Acquiring Data

Approximate Time

You can complete this exercise in approximately 15 minutes.

Background

NI-DAQmx is a programming interface you can use to communicate with data acquisition devices. Measurement & Automation Explorer (MAX) is a tool automatically installed with NI-DAQmx and used to configure National Instruments hardware and software.

This exercise introduces you to the Express VIs used to acquire data using NI-DAQmx. With NI-DAQmx you can create simulated devices in MAX, so no data acquisition hardware is required to complete this exercise. An NI-DAQmx simulated device is a software replica of data acquisition hardware.

Overview

In the following exercise, you will create an NI-DAQmx task that continuously takes a voltage reading and plots the data on a waveform graph. You will use the DAQ Assistant Express VI to create the voltage task. The voltage task will acquire data from an NI-DAQmx simulated device that you configure in MAX.
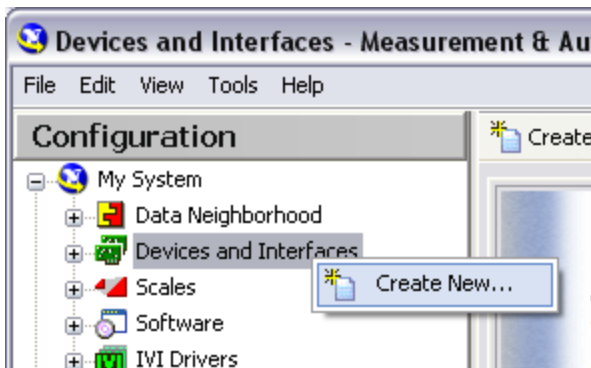
Completed Exercise
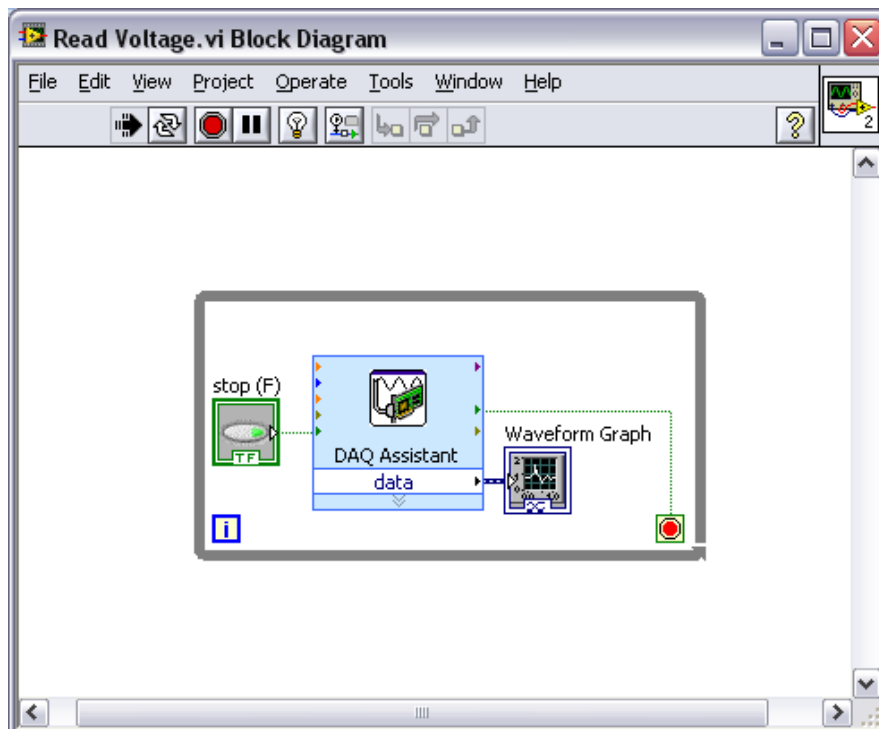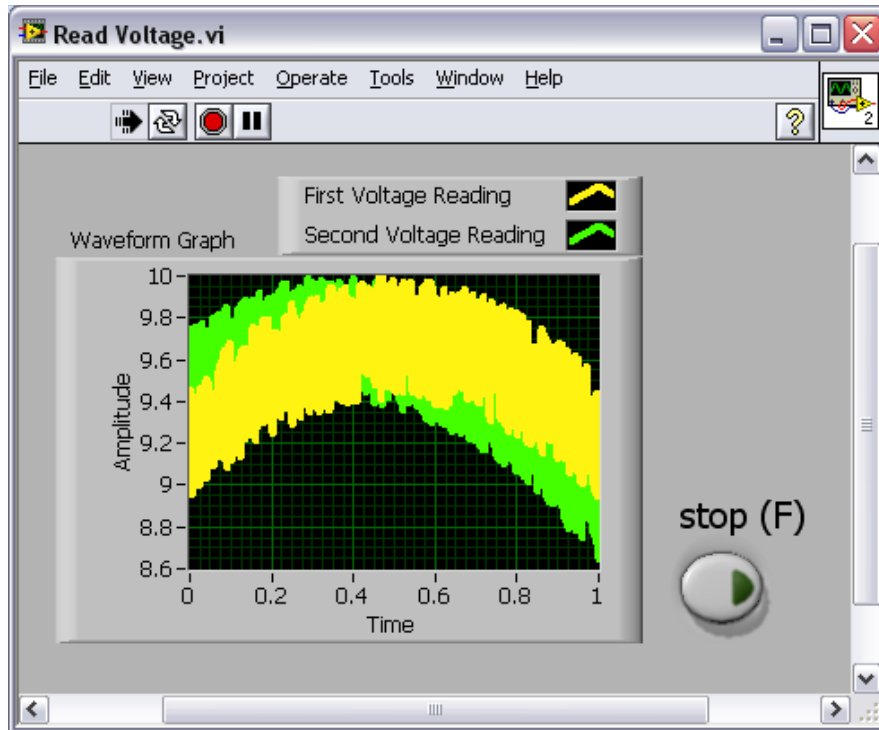
Step-by-Step Instructions

1. Creating a Simulated DAQ Device

Complete the following steps to configure a simulated data acquisition device in MAX.

a. Launch Measurement & Automation Explorer (MAX) by double-clicking the desktop icon or navigation to **Programs » National Instruments » Measurement and Automation** in the Windows **Start** menu.

b. Click **My System** and right-click **Devices and Interfaces**.

c. Select **Create New...** in the drop-down menu.

d. Select **NI-DAQmx Device » NI-DAQmx Simulated Device** and click **Finish**. The **Choose Device** window will prompt you to select a device.

With this list, you can browse hundreds of devices supported by NI-DAQmx. You can create an NI-DAQmx simulated device of nearly any NI-DAQmx-supported device.
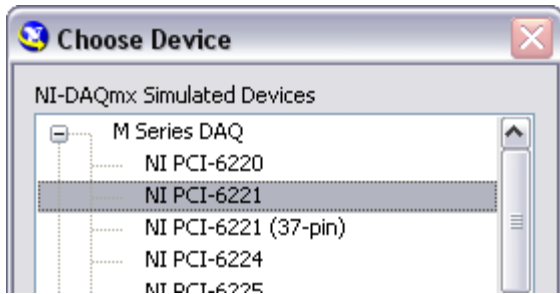
e. Select **M Series DAQ » NI PCI-6221** and click **OK**. The NI-DAQmx simulated device will appear in the **Configuration** pane within MAX.


The icon color of the NI-DAQmx simulated devices differs from real devices. Real devices are green and chassis are grey ; simulated devices and chassis are yellow.

2. Creating an NI-DAQmx Task

In NI-DAQmx, a task is a collection of one or more channels, timing, triggering, and other properties. Conceptually, a task represents a measurement or generation you want to perform. For example, you can create a task to measure temperature from one or more channels on a DAQ device. Complete the following steps to create and configure a task that reads a voltage level from your NI-DAQmx simulated device:

a. Open a new VI.

b. On the block diagram, display the **Functions** palette and select **Express » Input** to display the **Input** palette.

c. Select the DAQ Assistant Express VI on the **Input** palette and place it on the block diagram. The DAQ Assistant launches and the **Create New** dialog box appears.
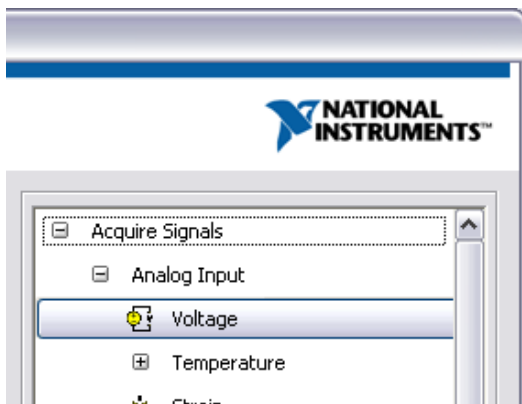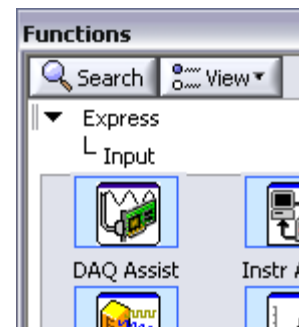

d. Click **Analog Input** under **Acquire Signals**.

e. Select **Voltage** to create a new voltage analog input task. The dialog box displays a list of channels for our simulated PCI-6221. The channels listed depends on the number of channels on the selected device.

f. In the **Supported Physical Channels** list, select **ai0** and click the **Finish** button. The DAQ Assistant opens a dialog displaying options for configuring the selected channel to complete a task.
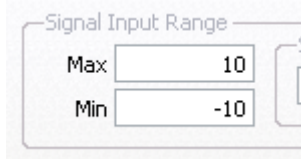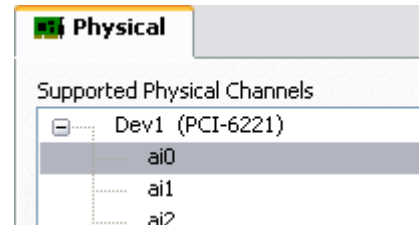
g. In the **Input Range** section of the **Settings** page, enter *10* for the **Max** value and enter *-10* for the **Min** value.

h. Select **N Samples** from the **Acquistion Mode** menu in the **Timing Settings** section.

i. Enter a value of *1000* in the **Samples To Read** box.
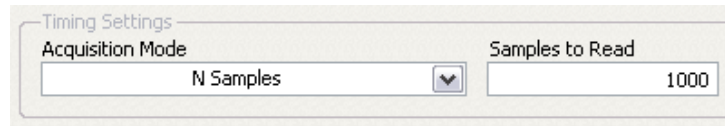
3. Testing the Task

You can test the task to verify that you correctly configured the channel. Complete the following steps to confirm that you are acquiring data:

a. Click the **Run** button once or twice to confirm that you are acquiring data.

b. Click the **OK** button to save the current configuration and close the DAQ Assistant. LabVIEW automatically builds the VI based on this configuration.

c. Save the VI as Read Voltage.vi on the desktop.

4. Graphing Data from a DAQ Device

You can use the task you created to graph the data acquired from a DAQ device. Complete the following steps to plot the data from the channel on a waveform graph and change the name of the signal:

a. On the block diagram, right-click the **data** output and select **Create » Graph Indicator** from the shortcut menu.

b. Display the front panel and run the VI three or four times. Observe the waveform graph. **Voltage** appears in the plot legend at the top of the waveform graph.
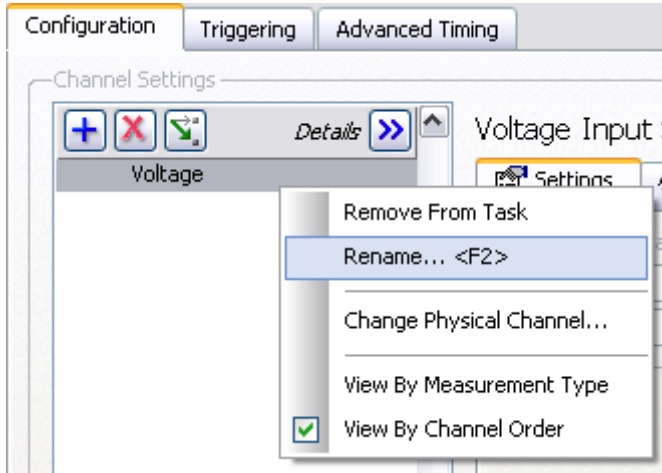

c. On the block diagram, right-click the DAQ Assistant Express VI and select **Properties** from the shortcut menu to open the DAQ Assistant.

d. Right-click **Voltage** in the list of channels and select **Rename** from the shortcut menu to display the **Rename a channel or channels** dialog box.

You also can select the name of the channel and press the <F2> key to display the **Rename a channel or channels** dialog box.

e. In the **New Name** text box, enter *First Voltage Reading*, and click the **OK** button.

f. In the **DAQ Assistant** dialog box, click the **OK** button to save the current configuration and close the DAQ Assistant.
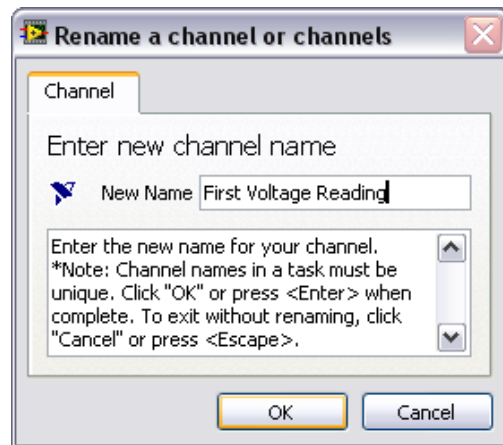
g. Display the front panel and run the VI. **First Voltage Reading** appears in the waveform graph plot legend.

h. Save the VI.

5. Editing an NI-DAQmx Task

You can add a channel to the task so you can compare two separate voltage readings. You also can customize the task to acquire the voltage readings continuously. Complete the following steps to add a new channel to the task and acquire data continuously:

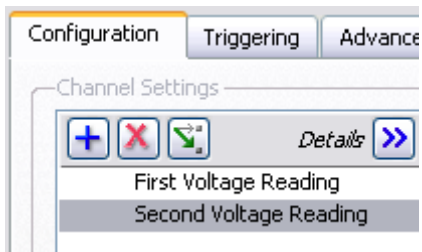   a. Double-click the DAQ Assistant Express VI to open the DAQ Assistant.

    b. Click the **Add Channels** button

    c. Select the Voltage channel from the **Add Channel** menu to display the **Add Channels To Task** dialog box.
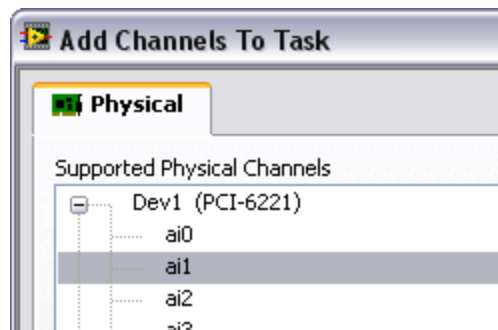
    d. Select any unused physical channel in the **Supported Physical Channels** list, and click the **OK** button to return to the DAQ Assistant.


    e. Click the **Add Channels** button Rename the channel *Second Voltage Reading*.

   f. Select **Continuous** from the **Acquistion Mode** menu in the **Timing Settings** section.

When you set timing and triggering options in the DAQ Assistant, these options apply to all the channels in the list of channels.

    g. Click the **OK** button to save the current configuration and close the DAQ Assistant. The **Confirm Auto Loop Creation** dialog box appears.
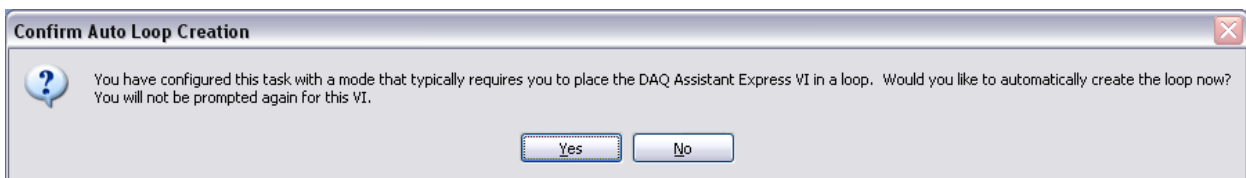
h. Click the **Yes** button. LabVIEW places a While Loop around the DAQ Assistant Express VI and the graph indicator. A stop button appears on the block diagram wired to the **stop** input of the DAQ Assistant Express VI. The **stopped** output of the Express VI is wired to the conditional terminal of the While Loop.
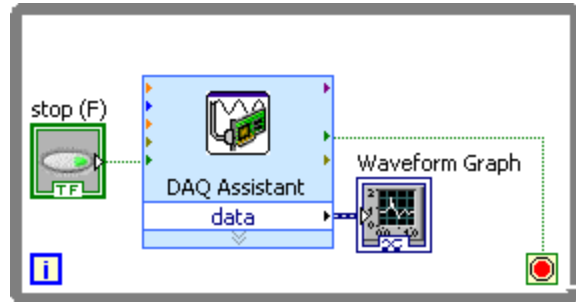
If an error occurs or you click the **stop** button while the VI is running, the DAQ Assistant Express VI stops reading data and the **stopped** output returns a TRUE value and stops the While Loop.

6. Visually Comparing Two Voltage Readings

Because you have two voltage readings displayed on a graph, you can customize the plots to distinguish between the two. Complete the following steps to customize the plot color on the waveform graph:

a. On the front panel, expand the plot legend to display two plots.

b. Run the VI. Two plots appear on the graph. The legend displays both plot names.

c. Right-click **First Voltage Reading** in the plot legend and select **Color** from the shortcut menu. Using the color picker, select a color such as yellow so the plot is easy to read. Change the plot color of **Second Voltage Reading**.

d. Stop the VI.

e. Save the VI.