NI Sound and Vibration Assistant

Getting Started with the NI Sound and Vibration Assistant

May 2013
371938D

NATIONAL INSTRUMENTS™

Worldwide Technical Support and Product Information
ni.com

Worldwide Offices
Visit 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.

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

For further support information, refer to the *Technical Support and Professional Services* appendix. To comment on National Instruments documentation, refer to the National Instruments 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

End-User License Agreements and Third-Party Legal Notices

You can find end-user license agreements (EULAs) and third-party legal notices in the following locations:

- Notices are located in the <National Instruments>_Legal Information and <National Instruments> directories.
- EULAs are located in the <National Instruments>\Shared\MDF\Legal\license directory.
- Review <National Instruments>_Legal Information.txt for more information on including legal information in installers built with NI products.

Trademarks

Refer to the *NI Trademarks and Logo Guidelines* at ni.com/trademarks for more information on National Instruments trademarks.

ARM, Keil, and µVision are trademarks or registered of ARM Ltd or its subsidiaries.

LEGO, the LEGO logo, WEDO, and MINDSTORMS are trademarks of the LEGO Group. ©2013 The LEGO Group.

TETRIX by Pitsco is a trademark of Pitsco, Inc.©2013

FIELDBUS FOUNDATION™ and FOUNDATION™ are trademarks of the Fieldbus Foundation.

EtherCAT® is a registered trademark of and licensed by Beckhoff Automation GmbH.

CANopen® is a registered Community Trademark of CAN in Automation e.V.

DeviceNet™ and EtherNet/IP™ are trademarks of ODVA.

Go!, SensorDAQ, and Vernier are registered trademarks of Vernier Software & Technology. Vernier Software & Technology and vernier.com are trademarks or trade dress.

Xilinx is the registered trademark of Xilinx, Inc.

Taptite and Trilobular are registered trademarks of Research Engineering & Manufacturing Inc.

FireWire® is the registered trademark of Apple Inc.

Linux® is the registered trademark of Linus Torvalds in the U.S. and other countries.

Handle Graphics®, MATLAB®, Real-Time Workshop®, Simulink®, Stateflow®, and xPC TargetBox® are registered trademarks, and TargetBox™ and Target Language Compiler™ are trademarks of The MathWorks, Inc.

Tektronix®, Tek, and Tektronix, Enabling Technology are registered trademarks of Tektronix, Inc.

The Bluetooth® word mark is a registered trademark owned by the Bluetooth SIG, Inc.

The ExpressCard™ word mark and logos are owned by PCMCIA and any use of such marks by National Instruments is under license.

Patents

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

Export Compliance Information

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

WARNING REGARDING USE OF NATIONAL INSTRUMENTS PRODUCTS

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

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

Contents

About This Manual

Chapter 1
Introduction to the NI Sound and Vibration Assistant

Chapter 2
Working with Projects

Chapter 3
Logging Data

Chapter 4
Performing Sound and Vibration Measurements

Chapter 5
Performing Sweep Measurements

Chapter 6
Performing Order Analysis

Chapter 7
Extending Sound and Vibration Assistant 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 the National Instruments Sound and Vibration Assistant. You can perform interactive measurements and use basic Sound and Vibration Assistant features to acquire and analyze signals.

This manual contains exercises that help you begin working with the Sound and Vibration Assistant. These exercises teach you how to run projects, configure steps, work with signals, perform sweep measurements, log data, and extend the Sound and Vibration Assistant with NI LabVIEW Graphical Programming.

The Sound and Vibration Assistant extends existing NI SignalExpress technology. Throughout this manual, you will notice references or examples in SignalExpress. These references and examples apply to both the Sound and Vibration Assistant and SignalExpress.

Required Software

The Sound and Vibration Assistant requires you to install the following software:

- If you use the Sound and Vibration Assistant with National Instruments data acquisition (DAQ) hardware, you must install NI-DAQmx 8.9.5 or later for NI-DAQmx support.
- If you want to convert Sound and Vibration Assistant projects to LabVIEW block diagrams, you must install LabVIEW 8.2 or later before you install the Sound and Vibration Assistant.

Related Documentation

In addition to the exercises in this manual, you can learn about the Sound and Vibration Assistant with the animated Sound and Vibration Assistant Tutorial. Complete the following steps to access this tutorial:

1. Select **Start»All Programs»National Instruments»Sound and Vibration»NI Sound and Vibration Assistant** to launch the Sound and Vibration Assistant.
2. Select **Help»Show Welcome Dialog**.
3. Scroll down to the **Sound and Vibration Assistant** heading.

4. Expand this heading.
5. Click the **View a five minute interactive tutorial** link.

You also can refer to the *Sound and Vibration Assistant Help*, available by selecting **Help»Sound and Vibration Assistant Help** and navigating on the **Contents** tab to **Step Reference»Sound and Vibration Assistant Steps**, for more information as you read this manual.

1

Introduction to the NI Sound and Vibration Assistant

The NI Sound and Vibration Assistant optimizes virtual instrumentation for sound and vibration engineers by offering instant interactive measurements that require no programming. You can use the Sound and Vibration Assistant interactively to acquire, generate, analyze, compare, import, log, and play back sound and vibration signals.

The Sound and Vibration Assistant extends NI SignalExpress technology to provide specific sound and vibration measurements and analysis. You can use the Sound and Vibration Assistant for audio testing, acoustic measurements, environmental noise testing, machine condition monitoring, vibration analysis, noise and vibration control, and noise, vibration, and harshness (NVH) measurements.

You also can extend the functionality of the Sound and Vibration Assistant by importing a custom virtual instrument (VI) created in NI LabVIEW or by converting a Sound and Vibration Assistant project to a LabVIEW block diagram so that you can continue development in LabVIEW.

2

Working with Projects

You can use the NI Sound and Vibration Assistant to perform measurements by adding and configuring steps in an interactive measurement environment. A step is a configurable function that acquires, generates, analyzes, loads, or saves signals. Most steps process input signals and produce output signals. A saved sequence of configured steps is a Sound and Vibration Assistant 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 an example project in the Sound and Vibration Assistant:

1. Select **Start»All Programs»National Instruments»Sound and Vibration»NI Sound and Vibration Assistant** to launch the Sound and Vibration Assistant.

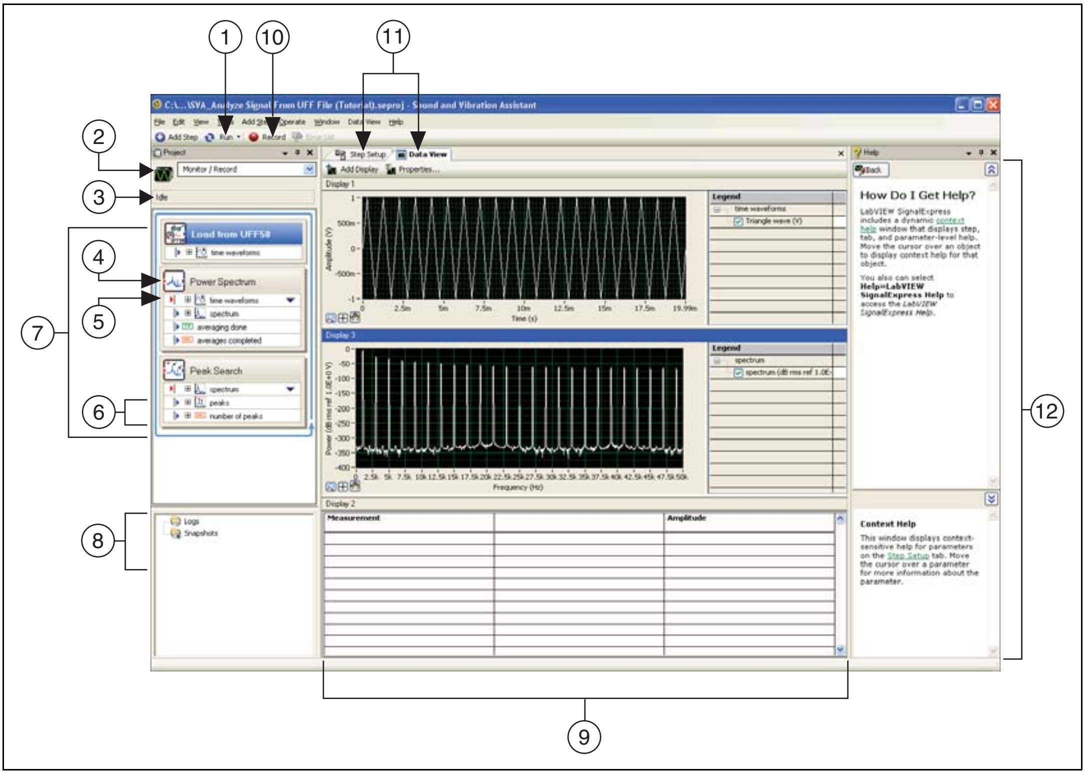
   Notice that the Sound and Vibration Assistant is split into views that display various types of information. The primary view appears in the center of the application window and contains tabs. If the Sound and Vibration Assistant opens in the default configuration, the **Data View** tab, the **Recording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

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

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

3. Select **Help»Open Example»Sound and Vibration Assistant**, double-click the `Tutorial` folder, and double-click the `SVA_Analyze Signal From UFF File (Tutorial).seproj` Sound and Vibration Assistant project.

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



1	Run button	5	Input	9	Primary view with Data View tab visible
2	Work area pull-down menu	6	Outputs	10	Record button
3	Project status	7	Project View	11	View tabs
4	Step	8	Logged data window	12	Context help

Figure 2-1. SVA_Analyze Signal from UFF File (Tutorial).seproj

The left pane displays the Project View, which presents the order of operations, or steps, for the project.

The center pane displays the **Data View** tab, which displays the signals that the project generates and analyzes.

The right pane displays the context help and updates when you hover the cursor over a new step, tab, parameter, or display. The upper half of the context help displays information about the steps and tabs, and the lower half of the context help displays information about the parameters.

Running a Project and Displaying Signals

The Sound and Vibration Assistant has two execution modes—Run Once and Run Continuously. When you click the down arrow on the **Run** button, shown at left, and select the **Run Once** option, the Sound and Vibration Assistant executes all steps in the project once. When you click the down arrow on the **Run** button and select the **Run Continuously** option, the Sound and Vibration Assistant executes all steps in the project continuously until you click the **Stop** button, shown at left.

The **Stop** button appears in place of the **Run** button as the project runs. While the steps in the project execute, the **Data View** tab updates continuously. You can change the measurement configurations while the project runs and view the results immediately.

You also can configure the run mode of a project. When you click the down arrow on the **Run** button and select **Configure Run**, you can set how long to run the project by setting the iterations, seconds, or setting the project to run continuously. If you click the **Run** button without selecting options from the pull-down menu, the project runs continuously until you click the **Stop** button.

Complete the following steps to run the example project and display signals:

1. Click the **Run** button to execute all steps in the project continously.

   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 UFF58 step loads a triangle wave from a UFF58 file, and the Power Spectrum step and the Peak Search step analyze the triangle wave and return new outputs. In the Project View, the Sound and Vibration Assistant designates inputs with red arrows and outputs with blue arrows.

2. Drag the **number of peaks** output signal of the Peak Search step from the Project View to the **Data View** tab to display the signal.

The Sound and Vibration Assistant creates a new graph on the **Data View** tab. The Sound and Vibration Assistant does not display the **number of peaks** signal in an existing table or graph because the **number of peaks** signal is a different type of data than the data in the existing graphs. The Sound and Vibration Assistant automatically recognizes different types of signals and renders them in the appropriate displays. The project appears as shown in Figure 2-2.

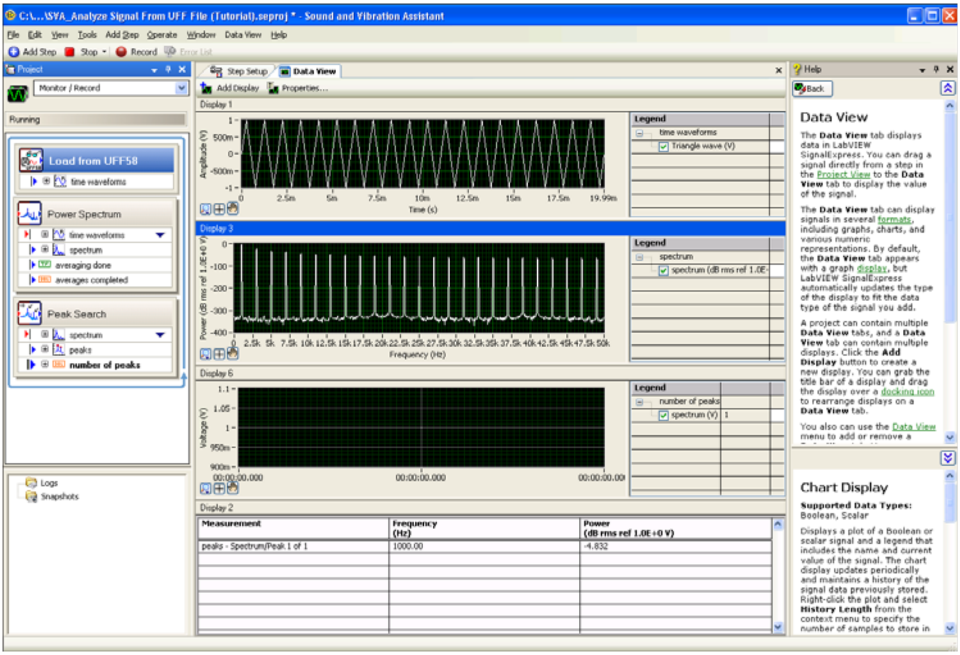


Figure 2-2. Outputs of SVA_Analyze Signal From UFF File (Tutorial).seproj

3. Select **Help»Sound and Vibration Assistant Help** and click the **Search** tab. Enter "`signal types`" and double-click on *Signal Types in SignalExpress* to find more information about signal types.

   The help provides information about using the Sound and Vibration Assistant and SignalExpress functionality such as projects, steps, and signals.

Configuring a Step

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

Complete the following steps to configure the Power Spectrum step and the Peak Search step:

1. Select **File»Save Project As** and save the project as `My First Project.seproj` in a convenient location.
2. Double-click the Power Spectrum step in the Project View. The **Step Setup** tab for the Power Spectrum step appears as shown in Figure 2-3. The graph on the **Step Setup** tab displays a preview of the input signal.

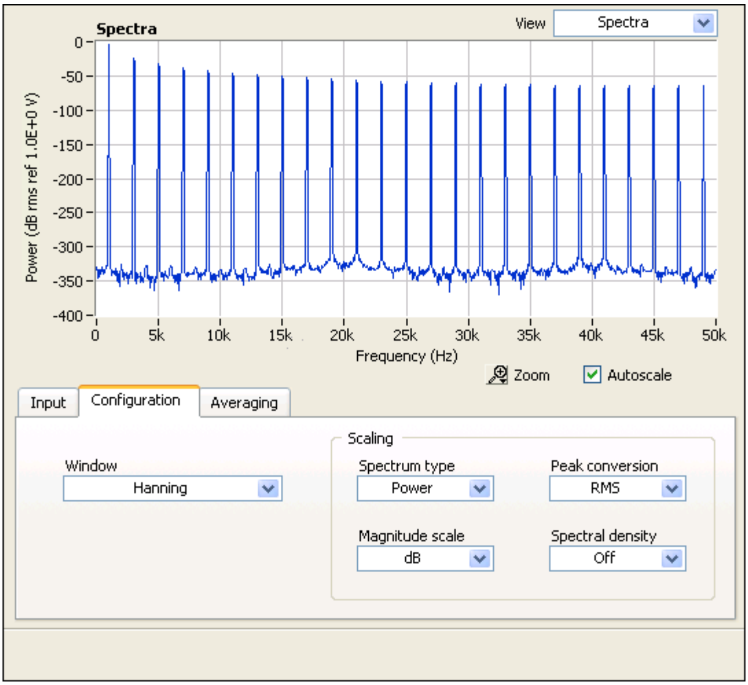


Figure 2-3. Power Spectrum Step Setup Tab

3. Click the **Input** tab in the **Step Setup** tab. Notice that the **Input signal** pull-down menu specifies a time waveform. The Load from UFF58 step produces the time waveform signal for the Power Spectrum step.
4. Click the Peak Search step in the Project View. The **Step Setup** tab changes from displaying the configuration of the Power Spectrum step to displaying the configuration of the Peak Search step.
5. Click the **Input** tab in the **Step Setup** tab. Notice that the **Input Signal** pull-down menu specifies a spectrum. The Power Spectrum step produces the spectrum signal for the Peak Search step.
6. Click the **Configuration** tab in the **Step Setup** tab. The settings indicate that the Peak Search step performs a peak search on the spectrum to locate a single maximum peak above a threshold of -40. The step generates two measurements as outputs—the frequency and amplitude of the peak and the number of peaks.
7. On the **Configuration** tab in the **Step Setup** tab, select **Multiple Peaks** from the **Search type** pull-down menu to measure every peak above the threshold of the input spectrum. The signals in the **Peaks** table on the **Step Setup** tab update to reflect the change you made.
8. Click the **Stop** button to stop the project.

   When you click the **Stop** button, the project stops running after completing the current iteration. You can click the down arrow on the **Stop** button and select the **Abort** option to completely stop the project without finishing the current iteration.
9. Select **File»Save Project** to save the project.
10. Select **File»Close Project** to close the project.

Moving Steps and Handling Errors

Most steps in Sound and Vibration Assistant projects depend on input data, which means steps can operate only on signals exported from previous steps in the Project View. The **Input signal** pull-down menu on the **Input** tab in the **Step Setup** tab of a step displays only compatible signals exported from previous steps. 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 executing.

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 the step name 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 input signals for other steps. This causes an error indicator to appear.

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.

Complete the following steps to move steps and handle an error:

1. Select **File»New Project** to open a new project.
2. Save the project as `My Error.seproj` in a convenient location.

3. Click the **Add Step** button, shown at left, and select **Processing»Standardized Filter** to add the Standardized Filter step.

Tip 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.

4. Click the **Cancel** button when the **Missing Inputs** dialog box appears, as shown in Figure 2-4.

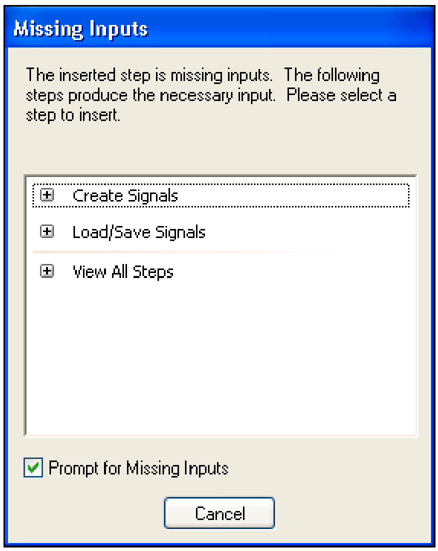


Figure 2-4. Missing Inputs Dialog Box

The **Missing Inputs** dialog box prompts you to select a step to produce the necessary input signal if an input signal does not exist.

5. Notice an error indicator, shown at left, appears at the bottom of the **Step Setup** tab and on the step in the Project View.

Note The Sound and Vibration Assistant logs all errors and warnings on the **Event Log** tab while a project runs. To display the Event Log, select **View»Event Log**. Refer to the *Sound and Vibration Assistant Help,* available by selecting **Help»Sound and Vibration Assistant Help**, for more information about errors and warnings. Click the **Search** tab and enter `errors`.

6. Click the **Details** button on the **Step Setup** tab to see the full description of the error in the **Details** dialog box. You also can click the error indicator in the Project View to display the full description. To handle this error, you can add a step to provide the missing input.
7. Close the **Details** dialog box.
8. Click the **Add Step** button and select **Create Signals»Create Analog Signal**. The Missing Input error still exists because the Create Analog Signal step does not provide the direct input signal to the Standardized Filter step.
9. Drag the Create Analog Signal step above the Standardized Filter step in the Project View. The Standardized Filter step uses the sine wave output of the Create Analog Signal step as an input signal, removing the error indicator.
10. Save and close the project.

3 Logging Data

You can use the Sound and Vibration Assistant to record measurements, including any time-domain, numeric, or Boolean step output.

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

Note References or examples of SignalExpress also apply to the Sound and Vibration Assistant.

Recording a Signal

Complete the following steps to record a signal:

1. Select **Help»Open Example»SignalExpress**, double-click the `Tutorial` folder, and double-click `Logging.seproj`.

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

2. Save the project as `My Logged Signal.seproj` in a convenient location.

3. 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.

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

5. 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.

6. Click the **Stop** button 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 window, below the Project View, displays a list of all logged data in the current project, as shown in Figure 3-1.

Figure 3-1. Logged Data Window

By default, the Sound and Vibration Assistant names the logged data according to the date and time you recorded the data. The Sound and Vibration Assistant saves logged data in the `.tdms` file format.

Note Visit `ni.com/info` and enter `tdms` for information about the `.tdms` file format.

7. Select **Tools»Options** and select the **Logging** option to specify the directory for the Sound and Vibration Assistant to save the logged data and to customize various preferences for the logged data.
8. Click the **OK** button to close the **Options** dialog box.
9. Save the project.

Viewing a Logged Signal

Complete the following steps to view the logged data:

1. Drag the data log from the Logged Data window to the **Data View** tab. The **Data View** tab displays the logged data in a graph and a preview graph beneath, as shown in Figure 3-2.

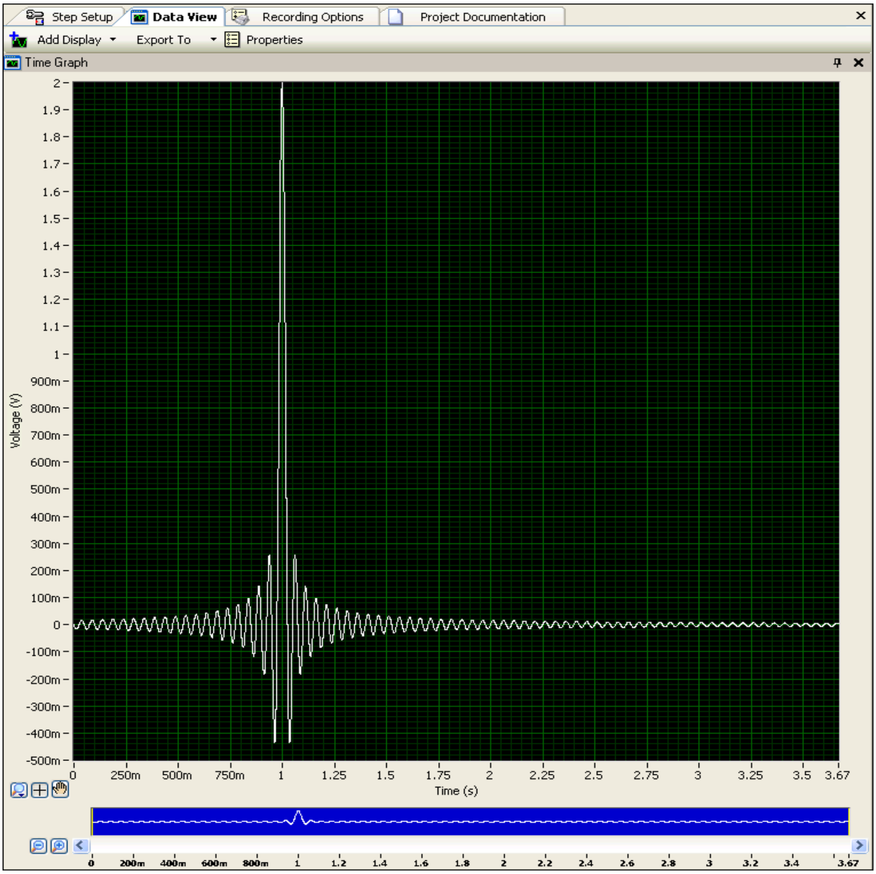


Figure 3-2. Logging.seproj

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

2. Click the **Zoom In** button, shown at left, next to the preview graph to zoom in on the logged signal. The cursors on the preview graph show the subset of the data currently displayed on the **Data View** tab. Click and drag the cursors to increase or decrease the subset of data you are viewing.
3. Use the scroll bar beneath the preview graph to scroll through the data.
4. Right-click the preview graph and select **Show All** from the shortcut menu to change the graph to display the entire signal.

Logging Signals with Predefined Start and Stop Conditions

You can configure start and stop conditions that signals must meet before the Sound and Vibration Assistant starts recording or stops recording the signals. Complete the following steps to log data based on start and stop conditions:

1. Click the **Recording Options** tab or select **View»Recording Options**.
2. Select **Signal Selection** from the **Category** list on the **Recording Options** tab.

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

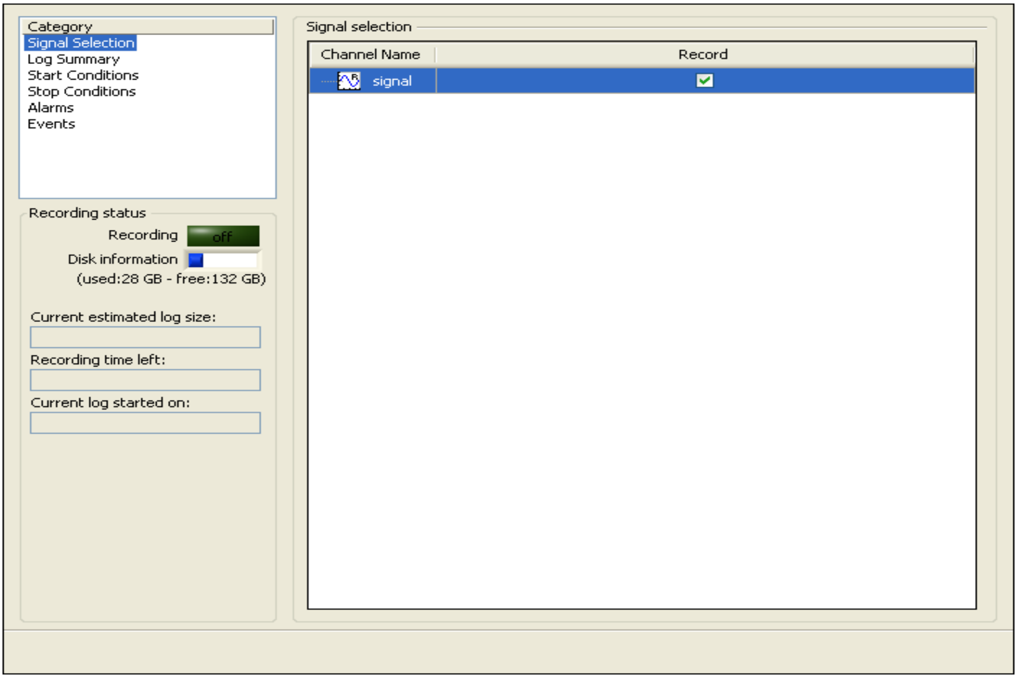
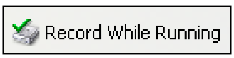


Figure 3-3. Signal Selection



The **Record** button changes to the **Record While Running** button, shown at left, in the enabled position with the green checkmark. When the **Record While Running** button is enabled, the Sound and Vibration Assistant records the selected signal when you click the **Run** button.

4. Select **Start Conditions** from the **Category** list on the **Recording Options** tab.
5. Click the **Add** button under the **Start condition list** to customize a start condition for the logging task.
   a. In the **Condition type** pull-down menu, verify that **Signal trigger** is selected to specify that the Sound and Vibration Assistant begins recording when the input signal meets the specified condition.
   b. In the **Signal** pull-down menu, verify that **signal** is selected.

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

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

The **Recording Options** tab appears as shown in Figure 3-4.

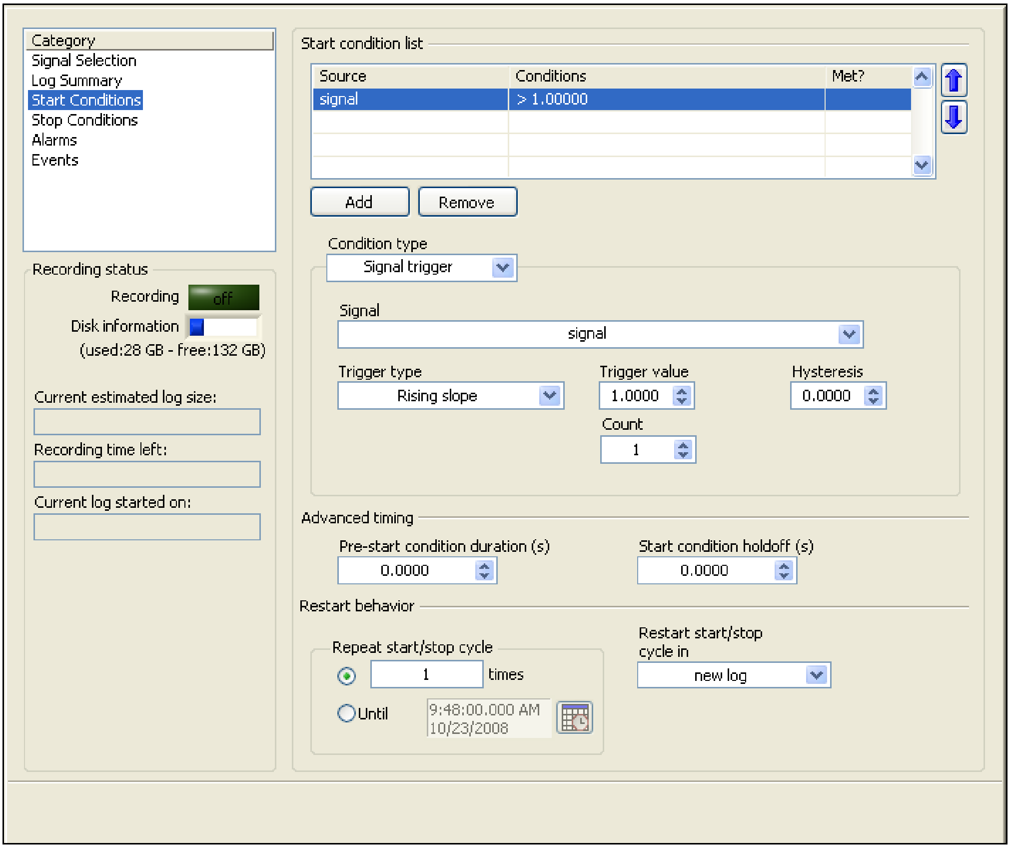


Figure 3-4. Recording Options Start Conditions

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

7. Click the **Add** button under the **Stop condition list** to customize a stop condition for the logging task.
    a. In the **Condition type** pull-down menu, verify that **Duration** is selected to specify that the Sound and Vibration Assistant stops recording after a specified amount of time passes.
    b. In the **Duration** control, verify that `5` appears to specify to record the signal for 5 seconds after the signal meets the start condition.
8. Click the **Run** button. The Sound and Vibration Assistant begins recording the signal when the signal crosses level 1 on a rising slope and continues recording the signal for 5 seconds.

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

    - **Recording** displays **on** when the signal meets the start condition and logging is in progress.
    - **Disk information** displays the available hard disk space on the computer for the log.
    - **Current estimated log size** displays the size of the log file on disk.
    - **Recording time available** displays the amount of time you can continue recording the log before running out of disk space.
    - **Current log started on** displays the start time of the current log.

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

4

Performing Sound and Vibration Measurements

You can use the Sound and Vibration Assistant to perform sound and vibration measurements interactively. You can analyze and process logged data by playing the logged data back through analysis steps. You also can analyze and process live data by using National Instruments data acquisition (DAQ) hardware, such as the NI dynamic signal acquisition (DSA) devices.

This chapter teaches you how to work with sound and vibration data in the Sound and Vibration Assistant, including how to analyze previously recorded data, plot measurements on graphs, and save signals to a file.

Opening Previously Recorded Data

You can analyze previously recorded data with the Sound and Vibration Assistant. You also can analyze data acquired using DAQ hardware.

Complete the following steps to open previously recorded sound pressure data:

1. Open a new project.
2. Select **File»Import»Logged Signals from SignalExpress TDMS Files**, navigate to the `National Instruments\Sound and Vibration Assistant\Examples\Log File - Sound Pressure\` directory, and click the **Current Folder** button.

   (Windows XP/2000) The `National Instruments` folder is located in the `C:\Documents and Settings\All Users\Shared Documents\` directory.

   (Windows Vista) The `National Instruments` folder is located in the `C:\Users\Public\Public Documents\` directory.
3. Notice that the logged data appears under **Logs** in the Logged Data window.

4. Select **Playback»Sound Pressure** from the work area pull-down menu at the top of the Project View, as shown in Figure 4-1.

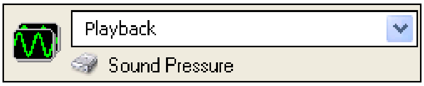


Figure 4-1. Work Area Pull-down Menu

Use work areas to perform multiple Sound and Vibration Assistant 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, the Sound and Vibration Assistant saves every work area within the project in the same project file.

The default work area, Monitor/Record, allows you to take measurements, analyze live data, and log data. The Playback work area allows you to use logged data that you record in the Monitor/Record work area as an input for an analysis step.

You now can analyze the sound pressure data in the `Sound Pressure.tdms` file.

Note You must have DAQ hardware and NI-DAQmx 8.9.5 or later installed to run projects with live sound pressure data. If you want to use live sound pressure data in the following exercises, create a new project, click the **Add Step** button, and select **Acquire Signals» DAQmx Acquire»Analog Input»Sound Pressure**. In the **Add Channels To Task** dialog box that appears, you can select the analog input channel(s) you want to analyze.

Playing Back Data

Complete the following steps to analyze the data in the `Sound Pressure.tdms` file:

1. Save the project as `My Analysis.seproj` in a convenient location.
2. Click the **Add Step** button and select **Analysis»Time-Domain Measurements»Sound Level**.
3. Click the **Input** tab in the **Step Setup** tab. Notice that the **Input signal** pull-down menu already contains the sound pressure signal from the `.tdms` file. The Sound and Vibration Assistant automatically selects the logged data as the input to the Sound Level step.

4. Click the **Weighting** tab in the **Step Setup** tab and select **A Weighting** from the **Weighting** pull-down menu to simulate the loudness of the low-level tones.
5. Click the **Averaging** tab in the **Step Setup** tab and remove the checkmarks from the **Leq** and **Peak** checkboxes to compute only the running equivalent continuous sound level and the exponential averaged sound level of the signal.
6. Drag the **Sound Pressure** signal from the Logged Data window to the **Data View** tab.

Note Viewing logged signals in Playback mode displays only one iteration of the data at a time. To view the complete logged data, select **Monitor/Record** from the work area pull-down menu to switch to the Monitor/Record work area. You also can view the complete logged data on the **Playback Options** tab.

7. Run the project.

   Notice that the recorded sound pressure data appears in the **Data View** tab with a timeline slider at the top of the view. The project runs until it reaches the end of the logged data file.

Tip When the project is not running, you can click and drag the slider in the timeline to view different portions of the previously recorded data.

8. Right-click the graph and select **Visible Items»Cursors** from the shortcut menu to display two interactive cursors, as shown in Figure 4-2.

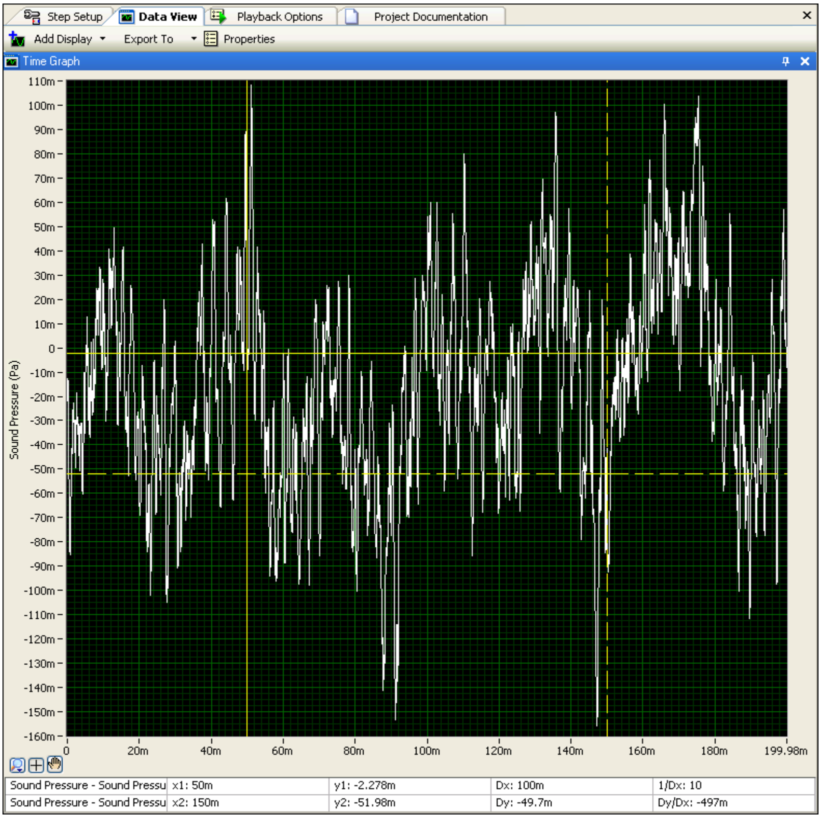


Figure 4-2. Displaying Cursors on the Graph

9. Click and drag the cursors to increase or decrease the subset of data you are viewing. As you drag the cursors, the Sound and Vibration Assistant displays the x and y values of the cursors in the cursor table below the graph in the **Data View** tab.
10. Save the project.

Analyzing Data

Complete the following steps to perform more measurements on the sound pressure data:

1. Drag the **running Leq** output of the Sound Level step to the **Data View** tab. The Sound and Vibration Assistant creates a new chart in the **Data View** tab.
2. Drag the **exponential** output of the Sound Level step to the existing chart on the **Data View** tab.
3. Run the project.

   Notice on the **Data View** tab that **running Leq** and **exponential** update in the chart as the playback data progresses.
4. Click the **Add Step** button and select **Analysis»Frequency-Domain Measurements»Octave Analysis**.
5. Notice that the warning **Frequency range coerced** appears at the bottom of the **Step Setup** tab. The Octave Analysis step automatically coerces the frequency range based on the sample rate of the input signal. Select **1k** from the **High frequency (Hz)** pull-down menu to change the frequency range and eliminate the warning.
6. Click the **Input** tab in the **Step Setup** tab and notice that the **Input Signal** is the same sound pressure signal you used with the Sound Level step.
7. Click the **Configuration** tab in the **Step Setup** tab and select **A Weighting** from the **Weighting** pull-down menu.

8. Drag the **octave** output of the Octave Analysis step to the **Data View** tab.

   The Sound and Vibration Assistant displays the sound pressure, running Leq, exponential, and octave charts, as shown in Figure 4-3.

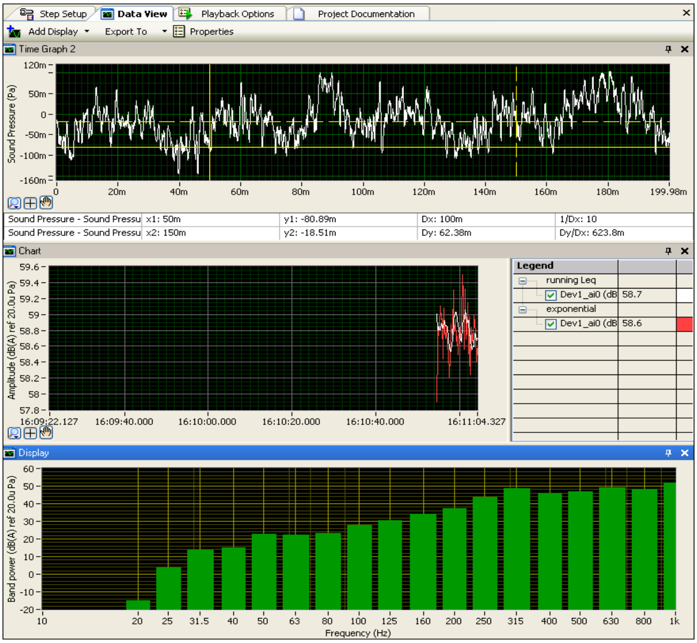


Figure 4-3. My Analysis.seproj with Octave Analysis

9. Drag the **total band power** output of the Octave Analysis step to the chart with **running Leq** and **exponential** on the **Data View** tab.
10. Right-click the chart and select **View As»Table Display** from the shortcut menu to view the numerical results as a table instead of a graph.
11. Run the project.

    Notice that the octave graph updates as the playback progresses. You have now performed sound level and octave analysis on the sound pressure data.
12. Save and close the project.

Advanced Playback

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

Complete the following steps to configure advanced playback options:

1. Select **File»Open Project**, navigate to `My Logged Signal.seproj`, and click the **Open** button.
2. Select **Playback** from the work area pull-down menu.
3. Click the **Playback Options** tab or select **View»Playback Options**. The **Playback Options** tab appears as shown in Figure 4-4.

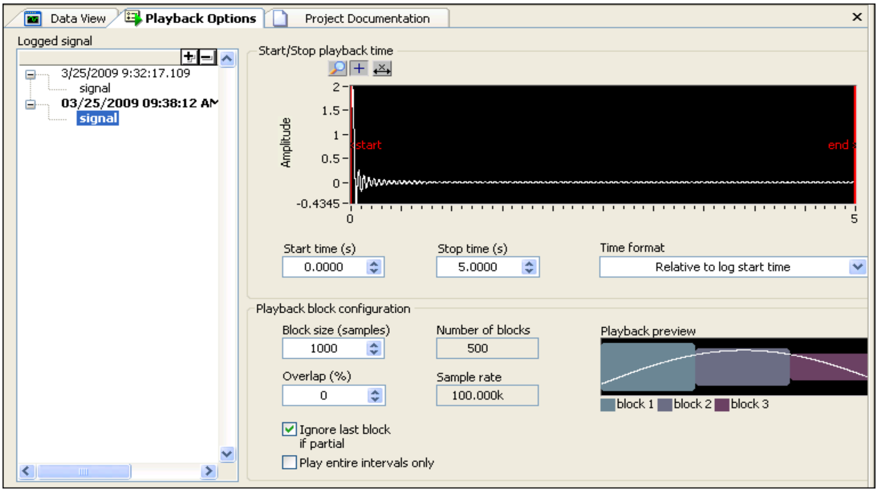


Figure 4-4. Playback Options Tab

4. Select **signal** from the second log in the **Logged signal** listbox.
5. Enter `1` in the **Start time (s)** field and `4` in the **Stop time (s)** field to play back or analyze a subset of the logged signal beginning one second and ending four seconds after the start of the log.
6. Click the **Add Step** button and select **Processing»Filter**.
7. Drag the **filtered data** output of the Filter step to the **Data View** tab.

8. Click the **Run** button. The Sound and Vibration Assistant filters the subset of the signal you specified on the **Playback Options** tab and displays the resulting filtered signal on the **Data View** tab. Notice that the signal starts at 1 second and stops at 4 seconds.
9. Save the project.

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

Signal Types in the Sound and Vibration Assistant

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 signals. 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, the Sound and Vibration Assistant adds only their amplitudes. However, if you add two frequency-domain phase signals, the Sound and Vibration Assistant adds the appropriate phase shift.

Saving, Exporting, and Printing Signals

You can use the Sound and Vibration Assistant to document signals or continue analysis in another software application. This section teaches you how to save signals to an ASCII file, export data to Microsoft Excel, print signals, and use the built-in project documentation feature to document the Sound and Vibration Assistant project.

Saving Signals to File

Complete the following steps to save a signal from `My Logged Signal.seproj` to a file:

1. Click the **Add Step** button and select **Load/Save Signals»Save to ASCII/LVM**. You can use a Load/Save Signals step to save data to a file every time the project runs.

2. Click the **File Settings** tab in the **Step Setup** tab and click the **Browse** button, shown at left, next to the **Export file path** field.
3. Navigate to the `National Instruments\Sound and Vibration Assistant\Examples\Tutorial\` directory and enter `octave.txt` as the filename.

4. Select **Overwrite** from the **If file already exists** pull-down menu.
5. Run the project to save the resulting signal to the specified ASCII file.
6. Save the project.
7. Navigate to the folder containing `octave.txt` and open the file in a text editor to view the saved data.

Exporting Signals to Microsoft Excel

To export signal data to Microsoft Excel, launch Excel and drag the output signal of a step in the Sound and Vibration Assistant to an Excel spreadsheet. The data is tab delimited.

You also can right-click a display on the **Data View** tab and select **Export To»Microsoft Excel** from the shortcut menu to export data from the selected graph on the **Data View** tab. You also can open saved data, such as `octave.txt`, in Microsoft Excel.

Printing Signals

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

Creating Reports in the Sound and Vibration Assistant

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

To print the documentation, open the **Project Documentation** tab and select **File»Print»Print Documentation**, or click the **Print Documentation** button.

Exporting Documentation to HTML

To export the documentation to HTML, display the **Project Documentation** tab and select **File»Export»Export Documentation to HTML**. You also can right-click on the **Project Documentation** tab and select **Export Documentation to HTML** from the shortcut menu to export the documentation.

5

Performing Sweep Measurements

You can create sweep operations in the Sound and Vibration Assistant to automate measurements in order to characterize and validate designs. You can use the sweep measurements to gather data from devices under test over a range of conditions in order to document the performance of the devices. 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.

This chapter teaches you how to set up sweep operations using the Sweep step in the Sound and Vibration Assistant. 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.

You can sweep physical signals generated from National Instruments arbitrary waveform generators, function generators, dynamic signal acquisition (DSA) devices, or multifunction I/O (MIO) devices.

Note You must have DAQ hardware with analog inputs and outputs and NI-DAQmx 8.9.5 or later installed to complete the following exercises with physical signals.

Defining Sweep Ranges and Outputs

You can use the Sweep step in the Sound and Vibration Assistant to define automated measurements for complex, repeatable sweep operations.

Creating a Project to Sweep

Complete the following steps to create a project to sweep:

1. Open a new project.
2. Save the project as `My Sweep.seproj` in a convenient location.
3. Add the Create Analog Signal step.

4. On the **Configuration** tab in the **Step Setup** tab, enter `10000` in the **Block size (samples)** field.
5. Add the Filter step.
6. On the **Configuration** tab in the **Step Setup** tab, select **Bandpass** from the **Type** pull-down menu. Notice that the **Filter Magnitude Response (dB)** graph updates as you change options in the **Filter Specifications** section.
7. Enter `2000` in the **Low cutoff (Hz)** field and `5000` in the **High cutoff (Hz)** field.
8. Click the **Add Step** button and select **Analysis»Frequency-Domain Measurements»Tone Measurements**.
9. Click the **Single-Tone Measurements** tab in the **Step Setup** tab and place checkmarks in the **THD+N** and **THD** checkboxes and remove checkmarks from the **SNR**, **SINAD**, and **SFDR** checkboxes in the **Distortion** and **Channel Evaluation** sections.
10. Save the project.

Defining the Sweep

Complete the following steps to define a frequency range to sweep through a filter:

1. Click the **Add Step** button and select **Execution Control»Sweep**.
2. Click the **Add** button in the **Sweep Configuration** tab in the **Step Setup** tab to display the list of sweepable parameters from each step in the project, as shown in Figure 5-1.

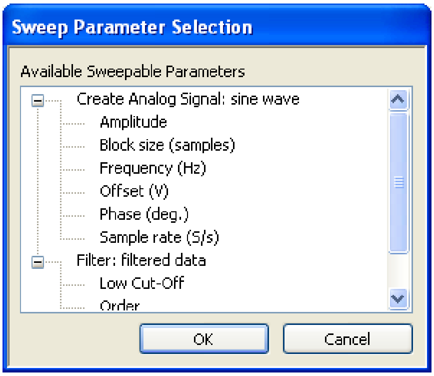


Figure 5-1. Sweep Parameter Selection Dialog Box

3. Select the **Frequency (Hz)** parameter under **Create Analog Signal: sine wave** and click the **OK** button.

   Notice in the Project View, as shown in Figure 5-2, that the Sweep step encloses the Create Analog Signal step, which provides the signal to sweep.

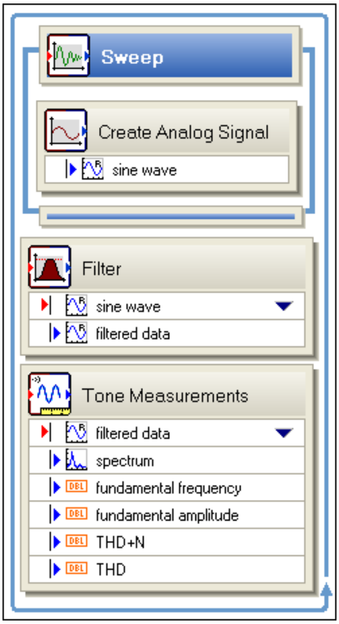


Figure 5-2. Sweep Step Enclosing Create Analog Signal Step

4. On the **Sweep Configuration** tab, select **Exponential** from the **Type** pull-down menu.
5. Enter `2000` in the **Start: Frequency (Hz)** field and `5000` in the **Stop: Frequency (Hz)** field.

6. Enter `20` in the **Number of points** field.

   The **Sweep Configuration** tab appears as shown in Figure 5-3.

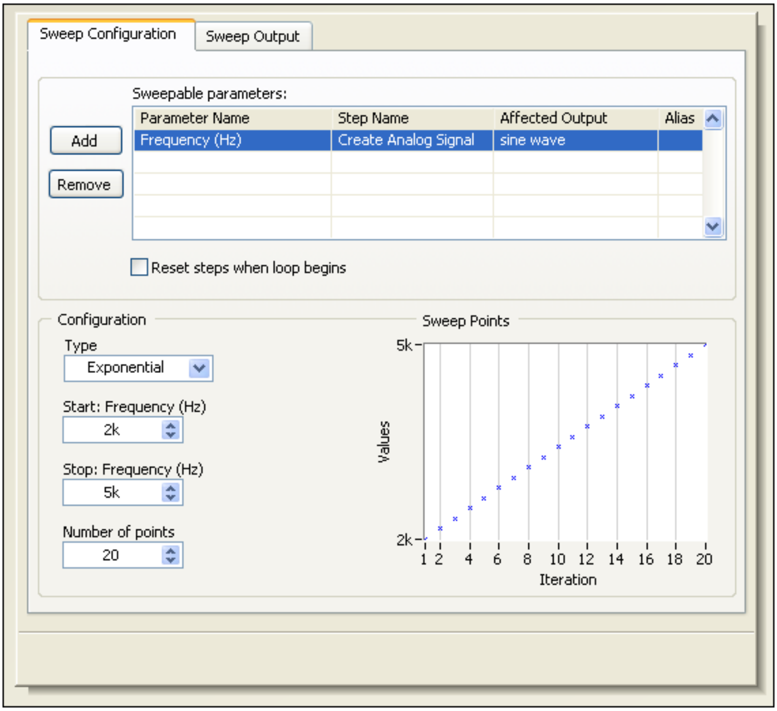


Figure 5-3. Sweep Configuration Tab

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

7. Click the **Sweep Output** tab in the **Step Setup** tab.
8. Click the **Add** button to display the list of sweepable outputs from each step in the project.

9. Select the **fundamental amplitude** output under **Tone Measurements** and click the **OK** button to plot this measurement against the swept **Frequency (Hz)** parameter.

   Notice that the Sweep step creates a loop around all the steps in the Project View, as shown in Figure 5-4, to include all the steps in the sweep operation.

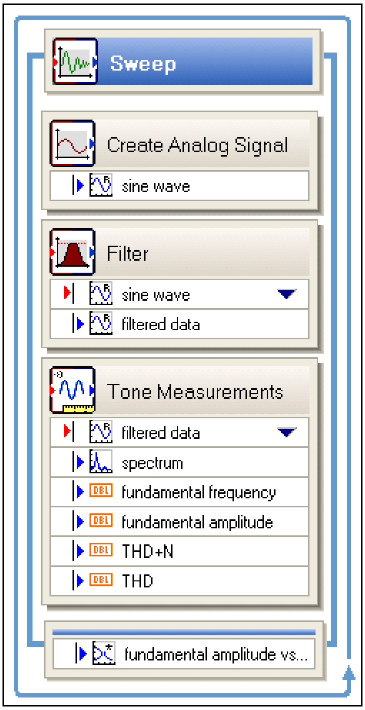


Figure 5-4. Sweep Step Including All Steps in the Sweep Operation

10. Save the project.

Plotting Sweep Results

Complete the following steps to run the sweep measurement:

1. Drag the **filtered data** output of the Filter step to the **Data View** tab.
2. Drag the **sine wave** output of the Create Analog Signal step to the **Data View** tab.
3. Select the **Run Once** option to execute the sweep measurement.

The project generates a sine wave stimulus signal using the Create Analog Signal step, passes it through a bandpass filter using the Filter step, and measures the THD+N and THD levels of the filter output using the Tone Measurements step. The Filter step acts as a simulated unit under test, so the project uses no hardware.

The **sine wave** output on the graph iterates through the specified range of frequencies.

4. Drag the **fundamental amplitude vs. Frequency (Hz)** signal from the Sweep loop to the **Data View** tab to display the output of the sweep.

   The Sound and Vibration Assistant creates a new graph. The data from the sweep operation is an XY array that requires a separate graph, as shown in Figure 5-5.

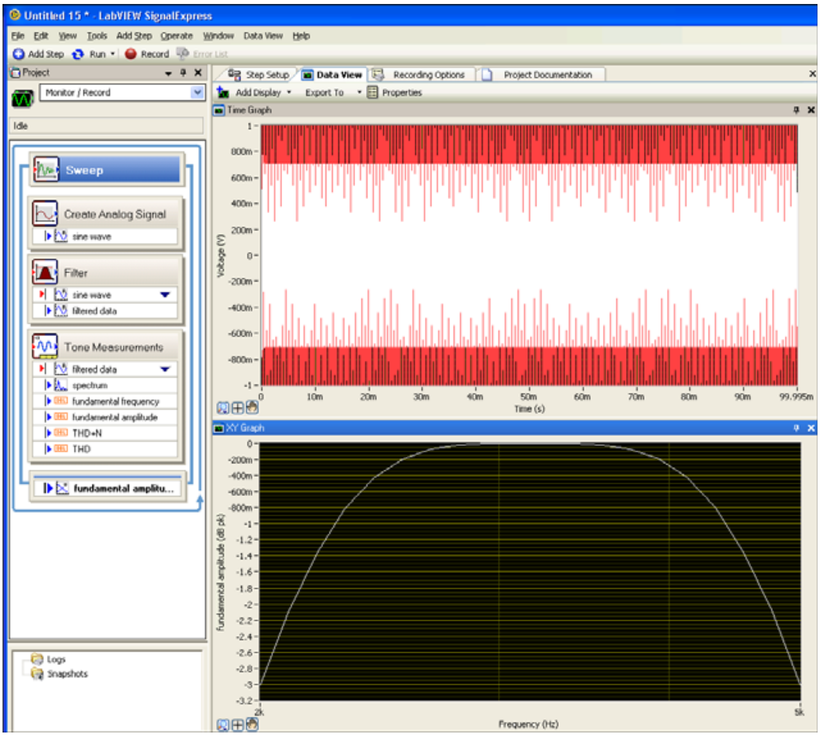


Figure 5-5. Sweep Project

5. Select the **Run Once** option to execute the sweep measurement.

   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, which is the fundamental amplitude output versus the frequency.

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

6. Save and close the project.

You can use the Sweep step to sweep multiple parameters simultaneously by adding additional parameters on the **Sweep Configuration** tab of the **Step Setup** tab. Sweeping two or more parameters simultaneously is called a parallel sweep. For example, if you want to vary the amplitude of a stimulus signal, you can run a parallel sweep to maximize the precision of the acquisition by varying the input range of a digitizer or DSA 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.

6

Performing Order Analysis

If you have the Sound and Vibration Measurement Suite, you can use the Sound and Vibration Assistant to perform order-domain measurements on a device under test (DUT). You can perform these measurements, also known as order analysis, either during the test or after the test. This chapter provides examples that demonstrate both situations.

In the first example, you perform order analysis on a universal file format binary (UFF58b) file that contains data acquired from a run-up test on a four-cylinder engine. This example demonstrates how you can use acquired test data to perform order analysis.

In the second example, you use the Playback feature of the Sound and Vibration Assistant to simulate a run-up test on a PC fan. During this simulated test, you perform order analysis on the acquired data. This example demonstrates how you can acquire and analyze data simultaneously.

Both examples involve run-up tests. Because run-up tests change the rotational speed of the DUT over the entire operating range, run-up tests are useful for identifying orders that cause significant noise or vibration in the DUT.

Performing Order Analysis on Logged Data

The example in this section uses logged data from a run-up test on a four-cylinder engine. You perform order analysis on this data to identify the orders that cause significant engine noise. This data includes the following two signals from the test:

- A tachometer pulse signal that measures the rotational speed of the engine. The tachometer generates one pulse per revolution of the engine.
- A microphone signal that measures the sound the engine emits. In this situation, the measured sound represents unwanted noise.

In the following sections, you complete these steps:

1. Load the acquired data from a UFF58b file.
2. Measure the rotational speed of the engine during the course of the test.
3. Identify the orders that cause significant engine noise.
4. Analyze the magnitude of these specific orders.

Loading the UFF58b Data File

Complete the following steps to open the example project:

1. Select **Help»Open Example»Sound and Vibration Assistant**, double-click the `Tutorial` folder, and double-click `SVA_Order Analysis From UFF File (Tutorial).seproj`.
2. Save the project as `My Engine.seproj` in a convenient location.

The **Data View** tab shows two signals: tachometer and sound pressure. The tachometer signal in the upper-left corner shows how the tachometer voltage changes during the run-up test. The sound pressure signal in the upper-right corner shows how the sound pressure, or noise, changes during the same test.

Measuring the Rotational Speed of the Engine

Because order analysis measures the rotational speed of the DUT, the first step in any order analysis procedure is measuring the speed profile of the DUT. A speed profile illustrates how the rotational speed of the DUT changes over time. You measure rotational speed in revolutions per minute (RPM).

You can use the Analog Tacho Processing step to acquire a speed profile of the DUT. This step converts a tachometer pulse signal into RPM. Complete the following steps to measure the speed profile of the engine:

1. Click the **Add Step** button and select **Analysis»Order-Domain Measurements»Analog Tacho Processing**.
2. On the **Input** tab in the **Step Setup** tab, notice the Tachometer signal is the source of analysis, or **Input signal**. This signal appears in the **Tachometer signal** graph on the upper portion of the **Step Setup** tab.

   The middle portion of the **Step Setup** tab displays the **Speed profile** graph. This graph represents an RPM measurement this step makes on the Tachometer signal. This measurement shows how the engine RPM changes during the course of the run-up test.

Figure 6-1 shows the **Speed profile** graph in the **Step Setup** tab.

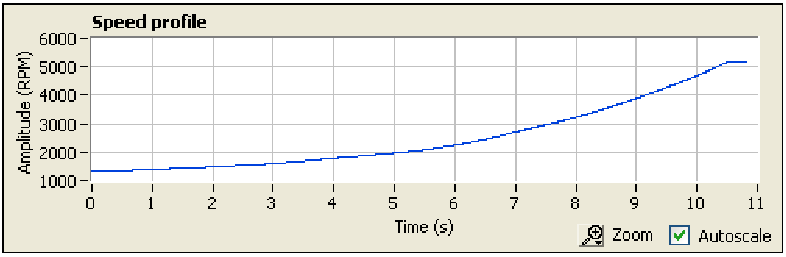


Figure 6-1. Speed Profile of the Engine

In Figure 6-1, notice how the engine RPM increases from about 1300 RPM to 5200 RPM over 10.84 seconds. You can use this speed profile as the basis for order analysis.

3. Drag the **speed profile** output of the Analog Tacho Processing step to the empty graph of the **Data View** tab.

   The **Data View** tab now displays three graphs as a function of time: the tachometer graph shows the tachometer voltage amplitude, the sound pressure graph shows the sound pressure level, and the speed profile graph shows the engine RPM.

4. Save the project.

Identifying Dominant Orders

Now that you have used the tachometer signal to acquire a speed profile, you can use the microphone signal to perform order analysis. A common tool for this analysis is a spectral map in the form of a colormap. A colormap shows you an overview of how the amplitude of a signal relates to either frequency or order. A colormap also shows how the amplitude of a signal relates to either time or speed.

You also can configure a colormap to display order as a function of RPM. An RPM-order colormap displays how the sound pressure levels at different orders change with engine RPM. You can use a colormap to identify the dominant orders, which are orders that cause high sound pressure levels.

You use the Spectral Map step to display a colormap. Complete the following steps to display a colormap of the recorded sound pressure levels:

1. Click the **Add Step** button and select **Analysis»Order-Domain Measurements»Spectral Map**.
2. On the **Input/Output** tab in the **Step Setup** tab, select **time waveforms»Sound Pressure** from the **Input signal** pull-down menu.

   The colormap now displays the noise frequency, in Hz, as a function of time. The colormap also displays the amplitude of the pressure level as a range of colors. Low sound pressure levels are in green and high sound pressure levels are in red.

   Notice that several high sound pressure levels occur between 600 to 750 Hz, as shown in Figure 6-2. This frequency range probably includes a resonant frequency of some parts of the engine.

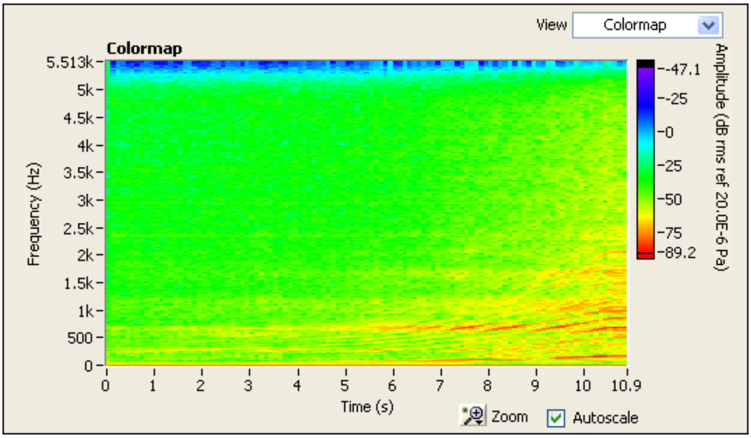


Figure 6-2. Frequency-Time Colormap Display

3. Click the **Configuration** tab in the **Step Setup** tab and select **RPM-Order** from the **Plot type** pull-down menu.

   The colormap still shows the amplitude of the sound pressure level in color. However, the colormap now shows how order changes with RPM instead of how frequency changes with time.

4. Double-click the maximum value on the x-axis and enter `20` as the new value. This adjusts the scale of the colormap to focus on orders 0 to 20.

   The **Colormap** display appears as shown in Figure 6-3.

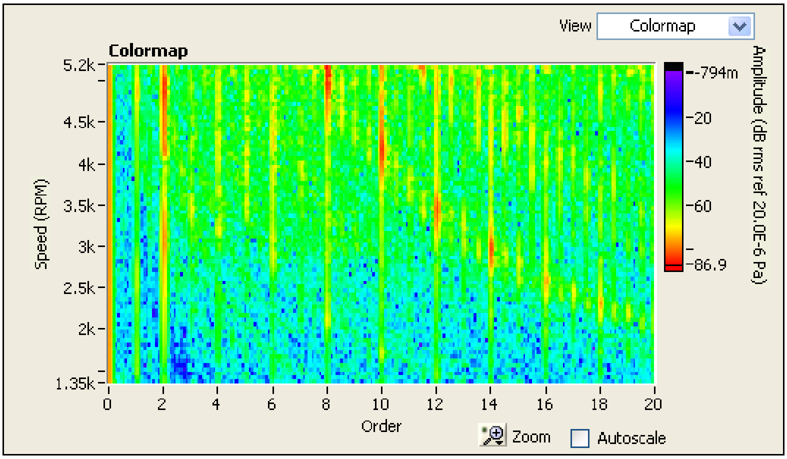


Figure 6-3. RPM-Order Colormap Display

In Figure 6-3, notice that the 2nd order contains red areas from 2500 to 3500 RPM and from 4100 to 5100 RPM. These red areas indicate high sound pressure levels that occur in these RPM ranges. This behavior at the 2nd order is expected because combustion, which causes noise, occurs twice per revolution in a four-cylinder engine.

Also notice that high sound pressure levels exist at the 8th, 10th, 12th, and 14th orders at different RPM ranges. These orders are harmonics, or multiples, of the 2nd order. Therefore, you also can expect these orders to cause significant noise. All of these orders are dominant orders.

5. Drag the **colormap** output of the Spectral Map step to the **Data View** tab. This tab now displays four graphs.
6. Repeat step 4 for the colormap display on the **Data View** tab by clicking the maximum value and replacing the current value with `20`.
7. Save the project.

Tracking the Magnitude of Specific Orders

The last step of the order analysis procedure is tracking the magnitude of two dominant orders. The colormap displays this data already. However, a colormap shows a broad overview of this data. A magnitude graph provides a simpler way of tracking the numeric data for specific orders.

You can use the Order Tracking step to show the magnitude of one or more orders as a function of RPM. Complete the following steps to display the magnitude of the 10^{th} and 12^{th} orders over the entire RPM range:

1. Click the **Add Step** button and select **Analysis»Order-Domain Measurements»Order Tracking**.
2. On the **Input/Output** tab in the **Step Setup** tab, select **time waveforms»Sound Pressure** from the **Input signal** pull-down menu.
3. Click the **Configuration** tab in the **Step Setup** tab and enter `10` and `12` in the **Orders to track** list.
4. Select **Speed** from the **X-axis selection** pull-down menu to change the x-axis to RPM.

Tip You can configure the Order Tracking step in the **Step Setup** tab to display the order magnitude as a function of time, speed, or number of revolutions.

5. Drag the **magnitudes** output of the Order Tracking step to the **Data View** tab.
6. Right-click the new magnitudes graph and select **Visible Items»Legend**.

   The magnitudes graph appears as shown in Figure 6-4.

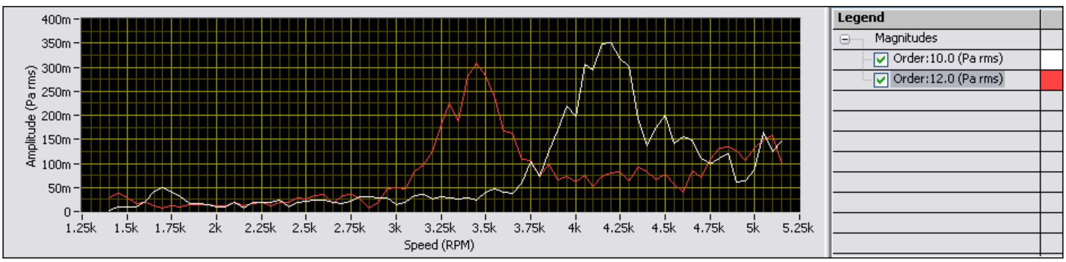


Figure 6-4. Magnitude Plot of 10^{th} and 12^{th} Orders

In Figure 6-4, notice the high magnitudes for the 12^{th} order exist around 3400 RPM. This peak corresponds to the red area for the 12^{th} order on the colormap. The magnitudes graph provides this information in a format that is simpler to read than a colormap.

The peak for the 10^{th} order exists around 4200 RPM. Again, the colormap confirms this analysis.

7. Save and close the project.

Performing Order Analysis During Data Acquisition

You can use the Sound and Vibration Assistant to perform order analysis while acquiring data from NI DAQ hardware, such as NI DSA devices.

In this section, you play back a file that contains logged data. You can perform order analysis on this data while the data plays back. In this way, the example simulates acquiring test data while performing order analysis on this data simultaneously. Refer to the *Playing Back Data* section of Chapter 4, *Performing Sound and Vibration Measurements*, for information about playing back logged data.

In the example in this section, the DUT is a PC fan with seven blades, mounted on a motor with four coils. The DSA device measures the following two signals from this fan:

- A tachometer pulse signal that measures the speed of the fan. The tachometer generates two pulses per revolution of the fan.
- An accelerometer signal that measures the vibration of the fan.

You perform order analysis on these signals to identify the orders that cause significant fan vibration.

In the following sections, you complete these steps:

1. Import and play back the acquired data from a Sound and Vibration Assistant log file to simulate real-time data acquisition.
2. Measure the rotational speed of the fan during the course of the test.
3. Identify the orders that cause significant fan vibration.
4. Analyze the magnitude of these specific orders.

Importing the Logged Data File

Complete the following steps to open the data from the PC fan run-up test:

1. Open a new project.
2. Select **File»Import»Logged Signals from SignalExpress TDMS Files**, navigate to the `National Instruments\Sound and Vibration Assistant\Examples\Log File - PC Fan Run-up\` directory, and click the **Current Folder** button.

3. Save the project as `My PC Fan.seproj` in a convenient location.
4. Notice that **PC Fan Run-up** appears under **Logs** in the Logged Data window.
5. Select **Playback»PC Fan Run-up** from the work area pull-down menu. The **Data View** tab now displays a timeline slide at the top of the tab.

Measuring the Rotational Speed of the PC Fan

As in the *Performing Order Analysis on Logged Data* section of this chapter, you must measure the speed profile of the DUT before you can perform any order analysis. Complete the following steps to measure the speed profile of the PC fan:

1. Add the Analog Tacho Processing step.
2. On the **Input** tab in the **Step Setup** tab, notice that the default **Input signal** is the **Analog Tachometer Signal**.
3. Click the **Configuration** tab in the **Step Setup** tab and enter `2` in the **Pulse/Revolution** field because for this PC fan, two pulses represent one revolution. Notice that the RPM values on the y-axis of the **Speed profile** graph reduce by half.
4. Drag the **speed profile** output of the Analog Tacho Processing step to the **Data View** tab.
5. Run the project to play back the data.

   Notice that the speed profile graph updates as the timeline slide moves. This movement shows how the Sound and Vibration Assistant calculates the fan speed as you acquire a tachometer signal from the DUT.

Tip While in the Playback work area, you can select **Operate»Set Playback Rate** to change the speed of the data playback.

6. Save the project.

Identifying Dominant Orders

As in the *Performing Order Analysis on Logged Data* section of this chapter, the next step is identifying the dominant orders. However, instead of using a colormap, the following steps use an order spectrum graph to identify these orders. Order spectrum graphs are useful for displaying numeric data, whereas colormaps are useful for obtaining a broad overview of this data.

You can use the Order Spectrum step to display an order spectrum graph. Complete the following steps to measure the order spectrum of the vibration signal:

1. Click the **Add Step** button and select **Analysis»Order-Domain Measurements»Order Spectrum**.
2. On the **Input** tab in the **Step Setup** tab, select **Active Log»Acquired Signals»Vibration Signal** from the **Input signal** pull-down menu. The **Order spectra** graph updates to display vibration amplitude as a function of order during a single iteration of the run-up test.
3. Click the **Spectrum Settings** tab in the **Step Setup** tab and select **Linear** from the **Magnitude scale** pull-down menu to provide a linear scale that is easier to read than decibels.

   On the **Order spectra** graph, notice that significant vibrations do not occur after the 10th order. You can change the scale of the graph to focus on the orders that do cause vibrations.
4. Enter `10` as the new value for the **Maximum order** field.

   The **Order spectra** graph appears as shown in Figure 6-5.

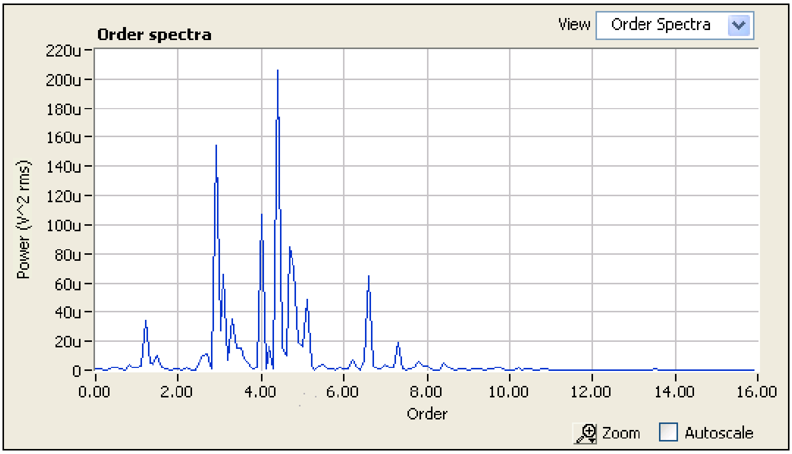


Figure 6-5. Order Spectra Graph of Vibration Signal

5. Drag the **order spectrum** output of the Order Spectrum step to the **Data View** tab. This tab now displays the speed profile graph and the order spectrum graph.
6. Run the project. Notice that the order spectrum graph updates as the project runs.

7. Right-click the order spectrum graph and select **View As»Waterfall Graph**. A waterfall graph is useful for showing a history of data throughout the test.
8. Run the project. Notice that the waterfall graph displays a history of the order spectra throughout the run-up test. The z-axis is the test iteration number.

   Figure 6-6 shows the waterfall graph.

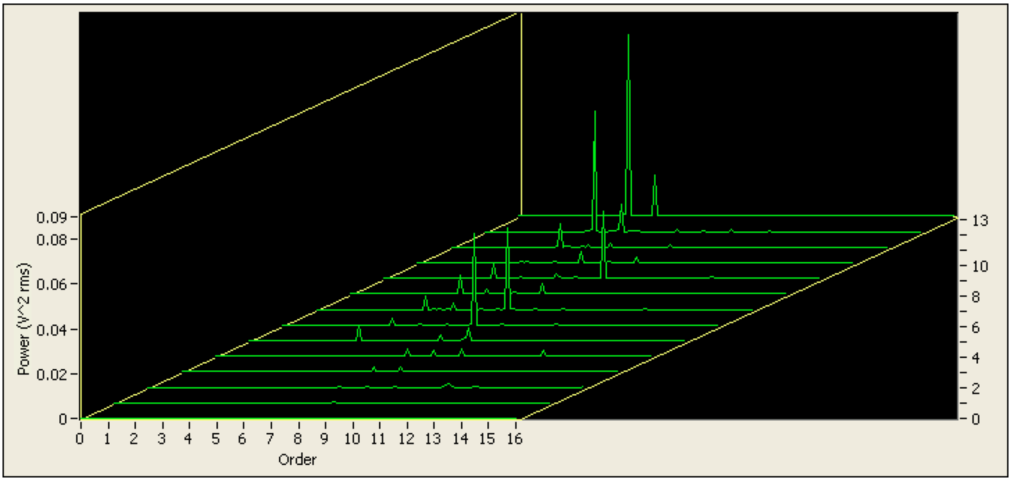


Figure 6-6. Waterfall Graph of the Order Spectra

The waterfall graph shows peaks at the 4th and 7th orders. These orders are the dominant orders. This behavior is expected because the four coils in the PC fan motor cause vibrations at the 4th order and the seven blades cause vibrations at the 7th order.

9. Save the project.

Tracking the Magnitude of Specific Orders

The last step is tracking the magnitude of the vibration at these orders throughout the test. Complete the following steps to display the magnitude of the vibration at the 4th and 7th orders:

1. Add the Order Tracking step.
2. On the **Input/Output** tab in the **Step Setup** tab, select **Active Log» Acquired Signals»Vibration Signal** from the **Input signal** pull-down menu.

3. Click the **Configuration** tab in the **Step Setup** tab and enter `4` and `7` in the **Orders to track** list to track the orders which correspond with the vibrations of the four coils in the PC fan motor and the seven fan blades.
4. Enter `0.5` in the **Bandwidth (order)** field and enter `500m` in the **Time interval (s)** field.
5. Drag the **magnitudes** output of the Order Tracking step to the **Data View** tab. The **Data View** tab now displays three graphs: the speed profile graph, the waterfall graph of the order spectra, and the magnitude graph of vibrations at the 4th and 7th orders.
6. Right-click the magnitudes graph and select **Visible Items»Legend**.
7. Run the project. The magnitudes graph displays the magnitude of vibrations at the 4th and 7th orders during the run-up test, as shown in Figure 6-7.

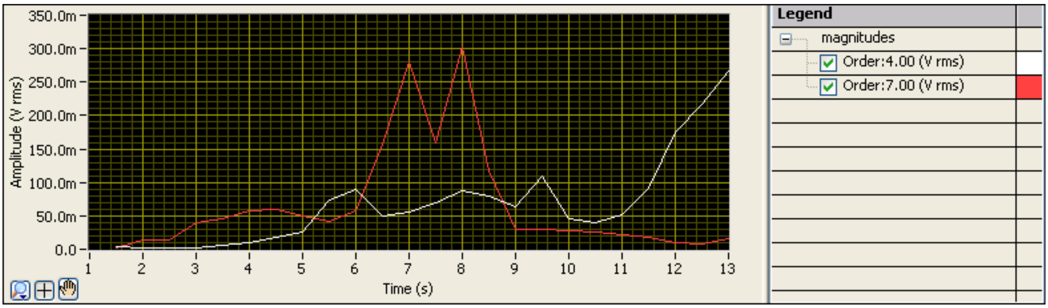


Figure 6-7. Magnitude Plot of 4th and 7th Orders

These peaks are the same as the peaks on the waterfall graph. The highest peak for the vibrations at the 4th order occurred at 13 seconds and the highest peak for the vibrations at the 7th order occurred at 8 seconds.

8. Save and close the project.

7

Extending Sound and Vibration Assistant Projects with LabVIEW

You can use the Sound and Vibration Assistant to define automated measurements by using built-in steps for acquiring, generating, analyzing, or logging signals. You can extend the functionality of the Sound and Vibration Assistant with LabVIEW Graphical Programming in the following ways:

- You can build VIs in LabVIEW and import them into the Sound and Vibration Assistant to expand the built-in measurement options and provide custom step functionality.
- You can convert Sound and Vibration Assistant projects to LabVIEW block diagrams to continue developing the application in LabVIEW. By doing so, you can use the interactive tools in the Sound and Vibration Assistant to define measurement procedures, and then convert the sequence into a LabVIEW block diagram to build automated test applications.

Note References or examples of SignalExpress also apply to the Sound and Vibration Assistant.

Importing LabVIEW VIs into the Sound and Vibration Assistant as Steps

Use the Run LabVIEW VI steps in the Sound and Vibration Assistant to call custom LabVIEW VIs. You can call a LabVIEW VI from the Sound and Vibration Assistant to do the following:

- Read or write data to more file formats
- Use the VI to further analyze data
- Define a measurement algorithm

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

Note You must have LabVIEW 8.6 or later to complete the exercises in this section.

1. Select **Help»Open Example»SignalExpress**, double-click the `Tutorial` folder, and double-click `User Step.seproj`.

   This project uses the Create Analog Signal step to generate a signal.
2. Save the project as `My Imported VI.seproj` in a convenient location.
3. Click the **Add Step** button and select **Run LabVIEW VI»Run LabVIEW 8.6 VI**.

   The VI you run in this exercise was saved in LabVIEW 8.6. You must use the version of the Run LabVIEW VI step that matches the version of LabVIEW you saved the VI in.
4. Click the **Browse** button in the **Select VI** section on the **Step Setup** tab and select `Limiter-LV86.vi` in the `SignalExpress\Examples\Tutorial` directory.

Note If you receive an error stating Unknown LabVIEW Version, open the VI in LabVIEW 8.6 and save the VI. This ensures that the VI is saved in the correct version of LabVIEW and executes with the Run LabVIEW 8.6 VI step in the Sound and Vibration Assistant without error. Then click the **Refresh VI** button on the **Step Setup** tab.

Verify that the **Step Setup** tab appears as shown in Figure 7-1.

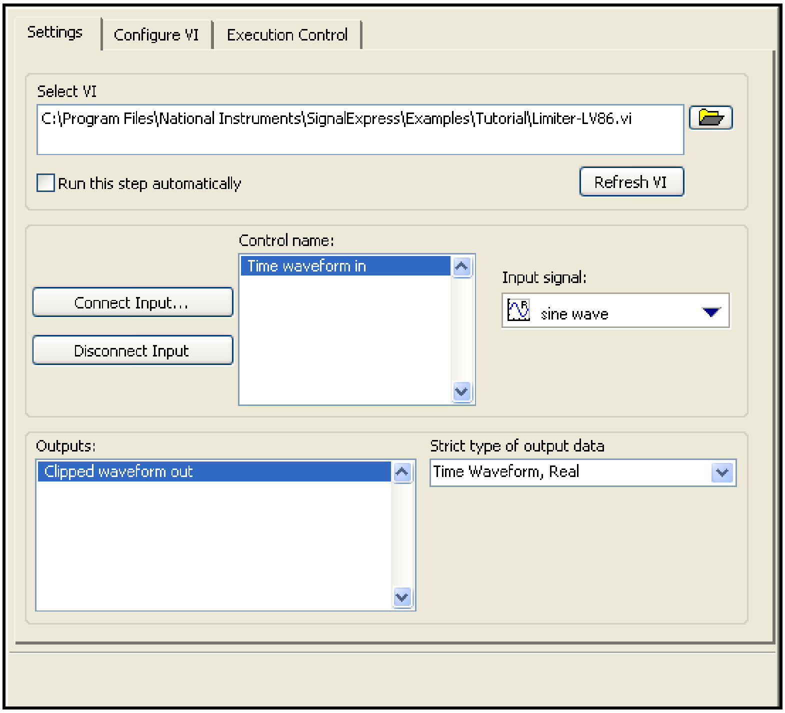


Figure 7-1. Limiter VI Step Setup Tab

The Limiter-LV86 VI accepts a time-domain waveform as an input, clips the signal above and below values that you specify in the **Step Setup** tab, and returns the clipped waveform as an output signal.

When you import a LabVIEW VI, the Sound and Vibration Assistant maps the inputs of the VI as input signals or parameters and the outputs of the VI as output signals.

An input signal appears in the Project View as an input to a step, which means that you can pass signals as inputs to a VI. A parameter is a value you can configure in 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**.

5. Click the **Configure VI** tab in the **Step Setup** tab to display the VI.
6. Enter `100` in the **Upper limit** field and `20` in the **Lower limit** field to specify the limits.

7. Drag the **Clipped waveform out** output of the Run LabVIEW 8.6 VI step to the **Data View** tab.
8. Drag the **sine wave** output of the Create Analog Signal step to the **Data View** tab. The graph on the **Data View** tab displays the two signals: the complete sine wave and the clipped waveform, as shown in Figure 7-2.

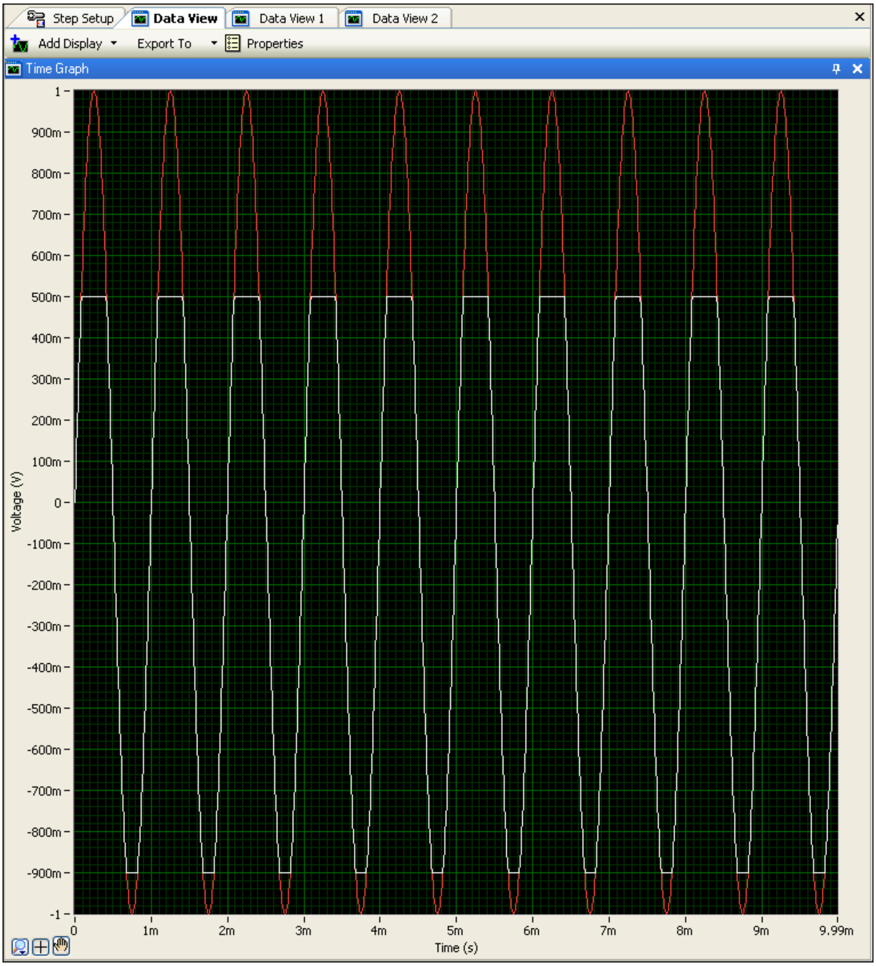


Figure 7-2. Sine Waveform with Clipped Waveform

9. Save and close the project.

Refer to the *Sound and Vibration Assistant Help*, available by selecting **Help»Sound and Vibration Assistant Help**, for more information about using LabVIEW VIs in the Sound and Vibration Assistant and developing VIs that work well in the Sound and Vibration Assistant.

Converting Sound and Vibration Assistant Projects to LabVIEW Block Diagrams

The Sound and Vibration Assistant can convert Sound and Vibration Assistant projects into LabVIEW block diagrams.

Note To convert a Sound and Vibration Assistant project to a LabVIEW VI, you must have the LabVIEW 8.2 Full Development System or later installed. You also must have the LabVIEW Support for Sound and Vibration Analysis component installed to the version of LabVIEW to which you want to generate code.

Complete the following steps to convert a Sound and Vibration Assistant project to a LabVIEW block diagram:

1. Select **Help»Open Example»Sound and Vibration Assistant**, double-click the `Tutorial` folder, and double-click `SVA_Analyze Signal From UFF File (Tutorial).seproj`.
2. Select **Tools»Generate Code»LabVIEW Diagram**.

Note If you have multiple versions of LabVIEW installed, this option generates code in the last version of LabVIEW you launched.

3. Save the new VI as `My LabVIEW VI.vi` and click the **OK** button. The Sound and Vibration Assistant generates the new VI and opens the VI in the last version of LabVIEW you opened.

   The resulting LabVIEW block diagram reflects the execution of the project in the Sound and Vibration Assistant. The LabVIEW block diagram consists of LabVIEW Express VIs wired together. Each Express VI correlates to a step in the Sound and Vibration Assistant project. You can double-click an Express VI to display the same **Step Setup** as in the Sound and Vibration Assistant. 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. When you convert an Express VI into a subVI, you cannot convert the subVI back into an Express VI. You can view the block diagram to see how the LabVIEW block diagram executes and modify the functionality of the VI.

   The LabVIEW front panel displays the **time waveforms** input signal, the **spectrum** input signal, the outputs of **averaging done** and

averages completed from the Power Spectrum step, and the outputs of **peaks** and **number of peaks** from the Peak Search step.

4. Double-click the UFF58 Read File Express VI on the block diagram. Verify that the UFF58 file in the **UFF58 filepath** field is `Triangle wave (1000 Hz).uff` and click the **OK** button.

5. Click the **Run** button, shown at left, to run the VI. On the front panel in LabVIEW, the **time waveforms** graph and the **spectrum** graph update continuously, as shown in Figure 7-3, until you stop the VI.

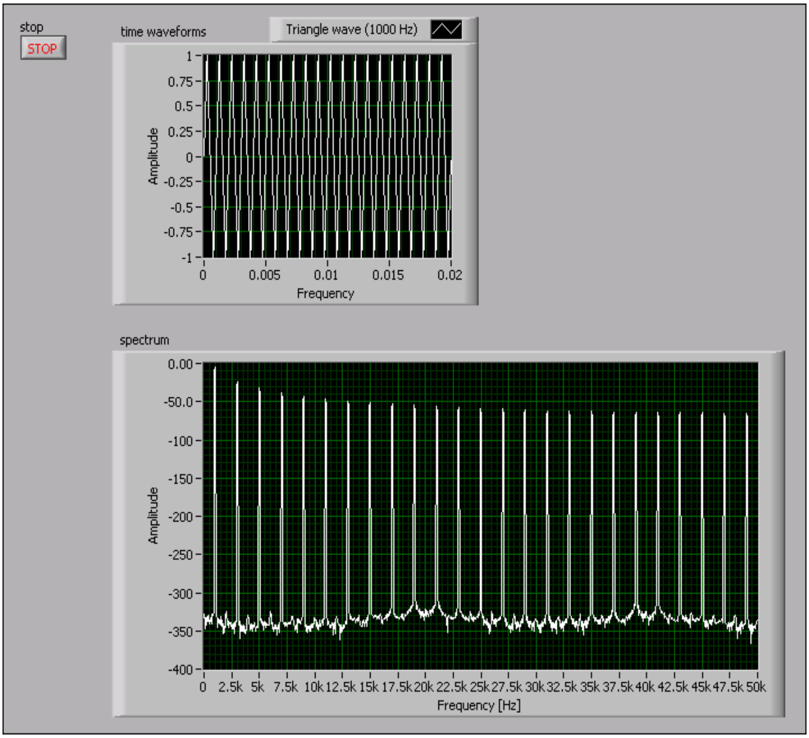


Figure 7-3. LabVIEW Front Panel

6. Click the **Stop** button, shown at left, to stop the VI.
7. Save the VI and close LabVIEW.

Note When you convert a Sound and Vibration Assistant project with logging or hardware configuration, the Sound and Vibration Assistant generates a LabVIEW block diagram with only one Express VI containing the entire contents of the project. 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 the Sound and Vibration Assistant.

Sound and Vibration Assistant Example Projects

The Sound and Vibration Assistant provides example projects that demonstrate capabilities of the Sound and Vibration Assistant with National Instruments hardware. Access these projects by selecting **Help»Open Example»Sound and Vibration Assistant**. Review these examples to learn more about the features of the Sound and Vibration Assistant or to start with a project that closely resembles your needs.

Opening a Sound and Vibration Example Solution

You can find completed solutions for each chapter of this manual by selecting **Help»Open Example»Sound and Vibration Assistant**. For example, open `SVA_Analyze Signal from UFF File.seproj` to view the completed solution for Chapter 2, *Working with Projects*.

Using Hardware with the Sound and Vibration Assistant

The Sound and Vibration Assistant supports a variety of National Instruments hardware for acquiring and generating signals. You can generate or acquire and log analog signals in the Sound and Vibration Assistant using National Instruments MIO devices, DSA 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. You must have NI-DAQmx 8.9.5 or later installed to use the Sound and Vibration Assistant with NI hardware.

Refer to the *Sound and Vibration Assistant Help*, available by selecting **Help»Sound and Vibration Assistant Help**, for more information about using hardware with the Sound and Vibration Assistant.

Web Resources

Refer to the National Instruments Web site at ni.com/soundandvibration for additional resources.

A

Technical Support and Professional Services

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

- **Support**—Technical support at `ni.com/support` includes the following resources:
  - **Self-Help Technical Resources**—For answers and solutions, visit `ni.com/support` for software drivers and updates, a searchable KnowledgeBase, product manuals, step-by-step troubleshooting wizards, thousands of example programs, tutorials, application notes, instrument drivers, and so on. Registered users also receive access to the NI Discussion Forums at `ni.com/forums`. NI Applications Engineers make sure every question submitted online receives an answer.
  - **Standard Service Program Membership**—This program entitles members to direct access to NI Applications Engineers via phone and email for one-to-one technical support, as well as exclusive access to self-paced online training modules at `ni.com/self-paced-training`. All customers automatically receive a one-year membership in the Standard Service Program (SSP) with the purchase of most software products and bundles including NI Developer Suite. NI also offers flexible extended contract options that guarantee your SSP benefits are available without interruption for as long as you need them. Visit `ni.com/ssp` for more information.

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

- **Training and Certification**—Visit ni.com/training for training and certification program information. You can also register for instructor-led, hands-on courses at locations around the world.
- **System Integration**—If you have time constraints, limited in-house technical resources, or other project challenges, National Instruments Alliance Partner members can help. To learn more, call your local NI office or visit ni.com/alliance.

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