InstrumentStudio Features

2023-02-16

Contents

InstrumentStudio Manual

The InstrumentStudio manual contains information on navigating, configuring, and using InstrumentStudio. In addition, the manual contains reference material describing concepts that you may need when using devices with InstrumentStudio.

To learn more about what you can accomplish with InstrumentStudio and National Instruments devices, refer to ni.com/instrumentstudio.

New Features

The following sections describe new features added to each release of InstrumentStudio.

2023 Q1

January 2023

- Added SMU measurement auto-range support for PXIe-4190 and PXIe-4190 (500 kHz).
- Added LCR Meter impedance measurement auto-range support in sweep mode.
- Added remote RF Signal Analyzer panel viewing as a preview feature.

> **Note** Preview features have limited functionality and are not fully supported.

Related information:

- SMU Measurement Autorange
- Enabling Preview Features
- Remote RF Signal Analyzer Panel

2022 Q4

October 2022

- Added oscilloscope **Manual Override** mode, enabling simultaneous manual configuration of Record length and Sample Size in oscilloscope settings .

Related information:

- Setting Sample Rate and Record Length Manually

2022 Q3

July 2022

- SMU measurement auto-range support for DC Power devices that support it.
- Single-channel sweep mode support for PXIe-4190 when used in LCR mode.
- LCR Meter mode usability improvements for PXIe-4190.
- RF Signal Analyzer instrument with support for RFmx personalities.

Related information:

- SMU Measurement Autorange
- SMU/VPS Modes of Operation
- Analyzing RF Signals with an RF Signal Analyzer

2022 Q2

April 2022

- Single-channel sweep mode measurement added as a preview feature for PXIe-4190 when used in LCR mode.

2022 Q1

January 2022

- LCR panel debug monitor functionality and usability improvements for the LCR PXIe-4190 module.

- Custom gating support for Scope measurements.

Related information:

- Customizing Oscilloscope Measurements

2021 Q4

October 2021

- Added new LCR panel supporting interactive use with the PXIe-4190 module.
- Added access and visibility to pin maps.
- Improved UI responsiveness and added pin filtering for systems with high channel counts.

Related information:

- SMU/VPS Modes of Operation
- Setting an Active Project Pin Map

2021

April 2021

- Added new RFSG Signal Generator panel supporting interactive use and debugging using PXIe-5820, PXIe-5830, PXIe-5831, PXIe-5832, PXIe-5840, and PXIe-5841.
- Added support for the PXIe-4135(40W) and PXIe-4137(40W) source measure unit, newly supported in NI-DCPOWER 20.7.

Related information:

- Generating RF Signals with an RF Signal Generator

2020

November 2020

- Added support for the PXIe-4139(40W) source measure unit, newly supported in NI-DCPower 20.5.

- Added support for multichannel session in the SMU/Power Supply panel, newly supported in NI-DCPower 20.6. Initializing an NI-DCPower session with independent channels allows you to use multiple instruments in the same session. With independent channels, you can configure multiple channels of the same instrument, or of multiple instruments independent of one another within the same session.
- Added support for monitoring merged channels in debug mode on the PXIe-4147. Merged channels are newly supported in NI-DCPower 20.6. Merging channels of an NI SMU allows you to use multiple channels in unison to increase the current you can source beyond the normal maximum for a single SMU channel.

Related information:

- NI-DCPower Independent Channel Sessions

2019 SP1

May 2020

- Added support for the PXIe-4147 source measure unit, newly supported in NI-DCPower 20.0.

2019

September 2019

- Multiple NI-SCOPE devices of the same model can now be synchronized in the same oscilloscope panel.
- Added support for selecting trigger sources and events from other NI devices in your system.
- Added waveform graphing, pulse current, pulse voltage, voltage sequence, and current sequence modes to SMU panels, including automatic hardware synchronization across multiple devices.
- Added asymmetric limits and software-protected ranges to SMU/Power Supply panels.
- Added NI-DCPower SourceAdapt configuration to SMU panels, useful for transient measurements.

- Added channel views to hide idle channels, show power calculations, and condense channel views to the SMU/Power Supply panels.
- Added inductance and capacitance measurements along with open and short cable compensation to digital multimeter (DMM) panels.
- Added the ability to generate waveforms from binary files to the waveform generator panel.
- Added debugging and monitoring support to the waveform generator panel.
- Added exporting configuration files from the waveform generator panel for use with the NI-FGEN driver.
- Added capability to export device configurations to TestStand for use in test step configuration.
- Added ability to use pin maps created in Digital Pattern Editor to group devices and channels by site and use pin names for channel aliases.

Related information:

- SMU/VPS Modes of Operation
- Configuring SourceAdapt Parameters to Measure Transients
- Performing Cable Compensation
- Generating Waveforms with a Waveform Generator
- Exporting a Device Configuration
- Exporting Configurations to TestStand
- Pin Maps

2018 SP2

January 2019

- Added support for the PXIe-5163 oscilloscope, newly supported in NI-SCOPE 18.7.
- Added CSV file format for data capture.
- Added support for exporting instrument events to PXI chassis trigger lines.
- Added support for arbitrary waveform generation mode to the Waveform Generator panels.

- Added support for frequency list and sweep modes to the Waveform Generator panels.
- Added FFT channels, frequency markers, and FFT channel measurements to the Oscilloscope panels.
- Added additional 2-channel measurements to the Oscilloscope panels.

Related information:

- Capturing Data
- Waveform Modes
- Adding FFT Channels and Markers

2018 SP1

September 2018

- Added support for the PXIe-5110, PXIe-5111, and PXIe-5113 oscilloscopes, newly supported in NI-SCOPE 18.6.

2018

May 2018

- This was the initial release of InstrumentStudio.

Supported Hardware

The following tables list hardware that can be used with InstrumentStudio, and the first available version of InstrumentStudio to support the hardware.

NI-DCPower

The following NI-DCPower hardware can be used with a SMU/Power Supply instrument in InstrumentStudio.

Model Name	First Available
PXIe-4112	2018
PXIe-4113	2018
PXIe-4135	2018

Model Name	First Available
PXIe-4135 (40W)	2021
PXIe-4136	2018
PXIe-4137	2018
PXIe-4137 (40W)	2021
PXIe-4138	2018
PXIe-4139	2018
PXIe-4139 (40W)	2020
PXIe-4140	2018
PXIe-4141	2018
PXIe-4142	2018
PXIe-4143	2018
PXIe-4144	2018
PXIe-4145	2018
PXIe-4147	2019 SP1
PXIe-4154	2018
PXIe-4162	2018
PXIe-4163	2018
PXIe-4190 [1]	2021 Q4
PXIe-4190 (500 kHz) [1]	2022 Q3
PXI-4110	2018
PXI-4130	2018
PXI-4132	2018

[1]The PXIe-4190 and PXIe-4190 (500 kHz) can be used with either an SMU/Power Supply instrument or an LCR meter in InstrumentStudio

NI-DMM

The following NI-DMM hardware can be used with a Digital Multimeter instrument in InstrumentStudio.

Model Name	First Available
PXIe-4080	2018

Model Name	First Available
PXIe-4081	2018
PXIe-4082	2018
PXI-4065	2018
PXI-4070	2018
PXI-4071	2018
PXI-4072	2018
PCIe-4065	2018
PCI-4065	2018
USB-4056	2018
PCI-4070	2018

NI-FGEN

The following NI-FGEN hardware can be used with a Waveform Generator instrument in InstrumentStudio.

Model Name	First Available
PXIe-5413 (1-channel)	2018
PXIe-5413 (2-channel)	2018
PXIe-5423 (1-channel)	2018
PXIe-5423 (2-channel)	2018
PXIe-5433 (1-channel)	2018
PXIe-5433 (2-channel)	2018
PXIe-5442	2018
PXIe-5450	2018
PXIe-5451	2018
PXI-5402	2018
PXI-5404	2018
PXI-5406	2018
PXI-5412	2018
PXI-5421	2018
PXI-5422	2018

Model Name	First Available
PXI-5441	2018
PCI-5402	2018
PCI-5406	2018
PCI-5412	2018
PCI-5421	2018

NI-SCOPE

The following NI-SCOPE hardware can be used with an Oscilloscope instrument in InstrumentStudio.

Model Name	First Available
PXIe-5105	2018
PXIe-5110	2018
PXIe-5111	2018
PXIe-5113	2018
PXIe-5113	2018
PXIe-5122	2018
PXIe-5160 (2-channel)	2018
PXIe-5160 (4-channel)	2018
PXIe-5162 (2-channel)	2018
PXIe-5162 (4-channel)	2018
PXIe-5163	2018 SP2
PXIe-5164	2018
PXIe-5170 (4-channel)	2018
PXIe-5170 (8-channel)	2018
PXIe-5171 (8-channel)	2018
PXIe-5172 (4-channel)	2018
PXIe-5172 (8-channel, 325T)	2018
PXIe-5172 (8-channel, 410T)	2018
PXIe-5185	2018
PXIe-5185 (1MΩ)	2018

Model Name	First Available
PXIe-5186	2018
PXIe-5186 (1MΩ)	2018
PXIe-5622	2018
PXIe-5622 (DD)	2018
PXI-5105	2018
PXI-5114	2018
PXI-5122	2018
PXI-5124	2018
PXI-5142	2018
PXI-5152	2018
PXI-5153	2018
PXI-5154	2018
PXI-5922	2018
PCI-5105	2018
PCI-5114	2018
PCI-5122	2018
PCI-5124	2018
PCI-5142	2018
PCI-5152	2018
PCI-5153	2018
PCI-5154	2018
PCI-5922	2018
PCIe-5155	2018
USB-5132	2018
USB-5133	2018

NI-RFSG

The following NI-RFSG hardware can be used with a RF Signal Generator instrument in InstrumentStudio.

Model Name	First Available
PXIe-5820	2021
PXIe-5830	2021
PXIe-5831	2021
PXIe-5832	2021
PXIe-5840	2021
PXIe-5841	2021

NI-RFSA

The following NI-RFSA hardware can be used with a RF Signal Analyzer instrument in InstrumentStudio.

Model Name	First Available
PXIe-5644	2022 Q3
PXIe-5645	2022 Q3
PXIe-5646	2022 Q3
PXIe-5663	2022 Q3
PXIe-5663E	2022 Q3
PXIe-5665	2022 Q3
PXIe-5668	2022 Q3
PXIe-5668 with PXIe-5698	2022 Q3
PXIe-5820	2022 Q3
PXIe-5830	2022 Q3
PXIe-5831	2022 Q3
PXIe-5832	2022 Q3
PXIe-5840	2022 Q3
PXIe-5841	2022 Q3

Configuring Panels and Layout

Adjust the arrangement of soft front panels and their associated devices with the Edit Layout window.

An InstrumentStudio layout consists of one large panel and up to four small panels. A large panel includes a measurement graph and is located in the center of the screen. Small panels feature channel settings and are located on the right sidebar. You can also combine multiple devices of the same type into a single panel. Layouts in InstrumentStudio must meet the following requirements:

- The layout must have exactly one large panel.
- You can place oscilloscopes only in a large panel. Only oscilloscopes of the same model may be added to the same panel.
- The layout can have up to four small panels. Small panels are not required.

Creating a Layout

Before you can take measurements and monitor devices in InstrumentStudio, you must create a layout. Create a layout from the Home screen, from the document toolbar, or from any instrument header menu.

Creating a Layout Automatically

When you create a layout automatically, InstrumentStudio detects devices you have installed and creates panels using those devices. Complete the following steps to create an automatic layout in InstrumentStudio:

> **Note** You can edit an automatic layout at any time by choosing **Add/ Remove devices** from the instrument header menu. Opening the Edit Layout window using **Add/Remove devices** automatically filters the device list based on the type of panel you selected Add/Remove devices from. Remove or change the filter using the drop-down menu in the upper left corner.

1. Install the devices you would like to use and make sure they appear in Measurement & Automation Explorer (MAX).
2. Open InstrumentStudio.
3. Select **Auto Create Layout** from the Home screen.

InstrumentStudio creates a large or small panel for each compatible device installed in the system.

Creating a Layout Manually

You can use the **Edit Layout** window to create a manual layout. In a manual layout, you can customize which devices are displayed and group devices of the same type into a single panel.

Refer to Configuring Panels and Layout to learn more about layout rules and restrictions.

Complete the following steps to create or edit a manual layout:

1. Access the Edit Layout window in one of three ways:
   - From the Home screen, select **Manual Layout**.
   - From any instrument header menu, select **Add/Remove devices**.

     > **Note**
     >
     > - Opening the Edit Layout window using **Add/Remove devices** automatically filters the device list based on the type of panel you selected **Add/Remove devices** from. Remove or change the filter using the drop-down menu in the upper left corner.
     > - Only oscilloscopes of the same model may be added to the same panel.

   - From the document toolbar, select the **Edit layout** icon ( ).
2. Search for the device(s) you would like to include in the layout in the left panel of the Edit Layout window. You can filter by device type using the drop-down menu in the upper left corner.
3. Add the device to a panel by selecting either **Create large panel** or **Create small panel** from the device's drop-down menu.
   The device is displayed in the panel you select.

> **Note** You can remove a device from a panel by clicking the × to the right of the device's name.

4. (Optional) Once a panel is created, you can group devices of the same type together in a single panel by selecting the group name from a device's drop-down menu.
5. Select **OK** to create the layout you specified.

You have created a manual layout.

Related concepts:

- Configuring Panels with the Instrument Header Menu

Configuring Panels with the Instrument Header Menu

You can use the instrument header menu to remove devices, export panels to a new tab, relink missing devices, and more.

Each panel in InstrumentStudio has its own instrument header menu, which you can access by clicking the instrument header menu icon in the upper-right corner of the panel. Within the instrument header menu(), you can select the following options:

- **Capture data**—Saves a timestamped screenshot and an NI-TDMS file containing a data snapshot. Refer to Capturing Data for more information.
- **Export configuration**—Saves model-specific configuration files to a specified folder. You can import a configuration file to LabVIEW to apply the device's configuration to a different device session. Refer to Exporting a Device Configuration for more information.
- **[Device name]**—Opens options for the specified device. From this menu, you can remove the device from the panel, relink devices, and access the device's documentation.
- **Add/Remove devices**—Opens the Edit Layout window. Refer to Creating a Layout for more information on the Edit Layout window.

- **Configure debug session**—Opens the Configure Debug Session window, which you can use to enable and disable debugging on devices. Refer to Debugging Programmatic Applications for more information.
- **Launch in new tab**—Opens the panel in a new tab with the same configuration.
- **Delete panel**—Removes the panel from the layout.

Related concepts:

- Capturing Data
- Debugging Programmatic Applications

Related tasks:

- Creating a Layout Manually
- Enabling Debugging with a C, C++, or .NET Application
- Exporting a Device Configuration
- Relinking or Replacing Missing Devices

Relinking or Replacing Missing Devices

Use relinking to re-establish a connection to a missing device, or replace the missing device with a different device of the same model in an existing panel.

If InstrumentStudio is closed and a device used in an InstrumentStudio project goes offline or has its name changed, InstrumentStudio flags that device as missing the next time you open the project. The panel's title bar is gray instead of blue and displays a "Missing hardware" message. The caution icon () appears over the instrument header menu of the affected panel and next to the missing device inside the instrument header menu. InstrumentStudio also notifies you of the missing device in the Edit Layout window.

Relink a missing device through the instrument header menu, or **replace** a missing device in the Edit Layout window. When you relink a missing device through the instrument header menu, InstrumentStudio applies the missing device's existing configuration to the relinked device. When you replace a missing device through the Edit Layout window, InstrumentStudio discards the existing device configuration and creates a new one for the replacement device.

Note You can also relink or locate a missing device through Measurement & Automation Explorer (MAX). If InstrumentStudio is open, changes made in MAX are automatically applied to InstrumentStudio panels. If a device went missing because it was renamed while InstrumentStudio was closed, relink the missing device by changing the device's name in MAX to match the name InstrumentStudio is looking for while the InstrumentStudio panel that contains the missing device is open.

Relinking Missing Devices from the Instrument Header Menu

Note You can also use this method to link a different device of the same model to an existing panel, even if the original device is not missing. Replacing a device in a panel via relinking preserves device settings from the original device and applies them to the replacement device.

1. Open the instrument header menu of the panel ( ) with the missing device.
2. Select the missing device. The missing device has a caution icon ( ) next to it.
3. Select **Relink**.
4. Select the name of the device to link to the panel.

   Note When relinking from the instrument header menu, the relinked device must be the same model as the missing device. InstrumentStudio applies the existing device configuration to the new device when it is linked.

5. Select **Run** on the relinked panel.

You relinked the missing device (or a device of the same model) to the panel.

Replacing Missing Devices from the Edit Layout Window

You cannot directly relink a missing device from the Edit Layout window, but you can remove a missing device and replace it with a different device by completing the following steps:

1. Open the Edit Layout window in one of two ways:
   - From the document toolbar, select the **Edit layout** icon ( ).
   - From any instrument header menu, select **Add/Remove devices**.

   > **Note** Opening the Edit Layout window using **Add/Remove devices** automatically filters the device list based on the type of panel you selected **Add/Remove devices** from. For best results, select **Add/Remove devices** from the panel that contains the device you want to replace.

2. Locate the missing device. The missing device has a caution icon ( ) next to it.
3. Select the × next to the caution icon to remove the missing device from the layout.

   > **Note** If the missing device was not part of a group, removing the device also removes the device's panel.

4. Add a replacement device of the same type to the panel (if adding to a group), or make a new panel with the replacement device.

   > **Note** When replacing missing devices from the Edit Layout window, the new device must follow the same InstrumentStudio layout guidelines as any other device. Refer to Configuring Panels and Layout for more information on layout guidelines.

5. Select **OK** to save your changes.
6. Select **Run** on the edited panel.

Related concepts:

- Configuring Panels with the Instrument Header Menu

Multi-Device Synchronization

In InstrumentStudio, device synchronization generally falls under two categories: software and hardware.

Software Synchronization	Hardware Synchronization
	
Channel acquisition and generation is sequenced in software.	Channel acquisition and generation is sequenced in hardware.
There is no strict timing guarantee between channels.	There is a strict timing guarantee between channels.

Panel Synchronization

The different panels within InstrumentStudio support various levels of device and channel synchronization:

DMM—All channels are always software synchronized.

Waveform generators—All channels are software synchronized by default.

> **Note** Channels within a 2-channel PXIe-5450 or PXIe-5451 waveform generator are always hardware synchronized.

Oscilloscopes

- All channels on a single device are always hardware synchronized.
- All channels within a single panel are hardware synchronized if every device is the same model and is in the same chassis. Channels across different physical devices are synchronized with NI-TClk technology.
- You can also trigger an oscilloscope from another instrument in the system by configuring a digital trigger and selecting the source based on the device name.

SMUs and Power Supplies

- Every channel configured for a sequence or pulse generation within a single SFP are automatically hardware synchronized. You can configure routing options by selecting the icon.
- Channels are software synchronized () in all other cases.
- You can also trigger an SMU from another instrument in the system by configuring a digital trigger and selecting the source based on the device name.

RFSG

- You can only have one RFSG device in a single panel.
- RFSG devices cannot be synchronized.

Remote RF Signal Analyzer Panel

This section describes how to add a remote RF Signal Analyzer to your project to use remote RFmx hardware.
This a preview feature in InstrumentStudio 2023 Q1. Preview features have limited functionality and are not fully supported. The following list describes important limitations to be aware of while using this preview feature.

> **Note**
>
> - The Spectral Analyzer personality is the only available personality in remote Signal Analyzer panels.
> - Remote hardware must be present on the same network in order to be remotely connected.
> - Remote panels perform best when the remote hardware is within your local facility topology.
> - Remotely debugging a remote session is not recommended.

- Enabling Preview Features
- Adding a Remote RFmx Panel to Your Layout
- Configuring Your System for Remote RF Signal Analyzer Panels

Adding a Remote RFmx Panel to Your Layout

Complete the following steps to view RFmx hardware measurements remotely using an InstrumentStudio panel.

Before you can view RFmx hardware measurements remotely, you must configure your system.

1. Enable the remote RFmx panel preview feature in InstrumentStudio settings.
2. Click **Add Remote Hardware**.
3. Enter the IP address or hostname and the port number of the server you want to connect to.
4. Click **OK** to add a panel for the remote device to your layout.
5. Continue setting up your layout as needed.

After adding the panel to your layout, the device name will appear as follows: Device name - Model number (Remote).

> **Note** If a session is running locally on the device, you will only be able to view the panel in debug mode. You must take over the device session in order to interact with the device in control mode.

Related information:

- Configuring Your System for Remote RF Signal Analyzer Panels
- Enabling Preview Features

Configuring Your System for Remote RF Signal Analyzer Panels

You must configure a gRPC server before remotely connecting to RF Signal Analyzer hardware using InstrumentStudio. The server must be set up on the computer that is connected to the hardware. Complete the following steps to enable remote connections to RF Signal Analyzer hardware.

1. Go to the NI gRPC device repo and follow the instructions there to install the NI gRPC server on the computer connected to the hardware.

2. Navigate to the install location, and open the server_config.json file in an editor.
3. Add the following parameter and value to your configuration file: "code_readiness": "RestrictedRelease"
4. Remove the line "address" : "[::1]". This allows any address to connect to the device.
5. Update the port number. This will require users to specify a port number when connecting from InstrumentStudio.
6. Run the server. Follow instructions in the NI gRPC device repo for specific instructions on running the server.

Your configuration file should now look like this:

```
{
  "port": 31763,
  "security" : {
     "server_cert": "",
     "server_key": "",
     "root_cert": ""
  },
  "code_readiness": "RestrictedRelease"
}
```

Your system is now configured to allow clients to create remote Signal Analyzer panels in InstrumentStudio.

Exporting a Device Configuration

Once you have configured a device in InstrumentStudio, you can export the device's settings to a device configuration file, which can then be imported to LabVIEW or another application to apply the device configuration to a different session.

Note

- You cannot import a device configuration file into InstrumentStudio. If you would like to save a layout or device configuration for future use in InstrumentStudio, you can either save the entire project (**File** » **Save all**)

> or an individual device's soft front panel (**File** » **Save [device name].sfp**).
>
> - The following waveform modes are not supported when exporting a configuration that includes a waveform generator:
>   - List and Sweep mode
>   - Abritrary waveform mode with a .CSV waveform type
>   - User-defined waveform mode

To export a device configuration, complete the following steps:

1. Configure the device's settings.
2. Open the instrument header menu and select **Export** » **Driver configuration...**
3. Select the location to save the configuration file.

The device configuration file is saved to the location you specified.

To apply device settings from a device configuration file to a device session in LabVIEW, you must first import the device configuration file to LabVIEW using a driver-specific LabVIEW VI called Import Attribute Configuration. Each driver has its own implementation of Import Attribute Configuration. Refer to the driver's documentation for more information:

Device Driver	Import Configuration VI
NI-SCOPE	NI-SCOPE Import Attribute Configuration File
NI-DCPower	NI-DCPower Import Attribute Configuration File
NI-DMM	NI-DMM Import Attribute Configuration File
NI-FGEN	NI-FGEN Import Attribute Configuration File

Related concepts:

- Configuring Panels with the Instrument Header Menu

Exporting Configurations to TestStand

Exporting configurations saves time when automating test sequences in TestStand. Once you configure your hardware in InstrumentStudio, you can export your configuration for use in TestStand sequences. You can export device and panel configurations to TestStand 2019 or later. You can then use these configurations with LabVIEW, C#, C, or CVI automation code in a TestStand sequence.

Refer to the following topics for more information on exporting device and panel configurations:

- Exporting an InstrumentStudio Configuration
- Inserting an InstrumentStudio Configuration in a TestStand Sequence
- Editing an Exported Configuration

Exporting an InstrumentStudio Configuration

Before exporting an InstrumentStudio configuration to TestStand, make sure instrument channels you want to include in the export are not set to idle. Idle instrument channels are not exported to TestStand.

1. From the top toolbar or the instrument header menu, select the Export to TestStand button ( ).

   Note You must have TestStand 2019 or later installed for the Export to TestStand button to appear.

   Note You cannot export an InstrumentStudio configuration to TestStand while the site filter is set to "All sites". Select a specific site in the site filter, or select "System pins" to only export system pins.

2. Specify a name for the exported configuration. You can configure the default TestStand export name format by selecting **File** » **Preferences** and navigating to the Export to TestStand section.
   InstrumentStudio creates an IO Configuration file, which contains both the driver configuration and the panel layout from InstrumentStudio.

Inserting an InstrumentStudio Configuration in a TestStand Sequence

When you insert an IO Configuration file into a TestStand sequence, TestStand creates a copy of the configuration file in the step you specify. When you run the sequence, the step creates sessions for the InstrumentStudio-configured instruments and applies the configuration to the new sessions.

These created sessions are automatically stored in local variables in TestStand. You can then pass the session from the variable to automation code in Action steps, apply a new configuration to the existing session using the variable, or use the session in a Sweep loop to configure the sequence of values for certain attribute in each instrument.

> **Note** For more information on using TestStand, refer to TestStand documentation on ni.com.

Editing an Exported Configuration

You can edit an exported configuration in a specific TestStand step. Complete the following steps to edit an exported configuration from TestStand:

1. Import an InstrumentStudio configuration to TestStand.
2. Navigate to the TestStand step you would like to edit and select **Edit**. InstrumentStudio opens in Edit mode and displays the instrument panels in the same layout as the document you exported the configuration from.
3. Edit the configuration in InstrumentStudio.
4. Click **OK**. Any changes you made to the configuration commit back to the TestStand step or IO Configuration file you edited.

Auto Range Modes

When specifying a range for a voltage or current measurement, you can select **Auto** or **Auto once** to measure a signal when you do not know which range to select. The digital multimeter (DMM) takes a series of measurements and adjusts the range until the measurement falls within the smallest range appropriate for that measurement.

- **Auto**—Takes a reading before each measurement to select the smallest appropriate range.
- **Auto once**—The first point measured determines the range for the rest of the measurements.

> **Note** For model-specific considerations when using Auto Range, refer to Configuring Auto Range in NI-DMM documentation.

Performing Cable Compensation

Typically, you connect the digital multimeter (DMM) to a device under test with switches, fixtures, or cables. These switches and fixtures can introduce undesired errors into the measurement. Open and short cable compensation minimizes these measurement errors when taking capacitance or inductance measurements.

Compensation consists of measuring the error and applying the measured error to the actual measurement to correct and minimize the errors introduced by the test system. The compensation values must be renewed before taking a measurement at a specified function and range. Any change in range or function defaults the compensation type to none, and you must perform compensation again for accurate measurements.

Performing Open Compensation for Capacitance and Inductance Measurements

1. Disconnect the device under test (DUT) from the DMM at the connection point on the DUT. Leave the cable connected to the DMM.
2. Configure the DMM for capacitance or inductance at the desired range.
3. Open the settings (···) for the DMM channel.
4. Click **Renew** under the Open cable compensation section of the settings window.

   > **Note** These settings are not available for simulated DMM devices.

5. Click **Continue** in the Open Cable Compensation dialog box.

> **Note** Check the device panel to make sure compensation has been applied ().

Performing Short Compensation for Capacitance and Inductance Measurements

1. Disconnect the DUT from the DMM at the connection point on the DUT. Leave the cable connected to the DMM.
2. Configure the DMM for capacitance or inductance at the desired range.
3. Set up a short condition using a low-impedance connection between the HI and LO terminals of the DMM. Cables and switches with low capacitance and low path resistance are recommended.
4. Open the settings (···) for the DMM channel.
5. Click **Renew** under the Short cable compensation section of the settings window.

> **Note** These settings are not available for simulated DMM devices.

6. Click **Continue** in the Short Cable Compensation dialog box.

> **Note** Check the device panel to make sure compensation has been applied ().

Calibrating an LCR Device

Self-calibrate an LCR device to ensure accurate measurements.

The accuracy of a device will naturally drift over time. Self-calibration ensures more accurate measurements by compensating for board level temperature variations and device degradation, as well as ensuring that the device operates within ranges defined in your test specifications.

1. In the LCR device instrument pane, open the instrument header menu ().
2. Click **Device Slot** » **Calibration** .

3. Click **Self-calibrate in MAX** to proceed to self-calibration in MAX.

Refer to When to Calibrate an LCR Device for information on how often to calibrate your LCR device.

- Performing Compensation on an LCR Device

When to Calibrate an LCR Device

Performing regular LCR device calibration ensures better accuracy in your test measurements.

Performing new LCR device self-calibration is recommended if any of the following occur.

- You are installing the module in a chassis for the first time.
- You install, uninstall, or move another module in the same chassis as your LCR device.
- The ambient temperature in your test environment has changed significantly.
- The onboard temperature of the LCR device has drifted significantly (+/- 5° C) since the last self-calibration.
- 24 hours have elapsed since the previous self-calibration.

Performing Compensation on an LCR Device

Typically, you connect the LCR device to a device under test with switches, fixtures, or cables. These introduce undesired errors into the measurement. Prevent measurement errors from switches, fixtures, or cables by applying compensation before taking measurements.

Perform the following steps to apply compensation to your LCR device.

> **Note** For detailed driver documentation on performing LCR device compensation, refer to Compensation of LCR Measurements with NI-DCPower.

1. Calibrate the LCR device.

2. Determine the type of cable compensation you want to perform. Click the option link for detailed instructions on performing the selected compensation.

Option	Recommended When
Standard cable compensation	You are using a standard NI cable that is represented with a Cable Length profile in the Instrument Settings dialog (…) and your test setup is simple—it excludes complex circuitry, like switches, that could affect LCR measurements.
Custom cable compensation	You are using non-NI cabling and your test setup includes complex circuitry like switches that could affect LCR measurements.

> **Note** NI recommends applying cable compensation for low-level LCR measurements where the effects of cabling may be significant relative to the quantity you are measuring.

> **Note** NI recommends first applying short LCR compensation when using custom non-NI cabling to ensure accurate measurements.

3. Determine the type of LCR compensation you want to perform using the following criteria. Click the option link for detailed instructions on performing the selected compensation.

Option	Recommended When
Open LCR compensation	Stray admittance in your test setup influences LCR measurements. **Note** Such stray admittance is present in most test setups.
Short LCR compensation	The impedance of your DUT is low ($<100\Omega$) and the residual impedance of cabling and fixtures may be a significant percentage of your DUT impedance.
Load LCR compensation	The reference load value is similar to your DUT and your test setup includes complex circuitry like switches that can affect LCR measurements.

> **Note** Open and short LCR compensations are prerequisites to performing load LCR compensation.

NI recommends performing cable and LCR device compensation after making changes to your test system set up or performing new LCR device calibration. Refer to When to Generate New Data for Custom Cable Compensation or LCR Compensation for more information on when to perform new compensations.

Applying Standard Cable Compensation

1. Click **Instrument Settings** (···) in your LCR device instrument panel.
2. Select your cable length from the **Cable type** drop-down.

Performing Custom Cable Compensation

Disconnect the device under test (DUT) from the LCR device at the connection point on the DUT. Leave the custom cables connected to the LCR device.

> **Note** The connection point may be a switch, fixture, or any other circuit used to connect LCR device cabling to the DUT. When performing custom cable compensation, only the custom cables should be connected to the LCR device.

1. Click **Custom Compensation** (⇆) in the instrument panel.
2. Click **Renew**.
3. Click **Compensate**.
4. Click **OK**.

Verify that your device is configured for custom cables by opening **Instrument Settings** (···) and selecting "Custom " from the **Cable type** dropdown.

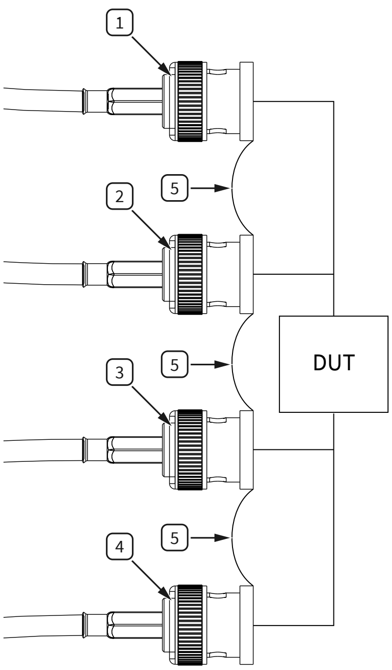
Verify compensation is applied. Click **Custom Compensation** (⇆) and check that **Apply compensation** is enabled.

Note For detailed driver documentation on performing LCR device compensation, refer to Compensation of LCR Measurements with NI-DCPower.

Performing LCR Load Compensation

Connect a reference load to the fixture that will contain your DUT.

Figure 1. Load Configuration Diagram. Refer to the diagram below when setting up a load configuration for LCR load compensation



1. HI CUR
2. HI POT
3. LO POT
4. LO CUR
5. Connection between outer conductors

1. Click **Load Compensation** () in the instrument panel to open the compensation dialog.
2. Click **Renew**.

3. **Optional:** Enter spots to compensate for.
4. Click **Compensate**.
5. Click **OK**.

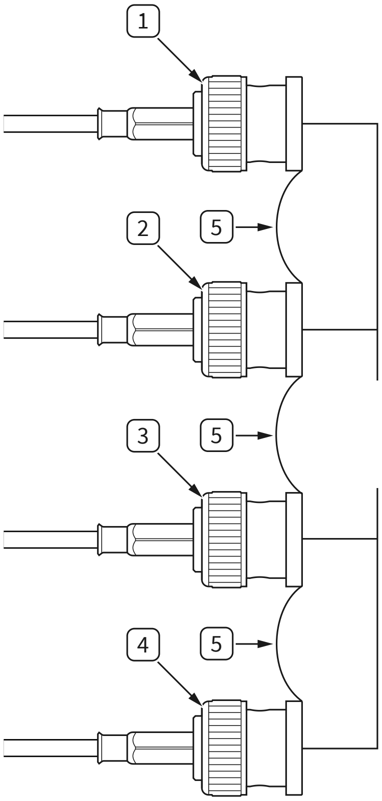
To verify compensation is applied, click **Load Compensation** () and check that **Apply compensation** is enabled.

> **Note** For detailed driver documentation on performing LCR device compensation, refer to Compensation of LCR Measurements with NI-DCPower.

Performing Open LCR Compensation

Disconnect the device under test (DUT) from the LCR instrument at the connection point on the DUT. Leave the cabling connected to the LCR device, and any other test system components such as switches and DUT fixtures.

Figure 2. Open Configuration Diagram. Refer to the diagram below when setting up an open configuration for LCR open compensation



1. HI CUR
2. HI POT
3. LO POT
4. LO CUR
5. Connection between outer conductors

1. Click **Open Compensation** ( ) in the instrument panel to open the compensation dialog.
2. Click **Renew**.
3. **Optional:** Enter additional frequencies to compensate for.
4. Click **Compensate**.
5. Click **OK**.

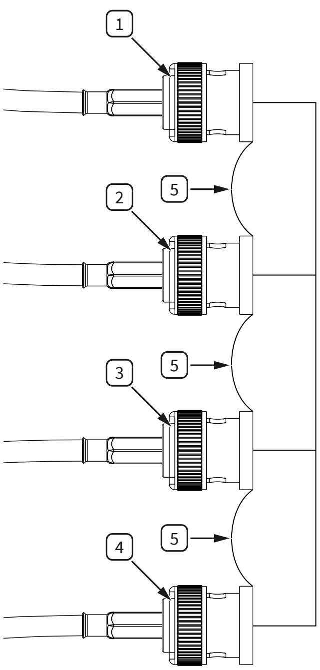
To verify compensation is applied, click **Open Compensation** ( ) and check that **Apply compensation** is enabled.

> **Note** For detailed driver documentation on performing LCR device compensation, refer to Compensation of LCR Measurements with NI-DCPower.

Performing Short LCR Compensation

Disconnect the device under test (DUT) from the LCR device at the connection point on the DUT. Leave cabling, switches, and DUT fixtures connected to the LCR device.

Figure 3. Short Configuration Diagram. Refer to the diagram below when setting up a short configuration LCR short compensation



1. HI CUR
2. HI POT
3. LO POT
4. LO CUR
5. Connection between outer conductors

1. Click **Short Compensation** () in the instrument panel to open the compensation dialog.
2. Click **Renew**.
3. **Optional:** Enter additional frequencies to compensate for.
4. Click **Compensate**.

> **Note** You must perform short LCR compensation when using custom non-NI cabling. Complete custom cable compensation before performing short LCR compensation.

5. Click **OK**.

To verify compensation is applied, click **Short Compensation** () and check that **Apply compensation** is enabled.

> **Note** For detailed driver documentation on performing LCR device compensation, refer to Compensation of LCR Measurements with NI-DCPower.

When to Generate New Data for Custom Cable Compensation or LCR Compensation

Performing new custom cable and LCR device compensation ensures better accuracy in test measurements after making changes to your test system setup.

Generate new custom cable compensation data or LCR compensation data if any of the following occur.

- You perform a new external calibration
- You perform a new self-calibration
- You add complex circuit elements, such as a switch, between the end of the cable and the fixture containing your DUT
- You change the length of the cabling in your test setup
- You change the physical orientation of the cabling in your test setup

- You begin testing a DUT that is different (on the basis of part number) from the DUT for which you originally generated compensation data;

> **Note** For load LCR compensation, choose a new reference load appropriate for the new DUT.

Measurement Types

The following measurement types are available when creating oscilloscope channels with InstrumentStudio.

Amplitude	Mid Ref Volts	Mean	Negative Duty Cycle
Peak-to-Peak	Low Ref Volts	Median	Positive Pulse Width
High	Positive Undershoot	Cycle RMS	Negative Pulse Width
Low	Negative Undershoot	Cycle Mean	Rise Time
Maximum	Positive Overshoot	Period	Fall Time
Minimum	Negative Overshoot	Frequency	Rise Rate
High Ref Volts	RMS	Positive Duty Cycle	Fall Rate
Delta Time	Setup Time	Hold Time	Crosspoint Voltage
Crosspoint Time	FFT Frequency	FFT Amplitude	

Related tasks:

- Customizing Oscilloscope Measurements

Amplitude

The voltage High measurement minus the voltage Low measurement.

Peak-to-Peak

The Maximum measurement minus the Minimum measurement.

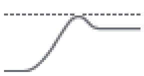
High

The High measurement is calculated using the selected High-Low method. If you select the Histogram High-Low method, the High measurement is calculated using the most common value found in the upper 40% of the waveform. If you select the Peak High-Low method, the High measurement is calculated using the waveform's Maximum measurement value.

Low

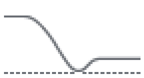
The Low measurement is calculated using the selected High-Low method. If you select the Histogram High-Low method, the Low measurement is calculated using the most common value found in the lower 40% of the waveform. If you select the Peak High-Low method, the Low measurement is calculated using the waveform's Minimum measurement value.

Maximum



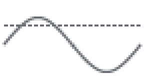
The maximum value found in the waveform.

Minimum



The minimum value found in the waveform.

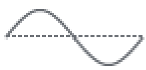
High Ref Volts



The voltage of the signal at the High reference level. If you set the reference level unit to Percentage, the High Ref Volts measurement is calculated with the selected

High-Low method. If you set the reference level unit to Volts, the voltage you set as the High reference level is used for the High Ref Volts measurement.

Mid Ref Volts



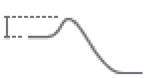
The voltage of the signal at the Mid reference level. If you set the reference level unit to Percentage, the Mid Ref Volts measurement is calculated with the selected High-Low method. If you set the reference level unit to Volts, the voltage you set as the Mid reference level is used for the Mid Ref Volts measurement.

Low Ref Volts

The voltage of the signal at the Low reference level. If you set the reference level unit to Percentage, the Low Ref Volts measurement is calculated with the selected High-Low method. If you set the reference level unit to Volts, the voltage you set as the Low reference level is used for the Low Ref Volts measurement.

Positive Undershoot

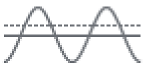
Negative Undershoot



Positive Overshoot

Negative Overshoot

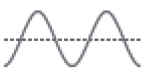
RMS



The true root mean square voltage over the entire waveform. This measurement uses the following formula:

RMS = sqrt[(sum(square(waveform[i]) / number of points)))]

Mean



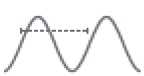
The mean over the entire waveform.

Median

The median over the entire waveform. The points in the waveform are sorted according to the following formulas:

Number of Points	Returned Value
Odd	waveform[(n-1)/2]
Even	(waveform[m/2] + waveform[n/2 +1]) / 2

Cycle RMS

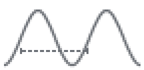


The true root mean square voltage over the first cycle of the waveform. The first cycle is determined using the values specified by the High, Mid, and Low reference levels. This measurement uses the following formula:

Cycle RMS = sqrt[(sum(square(waveform[i]) / number of points)))]

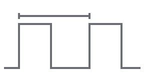
where **waveform** is all the points in the first cycle of the waveform.

Cycle Mean



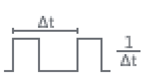
The voltage average over the first cycle of the waveform. The values you specify in the High, Mid, and Low reference level fields in the Settings tab of the Measurements window determine the first cycle.

Period



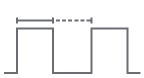
The time of the first two Mid reference level crossings in the same direction. A hysteresis is applied using the values specified by the High or Low reference levels.

Frequency



1.0 divided by the Period measurement, in hertz.

Positive Duty Cycle



The Positive Pulse Width divided by the Period times 100.

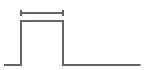
Positive Duty Cycle = (Positive Pulse Width/Period) × 100

Negative Duty Cycle

The Negative Pulse Width divided by the Period times 100.

Negative Duty Cycle = (Negative Pulse Width/Period) × 100

Positive Pulse Width

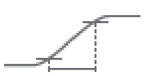


The time difference in seconds between the first two Mid reference level crossings, where the first slope is positive and the second is negative. The High or Low reference levels are used to apply a hysteresis.

Negative Pulse Width

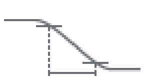
The time difference in seconds between the first two Mid reference level crossings, where the first slope is negative and the second is positive. The High or Low reference levels are used to apply a hysteresis.

Rise Time



The time span of the first rising edge of the waveform to cross the Low reference level until it crosses the High reference level. The time span starts from the Low reference level crossing immediately preceding the High reference level crossing.

Fall Time



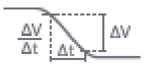
The time span between when the first falling edge of the waveform crosses the High reference level to when the same falling edge crosses the Low reference level.

Rise Rate

The Rise Time divided by the High reference level minus the Low reference level.

Rise Rate = Rise Time / (High reference level - Low reference level)

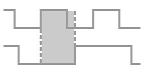
Fall Rate



The Fall Time divided by the High reference level minus the Low reference level.

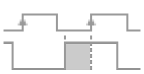
Fall Rate = Fall Time / (High reference level - Low reference level)

Delta Time



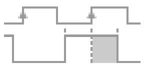
The time span from when the first edge of the waveform crosses the Mid reference level until the second edge of the waveform crosses the Mid reference level. The second edge can be configured to another channel.

Setup Time



The time span from when the waveform crosses the Mid reference level until the configured clock channel crosses the Mid reference level. The Setup Time measurement uses the crossing of the clock channel that is closest to the middle of the graph.

Hold Time



The time span from when the configured clock channel crosses the Mid reference level until the waveform crosses the Mid reference level. The Hold Time measurement uses the crossing of the clock channel that is closest to the middle of the graph.

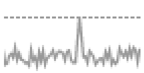
Crosspoint Voltage

The voltage at which two waveforms intersect. The second waveform can be configured to another channel. The crosspoint voltage measurement uses the intersection closest to the middle of the graph.

Crosspoint Time

The time at which two waveforms intersect. The second waveform can be configured to another channel. The crosspoint time measurement uses the intersection closest to the middle of the graph.

FFT Amplitude



Calculates a real FFT and returns the maximum amplitude. The search ignores the DC bin of the FFT; peaks close to DC (but not in the DC bin) are detected.

FFT Frequency

Calculates a real FFT and returns the frequency that corresponds to the maximum amplitude. The search ignores the DC bin of the FFT; peaks close to DC (but not in the DC bin) are detected.

Customizing Oscilloscope Measurements

You can use reference levels and gating to customize the measurements displayed in the measurement table. To customize these settings:

1. Click **Add/Remove** in the measurement table of an oscilloscope.
2. Click the Settings tab of the Measurements window to configure oscilloscope measurements.

Related reference:

- Measurement Types

Gating

Use gating options to select the region of captured data used to perform measurements displayed in the measurement table.

- **Screen**—Performs measurements only on the data visible in the large panel's Time domain graph.
- **Record**—Performs measurements on the full data record.
- **Custom**— Performs measurements only on the data between set positions. Configure custom gating positions using time values relative to the triggered data capture starting point.
  - **Position 1**—Sets the first position of gated data relative to the trigger to calculate measurements.
  - **Position 2**—Sets the second position of gated data relative to the trigger to calculate measurements.

Reference Levels

Customize the oscilloscope measurements displayed on the measurement table by adjusting the reference levels and the High-Low method from the Settings tab of the **Measurements** window.

High, Mid, and Low Reference Levels

Reference levels are range values that divide a waveform into high, middle, and low sections by intersecting the waveform at specified amplitude. Use reference levels to ascertain rise and fall times, calculate waveform cycles, and customize some oscilloscope measurement types. In InstrumentStudio, set reference levels by either absolute voltage levels or the percentage of the waveform to acquire. The following oscilloscope measurement types are calculated using reference levels:

▪ High Ref Volts ▪ Mid Ref Volts ▪ Low Ref Volts ▪ Cycle RMS	▪ Cycle Mean ▪ Period ▪ Rise Time	▪ Fall Time ▪ Rise Rate ▪ Fall Rate

Reference Level Unit

Use the reference level Unit settings to specify whether reference levels should be interpreted as percentages of the waveform or as absolute levels.

- **Percentage**—Interprets the reference levels as a percentage using the selected High-Low method.
- **Volts**—Interprets the reference levels as absolute levels, in units of volts.

High-Low Method

If you set the reference level Unit setting to **Percentage**, configure how the state levels are computed by selecting the High-Low method:

- **Auto-select**—Determines whether the histogram bins that correspond to the high and low state levels each have over 5% of the total hits. If so, the device returns those results. If not, the device uses the Peak High-Low method. The Auto-select High-Low method provides an answer for either a square wave (by ignoring the overshoot and undershoot) or a triangle wave (where the Histogram High-Low method fails).

- **Histogram**—Computes state levels using the histogram bins with the maximum number of hits in the upper and lower regions of the waveform. The upper and lower regions of the waveform include the upper and lower 40% of the peak-to-peak range of the waveform.
- **Peak**—Searches the waveform for its maximum and minimum levels.

Oscilloscope Setup Actions

You can configure the setup of an oscilloscope using two options: Auto setup and Default setup.

- **Default setup**—Returns the device to the default configuration.
- **Auto setup**—Enables all channels and automatically configures device settings.

Settings Changed by Auto Setup

Setting an oscilloscope to Auto setup changes the following settings:

General Settings	Value
Acquisition Mode	Normal
Reference Clock	Internal

Vertical Settings	Value
Vertical Coupling	Unchanged by Auto Setup.
Vertical Bandwidth	Full
Vertical Range	Changed by Auto Setup.[1]
Vertical Offset	0 V
Probe Attenuation	Unchanged by Auto Setup.
Input Impedance	Unchanged by Auto Setup.

Horizontal Settings	Value
Sample Rate	Changed by Auto Setup.[1]
Min Record Length	Changed by Auto Setup.[1]
Enforce Realtime	True
Number of Records	1

Triggering Settings	Value
Trigger Type	Edge if signal present, otherwise immediate.
Trigger Channel	Lowest numbered channel with a signal present.
Trigger Slope	Positive
Trigger Coupling	DC
Reference Position	50%
Trigger Level	50% of signal on trigger channel.
Trigger Delay	0
Trigger Holdoff	0
Trigger Output	None

[1] Auto Setup adjusts this setting to different values depending on the signal.

Setting Sample Rate and Record Length Manually

By default, InstrumentStudio optimizes either the record length or sample rate of an oscilloscope when taking measurements. When the thumbnail preview is enabled with **Data Display** set to **Live**, InstrumentStudio limits the maximum record length to 1 million samples. When set to **On demand**, InstrumentStudio increases the maximum record length to 10 million samples. To configure larger records, use **Manual Override** mode. Enable **Manual Override** mode to manually configure both **Sample rate** and **Record length** settings for an oscilloscope.

> **Note** NI recommends that you only enable **Manual Override** for test parameters that require specific sample rate and record length values. Otherwise, NI recommends leaving this setting off for the best interactive experience with an oscilloscope.

Complete the following steps to enable **Manual Override** mode.

1. Click **Instrument Settings** (···) in your oscilloscope device instrument panel.
2. Click the **Manual Override** mode toggle.
   The toggle should be in the **On** position after clicking.
3. Enter the desired values for **Sample Rate** and **Record Length**.

> **Note** Enabling **Manual Override** mode may lead to unexpected graph behavior if **Sample rate** and **Record length** are improperly configured. For example, if your device does not have enough onboard memory to accommodate measurement settings, your graphs may not render properly, or you may get an error message in InstrumentStudio. Other settings may either cause data to be unreadable, or take longer than expected to appear. For example, if you enter a **Sample rate** of 1 kS/s with a **Record length** of 3.5 MS, it would take around an hour before the measurement data appeared in your oscilloscope panel in InstrumentStudio.

Acquisition Status

You can determine the current state of an acquisition by looking at the acquisition status icon, located in the Horizontal & Acq. header of an oscilloscope panel.

- **Triggered**—The device met the trigger condition and performed an acquisition based on the specified trigger settings.
- **Auto**—After a certain amount of time has passed without a trigger firing, the device automatically triggers an acquisition. The Auto status displays briefly when this occurs.

  > **Note** The trigger mode must be set to **Auto** to enable automatic triggering.

- **Waiting**—The device is waiting to take an acquisition until the trigger condition has been met.
- **Stopped**—The instrument is not running.

Interpolation Method

You can set the interpolation method of an oscilloscope acquisition to one of the following settings from the Instrument Settings window:

- **No interpolation**—The oscilloscope does not interpolate data.
- **Auto**—The oscilloscope interpolates data when the visible time per division is too small for acquisitions performed at the maximum sample rate. If you set the sample rate of the oscilloscope to **Manual**, the oscilloscope treats the sample rate you enter as the maximum sample rate. The Auto interpolation method takes headroom into account when determining whether or not to perform interpolation; if there is low headroom between the oscilloscope's bandwidth and its sample rate, the oscilloscope does not interpolate the data.
- **On**—The oscilloscope interpolates data as often as possible without taking headroom into account. If you set the interpolation method to On, the oscilloscope will interpolate data in all cases except for the following:
  - The oscilloscope's sampling method is set to Random Interleaved Sampling (RIS).
  - The acquisition uses the peak detect sample mode.
  - The waveform has enough data and does not need interpolation.

Sampling Methods

Depending on the device, you can select different sampling methods from the Instrument Settings window of an oscilloscope:

- **Real time**—Gathers all the samples for a waveform in one acquisition with one trigger event.
- **RIS**—Random Interleaved Sampling (RIS) achieves a higher synthetic sample rate by combining several triggered waveforms.

> **Note** RIS is not supported on all oscilloscopes. You can select RIS only with supported devices. Refer to your device's documentation to determine whether or not it supports RIS.

Sample Modes

You can select one of the following sample modes from the Instrument Settings window when configuring an oscilloscope acquisition with InstrumentStudio:

- **Sample**—The oscilloscope measures the signal at a fixed time interval (the sample rate).
- **Peak detect**—The oscilloscope internally samples the signal at its maximum sample rate, then saves the maximum and minimum value of the input signal within the configured sample interval. Use Peak detect when you would like to acquire measurements for a longer time while still being able to detect transients.
- **Averaging**—The oscilloscope averages the waveform data of continuous acquisitions based on the averaging count. The oscilloscope performs an exponential moving average and shows the number of averages, which shows how many waveforms have been read and averaged.

Adding FFT Channels and Markers

Use FFT channels with an oscilloscope to measure amplitude and frequency of a sample. You can also fetch more detailed measurements and search for peaks using markers.

Creating FFT Channels

1. From a large oscilloscope panel, click the **FFT** button in the **Add Channels** section. A frequency graph displays.
2. Select a **Source** for the FFT channel. You can select any single existing channel.
3. Select a **Window** function for the FFT channel.
4. (Optional) Configure additional settings for the FFT channel by clicking the FFT channel's settings cog. Refer to Searching For and Computing Peaks for more information.

Creating Markers

A marker fetches the amplitude (y-location) of a particular sample at a specified frequency (x-location) of a particular sample. Complete the following steps to create a marker or markers:

1. After creating an FFT channel, select the **Markers** drop-down in the frequency graph header. Select **Markers: On** to enable markers and open the marker toolbar.
2. From the marker toolbar, select a marker to configure. By default, only Marker 0 is enabled.

> **Note** Enable more markers by selecting a marker from the marker toolbar and changing the marker mode from **Off** to **Normal**. You can enable up to 12 markers.

3. Select the FFT channel the selected marker should measure.
4. Enter the frequency where you want to place the marker. The amplitude of the signal at the frequency of the marker is displayed in the **Level** section of the marker toolbar.

> **Note** You can view and edit the frequency of all enabled markers in the Markers table below the frequency graph. The Markers table also displays the markers' current level, mode, and source channel.

Searching For and Computing Peaks

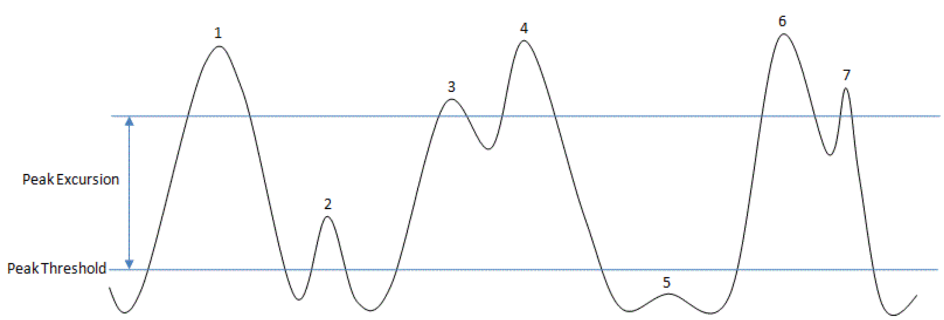
After enabling at least one FFT channel and markers, you can use the peak search functions to locate peaks. Peaks are the samples for which the amplitude rises and falls around a threshold. You can move the selected marker to the highest peak, the next highest peak, or an adjacent peak to the left or right of the marker. You can configure the peak threshold and excursion settings by selecting the cog icon next to the peak search functions.

Computing Peaks Using Peak Threshold and Excursion

The peak threshold is the minimum amplitude level a sample must rise above to be considered a peak. If peak threshold is enabled but peak excursion is not, every amplitude measurement above the threshold is considered a peak. Set the peak threshold from the peak search settings.

Peak excursion specifies the minimum amplitude variation required in a signal to be considered as a peak. Peak excursion is always specified with respect to a threshold value. A signal must rise and fall above the threshold level by at least the peak excursion value to be considered as an eligible peak.

The following figure shows how to identify a valid peak that meets excursion criteria:



Peak 1 rises and falls by at least the peak excursion value above the threshold. Hence, it follows the excursion criteria.

Peak 2 is above the threshold level, but it does not rise and fall by the excursion value above the threshold level. Hence, it does not satisfy excursion criteria.

Peak 3 rises above the threshold value by at least the excursion value, but it does not completely fall by the excursion value. The signal rises and crosses the Peak 3 amplitude level. Hence, it is not an eligible peak.

Peak 4 rises and falls by at least the peak excursion value above the threshold. Although during the rise it slightly falls (Peak 3) the net rise from the threshold level exceeds the excursion value. Hence, it is considered an eligible peak.

Peak 5 is below the threshold level. Hence, it is not detected as a peak.

Peak 6 rises by at least the peak excursion value above the threshold. During the fall, the signal slightly rises (as Peak 7), but it does not rise above peak 6 amplitude level before falling again. The total fall, which starts at the peak 6 amplitude level, is more than the excursion value. Hence, it is an eligible peak.

Peak 7 falls by the excursion value, but it does not rise by the excursion value. Hence, it does not satisfy excursion criteria.

FFT Averaging Modes

You can perform averaged measurements on an FFT channel to improve measurement accuracy or to help compensate for a low signal-to-noise ratio.

- **Disabled**—Disables FFT averaging.
- **RMS**—Root mean square. RMS averaging averages the energy (or power) of the signal, so it reduces signal fluctuations, but not the noise floor.
- **Peakhold**—Performs a peak hold averaging measurement at each individual frequency line and retains the RMS peak levels of the averaged quantities from one FFT record to the next. Peak-hold averaging is the most useful when configuring a measurement system or when applying limit or upper limit testing to a frequency spectrum.

Measuring Spectral Density

A power spectral density is the measure of a signal's power content versus frequency. You can view the power spectral density of a signal while analysing data in the frequency domain; this is helpful for determining which frequency ranges have strong or weak variations in power.

Complete the following steps to measure spectral density in InstrumentStudio:

1. Add an oscilloscope to the large panel.
2. Create an FFT channel by selecting the **FFT** button in the **Add Channels** section of the large panel.
   A frequency chart opens underneath the large panel time chart.
3. Configure the FFT Axis settings:
   1. In the header of the frequency chart, select the Chart Options button.
   2. In the Y-axis section, change Units to **V/√(Hz)** or **dBm/Hz.**

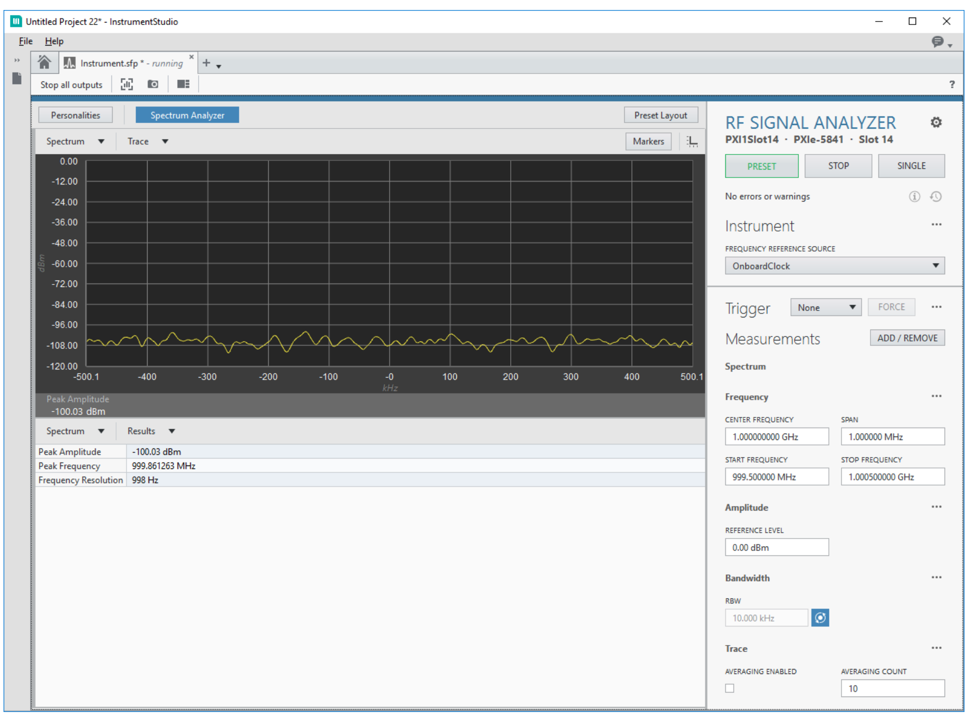
3. In the X-axis section, set Scaling to **Logarithmic**.
4. Run the panel if it is not already running.
   InstrumentStudio plots power spectral density data on the Frequency chart.

For better frequency resolution, you can configure averaging through the FFT channel settings, or you can modify the RBW through the axis settings in Chart Options.

Analyzing RF Signals with an RF Signal Analyzer

Create an RF signal analyzer panel to monitor and configure RF signal analyzer settings in InstrumentStudio.

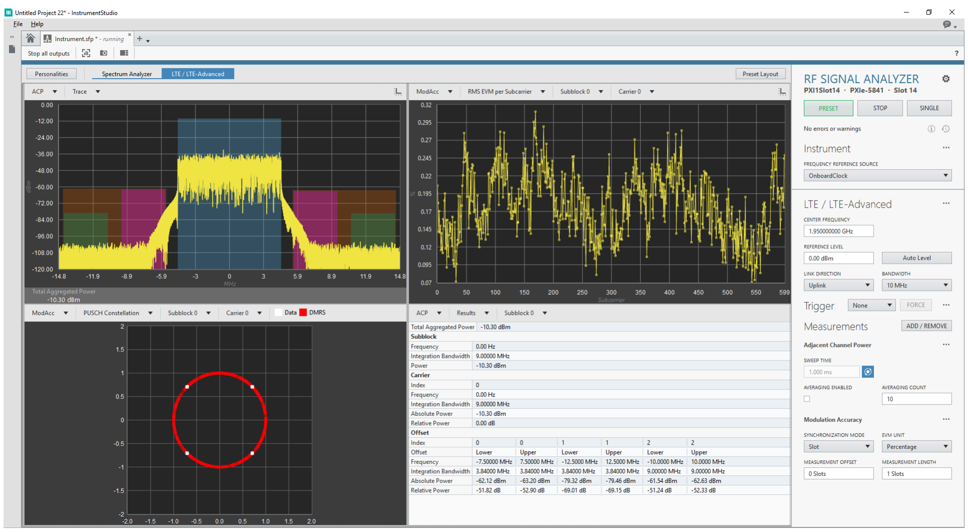
1. Create an RF Signal Analzyer panel.
   The default Spectrum Analyzer view will appear in the instrument panel.



2. Select the personality you want to use to measure from the RFmx Personalities dropdown.

> **Note** If the desired personality does not appear in the dropdown, make sure the personality is supported in InstrumentStudio, and make sure the driver for that personality is installed.

The instrument panel and highlighted settings in the instrument configuration panel will change based on selected personality. Shown below is the view for the LTE personality.



3. Select the desired measurements from the settings panel using the **Add/ Remove** button.
4. Configure center frequency, reference level, and any other important values for the signal being analyzed in the settings panel.
5. View the results and traces in the measurement pane.

Supported RFmx Personalities

The following list contains the RFmx personalities supported in InstrumentStudio for use with RF Signal Analyzer panels.

- Spectrum Analyzer
- GSM
- LTE / LTE-Advanced

- NR
- TD-SCDMA
- WCDMA
- Bluetooth
- WLAN

De-Embedding RFSA Devices

De-embedding refers to the process of removing the effects of test fixture cabling and components. Applying de-embedding to an RF signal ensures more accurate results when reading data from your RF device. Once applied, the de-embedded signal you see at the device port matches the requested signal settings.

De-embedding

To access de-embedding properties, open the RF device **Instrument Settings** dialog box (···), click the **RF** tab, and scroll down to De-embedding. The following are options you can configure to apply de-embedding your RF device:

- **Type**—Specifies the type of de-embedding you want to use.
  - None—De-embedding is disabled.
  - Scalar—De-embeds the signal using the gain term.
  - Vector—De-embeds the signal using the gain term and reflection term.
- **Status**—Displays whether an S2P table is loaded into the device session.
- **Load table from S2P file**—Loads an S2P file into the device session to use for de-embedding.

> **Note**
>
> - S2P tables are used to incorporate S-parameters into signal de-embedding. S-parameters characterize the effects of a linear network on a signal when it passes from one port to another. Refer to the installed RF driver documentation for more information.

- You must configure de-embedding for each port when using multi-port modules.

Performing RF Signal Analyzer Self-Calibration

Self-calibrate an RF signal analyzer device to ensure accurate measurements.

The accuracy of a device will naturally drift over time. Self-calibration ensures more accurate measurements by compensating for board level temperature variations and device degradation, as well as ensuring that the device operates within ranges defined in your test specifications.

Performing Self-Calibration

1. In the device panel, open the instrument header menu ( ).
2. Click **Device Slot** » **Calibration** » **Self Calibrate...** to begin self-calibration.

> **Note** InstrumentStudio will warn you that the process may take several minutes to finish, and will lock the instrument until the process completes.

3. Click **Continue** to proceed.

Performing Self-Calibration Range

This mode of calibration completes more quickly than general self-calibration.

> **Note** This calibration is only valid until you restart your system.

1. In the device panel, open the instrument header menu ( ).
2. Click **Device Slot** » **Calibration** » **Self Calibrate Range...**.
3. Under **Steps to omit** in the dialog box, select any steps you want InstrumentStudio to skip during the self-calibration process.
4. Under **Ranges**, enter the values for the desired ranges of self-calibration.
5. Click **Continue** to proceed, or **Cancel** to cancel the operation.

Checking Calibration History

1. In the device panel, open the instrument header menu (⚙).
2. Click **Device Slot** » **Calibration** » **History**.

Generating RF Signals with an RF Signal Generator

Create an RF signal generator panel to monitor and configure RF signal generator settings in InstrumentStudio.

1. Add an RF signal generator device to your panel.
2. Select a mode of operation within the device panel.
   - CW—Configures the RF signal generator to generate a continuous waveform signal.
   - Arb—Configures the RF signal generator to generate an arbitrary waveform using the settings specified in an external file and any additional settings you specify.
3. Specify the type of signal to generate.
   - If you want to generate a CW signal, specify the signal settings by opening the **Instrument Settings** (···) dialog box.
   - If you want to generate an arbitrary waveform, click the **Add Waveform** button next to the **Select Waveform** drop-down listbox. Then select a waveform file. Specify the signal settings by opening the **Instrument Settings** (···) dialog box.

   > **Note** Arbitrary waveform files must be in the .tdms format.

4. Select **Run** and set the output to **On**. The signal generates with the settings you specify.

   > **Note**

- The panel will already be running in CW mode, and you only need to set output to **On** to output a generated waveform. In Arb mode, you must first select a waveform, then press **Run**.
- The panel will continue running and generate the waveform continuously unless you set **Waveform repetition mode** is to Finite when configuring your signal settings. When set to Finite, the panel will output the specified number of repetitions and then stop.

Arbitrary Waveform Properties

Waveform properties are settings and values associated with waveforms generated in Arbitrary Waveform mode.
When changing a waveform, waveform property values update to reflect the settings for that waveform. When downloading a waveform from file, the property values are automatically populated with the values found in the file. Any changes made to the waveform properties only affect the selected waveform.

Debugging Test Programs

Waveform properties were introduced in the NI-RFSG 20.7 driver. Prior to that release, only a subset of these properties was available, and they applied globally to all waveforms. When debugging a test program that still utilizes these global properties, the global values will be shown for the selected waveform. Any subsequent edits in InstrumentStudio will only apply to the selected waveform.

The following table shows legacy property names that map to new RFSG property names:

Global Property Name	RFSG Arbitrary Waveform Property Name
IQ Rate	IQ Rate
Peak Power Adjustment	PAPR
Pre-filter Gain	Run time Scaling
Signal Bandwidth	Signal Bandwidth

Performing RF Signal Generator Self-Calibration

Self-calibrate an RF signal generator device to ensure accurate measurements.

The accuracy of a device will naturally drift over time. Self-calibration ensures more accurate measurements by compensating for board level temperature variations and device degradation, as well as ensuring that the device operates within ranges defined in your test specifications.

Performing Self-Calibration

1. In the device panel, open the instrument header menu (⚙).
2. Click **Device Slot** » **Calibration** » **Self Calibrate...** to begin self-calibration.

   > **Note** InstrumentStudio will warn you that the process may take several minutes to finish, and will lock the instrument until the process completes.

3. Click **Continue** to proceed.

Performing Self-Calibration Range

This mode of calibration completes more quickly than general self-calibration.

> **Note** This calibration is only valid until you restart your system.

1. In the device panel, open the instrument header menu (⚙).
2. Click **Device Slot** » **Calibration** » **Self Calibrate Range...**.
3. Under **Steps to omit** in the dialog box, select any steps you want InstrumentStudio to skip during the self-calibration process.
4. Under **Ranges**, enter the values for the desired ranges of self-calibration.
5. Click **Continue** to proceed, or **Cancel** to cancel the operation.

Checking Calibration History

1. In the device panel, open the instrument header menu (⚙).

2. Click **Device Slot** » **Calibration** » **History**.

RF Signal Generator Status

An RF signal generator can be in one of the following states:

- **Generating Output**—Signal generation is running and being output on this channel. Generating can also mean the channel is waiting for a trigger.
- **Not Generating Output**—Signal generation is stopped on this channel.
- **Unknown**—Signal generation status is unknown.

> **Note** The panel must be running and the output set to On for the RF signal generator to output the waveform. Output stops when closing an RF signal generator panel except when monitoring an external session.

De-embedding RFSG Devices

De-embedding refers to the process of removing the effects of test fixture cabling and components. Applying de-embedding to an RF signal ensures more accurate results when reading data from your RF device. Once applied, the de-embedded signal you see at the device port matches the requested signal settings.

De-embedding

To access de-embedding properties, open the RF device **Instrument Settings** dialog box (···), click the **RF** tab, and scroll down to De-embedding. The following are options you can configure to apply de-embedding your RF device:

- **Type**—Specifies the type of de-embedding you want to use.
  - None—De-embedding is disabled.
  - Scalar—De-embeds the signal using the gain term.
  - Vector—De-embeds the signal using the gain term and reflection term.
- **Status**—Displays whether an S2P table is loaded into the device session.
- **Load table from S2P file**—Loads an S2P file into the device session to use for de-embedding.

Note

- S2P tables are used to incorporate S-parameters into signal de-embedding. S-parameters characterize the effects of a linear network on a signal when it passes from one port to another. Refer to the installed RF driver documentation for more information.
- You must configure de-embedding for each port when using multi-port modules.

RF Signal Generator Modes

You can generate an RF signal using one of the following three modes:

- Continuous Waveform (CW)—Configures an RF signal generator to output a continuous waveform at the specified frequency and level.
- Arbitrary Waveform (Arb)—Configures an RF signal generator to output a complex signal according to the waveform and waveform settings specified in an external file.
- Script—Configures an RF signal generator to output a sequence of multiple waveforms according to settings specified in an external script. This option is only available in debug mode, and is only supported when controlling an external session configured to script mode. For more information on scripting, refer to NI-RFSG driver documentation at ni.com.

SMU/VPS Modes of Operation

Device Operating Modes

You can use one of the following modes for acquiring measurements using an SMU or VPS device in InstrumentStudio: Chart, Waveform, and Sweep.

Operating Mode	Sourcing Behavior	Large Panel Plotting Behavior	Pause Behavior	Potential Uses	Notes
Chart	Allows one or more channels to	Appends samples to the chart as	Channels continue to source at the last	Observe signals over an	

Operating Mode	Sourcing Behavior	Large Panel Plotting Behavior	Pause Behavior	Potential Uses	Notes
	source current or voltage.	they are measured.	configured level when you pause measurements.	infinite duration.	
Waveform	Allows one or more channels to source current or voltage.	Plots samples to the graph all at once; the length of the graph is determined by capture time.	Channels continue to source at the last configured level when you pause measurements.	Observe signals over a shorter, finite duration, similar to an oscilloscope.	Unless you press **Single**, InstrumentStudio automatically configures the device for a faster sample rate and runs the output continuously in a loop.
Sweep	Sweeps through configured current or voltage steps for a single channel (Single-Channel Sweep) or two channels (Two-Channel Sweep).	Plots sample step results on the graph as they are measured.	Both measurement and output channels are paused when you pause measurements.	Allows you to plot the relationship between current and voltage in an I-V curve.	When you are using Two-Channel Sweep, the sweep acts as a nested loop, where the inner channel executes all the steps for each step configured in the outer channel.
Sweep (LCR Mode)	Sweeps through configured frequency or DC bias steps for a single channel (Single Channel Sweep)	Plots sample step results on the graph as they are measured.	Both measurement and output channels are paused when you pause measurements.	Allows you to plot the relationship between frequency or DC bias and enabled LCR readbacks.	

Note **Waveform** and **Sweep** modes are not available on PXI-4110, PXIe-4112, PXIe-4113, PXI-4130, or PXI-4132 devices.

Note **Sweep** mode is available on PXIe-4190 device when operating in LCR mode. To change to LCR mode, go to the Channel settings within the device panel, click the **Mode** drop-down, and select **LCR meter**.

Channel Operating Modes

In addition to device operating modes, you can also specify an operating mode for each channel. Some channel operating modes are only available on devices with certain capabilities.

If you change the operating mode of a channel, all panel measurements are paused. Any enabled outputs continue to generate signal at the last configured level. The new operating mode of the channel will not take effect until you run the panel.

- Idle—The channel is not programmed. If you change a channel to Idle from another mode, the channel continues to generate signal according to its last configuration.
- Voltage or Current—Sources DC voltage or DC current at a single configured level while the output is enabled.
- Pulse voltage or Pulse current—Generates a single pulse at the pulse level, then returns to the bias level.

  Note

  - For more information on pulsing and devices that support pulse measurements, refer to Pulsing in the NI-DCPower help.
  - InstrumentStudio currently supports single point pulsing only; InstrumentStudio 2019 does not support pulse sequencing.

- Voltage sequence or Current sequence—Allows you to configure a level and a duration for each step in a list. You can configure the list of steps on the device panel, or import steps from a .CSV file. For more information on creating voltage or current sequences, refer to Configuring Sequences.

> **Note**
>
> - If you import a step sequence from a .CSV file, the .CSV file must have two columns: the first for the level, and the second for the duration.
> - Voltage sequence and Current sequence modes are not available on the PXI-4110, PXIe-4112, PXIe-4113, PXI-4130, or PXI-4132 devices.

SMU Measurement Autorange

When the range for a signal is unknown, using measurement autorange enables the signal range to be automatically determined.
When a measured signal exceeds the specified range in InstrumentStudio, InstrumentStudio will not be able to accurately measure that signal, and will warn you that the measurement is over range. When you enable measurement autorange, InstrumentStudio uses incoming signal data to automatically set measurement ranges, ensuring more accurate measurements.

Enabling Autorange

1. To enable autorange mode for an SMU or VPS device, click the **Measurement Autorange** toggle within the device panel and ensure it is in the **On** position.
2. **Optional:** Use the device panel to configure autorange settings.
3. **Optional:** To disable autorange, click the **Measurement Autorange** toggle again and ensure it is in the **Off** position.

Configuring Autorange

Use the following controls in the SMU or VPS device panel to specify how you want autorange to respond to incoming signals from your device.

> **Note** Autorange mode must be enabled to access these settings in your device panel.

- **Aperture time mode** — Specifies whether the aperture time used for measurement autorange is determined automatically or customized using the value specified by the **Minimum aperture time** setting.
- **Minimum aperture time** — Specifies the aperture time used by the measurement autorange algorithm.

  > **Note** This is a minimum value, and will be scaled up to optimize the algorithm for different ranges. For smaller ranges, the value will be scaled up to account for signal noise.

- **Minimum aperture time units** — Specifies the units of the **Minimum aperture time** setting. Aperture time can be specified in seconds or power line cycles (PLCs).
- **Autorange behavior** — Specifies the algorithm the hardware uses for measurement autoranging. Choose from the following behaviors.
  - **Range up to limit then down** — go to range limit the range down until measured value is within thresholds.
  - **Range up** — go up one range when upper threshold is reached.
  - **Range up and down** — go up or down one range when the upper or lower threshold is reached.
- **Minimum current range** — Specifies the lowest range used during measurement autoranging.

  > **Note** Limiting the lowest range can improve the speed of autoranging and minimize thrashing between ranges when measuring a noisy signal.

NI-DCPower Independent Channel Sessions

InstrumentStudio creates a single multi-channel session for all devices in a single SMU device by default.

> **Note** This is not default behavior in some versions of InstrumentStudio.

Considerations when configuring sessions with SMU devices

Using independent channels allows you to configure multiple channels of the same instrument, or multiple instruments, within the same session independently of one another. InstrumentStudio exports a single session configuration for all channels by default. In certain cases, you must change your session type to Single channel (compatibility) mode.

> **Note** To manually select the session type you want to use, open the instrument header menu (⚙) of the device in your panel, and click **Session Type**.

You should select **Session Type** » **Single channel (compatibility)** when:

- Exporting a session to platforms which do not support NI-DCPower independent channel sessions. These include Python, LabWindows/CVI, LabVIEW NXG, and Linux desktop.
- Exporting a session to NI-DCPower version 20.5 and earlier.
- Exporting the session to end-user code which uses deprecated NI-DCPower Initialize Functions in C/LabVIEW or deprecated constructors in .NET.
-

> **Note** Exports from sessions created in InstrumentStudio with independent channels cannot be imported into sessions created with single channels. To verify your session channel configuration, go to the **instrument header** menu (⚙) and click **Session Type**.

Output Status

You can determine the current output status of non-idle channels with the output status icon, located at the top of the **Channels** section of an SMU/Power Supply panel.

Icon	Status	Description
	Generating Output	One or more channels in the panel are sourcing voltage or current.
	Not Generating Output	No channels in the panel are sourcing voltage or current.
	Waiting for Trigger	Channels in the panel are waiting until the trigger condition has been met to start an operation.
	Unknown Status	The output status of the channel(s) in the panel is unknown. Run the panel to refresh the output status.

Table 1. SMU/Power Supply Channel Output Status

Note Output continues when closing a panel.

Configuring SourceAdapt Parameters to Measure Transients

You can use SourceAdapt technology to customize the source measure unit (SMU) response to any load. For more information on National Instruments' SourceAdapt technology, refer to NI SourceAdapt Next-Generation SMU Technology on ni.com.

Note NI SourceAdapt technology is available only on the following SMUs: PXIe-4135, PXIe-4137, and PXIe-4139.

Selecting a SourceAdapt Preset

1. Place a SourceAdapt-compatible SMU in the large panel.
2. Place the SMU panel into Waveform mode.

3. Open the Channel Settings window.
4. Scroll down to the SourceAdapt tuning parameters section of the Channel Settings window.
5. Select a SourceAdapt preset from the list:
   - **Slow**—Slows the control loop for a more stable response.
   - **Normal**—The default SourceAdapt preset, a balance between speed and stability.
   - **Fast**—Speeds up the control loop for a faster response.
6. (Optional) Make custom adjustments to the response using the fields in the SourceAdapt tuning parameters section.

Configuring Sequences

You can change the channel operating mode of an SMU/Power Supply channel to Voltage Sequence or Current Sequence mode to enable power sequencing. Power sequencing allows you to output the desired power-up sequence for a DUT with multiple power sources by configuring the order, timing, and level of each power supply channel.

> **Note**
>
> - Channels within the same panel operating in Voltage Sequence, Current Sequence, Pulse Voltage, or Pulse Current mode are automatically hardware-synchronized. Refer to Multi-Device Synchronization for more information on hardware and software synchronization within panels.
> - Sequence channel operating modes are supported only on power supplies that support advanced sequencing. For more information on advanced sequencing and supported devices, refer to NI-DCPower Advanced Sequencing documentation.

1. Place an SMU or power supply in a panel.

If you place the device in a large panel, the graph shows a preview of the output sequence the channel will generate.

2. Set the operating mode of the panel to **Waveform** or **Charting**.
   Placing the panel in **Waveform** mode enables fast sampling and allows you to visualize the sequence in more detail.
3. Set the channel operating mode to **Voltage sequence** or **Current sequence**.
   A step table appears within the channel.
4. Edit the steps in the step table by entering the values manually, importing data from a file, or pasting data from the clipboard.
5. (Optional) Tune the output of the sequence using SourceAdapt parameters, found in the Channel Settings window.
6. Enable the channel output and run the panel to begin the sequence.

Generating Waveforms with a Waveform Generator

Use a waveform generator to monitor and configure waveforms in InstrumentStudio.

1. Add a waveform generator device to the large panel.
2. Select a waveform mode from the drop-down menu in the upper-right corner of the large panel.
   - Standard waveform—Generates a waveform using the channel settings you specify.
   - Arbitrary waveform—Generates a waveform using settings specified in a waveform file and any additional channel settings you specify.
3. Specify the type of waveform to generate.
   - If you are generating a standard waveform, select the waveform type using the **Waveform** drop-down menu.

   > **Note** If you select a **User-defined** waveform, you must load a waveform file. User-defined waveform files must be in CSV format, with the exact number of samples listed in your device's

> specifications document. For example, the PXIe-5433 requires 8192 samples for a user-defined waveform. The waveform samples may be in a single row or single column.

- If you are generating an arbitrary waveform, load a waveform file using the **Filename** field. The waveform file specifies the type of waveform to generate.

> **Note** Arbitrary waveform files must be in CSV format or binary format:
>
> - CSV—For single-channel instruments, waveform samples must be in a single row or a single column. For two-channel instruments, each channel's waveform samples must be in a separate column.
> - Binary—For single-channel instruments, waveform samples may be 8-byte double-prevision or 2-byte half-precision floating point, and the byte order may be little-endian or big-endian. For two-channel instruments, both channels' samples must be interleaved.

4. Configure waveform channel settings using either the controls on the large panel or the large panel's Channel Settings window. Customizable settings change depending on the waveform mode and type you select.

   > **Note** If you are generating an arbitrary waveform, you must select a triggering mode for the waveform using the **Mode** setting. For more information on arbitrary waveform trigger modes, refer to Advanced Waveform Sequencing and Triggering on Arbitrary Waveform Generators at ni.com.

5. Select **Run**.
   The waveform generates with the settings you specified.

Custom Waveform Channels and Triggers

Use the following settings to configure channels, triggers, and other parameters of the waveform generator.

Waveform Channel Settings

- Output Impedance—Sets the output impedance value to either 50 Ω or 75 Ω.
- Load Impedance—Sets the load impedance value to one of the following settings:
  - **Match output**—Matches the load impedance to the output impedance.
  - **High-Z**—Sets the load impedance to high impedance.
  - **Custom**—Sets the load impedance to a custom value, in ohms.
- Digital Filter—Enables or disables the digital filter. The digital filter increases the effective sampling rate by providing points that interpolate between generated samples.
- Analog Filter—Enables or disables the analog filter. The analog filter is applied after the interpolated signal, and, when combined with the digital filter, can remove high-frequency images from the frequency domain.
- Terminal config.—Returns the terminal configuration for the waveform generator. Most waveform generators support only one option (differential or single-ended).

Waveform General Settings

- Waveform—Sets the type of waveform to generate if you are generating a standard waveform.

> **Note**
>
> - If you select **User-defined**, you must load a waveform file. User-defined waveform file must be in CSV format.
> - Different waveform types have different configurable settings.

- Sample Rate—Sets the sample rate of an arbitrary waveform.

Waveform Trigger Settings

- Trigger Mode—Sets the mode for triggering arbitrary waveform generation.

> **Note** For more information on arbitrary waveform trigger modes, refer to Advanced Waveform Sequencing and Triggering on Arbitrary Waveform Generators at ni.com.

- Trigger Type—Sets the trigger type to one of the following values:
  - **Immediate**—Triggers waveform generation as soon as you run the waveform generator.
  - **Software**—Triggers waveform generation according to settings in an external application.
  - **Digital edge**—Triggers waveform generation on the rising edge of a specified source terminal. The source terminal must export a signal to use digital edge triggering.

Hardware Event Output Terminals

These settings determine to which chassis terminal(s) hardware events are exported.

Waveform Modes

You can generate a waveform using one of the following two modes:

- **Standard Waveform**—Outputs waveforms according to the channel settings you specify in InstrumentStudio or an external application. Using a standard waveform allows you to generate several different standard waveform types (sine, square, etc.) at precise frequencies.

> **Note** The user-defined standard waveform type allows you to generate a periodic waveform with a finite, specified number of

> points using Direct Digital Synthesis (DDS). To generate a user-defined waveform of any size, use Arbitrary Waveform mode.

You can also change the output waveform frequency of a standard waveform during generation with a short response time. You can also use the Sweep operating mode to configure a range of frequences to generate over a specified number of steps and time duration, or use the List operating mode to configure a sequence of frequencies to generate for specified durations.

- **Arbitrary Waveform**—Outputs waveforms according to the settings specified in an external file. Using an arbitrary waveform allows you to define large, complex waveforms using a waveform settings file. While you can generate more complex waveforms using an arbitrary waveform, changes to arbitrary waveform settings have a longer response time than standard waveforms.

Waveform Generator Status

A waveform generator channel can be in one of the following states:

- **Stopped**—Waveform generation is stopped on this channel.
- **Running**—Waveform generation is running on this channel.
- **Waiting for trigger**—Waveform generation is waiting until the trigger condition has been met.

> **Note** Generation stops when closing a panel.

Capturing Data

Using InstrumentStudio, you can capture data from an entire layout or a single panel. You can also take a screenshot without capturing data.

Adjust data capture preferences by selecting **File** » **Preferences** and navigating to the Capture data tab. In this tab, you can set the destination directory and change the filename and format for both images and data. By default, images are captured in PNG and data is captured in the NI-TDMS format, but you can also capture data in CSV format.

Capturing Data from a Layout

Capture data from an entire layout by selecting the **Capture data** icon ( ) from the document toolbar, located just above the large panel. InstrumentStudio saves a timestamped screenshot and a data file to the destination directory. The screenshot contains the view of the entire layout at the time data was captured, and the data file contains detailed information about every device in the layout.

Capturing Data from a Panel

Capture data from a single panel by opening the instrument header menu ( ) and selecting **Capture data**. InstrumentStudio saves a timestamped screenshot and a data file to the destination directory. The screenshot contains the view of the panel at the time data was captured, and the data file contains detailed information about the device or devices in the panel.

Capturing a Separate Screenshot

If you want to capture a screenshot but not a data file, select the **Capture screenshot** icon ( ) from the document toolbar, located just above the large panel. InstrumentStudio attaches the screenshot to the clipboard.

Parsing Captured Data

Parse NI-TDMS data files using LabVIEW, LabWindows/CVI, DIAdem, VeriStand, or one of several third-party applications. For more information on using the NI-TDMS file format, visit ni.com/info and enter the Info Code tdmsfileformat.

Parse CSV files using a spreadsheet application for best results. CSV files sort channel, device, and measurement type into columns. Individual measurements and statistics—including mean, range, and standard deviation—are sorted into rows.

Related concepts:

- Configuring Panels with the Instrument Header Menu

Debugging Programmatic Applications

Use InstrumentStudio's debug mode to monitor and control devices in use by an external application.

You can use InstrumentStudio to troubleshoot devices currently in use by an external application. An external application is any application running outside of InstrumentStudio, including LabVIEW, TestStand, or a C/C++/.NET application.

If you receive unexpected results from a device in an external application, you can monitor the device in InstrumentStudio to assess the issue, troubleshoot the issue by changing the device's settings in InstrumentStudio, then continue with the external application. For example, you can pause the external application where you suspect an error, edit the device settings in InstrumentStudio, then resume the application with the new settings.

> **Note**
>
> - Debugging features are not supported when using sweep mode in an SMU (Source Measure Unit) panel.
> - When monitoring a device in debug mode, InstrumentStudio establishes a connection to the device session in the external application, not the device itself. Therefore, InstrumentStudio will only monitor data that is fetched or read from the currently running external application.

Waveform Generator Debug Mode Limitations

When using debug mode with a waveform generator, be aware of the following limitations:

- Debug mode does not support arbitrary sequence or script modes. Controlling abritrary sequence or script mode sessions is not allowed.
- List mode control is disabled when InstrumentStudio is in monitor or controller mode.
- When InstrumentStudio is in controller mode, you cannot switch to list or sweep mode from another operation mode.

- When InstrumentStudio is in monitor mode, you cannot take control of a panel with a mix of standard and arbitrary waveform channels.

RF Signal Generator Debug Mode Limitations

When using debug mode with an RF signal generator, be aware of the following limitations:

- You cannot take control of an RF signal generator panel if streaming is enabled, or if there is an active configuration list.
- You cannot return to script mode after clicking the **Preset** button.
- Waveform-specific controls are read-only in Script mode.

Related concepts:

- Configuring Panels with the Instrument Header Menu

Entering Debug Mode

You can enter debug mode from the Home screen by selecting **Debug**. Using this method, InstrumentStudio searches for open device driver sessions and then creates a layout based on the devices it discovers. The devices in the newly created layout automatically enter debug mode.

You can also enter debug mode while an InstrumentStudio project is already open. A device in use by an external session automatically enters debug mode if debugging is enabled and the device is placed in a layout or is already part of an active layout. For example, if you are running an oscilloscope from InstrumentStudio and you run a LabVIEW application that acquires measurements from that oscilloscope, the InstrumentStudio panel containing the oscilloscope enters debug mode until the device's external session (in this case, the LabVIEW application) closes.

Devices currently in use by an external application display the external session icon ( ) in the Edit Layout window.

> **Note** Debugging is enabled by default. If a device fails to enter debug mode, confirm that debugging is enabled by opening the Configure Debug Session window from the instrument header menu and checking the

Debug Enabled box for the device you want to debug. If you change settings in the Configure Debug Session window while an external session is already open, you must restart the external session before changes will take effect.

Related tasks:

- Enabling Debugging with a C, C++, or .NET Application
- Disabling Debugging

Enabling Debugging with a C, C++, or .NET Application

Complete the following steps to debug a C, C++, or .NET application using InstrumentStudio.

> **Note** Supporting communication with a device in C, C++, or .NET environments at breakpoints requires additional overhead that may affect performance.

1. Select **Configure debug settings** from the instrument header menu.
2. Select the **Using Breakpoints in C/C++/.NET Applications** checkbox next to the device you want to debug. Click **OK**.
   InstrumentStudio enables breakpoints in C, C++, and .NET external applications.
3. Debug the application and/or device.
4. When you finish debugging, deselect the **Using Breakpoints in C/C++/.NET Applications** checkbox in the Configure Device Settings window and restart the external application to apply the changes.

Refer to Monitoring and Controlling Devices in External Sessions for more information on using debug mode.

Related concepts:

- Entering Debug Mode

- Configuring Panels with the Instrument Header Menu

Monitoring and Controlling Devices Used in an External Session

You must enter debug mode to access the **Monitor** and **Control** buttons. Refer to Entering Debug Mode for more information.

Complete the following steps to monitor and/or control a device in InstrumentStudio when an external application is using the device:

1. Select **Monitor**.
   InstrumentStudio obtains read-only access to the external session, and the device's controls are disabled. You are now monitoring the device.
2. Select **Control**.
   InstrumentStudio takes control of the device if an external application is not currently accessing the device. If you select **Control** and you are not monitoring any external application sessions or the external application is paused, InstrumentStudio takes ownership of the device.

   > **Note** Selecting **Control** takes control of the device only until the next time the external application calls the device, at which point control reverts to the external application and InstrumentStudio returns to monitoring the session. You can use breakpoints to pause the external application, allowing InstrumentStudio to retain control of the device until you have made your changes and you are ready to continue the application.

3. Edit the device settings.
4. Click **Monitor** to return to monitoring the session and revert control of the device to the external session.
   The edited device settings are applied to the external session.

Live Measurement View

The following topic contains information about monitoring live device data in InstrumentStudio.

When monitoring a device in debug mode, certain devices will instantly display measurements of channel data, even if external applications are not fetching device data. InstrumentStudio currently supports live data monitoring for the following devices:

▪ PXIe-4135	▪ PXIe-4139	▪ PXIe-4143
▪ PXIe-4136	▪ PXIe-4140	▪ PXIe-4144
▪ PXIe-4137	▪ PXIe-4141	▪ PXIe-4145
▪ PXIe-4138	▪ PXIe-4142	▪ PXIe-4147

Note For devices not listed here, test code must call measure or fetch for measurements to be updated when in monitor mode.

Disabling Debugging

Complete the following steps to disable debugging for a device:

1. Open the Configure Debug Session window by selecting **Configure debug session** from the instrument header menu.
2. Deselect the **Debug Enabled** checkbox to disable debugging for a specific device.
3. Click **OK**.
4. Restart the external application to apply the changes.

Debugging is disabled for the specified device. This device cannot enter debug mode until debugging is enabled again.

Related concepts:

- Entering Debug Mode

InstrumentStudio Plugins

You can use InstrumentStudio to host plugins written in LabVIEW or C#. These plugins run alongside the InstrumentStudio panels used to configure PXI instruments. Plugins have a runtime configuration and an edit-time configuration. When exporting a plugin configuration to TestStand and using it in a step, the runtime configuration is provided to the sequence as a variable. Both of these configurations persist when the project containing them is saved.

Support for Plugins

The following limitations apply when hosting plugins in InstrumentStudio:

- Hosting a plugin created in C# requires Visual Studio 2019 or newer and .NET 6.0.
- Plugins built in older versions of InstrumentStudio (before version 22Q3) are no longer compatible and must be re-built.
- Plugins cannot be created from the **File** » **New** menu.
- You cannot use MeasurementStudio controls when hosting plugins created in C#.
- When hosting a plugin in InstrumentStudio, you are limited to the following interactions:
  - Saving and loading
  - Resizing the window
  - Pulling out tabs
  - Exporting the session to TestStand
  - Stopping all outputs with LabVIEW plugins

Pin Maps

Pin maps define test instruments and hardware, sites and pins connected to a DUT, and how the test instruments and hardware connect to DUT pins for each test site. InstrumentStudio preserves mappings created in TestStand Semiconductor Module (TSM) pin maps.

Refer to the other topics in this section of the manual for specific information on how to use and modify pin maps in InstrumentStudio.

Setting an Active Project Pin Map

Set an active pin map to filter your view of connected sites, pins, and devices within your test system in InstrumentStudio.

1. Open your InstrumentStudio project.
2. Decide how you want to add a pin map to your project.

Option	Description
Create a new pin map	Use the Pin Map Editor to create a new pin map for your project.
Set a default pin map	Automatically add a selected pin map to new InstrumentStudio projects and set it as the active pin map.
Add a pin map file to your project files	Select **File** » **New** » **Pin Map** to add a pin map file (.pinmap) to your project.

> **Note** InstrumentStudio automatically sets the first pin map file you add as the active project pin map. The name of the active project pin map appears in bold in the project files pane.

> **Note** Only one pin map can be active within a project.

3. Confirm that the pin map is valid. An error icon will appear next to the pin map file in the project files pane if InstrumentStudio detects a problem with it. Refer to Pin Map Errors and Warnings for more information.
4. Activate the pin map. Right-click the pin map file in the project files pane and select **Make active**.
5. Apply site filtering to view site pins in your panels.
6. Apply pin filtering to view selected pins within a site in your panels.

To deactivate the pin map, right-click the active pin map in the project files pane and select **Make inactive**.

> **Note** When you launch InstrumentStudio within TestStand Semiconductor Module (TSM), InstrumentStudio automatically uses the pin map configured in TSM.

Selecting a Default Pin Map

Choose a default pin map to add the selected pin map to all new projects created in InstrumentStudio.

1. Go to **File** » **Preferences** » **Pin and site mapping**.
2. Click the **browse** button to open the **Default pin map file** dialog and select a pin map file.
3. To remove the default pin map preference, return to the Preferences menu, go to the Pin and site mapping section, and click the **Clear** button to remove the default pin map.

InstrumentStudio will add the default pin map to project files when creating a new project, and automatically use the default pin map as the active project pin map.

Creating a New Pin Map

Use the Pin Map Editor to create new pin maps in InstrumentStudio.

1. Select **File** » **New** » **Pin Map** or click **Add** in the project files pane toolbar to create a new pin map file (.pinmap).
2. Use the Pin Map Editor to configure your pin mappings. The Pin Map tab displays mappable items. Use the left pane to browse through and configure the mappings you want to include in your pin map. To manually edit the pin map file text, click the **XML** tab.

   > **Note** If you are working in an unsaved project, InstrumentStudio will prompt you to save the project before creating a new pin map.

3. Click **OK** to close the editor and return to your project. The new pin map is now the active project pin map.

Adding Pin Map Files to a Project

Add existing pin map files to your project to use pin maps you create outside of InstrumentStudio.

1. Select **File** » **New** » **Pin Map**.
2. Select a pin map file (.pinmap).
3. Click **OK** to add the file to your project.
4. **Optional:** To open the Pin Map Editor and update the pin map, double-click the file in the project files pane.

InstrumentStudio will automatically make the first pin map file added to a project the active pin map for that project. To activate another pin map, right click it in the project files pane and select **Make active**.

Editing a Pin Map

Use the Pin Map Editor to make changes to a project pin map.

1. Double-click a pin map (.pinmap) in the project files pane to open the Pin Map Editor.
2. Make the desired changes to your instruments, connections, and pins. Use the Pin map tab to view mappable items and change mappings. To manually change the pin map file, click the **XML** tab and to edit the pin map.

When you have completed your edits, click **OK** to close the Pin Map Editor.

Using the Site Filter

After setting an active pin map in InstrumentStudio, devices in your panel appear as pins grouped by sites, system pins, and unmapped channels. Use the site filter in the document toolbar to display pins from a selected site.

1. In your instrument panel, click the **site filter** drop-down in the toolbar at the top of the document window.
2. Select a site from the list to filter the view in the panel to the selected site.

> **Note**

- You must select **All sites** in the site filter to view unmapped channels. Site filtering does not apply to RF signal generators because each device channel can be mapped to multiple pins on multiple sites. You must select sites and pins in the RF signal generator panel settings (···).

Note You cannot export an InstrumentStudio configuration to TestStand while the site filter is set to "All sites". Select a specific site in the site filter, or select "System pins" to only export system pins.

Related tasks:

- Applying a Pin Filter
- Exporting an InstrumentStudio Configuration
- Applying a Pin Filter

Applying a Pin Filter

Use a pin filter to select site and system pins to view in an instrument panel.

Before applying pin filtering, you must select an active project pin map.

Pin maps often contain many pins. Pin filtering enables you to view selected pins to improve navigating within a panel.

1. Click the pin filter button (☰) within your device or instrument panel.
2. Enable the pins you want to view in your panel.
3. Click **OK**.

Your panels will now display configuration settings for pins selected in the pin filter, and measurement panels will display measurements for those pins. Information about hidden and non-idle pins appears underneath the Pins section of the panel.

Note Non-idle pins are pins that are not visible within the filtered view, but are still active.

> **Note** The pin filter state is saved in the instrument panel, and will be restored when the panel is re-opened.

Related tasks:

- Using the Site Filter

Pin Map Errors and Warnings

InstrumentStudio highlights errors and warnings in project pin map files (.pinmap).

Error and Warning Behavior

Common causes of pin map errors and warnings include improperly formatted pin map files, and pin map settings that do not correspond to configurations within your test system.

To determine the cause of the error, double-click the pin map file to open the Pin Map Editor. The dialog will display a message with specific information about the cause of the error or warning. Refer to the following table for information on pin map errors and warnings.

Notification Type	Expected Behavior
Error	Panels will not work using the selected pin map. InstrumentStudio reverts to using the default instrument and channel views. You must address the error before using the pin map file.
Warning	You can use the pin map, but errors may occur at run time. Determine the source of the warning and make the necessary corrections using the Pin Map Editor before run time.

For detailed information on pin map files, refer to Pin Map XML File Structure.

Enabling Preview Features

Preview features are new features in InstrumentStudio that support specific workflows. They are not release-quality features, and may have functionality gaps. They are only supported for specific uses. You must enable preview features in the InstrumentStudio settings menu to use them.

Follow the steps below to enable preview features.

1. Go to **File** » **Preferences** » **Preview Features**.
2. Click the toggle to enable the desired preview features.
3. Close the **Preferences** panel.

Once you have selected the preview features you want to enable, you must restart InstrumentStudio.