

Getting Started with the LabVIEW™ DSP Module

Version 2.0

Contents

Introduction	1
Example 1—Getting Started with DSP VIs and Projects	2
Launching LabVIEW and Opening a Project	2
Opening a VI for a DSP Target	5
Building, Downloading, and Running a DSP Application	7
Example 2—Creating a New Project to Acquire and Analyze Signals.....	8
Creating the Project	8
Saving the DSP VI and the Project	12
Creating the Front Panel	13
Adding Elemental I/O Nodes to the Block Diagram	15
Completing the Block Diagram	17
Building, Downloading, and Running a DSP Application	19
Where to Go for Support.....	20

Introduction

The LabVIEW DSP Module 2.0 uses a project framework to help you manage and implement the components of a DSP application. Specifically, you must use a project to define a DSP target for which you can build and run a DSP application.

The two examples in this manual assume you are familiar with basic LabVIEW concepts. Refer to the *Getting Started with LabVIEW* manual, available by selecting **Start»All Programs»National Instruments»LabVIEW»LabVIEW Manuals** and opening `LV_Getting_Started.pdf` for exercises that teach you basic LabVIEW concepts.

Example 1—Getting Started with DSP VIs and Projects

This example uses the Heterodyne project, located in the `labview\examples\EmbeddedDSP` directory, and an NI SPEEDY-33 board. The VI in this project shows double-sideband modulation, also called signal heterodyning or signal mixing.



Note You also can use a Texas Instruments 6711 DSK, a Spectrum Digital 6713 DSK, or a Spectrum Digital 6416 DSK target with this example.

Launching LabVIEW and Opening a Project

You use a project to group together LabVIEW and non-LabVIEW files, to specify the hardware resources available on the DSP target, and to download a DSP application to a DSP target. When you save a project, LabVIEW creates a project file (`.lvproj`) that includes the target configuration, a link to the DSP VI, and a list of any subVIs.



Note Saving the project file does not save the DSP VI. You must save the DSP VI separately or select **File»Save All** in the **Project Explorer** window to save the project and any open VIs.

Complete the following steps to launch LabVIEW and open the LabVIEW project, `Heterodyne.lvproj`.

1. Launch LabVIEW.
2. In the **Getting Started** window, shown in Figure 1, select **File»Open Project** to open the **Select the Project to Open** dialog box.



Tip You also can click **Open** in the **Files** section of the **Getting Started** window to open a project or VI.

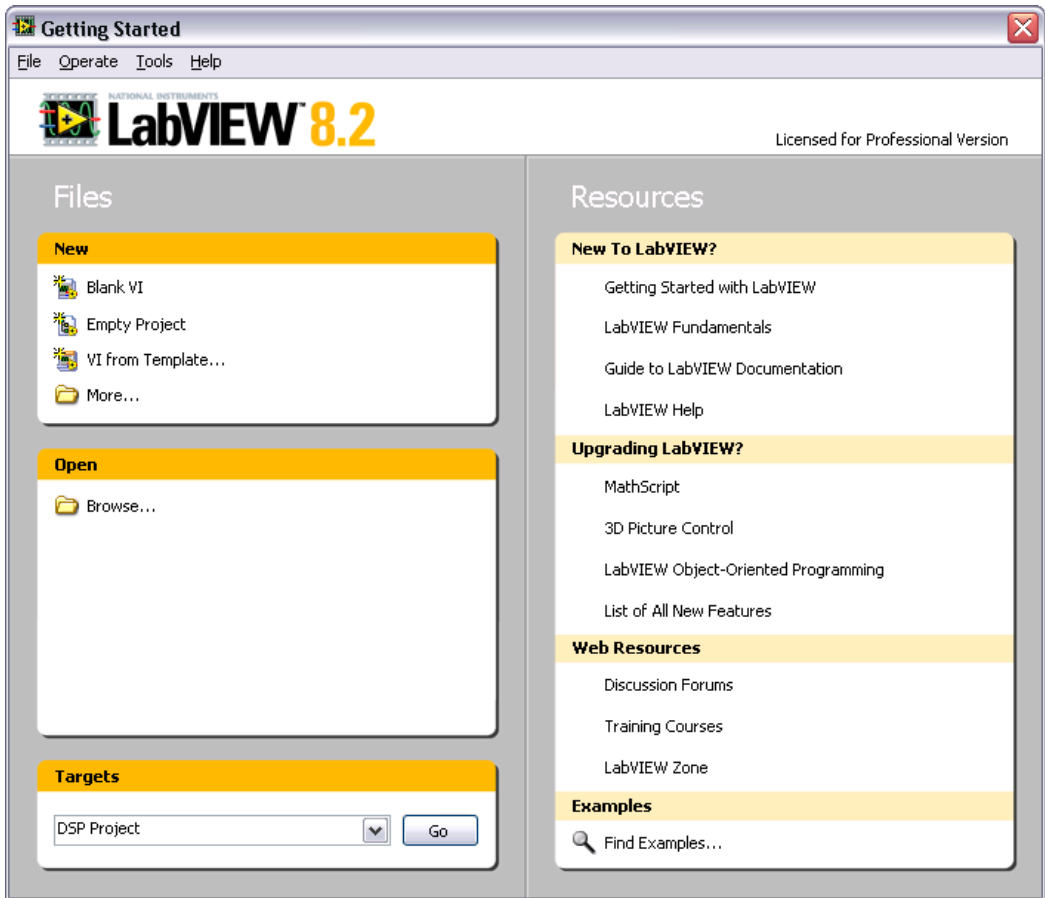


Figure 1. Getting Started Window

3. Navigate to the `labview\examples\EmbeddedDSP` directory and open `Heterodyne.lvproj`.



Note The same LabVIEW project file can contain information about more than one target. Notice that the Heterodyne project contains target information for five targets: My Computer, DSK6416, DSK6711, DSK6713, and SPEEDY-33.

- Expand the target you are using in the **Project Explorer** window to see the items and VIs the project associates with that target, as shown in Figure 2.

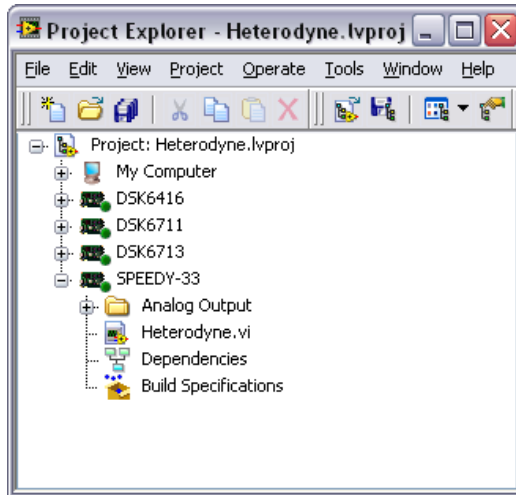


Figure 2. Targeting the SPEEDY-33 with the Heterodyne VI

The following items appear under each DSP target in this example: **Analog Output**, **Heterodyne.vi**, **Dependencies**, and **Build Specifications**.

The **Analog Output** folder contains the analog output resources available on the specific DSP target. Refer to the [Adding Elemental I/O Nodes to the Block Diagram](#) section of this manual for more information about adding input and output (I/O) resources for a DSP target. The Heterodyne VI, which contains the graphical code for the DSP application, only uses one analog output resource.

The **Dependencies** item automatically lists any subVIs necessary for the DSP application to build and run properly on the DSP target. The **Build Specifications** item contains options for downloading to the flash memory on the DSP target and distributing the DSP application. Refer to the *Using LabVIEW Projects* topic in the *LabVIEW Help*, available by selecting **Help»Search the LabVIEW Help**, for more information about dependencies and build specifications.



Tip When searching for a specific topic in the *LabVIEW Help* from the **Search** tab, place a checkmark in the **Search titles only** checkbox and enter the topic title in the **Type in the word(s) to search for** textbox. This limits the number of topics that the *LabVIEW Help* finds so you can locate the topic more quickly.

Opening a VI for a DSP Target

You must open a DSP VI within a project to use the hardware resources available on the DSP target and to build and download the DSP application to the target.

Complete the following steps to open the Heterodyne VI in a DSP target application instance.



Tip Refer to the *Working with Application Instances* topic in the *LabVIEW Help* for more information about working with different targets.

1. Double-click **Heterodyne.vi** under the DSP target you are using in the **Project Explorer** window to open the front panel of the VI, shown in Figure 3. You create user interfaces for DSP VIs in the same way you create user interfaces in LabVIEW for Windows.

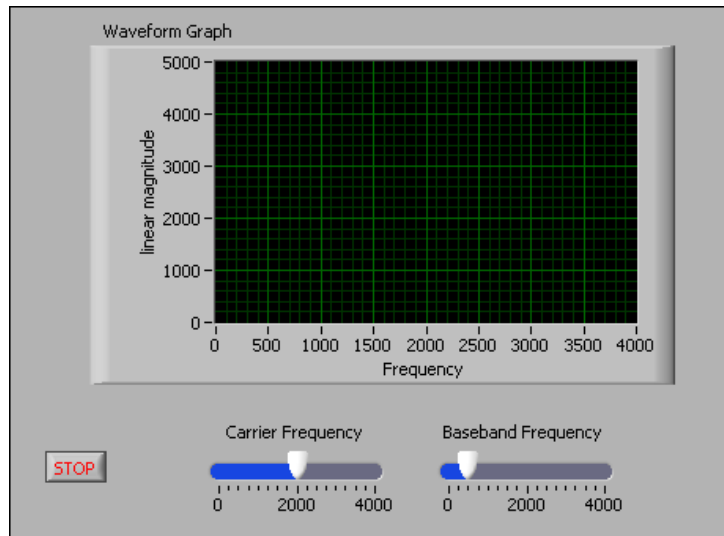


Figure 3. Looking at the Heterodyne VI Front Panel

The waveform graph on the front panel displays the heterodyned signal. You can use the slider controls on the front panel to modify the carrier frequency and baseband frequency of the signal.

2. Select **Window»Show Block Diagram** and look at the VIs that the Heterodyne VI uses.



Tip Press the <Ctrl-E> keys to switch from the front panel to the block diagram or from the block diagram to the front panel.

Figure 4 shows the block diagram of the Heterodyne VI. The following VIs are used in the Heterodyne VI:

- Simulate Signal Express VI—Generates sine waves. One instance generates the carrier frequency, and one instance generates the baseband frequency. The two sine waves are multiplied together, which results in a mixed signal. The product of these two signals is the input to the analog output Elemental I/O Node.
- Spectral Measurements Express VI—Computes the FFT with linear magnitude processing and displays the signal on a waveform graph on the front panel.
- Analog Output Elemental I/O Node—Writes data to the Digital-to-Analog (D/A) converter, also known as a CODEC (coder-decoder), on the SPEEDY-33 target. You can configure how the VI writes data to the analog output by right-clicking the **AO0** item name in the Elemental I/O Node and selecting **Properties** from the shortcut menu.

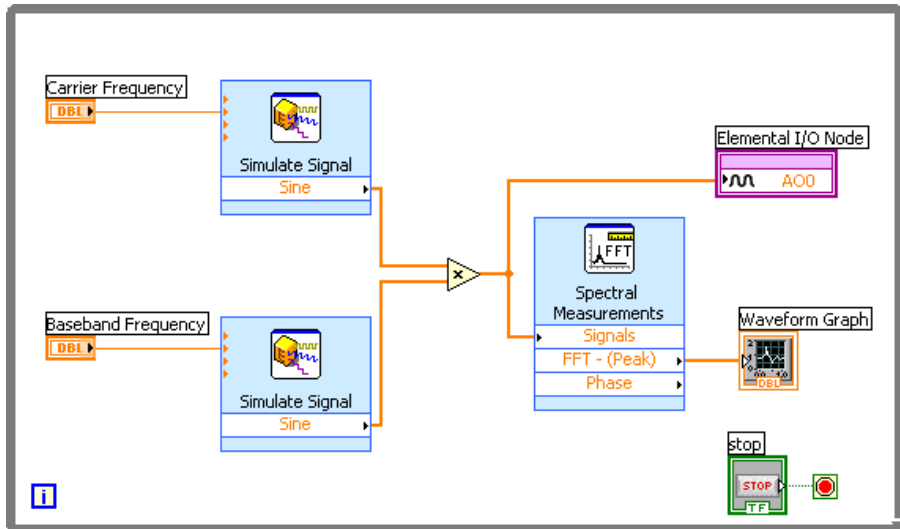


Figure 4. Looking at the Heterodyne VI Block Diagram

Building, Downloading, and Running a DSP Application

Complete the following steps to build the Heterodyne VI into a DSP application and download and run the DSP application on the target.

1. Make sure the DSP target is connected to the host computer.
2. Make sure the DSP VI targets the correct application instance by looking in the bottom left corner of the front panel or block diagram window, as shown in Figure 5.



Tip If you have more than one project open, the application instance shortcut menu displays the project name as well as the target.

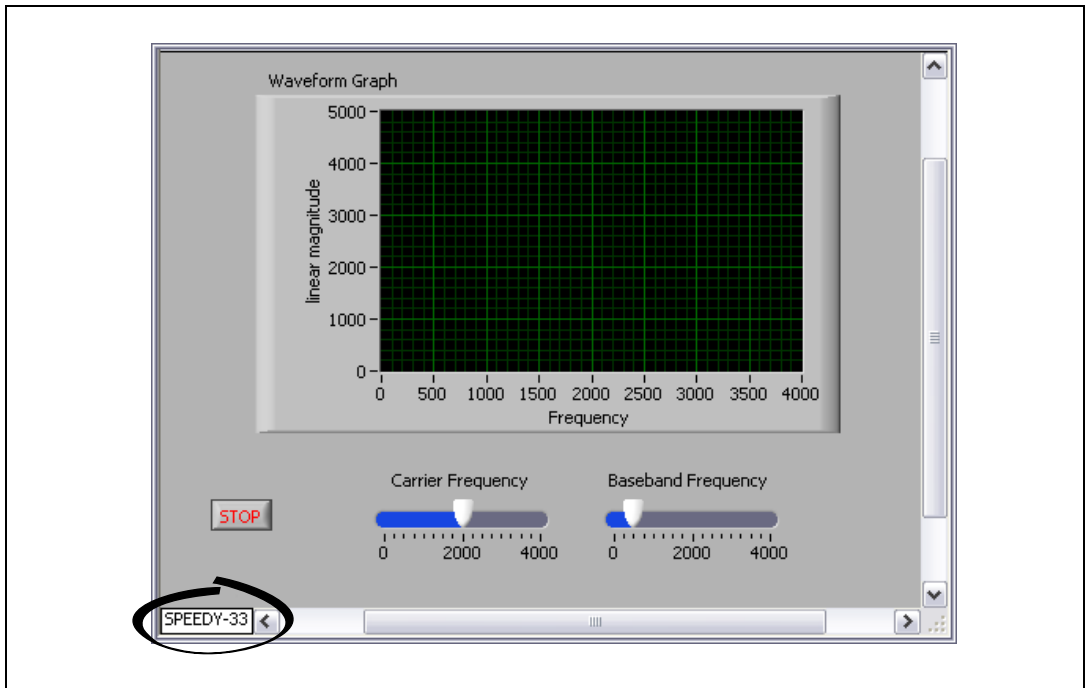


Figure 5. Checking the Application Instance for the Correct Target



3. Click the **Run** button, shown at left, to build, download, and run the DSP application on the SPEEDY-33 target. When you click the **Run** button, the **Build Status** dialog box, shown in Figure 6, appears and displays the progress of the building and downloading of the DSP application on the SPEEDY-33 target.



Note The building and downloading process takes extra time the first time you build and download a DSP application because LabVIEW must initialize and load the compiler.

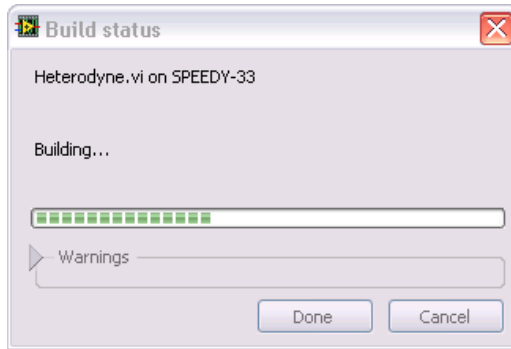


Figure 6. Building the DSP VI into a DSP Application

When the application is running on the DSP target, you can modify the carrier frequency and baseband frequency using the slider controls on the front panel.

The waveform graph on the front panel displays the frequency response of the signal. If you plug in speakers or headphones to the analog output on the SPEEDY-33 target, you also can hear the changes to the carrier frequency and baseband frequency.

4. Click the **Stop** button on the front panel to stop the DSP application.

Example 2—Creating a New Project to Acquire and Analyze Signals

In this example, you will create a new project and program the SPEEDY-33 to measure an analog input signal and turn on an LED when the DSP target reads a signal higher than a value you specify. Because this DSP VI uses Elemental Input/Output (I/O) Nodes to acquire the signal and light up the LED, you must add analog and digital I/O items to the DSP project.



Note This example uses analog input so you need an external microphone if the target is not a SPEEDY-33. Alternatively, you can simulate an input signal using the Simulate Signal Express VI instead of an analog input Elemental I/O Node.

Creating the Project

This example uses the New DSP Project Wizard to guide you through creating a project. Complete the following steps to create a DSP project before you create the DSP VI.

1. Launch LabVIEW.

2. In the **Getting Started** window, shown in Figure 7, select **DSP Project** from the **Targets** pull-down menu and click the **Go** button to open the New DSP Project Wizard.



Tip You also can select **File»New**, then select **Project»Project from Wizard»DSP Project** to open the New DSP Project Wizard.

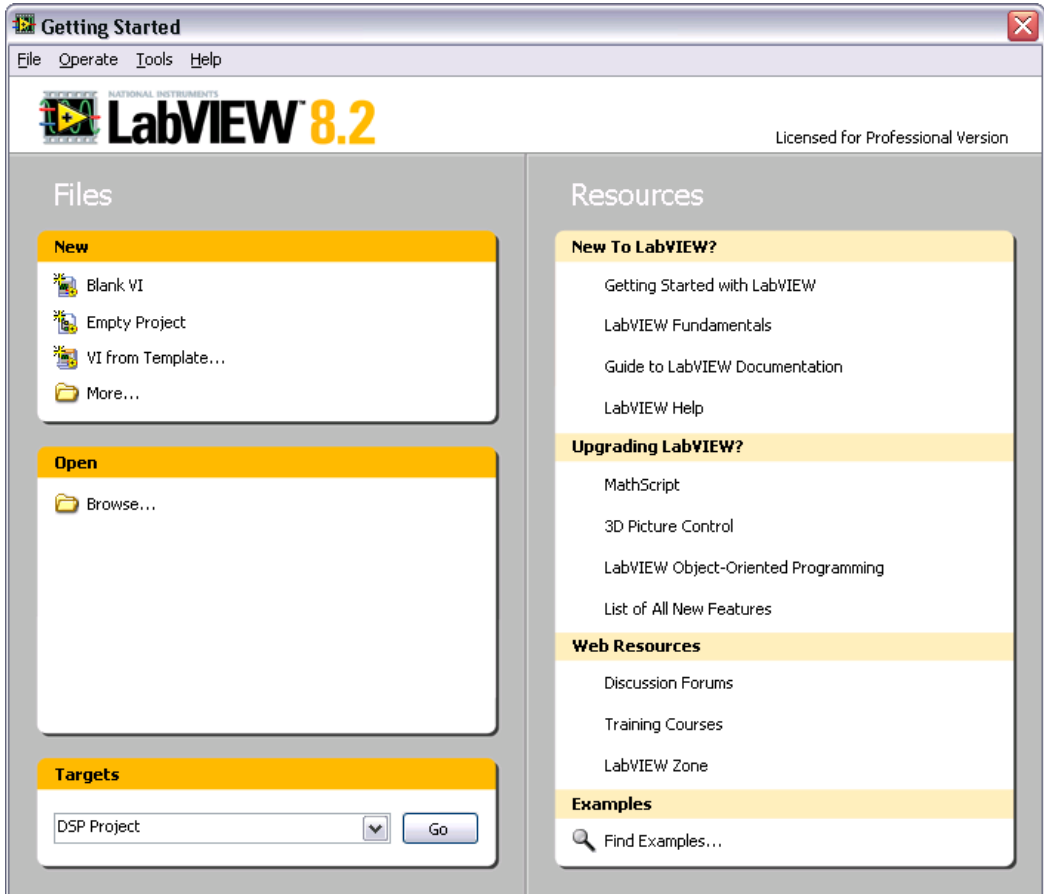


Figure 7. Getting Started Window

3. Select the **New DSP project, blank VI** from the **Project type** pull-down menu on the **Define project information** page, shown in Figure 8, to create a new project with an empty VI.



Tip Select **New DSP project, existing VI** from the menu if you want to create a project for an existing DSP VI. When you create a project with an existing VI, you must enter the path of the VI in the **VI path** text box or browse for the existing VI.

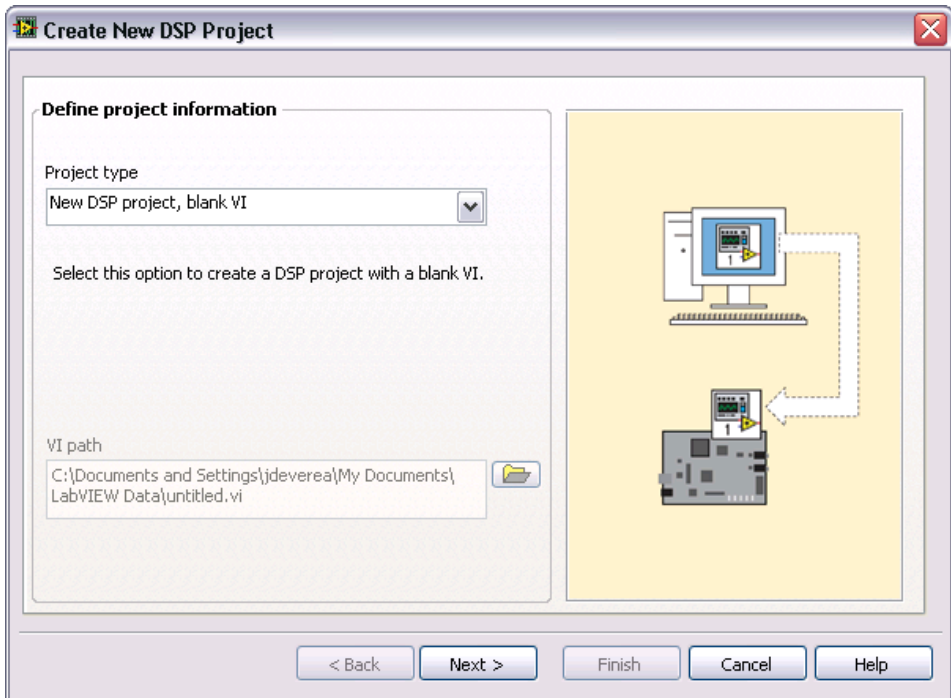


Figure 8. Creating a New Project with a New VI

4. Click the **Next** button.
5. Select the target hardware you are using from the **Target type** pull-down menu on the **Select target type and input/output resources** page.

6. Make sure there is a checkmark in the **Analog Input** and **Digital Output** checkboxes. By default, the wizard adds all the resources available to the target. You can remove the checkmarks from any of the input/output resources checkboxes that you do not need, as shown in Figure 9.

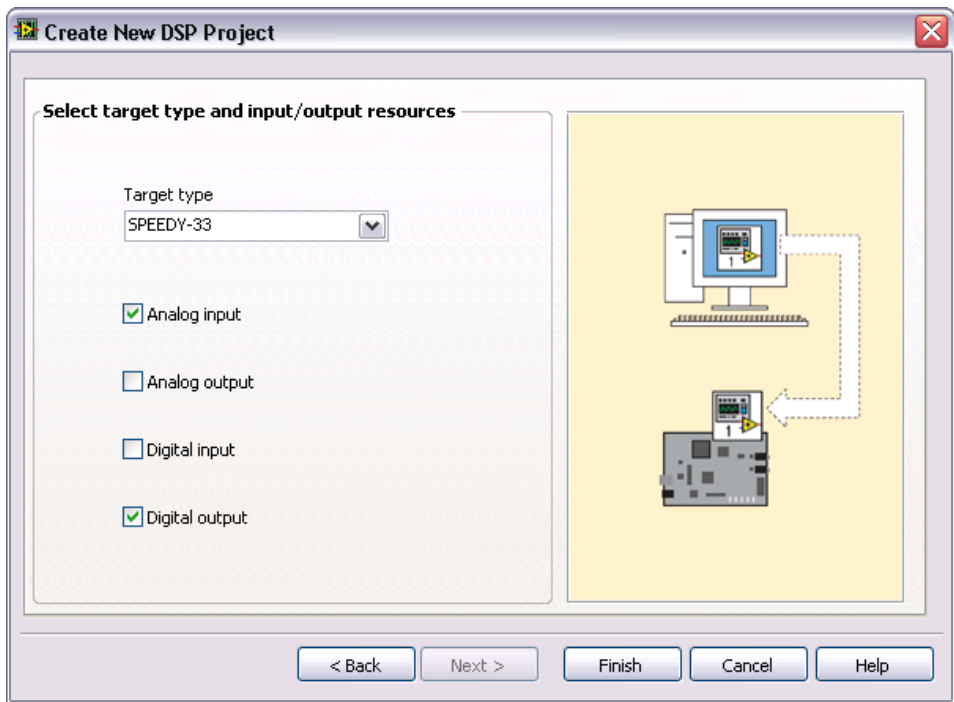


Figure 9. Specifying the DSP Target Hardware and Resources



Tip You can remove analog and digital input/output items from the **Project Explorer** window after you create the project if the VI does not need to access those resources. Refer to the *Using Analog and Digital I/O Resources* topic in the *LabVIEW Help* for more information about working with Elemental I/O resources.

7. Click the **Finish** button to close the wizard and create the DSP project with a blank VI.

The **Project Explorer** window that appears contains the DSP target you selected in step 5. The blank VI appears under the DSP target in the **Project Explorer** window as **Untitled x**.

Saving the DSP VI and the Project

Complete the following steps to create a DSP VI that reads a signal from an analog input channel on the DSP target and turns on an LED if the maximum value the DSP target reads is higher than a constant you specify.

1. Right-click the DSP target in the **Project Explorer** window and select **Rename** from the shortcut menu.
2. Enter `DSP Target` in the text box and press the <Enter> key to change the name of the target, as shown in Figure 10.

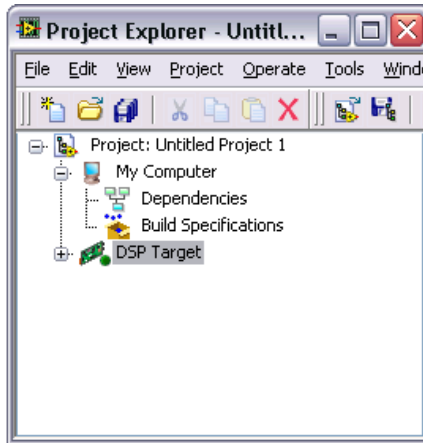


Figure 10. Renaming the Target in the Project

3. Expand **DSP Target** in the **Project Explorer** window to access the new, untitled VI.
4. Right-click **Untitled x** in the **Project Explorer** window and select **Save** from the shortcut menu to open the **Name the VI** dialog box. The number *x* varies depending on how many new VIs you open after you first launch LabVIEW.
5. Save the VI as `Tutorial.vi`, and click the **OK** button. You can save the VI anywhere on the computer.
6. Select **File»Save** in the **Project Explorer** window and save the project as `Tutorial.lvproj`.



Note Saving the project does not save the VI. The project file and the VI are separate files. Select **File»Save All** to save the project and any open VIs.

The **Project Explorer** window now appears like Figure 11.

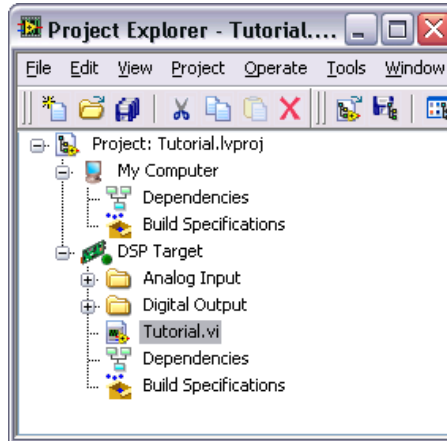


Figure 11. Saving the DSP VI and the LabVIEW Project



Tip If you want to add additional blank DSP VIs to the project, right-click the DSP target in the **Project Explorer** window and select **New>VI** from the shortcut menu. To add a new VI to run on Windows, right-click **My Computer** and select **New>VI**.

Creating the Front Panel

Complete the following steps to create the DSP VI front panel.

1. Select **View>Controls Palette** in the front panel window to open the **Controls** palette.



Tip You also can right-click in the front panel window and click the thumbtack in the upper left corner to open and pin the **Controls** palette.

2. Place the following controls on the front panel window as shown in Figure 12.
 - A waveform graph located on the **Modern>Graph** palette.
 - A numeric control and numeric indicator located on the **Modern>Numeric** palette.
 - A round LED indicator located on the **Modern>Boolean** palette.

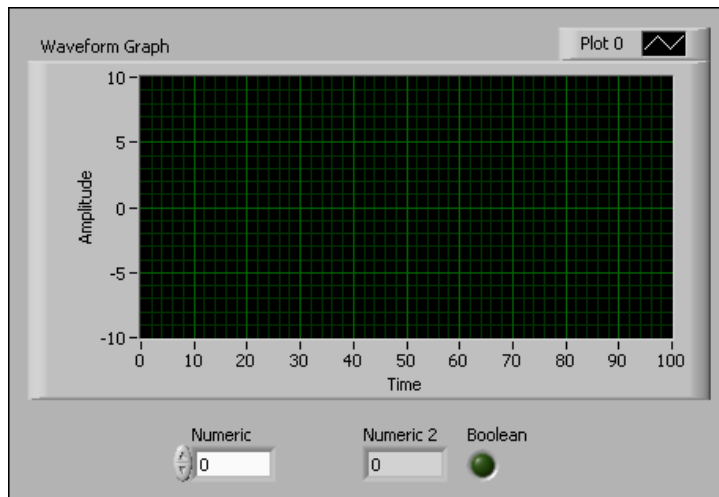


Figure 12. Creating a Front Panel



Tip If you cannot find the object you are looking for, click the **Search** button on the **Controls** or **Functions** palette toolbar. Type the name of the object for which you want to search. LabVIEW searches as you type and displays any matches in the search results text box.

3. Rename the controls by double-clicking the labels and entering new names.
 - Change the numeric control to **Max Limit**.
 - Change the numeric indicator to **Max Value**.
 - Change the round LED to **Warning**.
4. Right-click the y-axis of the waveform graph and remove the checkmark from **AutoScale Y** in the shortcut menu.
5. Click the **10** on the y-axis of the waveform graph and enter 4000.
6. Click the **-10** on the y-axis of the waveform graph and enter -1000.

The waveform graph displays the analog signal the DSP target acquires and the **Max Value** numeric indicator displays the maximum value of the signal. You can use the **Max Limit** numeric control to specify the input signal level that turns on the LED on the DSP target and the front panel.

- Click the **100** on the x-axis of the waveform graph and enter 128. The front panel window now appears as shown in Figure 13.

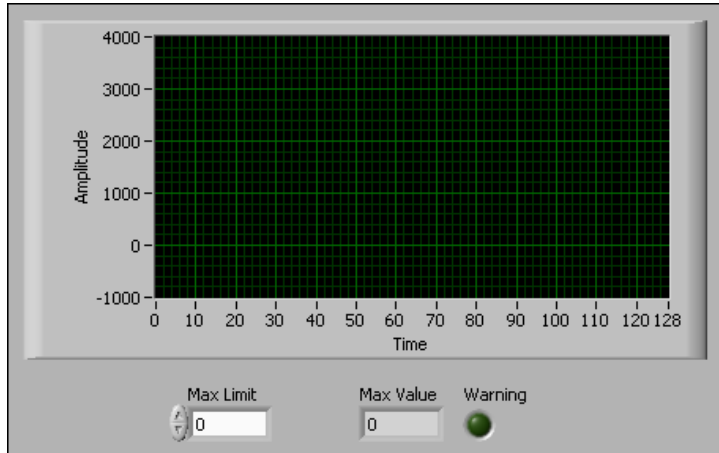


Figure 13. Renaming Controls and Indicators on the Front Panel

Adding Elemental I/O Nodes to the Block Diagram

Use Elemental I/O Nodes to access the analog input capabilities or to use the LEDs on the DSP target. Complete the following steps to add Elemental I/O Nodes to the DSP VI block diagram.

- Press the <Ctrl-E> keys to switch to the block diagram window.
- Select **View»Functions Palette** to open the **Functions** palette.



Note The items on the **Functions** palette change depending on the application instance.



Tip If you use the Simulate Signal Express VI instead of an analog input Elemental I/O Node to create a signal, skip to step 5.

- Place an Analog Input node located on the **Programming»Elemental IO** palette on the block diagram.

The analog input item in the Elemental I/O Node appears on the block diagram with orange text because analog input is a floating-point numeric data type. The text is not black because you selected analog input resources when you created the new project in step 6 of the [Creating the Project](#) section.



Note If the terminal text appears in black, you must add the Elemental I/O item to the project by right-clicking **DSP Target** in the **Project Explorer** window and selecting **New»Elemental I/O** from the shortcut menu. You can add the appropriate resources in the **New Elemental I/O** dialog box.

4. You can configure the analog input by right-clicking the analog input item in the Elemental I/O Node on the block diagram and selecting **Properties** from the shortcut menu.

Use the **Elemental I/O Node Properties** dialog box to configure analog input to the DSP target. Configuration options differ between DSP targets and can include the sample rate, framesize, and gain.

Leave the default values in the **Elemental I/O Node Properties** dialog box, as shown in Figure 14, and click the **OK** button.

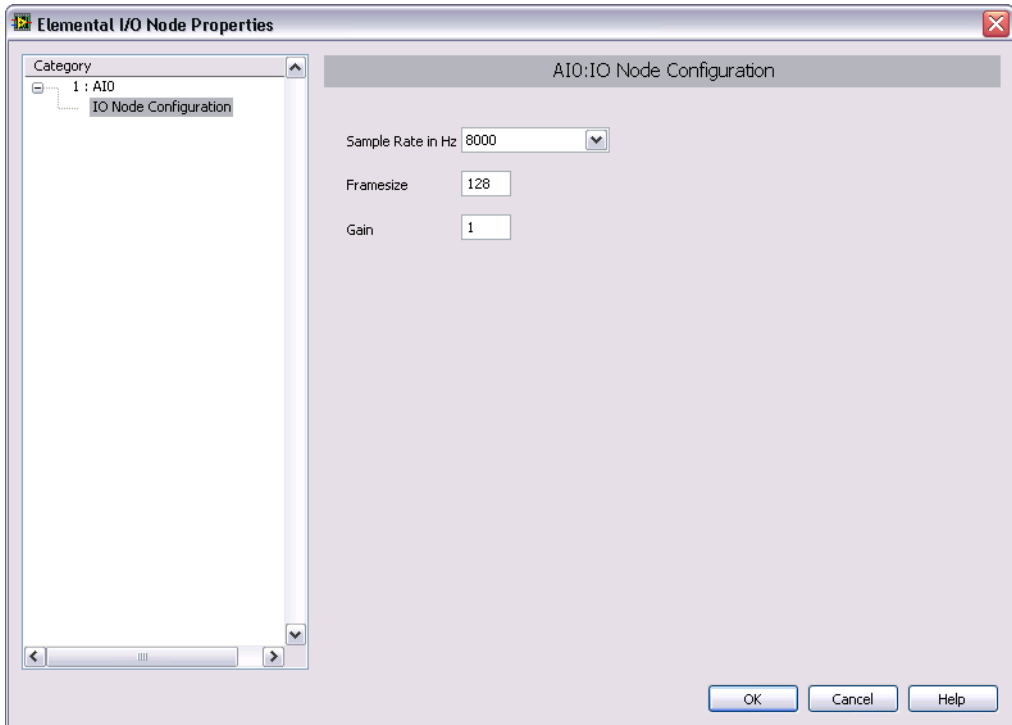


Figure 14. Configuring the Analog Input for SPEEDY-33

5. (Optional) If you use the Simulate Signal Express VI instead of the analog input on the DSP target, you must configure the Simulate Signal Express VI so you can see the signal on the waveform graph. Complete the following to configure the Simulate Signal Express VI:
 - a. Double-click the Simulate Signal Express VI to open the **Configure Simulate Signal** dialog box.
 - b. Enter 3 000 in the **Amplitude** text box.
 - c. Place a checkmark in the **Add noise** checkbox.
 - d. Select **Uniform White Noise** in the **Noise type** drop down menu.
 - e. Enter 1 000 in the **Noise amplitude** textbox.

- Place a DSP LED node located on the **Programming»Elemental I/O** palette on the block diagram as shown in Figure 15.

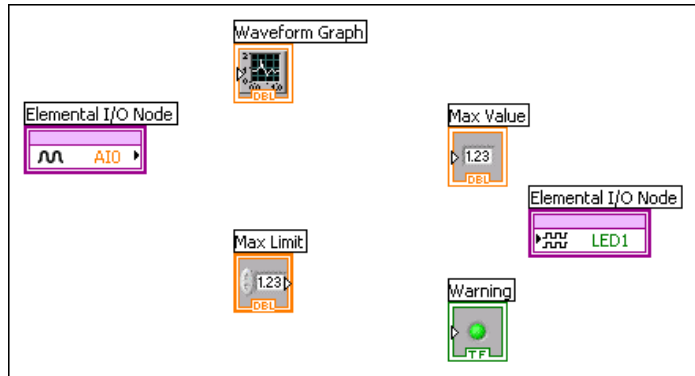


Figure 15. Adding the Elemental I/O Nodes to the Block Diagram



Tip You can change which LED lights up when you run the application by dragging the top or bottom handle of the Elemental I/O Node to expose different LEDs. You also can click the terminal name and select a different LED from the **Digital Output** shortcut menu.

Completing the Block Diagram

Complete the following steps to finish the block diagram as shown in Figure 16.

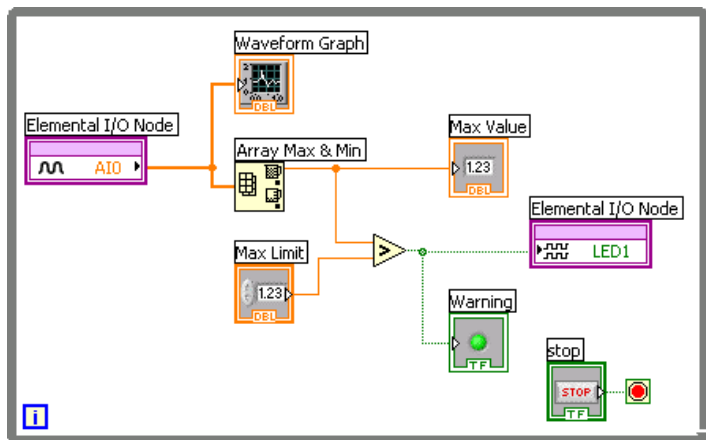


Figure 16. Finishing the Block Diagram

- Select **Help»Show Context Help** to display the **Context Help** window. The **Context Help** window displays basic information about LabVIEW objects when you move the cursor over each object.



Tip You also can press <Ctrl-H> to open and close the **Context Help** window.

2. Place a While Loop located on the **Programming»Structures** palette around the controls and indicators on the block diagram.

While Loops repeat the subdiagram inside it until the conditional terminal, which is an input terminal, receives a particular Boolean value. Notice that when you place the While Loop from the **Structures** palette, there is a Stop Boolean already wired to the conditional terminal.

- a. Double-click the Stop Boolean to locate the **Stop** button on the front panel.
 - b. Move the **Stop** button on the front panel to the right of the **Warning LED**.
 - c. Press <Ctrl-E> to return to the block diagram.
3. Place an Array Max & Min function located on the **Programming»Array** palette on the block diagram.
 4. Wire the analog input item in the Elemental I/O Node to the waveform graph and the **array** input of the Array Max & Min function.



Note Wire the signal output from the Simulate Signal Express VI to the waveform graph if you use the Simulate Signal Express VI instead of the Elemental I/O Node.

5. Place a Greater? function located on the **Programming»Comparison** palette on the block diagram.
6. Wire the **max value** output of the Array Max & Min function to the **Max Value** indicator and the **x** input of the Greater? function.
7. Wire the **Max Limit** control to the **y** input of the Greater? function.
8. Wire the **x > y?** output of the Greater? function to the **Warning** indicator and the DSP LED item in the Elemental I/O Node.
9. Save the VI.

Building, Downloading, and Running a DSP Application

Complete the following steps to build, download, and run the DSP application on the DSP target and interact with the DSP target application using controls and indicators on the front panel.



1. Click the **Run** button, shown at left, on the front panel window or block diagram window to build, download, and run the DSP application on the DSP target.



Tip You also can right-click the VI in the **Project Explorer** window and select **Run** from the shortcut menu.

2. Whistle or snap your fingers into the microphone on the DSP target and watch the waveform graph update. If the amplitude of the sound goes higher than the maximum limit you set in **Max Limit**, the LED lights up on the DSP target and the front panel.



Note If you use the Simulate Signal Express VI instead of the Elemental I/O Node, right-click the y-axis of the waveform graph while the VI is running and place a checkmark next to **AutoScale Y** in the shortcut menu.

3. Enter a new value in **Max Limit** and repeat step 2.
4. While the DSP application is running, right-click **DSP Target** in the **Project Explorer** window and select **Utilities»Target Information** from the shortcut menu to open the **Target Information** dialog box shown in Figure 17. The **Target Information** dialog box provides information about memory usage on the DSP target.

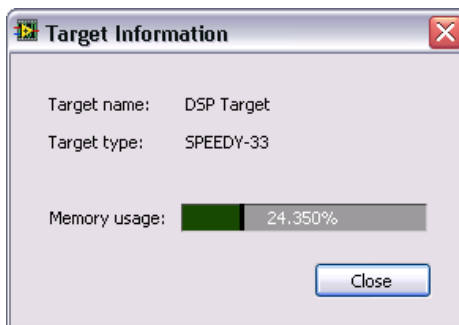


Figure 17. Viewing Memory Usage While the DSP Application Is Running

5. Click the **Close** button to close the **Target Information** dialog box.
6. Click the **Stop** button on the front panel to stop the application.

Where to Go for Support

The National Instruments Web site is your complete resource for technical support. At ni.com/support you have access to everything from troubleshooting and application development self-help resources to email and phone assistance from NI Application Engineers.

National Instruments corporate headquarters is located at 11500 North Mopac Expressway, Austin, Texas, 78759-3504. National Instruments also has offices located around the world to help address your support needs. For telephone support in the United States, create your service request at ni.com/support and follow the calling instructions or dial 512 795 8248. For telephone support outside the United States, contact your local branch office:

Australia 1800 300 800, Austria 43 0 662 45 79 90 0,
Belgium 32 0 2 757 00 20, Brazil 55 11 3262 3599,
Canada 800 433 3488, China 86 21 6555 7838,
Czech Republic 420 224 235 774, Denmark 45 45 76 26 00,
Finland 385 0 9 725 725 11, France 33 0 1 48 14 24 24,
Germany 49 0 89 741 31 30, India 91 80 41190000,
Israel 972 0 3 6393737, Italy 39 02 413091, Japan 81 3 5472 2970,
Korea 82 02 3451 3400, Lebanon 961 0 1 33 28 28,
Malaysia 1800 887710, Mexico 01 800 010 0793,
Netherlands 31 0 348 433 466, New Zealand 0800 553 322,
Norway 47 0 66 90 76 60, Poland 48 22 3390150,
Portugal 351 210 311 210, Russia 7 095 783 68 51,
Singapore 1800 226 5886, Slovenia 386 3 425 4200,
South Africa 27 0 11 805 8197, Spain 34 91 640 0085,
Sweden 46 0 8 587 895 00, Switzerland 41 56 200 51 51,
Taiwan 886 02 2377 2222, Thailand 662 278 6777,
United Kingdom 44 0 1635 523545

National Instruments, NI, ni.com, and LabVIEW are trademarks of National Instruments Corporation. Refer to the *Terms of Use* section on ni.com/legal for more information about National Instruments trademarks. Other product and company names mentioned herein are trademarks or trade names of their respective companies. For patents covering National Instruments products, refer to the appropriate location: **Help»Patents** in your software, the `patents.txt` file on your CD, or ni.com/patents.