

	Full	LE
Instrument Support		
DPO2000 Digital Phosphor Oscilloscopes	✓	✓
DPO3000 Digital Phosphor Oscilloscopes	✓	✓
DPO4000 Digital Phosphor Oscilloscopes	✓	✓
MSO2000 Mixed Signal Oscilloscopes	✓	✓
MSO4000 Mixed Signal Oscilloscopes	✓	✓
TDS1000 Digital Storage Oscilloscopes	✓	✓
TDS2000 Digital Storage Oscilloscopes	✓	✓
TDS3000 Digital Phosphor Oscilloscopes	✓	✓

Table 2-1. Differences in LabVIEW SignalExpress Tektronix Edition Full and LE Versions (Continued)

	Full	LE
AFG3000 Arbitrary/Function Generators	✓	✓
Over 400 common standalone instruments	✓	✓
Visualization and Documentation		
Customizable graphing	✓	✓
Interactive cursors	✓	✓
Save signals to file	✓	✓
Print and export graphs	✓	✓
Drag and drop data into Microsoft Excel, Word, and WordPad	✓	✓
Interactive Project Documentation view	✓	—
Signal Processing		
Software filters	✓	—
Scalar and waveform math	✓	—
Analog and digital conversion	✓	—
Interactive signal comparisons	✓	—
Load simulation data from PSPICE, Multisim, and other SPICE packages	✓	—
Time and Frequency Measurements		
Amplitude and level	✓	—
Timing and transition	✓	—
Power spectrum	✓	—
Frequency response	✓	—
Distortion measurements	✓	—
Tone extraction	✓	—

Table 2-1. Differences in LabVIEW SignalExpress Tektronix Edition Full and LE Versions (Continued)

	Full	LE
Data Logging		
Limited data logging (one log per project)	✓	✓
Unlimited data logging	✓	—
Logging alarms and events	✓	—
Logging with start or stop conditions	✓	—
Measurement Automation		
Parameter sweeping	✓	—
Limit testing	✓	—
Software triggering	✓	—

LabVIEW SignalExpress Tektronix Edition Licensing Options

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

LabVIEW SignalExpress Tektronix Edition LE

The limited, unlicensed LE version of LabVIEW SignalExpress Tektronix Edition gives you access to all features in Table 2-1 for 30 days. After that period, LabVIEW SignalExpress Tektronix Edition runs in unlicensed mode. If you do not activate a valid license within 30 days and you create a project that contains licensed steps, the following occurs:

- Each time you drop a step that is licensed or not Tektronix-specific, a dialog box prompts you to activate the software.
- You cannot save a project.
- Projects close after 10 minutes.

Note Tektronix-specific steps are not licensed steps.

Refer to the National Instruments Web site at ni.com/tek to purchase the LabVIEW SignalExpress Tektronix Edition Full version.

LabVIEW SignalExpress Tektronix Edition Full Version

The full version of LabVIEW SignalExpress Tektronix Edition provides you with the full functionality shown in Table 2-1. You can activate the full version of LabVIEW SignalExpress Tektronix Edition using the National Instruments License Manager or the LabVIEW SignalExpress Tektronix Edition installer.

3

Connecting and Controlling Instruments

LabVIEW SignalExpress Tektronix Edition provides native support for the Tektronix DPO2000, DPO3000, DPO4000, MSO2000, MSO4000, TDS1000, TDS2000, TDS3000 series oscilloscopes, and AFG3000 series signal generators. This chapter covers the process of connecting instruments to the PC and performing signal generation and acquisition from within LabVIEW SignalExpress Tektronix Edition. Refer to Chapter 4, *Working with Projects*, for more information about the LabVIEW SignalExpress Tektronix Edition environment.

LabVIEW SignalExpress Tektronix Edition also supports over 400 of the most common standalone/modular instruments, including the majority of instruments in the Tektronix oscilloscope and signal generator product lines. Refer to the *Using Hardware with LabVIEW SignalExpress Tektronix Edition* section in Chapter 9, *Where to Go from Here*, for more information about controlling other instruments.

Connecting Instruments Using a USB Device Cable

You can use a USB device cable to connect the following Tektronix instruments to a PC:

- DPO2000 Digital Phosphor Oscilloscopes
- DPO3000 Digital Phosphor Oscilloscopes
- DPO4000 Digital Phosphor Oscilloscopes
- MSO2000 Mixed Signal Oscilloscopes
- MSO4000 Mixed Signal Oscilloscopes
- TDS1000B Digital Storage Oscilloscopes
- TDS2000B Digital Storage Oscilloscopes
- AFG3000 Arbitrary/Function Generators

Connecting over USB

With built-in USB plug-and-play capabilities, if you use a USB device cable to connect a Tektronix instrument to a PC, LabVIEW SignalExpress Tektronix Edition automatically detects the instrument and allows you to control the oscilloscope or arbitrary/function generator and acquire or generate signals immediately. Complete the following steps to connect a Tektronix oscilloscope or arbitrary/function generator to a PC using a USB device cable.

1. Turn on the oscilloscope or arbitrary/function generator.
2. Using a USB device cable, plug the Type B, device-side connector into the USB device port on the back of the oscilloscope or arbitrary/function generator, as shown in Figures 3-1 and 3-2. Connect the Type A, host-side connector to the computer.

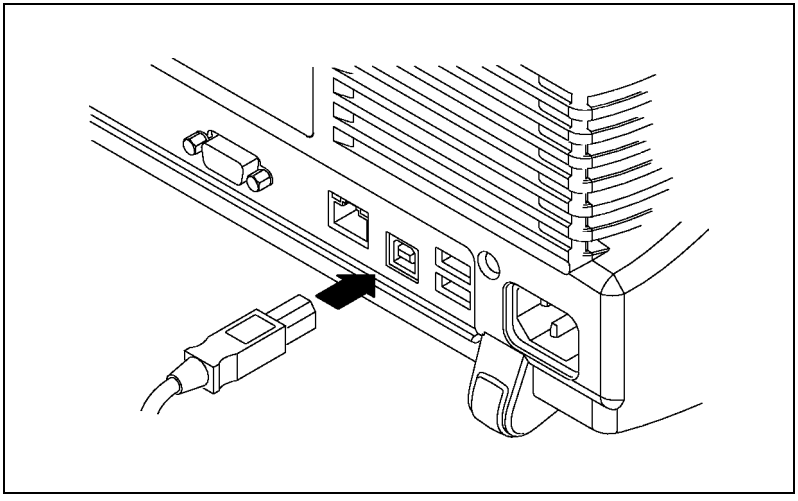


Figure 3-1. Connecting a USB Device Cable to the Back of a DPO4000 Series Oscilloscope

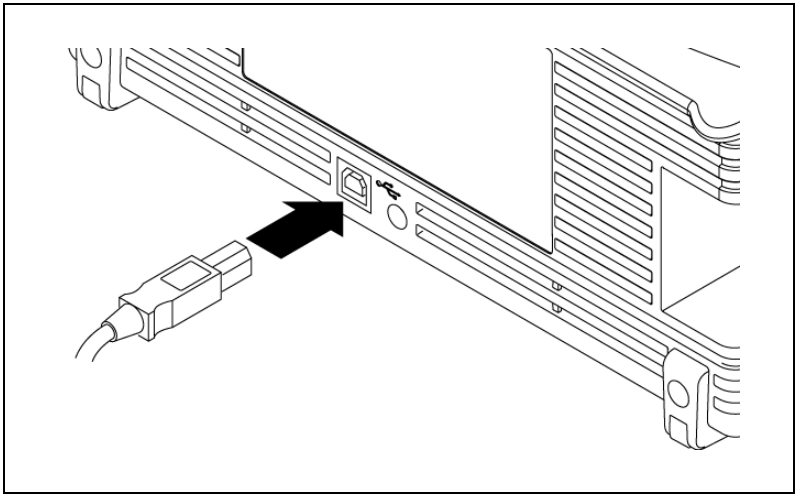


Figure 3-2. Connecting a USB Device Cable to the Back of a TDS1000B or TDS2000B Series Oscilloscope

After you complete the connections, Windows detects that you have connected a new device to the computer and prompts you to install the driver for the instrument.

3. Follow the instructions on screen and click the **Next** button on the dialog boxes that appear to finish installing the instrument driver.

Note You only need to perform step 3 the first time you plug the oscilloscope or arbitrary/function generator into the computer. Subsequent connections immediately launch the dialog box discussed in step 4.

4. Windows detects the instrument and a dialog box appears that prompts you for the action to take, as shown in Figure 3-3. Select **Control Instrument using LabVIEW SignalExpress** and click the **OK** button.

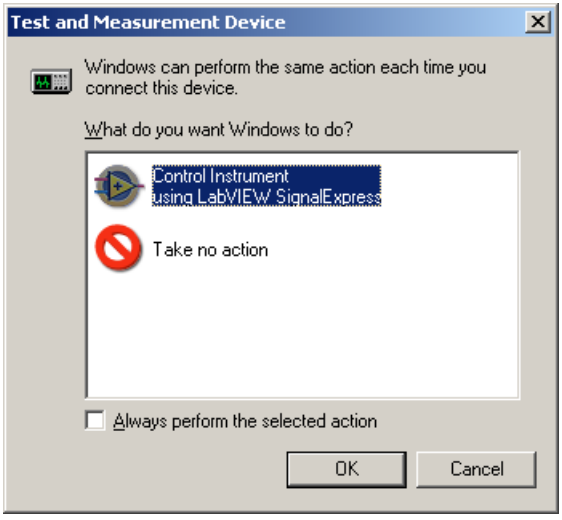


Figure 3-3. Windows Test and Measurement Device Dialog Box

5. LabVIEW SignalExpress Tektronix Edition launches and provides a live view of the signal from the oscilloscope or arbitrary/function generator. Figure 3-4 shows an example signal acquired from a DPO4000 series oscilloscope.

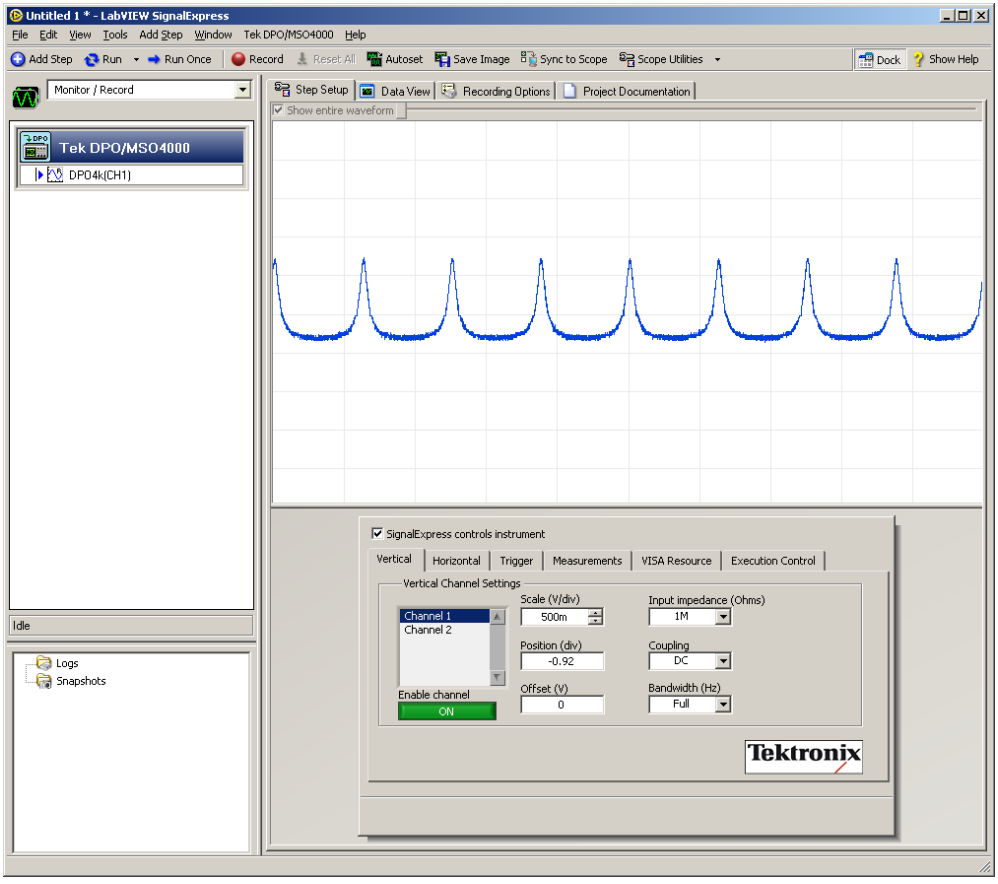


Figure 3-4. Live View of a Signal from a Tektronix DPO4000 Series Oscilloscope

Figure 3-5 shows an example signal acquired from an AFG3000 series arbitrary/function generator.

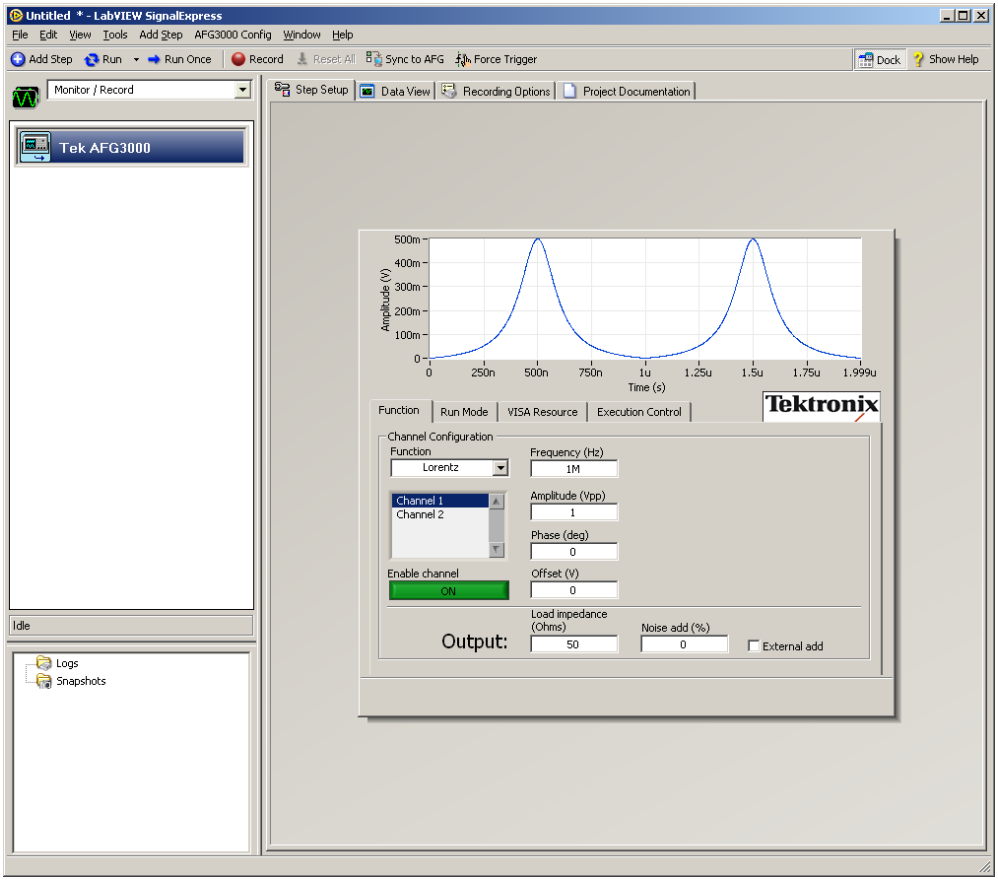


Figure 3-5. Live View of a Signal from a Tektronix AFG3000 Series Arbitrary/Function Generator

The settings on the **Step Setup** tab reflect the current configuration of the instrument. Adjust these settings to control the instrument from LabVIEW SignalExpress Tektronix Edition.

Connecting Instruments Using an Ethernet Cable

You can use an Ethernet cable to connect the following Tektronix instruments to a PC:

- DPO2000 Digital Phosphor Oscilloscopes
- DPO3000 Digital Phosphor Oscilloscopes
- DPO4000 Digital Phosphor Oscilloscopes
- MSO2000 Mixed Signal Oscilloscopes
- MSO4000 Mixed Signal Oscilloscopes
- TDS3000 Digital Phosphor Oscilloscopes
- AFG3000 Arbitrary/Function Generators

Refer to the user manual for the instrument to connect the instrument to a PC and configure the instrument for communication over Ethernet.

Controlling an Instrument over Ethernet

After you connect and configure an oscilloscope, complete the following steps to access the oscilloscope from within LabVIEW SignalExpress Tektronix Edition.

1. Launch LabVIEW SignalExpress Tektronix Edition.

2. Click the **Add Step** button, shown at left, to display the **Add Step** palette.
3. On the **Add Step** palette, select **Textronix»Acquire Signals»** ***StepName***, where ***StepName*** is the name of the step that corresponds to the instrument you are using. The **Step Setup** tab for the step appears, as shown in Figure 3-6 for the DPO/MSO4000 step.

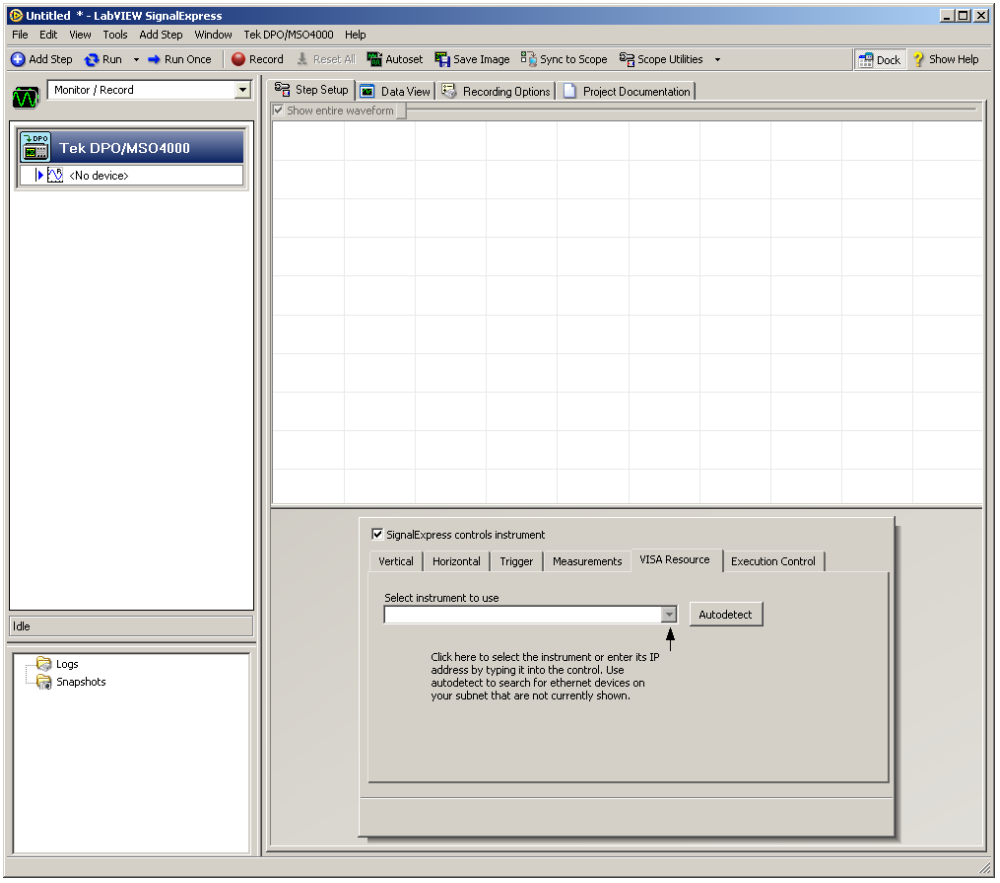


Figure 3-6. Step Setup Tab for Tektronix DPO4000 and MSO4000 Series Oscilloscopes

4. If you know the IP address of the connected instrument, enter the IP address in the **Select instrument to use** field on the **VISA Resource** page of the **Step Setup** tab.

   If you do not know the IP address, click the **Autodetect** button on the **VISA Resource** page to scan the local subnet for a connected instrument. If LabVIEW SignalExpress Tektronix Edition detects an instrument, the application displays the IP address and connects to the instrument. If the instrument is an oscilloscope, LabVIEW SignalExpress Tektronix Edition performs one waveform acquisition.

Note You must enter the IP address manually when connecting to an instrument that is not on the local subnet. Refer to the *LabVIEW SignalExpress Help* for information about how to manually connect to a Tektronix DPO2000, DPO3000, DPO4000, MSO2000, MSO4000, or TDS3000 series oscilloscope or an AFG3000 series arbitrary/function generator by selecting **Help»LabVIEW SignalExpress Help**, clicking the **Search** tab, and entering the name of the instrument you are using, such as "`DPO4000`".

5. Click the **Run** button, shown at left, to continually acquire a signal from an oscilloscope or to start generating a signal using an arbitrary/function generator.
6. Use the parameters on the **Step Setup** tab to control the instrument from LabVIEW SignalExpress Tektronix Edition.

Connecting Instruments Using a GPIB Cable

You can use a GPIB cable to connect the following Tektronix instruments to a PC:

- TDS3000 Digital Phosphor Oscilloscopes
- AFG3000 Arbitrary/Function Generators

Connecting over GPIB

Complete the following steps to connect and control an instrument over GPIB.

1. Connect the instrument to the computer using a GPIB cable.
2. Launch LabVIEW SignalExpress Tektronix Edition.

3. Click the **Add Step** button, shown at left, to display the **Add Step** palette.
   - If you are using a TDS3000 series oscilloscope, select **Tektronix»Acquire Signals»Tek TDS3000**. The **Step Setup** tab for the Tek TDS3000 step appears, as shown in Figure 3-7.

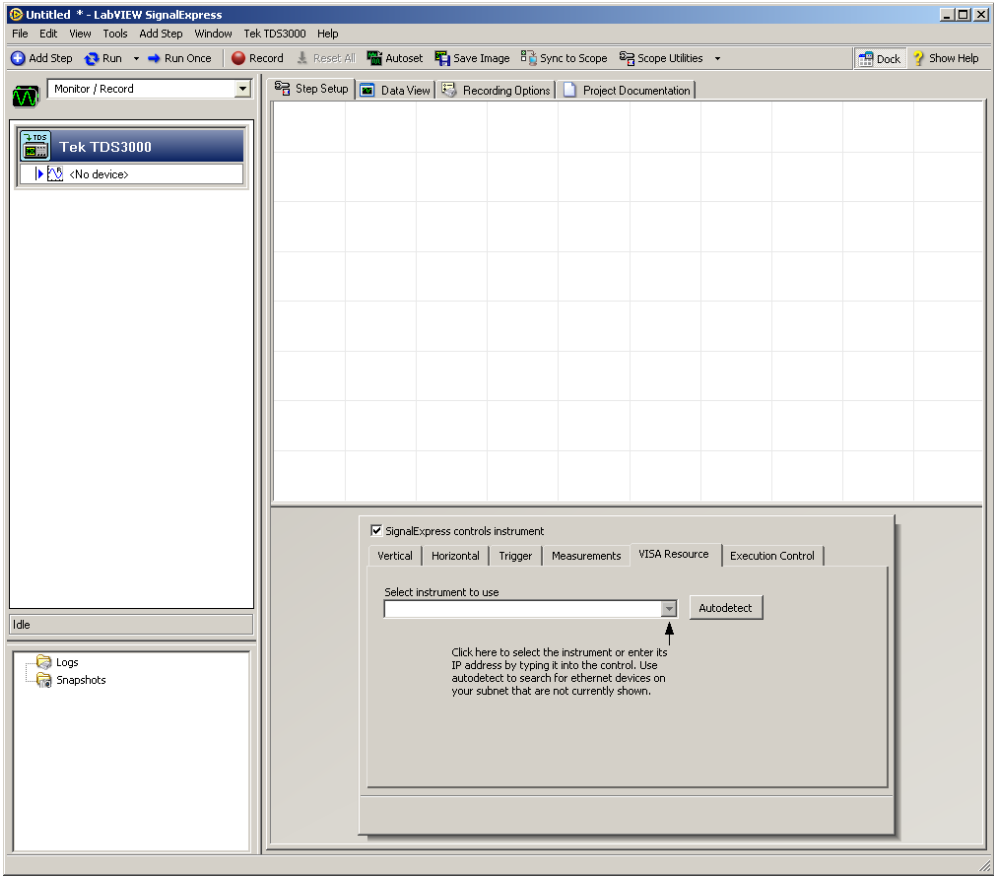


Figure 3-7. Step Setup tab for Tektronix TDS3000 Series Oscilloscopes

If you are using an AFG3000 series arbitrary/function generator, select **Tektronix»Generate Signals»Tek AFG3000**.

4

Working with Projects

You can use LabVIEW SignalExpress Tektronix Edition 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 LabVIEW SignalExpress Tektronix Edition.

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 LabVIEW SignalExpress Tektronix Edition project.

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

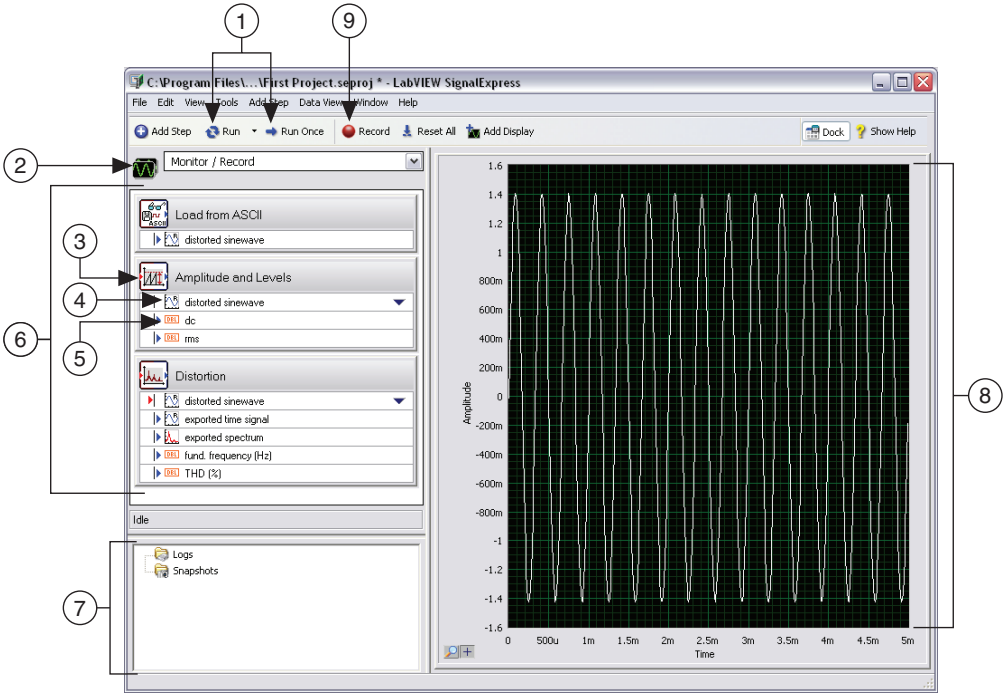
Opening a Project

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

1. Launch LabVIEW SignalExpress Tektronix Edition.

   Notice that LabVIEW SignalExpress Tektronix Edition is split into three main views: the Project View on the left, tabs in the middle, and the context help on the right. If LabVIEW SignalExpress Tektronix Edition opens in the default configuration, the **Data View** tab, the **Recording Options** tab, and the **Project Documentation** tab appear in the middle view.
2. If LabVIEW SignalExpress Tektronix Edition does not open in the default configuration, select **View»Data View** to display the Data View. You can use the **View** menu to show or hide tabs at any time.
3. Select **Help»Open Example**, navigate to the `SignalExpress\Examples\Tutorial` directory, and double-click the `First Project.seproj` LabVIEW SignalExpress Tektronix Edition project to open the project.

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



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 4-1. 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 Tektronix Edition has two execution modes—Run and Run Once. When you click the **Run** button, shown at left, LabVIEW SignalExpress Tektronix Edition 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.

The Data View updates 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. When you click the **Run Once** button, shown at left, LabVIEW SignalExpress Tektronix Edition executes all the steps in the project one time.

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

1. Click the **Run** button 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, LabVIEW SignalExpress Tektronix Edition 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 XY signals.

2. Drag the **exported spectrum** output signal of the Distortion step from the Project View to the Data View to display the signal.

   LabVIEW SignalExpress Tektronix Edition creates a new graph in the Data View. LabVIEW SignalExpress Tektronix Edition 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 Tektronix Edition automatically recognizes different types of signals and renders them in the appropriate displays.

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

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

   LabVIEW SignalExpress Tektronix Edition creates 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

LabVIEW SignalExpress Tektronix Edition uses to plot the output on the adjoining graph.

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

   LabVIEW SignalExpress Tektronix Edition creates a new row in the table to display the second measurement. The project appears as shown in Figure 4-2.

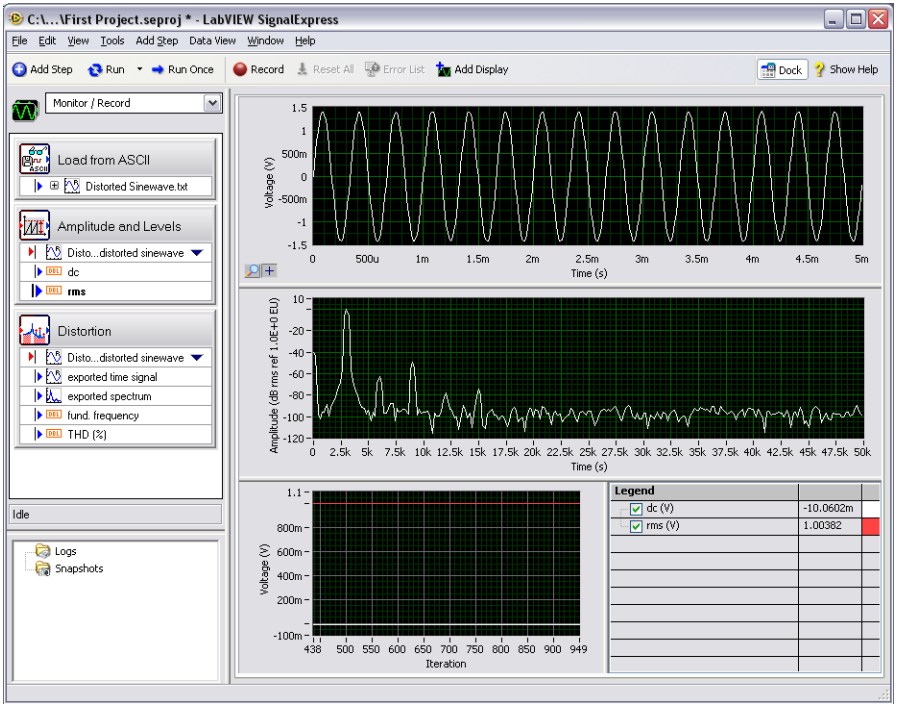


Figure 4-2. Outputs of First Project.seproj

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 LabVIEW SignalExpress Tektronix Edition 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 in the Data View, 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. LabVIEW SignalExpress Tektronix Edition displays the **Step Setup** tab for the Distortion step, which appears as shown in Figure 4-3.

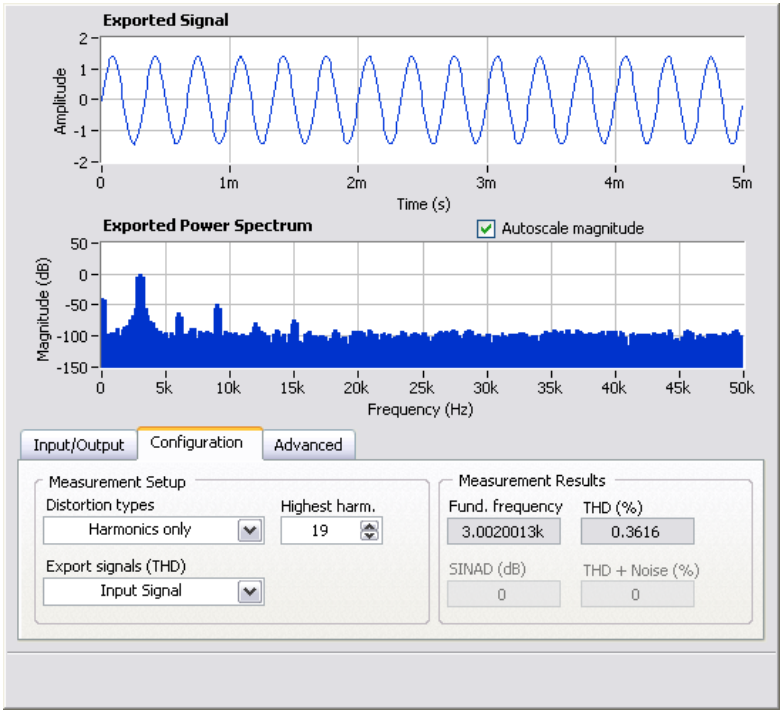


Figure 4-3. Distortion Step Setup Tab

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 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 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 of the **exported spectrum** output on the Data View 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, the **exported spectrum** output, and the graph of the output on the Data View all 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 4-4.

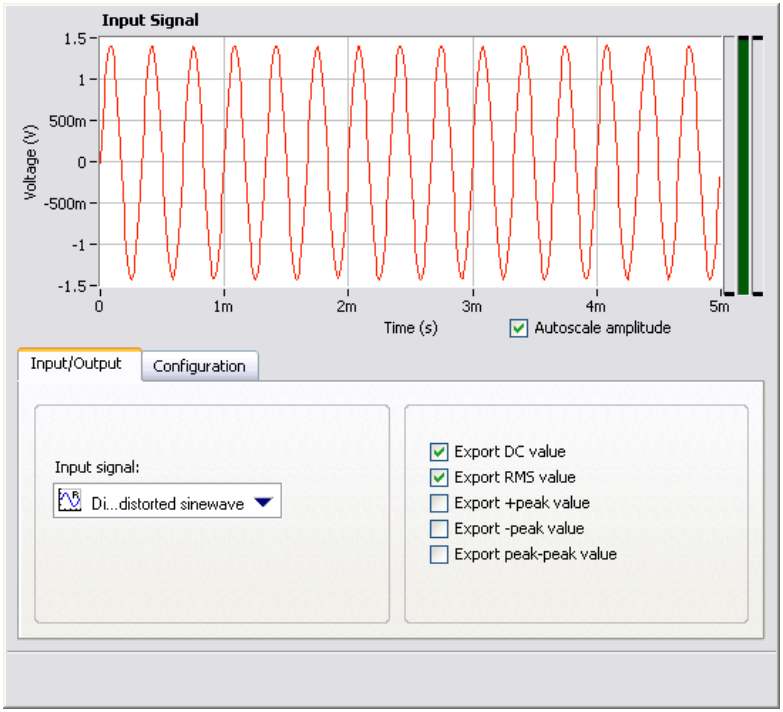


Figure 4-4. Amplitude and Levels Step Setup Tab

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

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

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

Most steps in LabVIEW SignalExpress Tektronix Edition projects require input data, which means steps can operate only on signals exported from previous steps in the Project View. 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 shortcut 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 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** tab. Click the **Details** button to the right of the error description to display the full error description.

LabVIEW SignalExpress Tektronix Edition logs all errors and warnings in the Event Log while a project runs. To display the Event Log, select **View»Event Log**. 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`.

5

Working with Signals

You can use LabVIEW SignalExpress Tektronix Edition to generate and analyze signals without programming. This chapter teaches you how to work with signals in LabVIEW SignalExpress Tektronix Edition, 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 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.
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 Tektronix Edition creates a new graph.

4. Click the **Add Display** button, shown at left, 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 5-1.

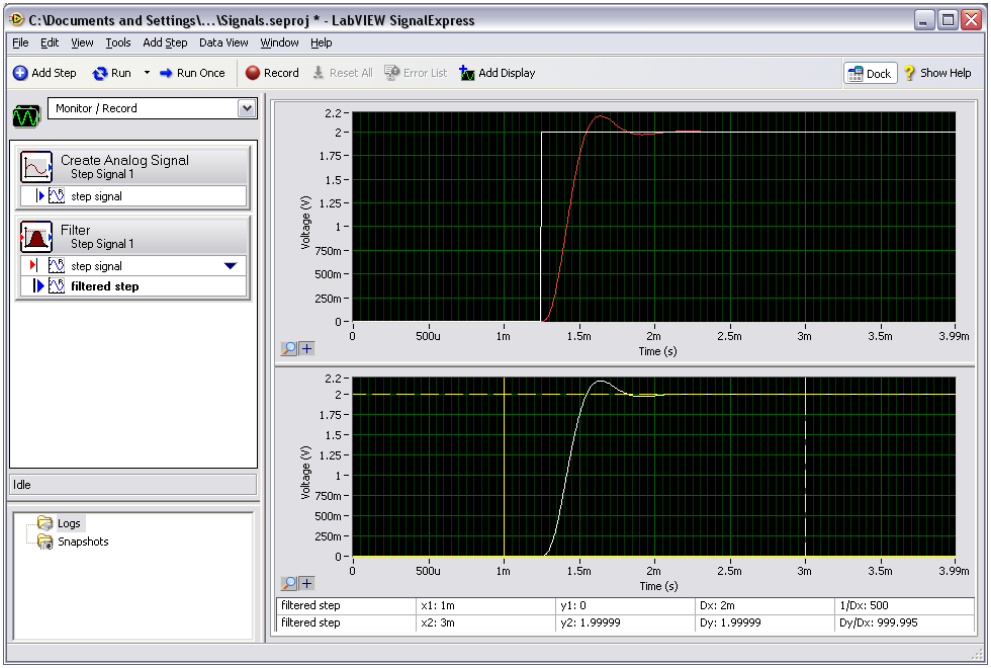


Figure 5-1. Signals.seproj

As you drag the cursors, LabVIEW SignalExpress Tektronix Edition 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 electronic design automation 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 to add the Load from ASCII step to the Project View. The **Step Setup** tab for the Load from ASCII step appears.

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

2. On the **Step Setup** tab, click the browse button, shown at left, 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. Switch to the **Import Signals** page of the **Step Setup** 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** tab displays a preview of the signal on the **Imported Signal** graph.
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. Switch to the Data View.
7. In the Project View, expand the **Step Response** output of the Load from ASCII step.
8. Right-click the **Column 2** output and select **Rename** from the shortcut menu.
9. Enter `step response` and press the <Enter> key to rename the output.
10. Drag the new **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 5-2.

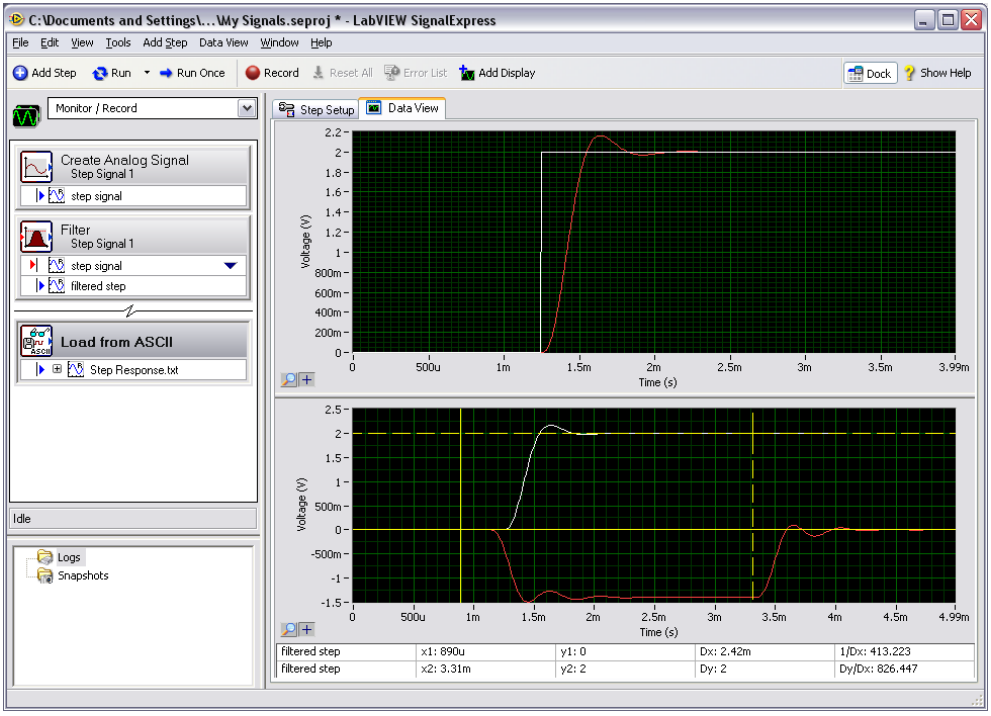


Figure 5-2. Signals of Signals.seproj

11. 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 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 on the **Step Setup** tab, as shown in Figure 5-3.

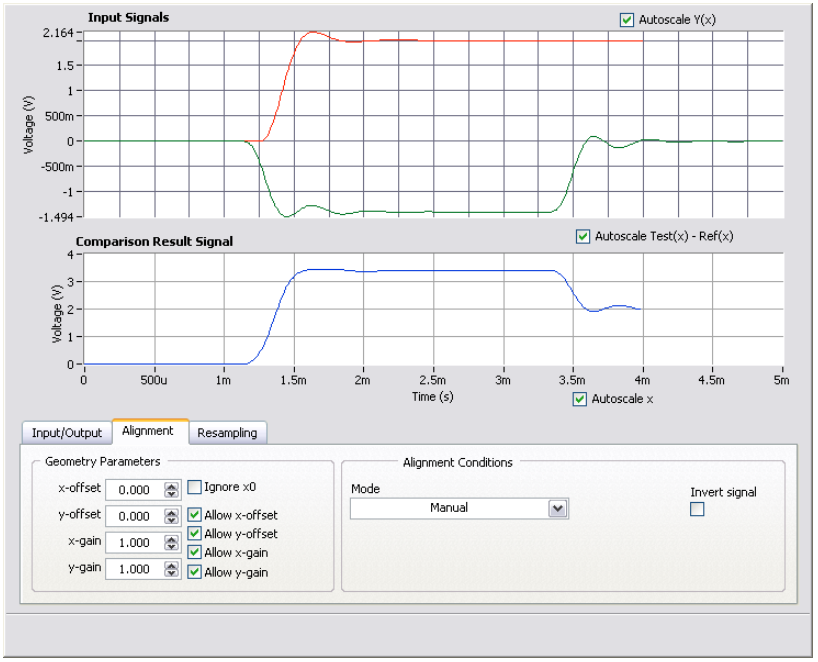


Figure 5-3. Interactive Alignment Step Setup Tab

When you add a step to a project, LabVIEW SignalExpress Tektronix Edition 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 on the **Input/Output** page of the **Step Setup** tab.

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 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. LabVIEW SignalExpress Tektronix Edition bases this alignment mode on 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 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 Tektronix Edition

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 Tektronix Edition adds only their amplitudes. However, if you add two frequency-domain phase signals, LabVIEW SignalExpress Tektronix Edition adds the appropriate phase shift.

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

Exporting and Printing Signals

You can use LabVIEW SignalExpress Tektronix Edition 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 Tektronix Edition project.

Saving Signals to File

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

1. Click the **Add Step** button, shown at left, to display the **Add Step** palette.
2. Select **Load/Save 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 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 new file by entering `SignalExpress\Examples\Tutorial\filtered signal.txt` in the **Export file path** field.
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 **Run Once** 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 Tektronix Edition 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 Tektronix Edition

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

6

Logging Data

You can use LabVIEW SignalExpress Tektronix Edition 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 Tektronix Edition. 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 Analog 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 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.

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, as shown in Figure 6-1.

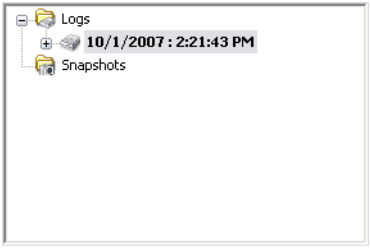


Figure 6-1. Logged Data Window

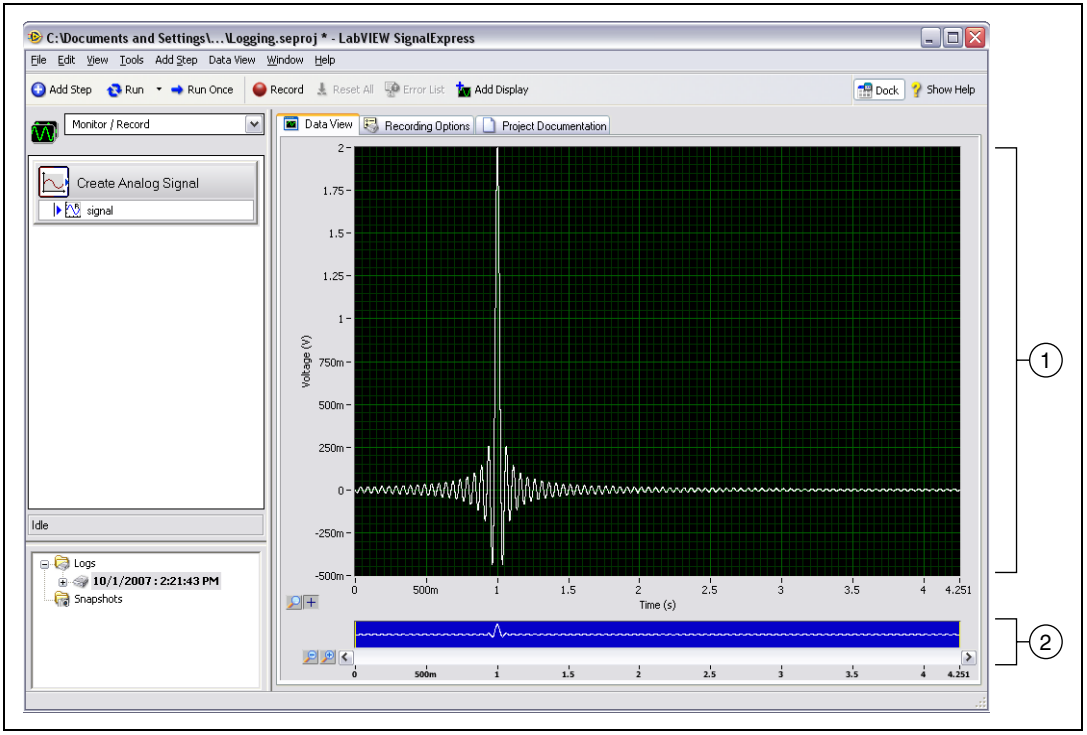
   By default, LabVIEW SignalExpress Tektronix Edition names the logged data according to the date and time you recorded the data. LabVIEW SignalExpress Tektronix Edition saves logged data in the `.tdms` file format in the directory you specify in the **Options** dialog box.

6. Select **Tools»Options** and select the **Logging** option to specify the directory for LabVIEW SignalExpress Tektronix Edition 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 6-2. The signal displayed in the Data View might differ from the signal displayed in Figure 6-2, depending on how long you record the signal.



1	Logged Data	2	Preview Graph

Figure 6-2. 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** from the shortcut menu 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 signals must meet before LabVIEW SignalExpress Tektronix Edition 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 6-3.

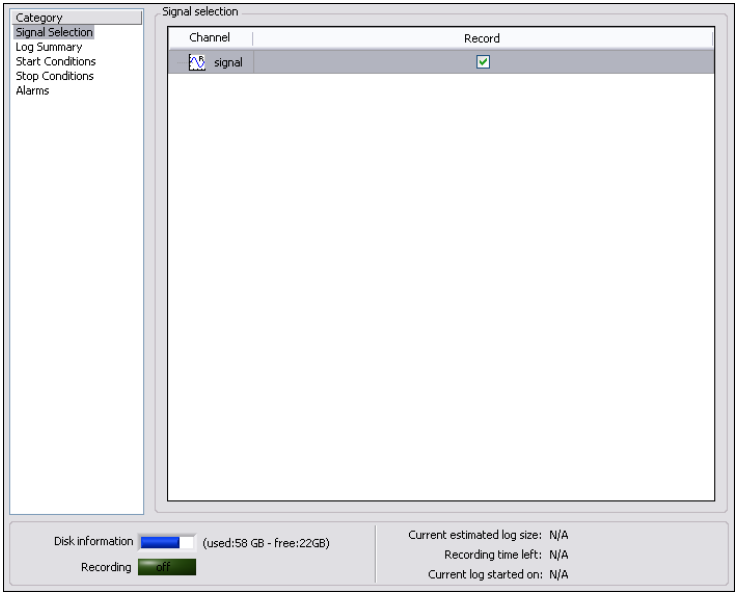
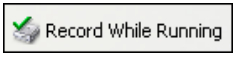


Figure 6-3. Signal Selection



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 Tektronix Edition 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** section to customize a start condition for the logging task.
   a. In the **Condition source** pull-down menu, select **Signal** to specify that LabVIEW SignalExpress Tektronix Edition begin recording when the input signal meets the specified condition.
   b. Select **signal** from the **Signal** pull-down menu that appears.
   c. Select **Rising slope** from the **Condition** pull-down menu 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** field 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** section to customize a stop condition for the logging task.
   a. In the **Condition source** pull-down menu, select **Duration** to specify that LabVIEW SignalExpress Tektronix Edition stop recording after a specified amount of time passes.
   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 Tektronix Edition 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 displays **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 Tektronix Edition window, as shown in Figure 6-4. Click the down arrow and select **Playback** to switch to the Playback work area.

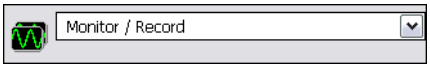
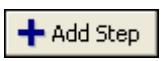


Figure 6-4. Work Area pull-down menu

Use work areas to perform multiple LabVIEW SignalExpress Tektronix Edition 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, LabVIEW SignalExpress Tektronix Edition 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 Tektronix Edition 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 to the Data View to view the resulting signal.
4. Click the **Run** button. The Data View displays the resulting filtered signal and LabVIEW SignalExpress Tektronix Edition 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 display the Playback Options View, as shown in Figure 6-5. The Playback Options View allows you to preview the logged data, and specify a subset of the logged data to play back or send to analysis steps.

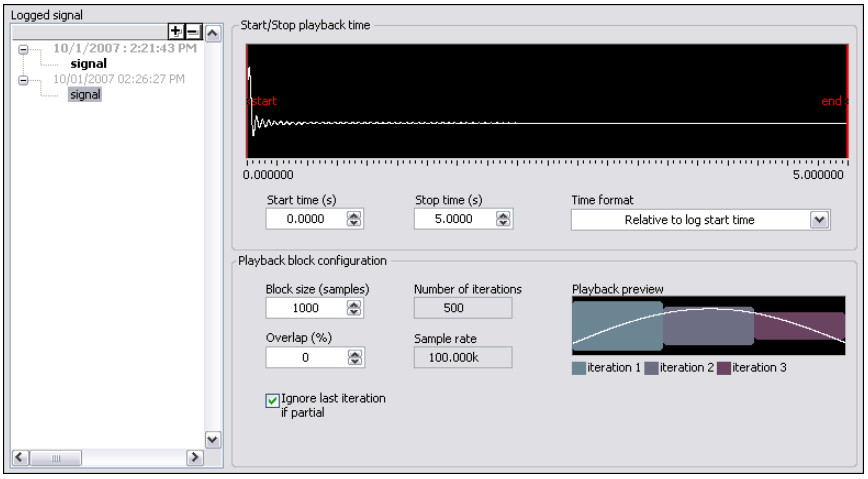


Figure 6-5. Playback Options View

2. Select **signal** from the second log you created in the **Logged signal** listbox.
3. Enter `1` in the **Start time (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.
5. In the **Logged signal** listbox, right-click the **signal** from the second log and select **Activate** from the shortcut 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 shortcut menu to make that log the active log.

6. Switch to the Data View and drag the filtered data output of the Filter step into the Data View.

7. Click the **Run** button, shown at left. LabVIEW SignalExpress Tektronix Edition filters the subset of the signal you specified in the Playback Options View and displays the resulting filtered signal in the Data View.
8. 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.

7

Performing Sweep Measurements

You can use LabVIEW SignalExpress Tektronix Edition 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 LabVIEW SignalExpress Tektronix Edition, such as a circuit board. 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 Tektronix Edition. 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 Tektronix Edition 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 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 Tektronix signal generator or 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.

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 sweep parameters from each step in the project, as shown in Figure 7-1.

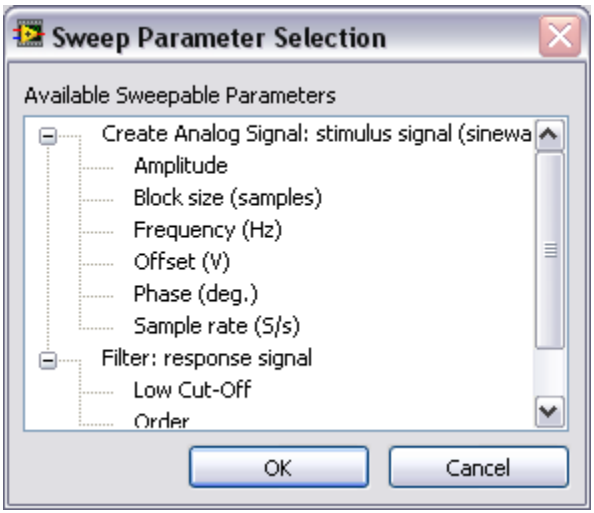


Figure 7-1. Sweep Parameter Selection Dialog Box

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 7-2.

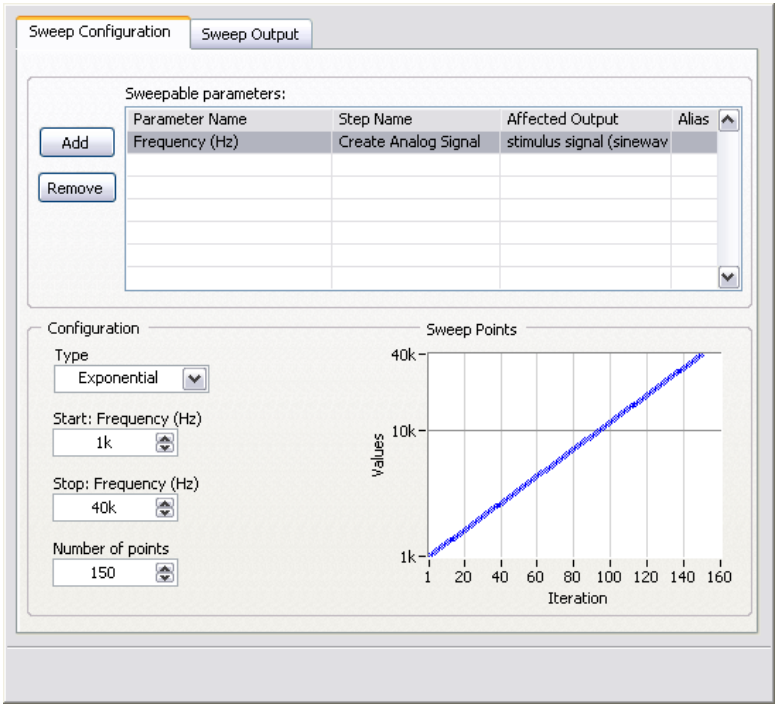


Figure 7-2. Sweep Step Setup Tab

You used 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. However, you can use the Sweep step to sweep 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 sweep 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 and 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 Tektronix Edition creates a new graph. The data from a sweep operation is an XY array that requires a separate graph, as shown in Figure 7-3.

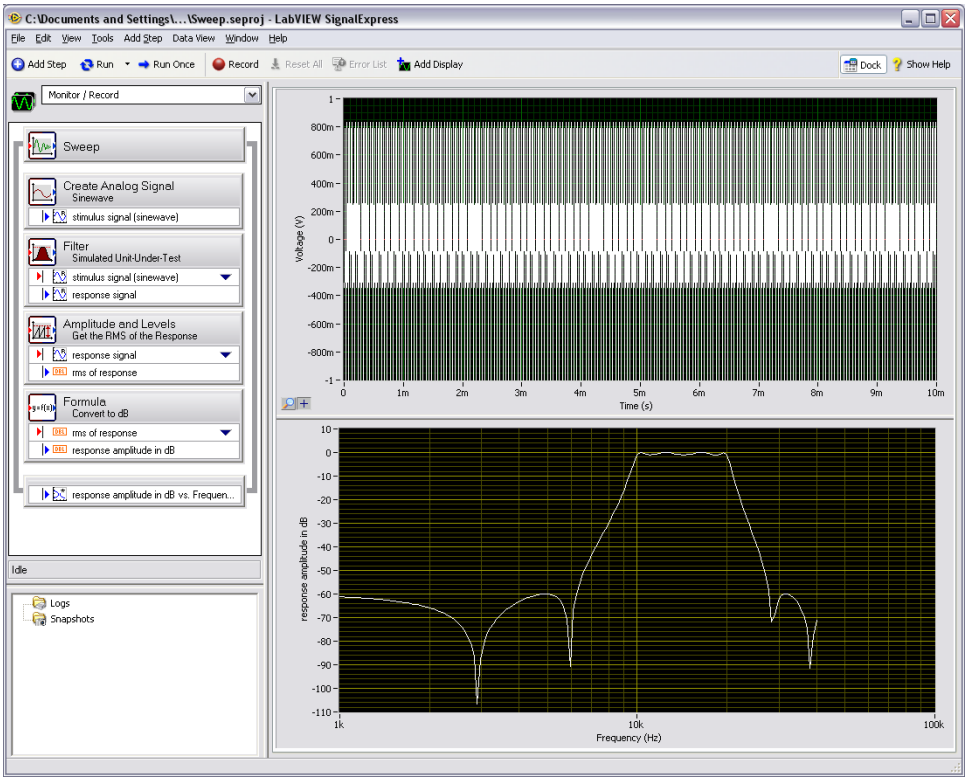


Figure 7-3. Sweep.seproj

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 on the **Step Setup** tab.

   The frequency response of the filter in the **Filter Magnitude Response (dB)** 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** 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. You can use a parallel sweep to maximize the precision of the acquisition by varying the input range of a Tektronix oscilloscope or 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 nested sweep** 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 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 7-4.

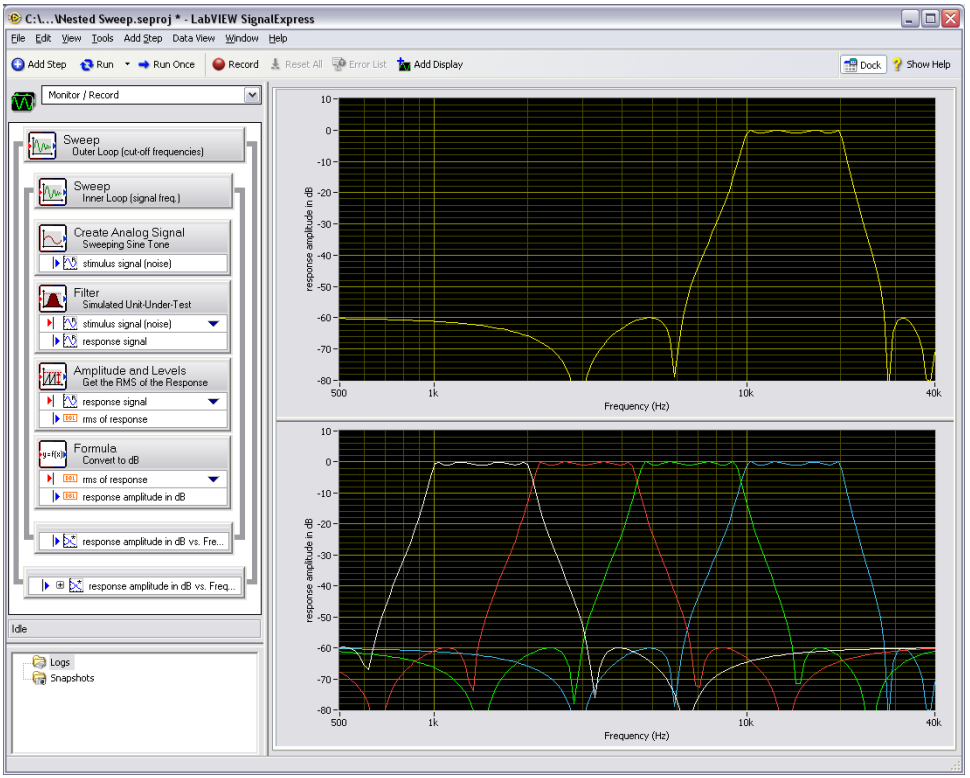


Figure 7-4. Nested Sweep.seproj

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

8

Extending LabVIEW SignalExpress Tektronix Edition Projects with LabVIEW

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

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

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

- Control GPIB instruments
- Control National Instruments hardware that LabVIEW SignalExpress Tektronix Edition 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 Analog Signal step to generate a signal.

2. Click the **Add Step** button, shown at left, 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 in which you saved the VI.

3. On the **Step Setup** tab, click the browse button in the **Select VI** section 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 on the **Step Setup** tab, and returns the clipped waveform as an output signal.

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

   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 8-1.

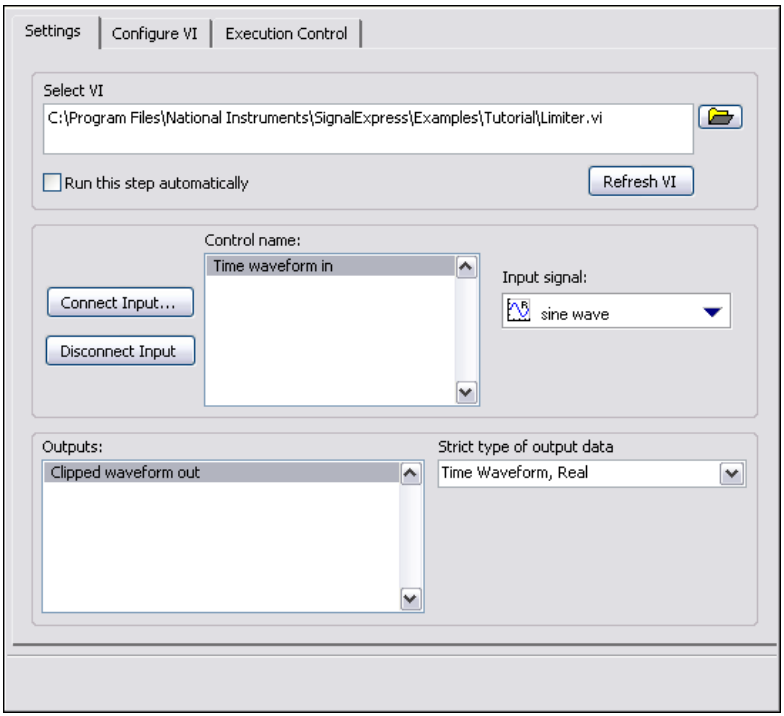


Figure 8-1. Limiter VI Step Setup Tab

4. Click the **Run** button, shown at left, to run the project.
5. Switch to the Data View and drag the **Clipped waveform out** output signal from the Limiter-LV82.vi step to the Data View.
6. Double-click the Limiter-LV82.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. 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 Project** to save the project.
12. Select **File»Close Project** to close the project.

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

Converting LabVIEW SignalExpress Tektronix Edition Projects to LabVIEW Block Diagrams

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

Note To convert a LabVIEW SignalExpress Tektronix Edition 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 Tektronix Edition 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 Tektronix Edition. The LabVIEW block diagram consists of LabVIEW Express VIs wired together. Each Express VI correlates to a step in the LabVIEW SignalExpress Tektronix Edition project. You can double-click an Express VI to display a configuration dialog box that is identical to the **Step Setup** tab in LabVIEW SignalExpress Tektronix Edition. 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 Tektronix Edition project with logging, LabVIEW SignalExpress Tektronix Edition generates a LabVIEW block diagram with one Express VI. You cannot convert the generated Express VI into a subVI.

9

Where to Go from Here

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

LabVIEW SignalExpress Tektronix Edition Sample Projects

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

Using Hardware with LabVIEW SignalExpress Tektronix Edition

LabVIEW SignalExpress Tektronix Edition supports a variety of National Instruments hardware for acquiring and generating signals. You can generate or acquire and log analog signals in LabVIEW SignalExpress Tektronix Edition 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 Tektronix Edition.

Web Resources

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

A

Technical Support and Professional Services

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

- **Support**—Technical support resources at ni.com/support include the following:
  - **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 on demand training modules via the Services Resource Center. NI offers complementary membership for a full year after purchase, after which you may renew to continue your benefits.

    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 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.