InstrumentStudio User Manual

dataTec
Mess- und Prüftechnik. Die Experten.

Ihr Ansprechpartner / Your Partner:

dataTec AG
E-Mail: info@datatec.eu
datatec.eu

Authorized Distributor

2026-07-23

Contents

InstrumentStudio User Manual

The InstrumentStudio User Manual provides detailed descriptions of the product functionality and the step by step processes for use.

Looking for Something Else?

For information not found in the User Manual for your product, such as specifications and API reference, browse *Related Information*.

Related information:

- What is InstrumentStudio?
- Download InstrumentStudio
- Release Notes
- Interactive Activation Guide
- Discussion Forums
- InstrumentStudio Plug-Ins on GitHub
- NI Learning Center

InstrumentStudio Overview

InstrumentStudio provides an integrated approach to interactive PXI measurements. Use InstrumentStudio to unify your display, export instrument configurations to code, and monitor and debug an automated test system. You can use InstrumentStudio for free. You can also purchase an InstrumentStudio Professional license to access additional features.

InstrumentStudio Key Features

InstrumentStudio has the following key features and capabilities:

InstrumentStudio

- View PXI instrument data on unified displays
- Capture multi-instrument screenshots and measurement results
- Save project-level configurations for specific DUTs
- Export instrument configurations to different programming environments
- Debug running test applications
- NI Nigel AI for context-aware assistance, supported application actions, and measurement data exploration.

InstrumentStudio Professional

For more information on the InstrumentStudio Professional license, see *Select Your InstrumentStudio Software Edition* in the *Related Information* section.

- Support for SCPI instruments and SCPI instrument panels
- Create, edit, and configure sequences with the *Sequencer Tool*
- Connect to plug-in libraries
- Configure, control, and monitor measurement plug-ins
- Integration with TestStand through *Measurement Plug-Ins*

Note

> Features that are only available with an InstrumentStudio Professional license are marked with the symbol.

Related concepts:

- Sequencer Tool
- NI Nigel AI in InstrumentStudio

Related information:

- Select Your InstrumentStudio Software Edition
- Measurement Plug-Ins User Manual

InstrumentStudio New Features and Changes

Learn about updates, including new features and behavior changes, introduced in each version of InstrumentStudio.

Discover what is new in the latest releases of InstrumentStudio.

> **Note** If you cannot find new features and changes for your version, it might not include user-facing updates. However, your version might include non-visible changes such as bug fixes and compatibility updates. For information about non-visible changes, refer to your product *Release Notes*.

Related information:

- Release Notes

InstrumentStudio 2026 Q3 Changes

Introducing Nigel AI, Measurement Data Services workflows, RF Signal Analyzer and Network Analyzer enhancements, and DAQ debugging capabilities in InstrumentStudio.

This version of InstrumentStudio adds support for the following features and improvements.

Introducing NI Nigel™ AI

- Added support for retrieving published measurement data by using NI Nigel AI and Measurement Data Services APIs.
- Added NI Nigel AI guidance features for supported workflows.

Measurement Data Services (MDS)

- Added support for capturing measurement data from layouts and panels. The

captured measurement data is published to Measurement Data Services.

RF Signal Analyzer and Network Analyzer Enhancements

- Added support for maximizing RF Signal Analyzer and Network Analyzer displays, making it easier to focus on specific results.
- Added support for embedding fixture information in VNA calsets.
- Added enhancements to demodulation workflows and result visualization.
- Added support for custom symbol maps in the Demod personality.

DAQ Analog Input and DAQ Analog Output Enhancements

- Added support for DAQ Analog Input and DAQ Analog Output debugging workflows.
- Added a strip chart display for DAQ Analog Output panels when using software timing.
- Added support for the Start and Reference trigger mode in DAQ Analog Input panels.
- Added support for independent sensitivity units and expanded sensitivity unit options for supported DAQ Analog Input signal types.

New Hardware Support

This version of InstrumentStudio adds support for the following hardware:

- PXIe-4311

Related concepts:

- NI Nigel AI in InstrumentStudio
- Debugging Measurements from External Applications in InstrumentStudio
- Capturing Data in InstrumentStudio
- Measurement Plug-Ins in InstrumentStudio
- Panels and Layouts in InstrumentStudio
- DAQmx Task Selection in InstrumentStudio Debug Mode

Related reference:

InstrumentStudio 2026 Q2 Changes

Learn about new features, behavior changes, and other updates in InstrumentStudio 2026 Q2.

New Hardware Support

This version of InstrumentStudio adds support for the following hardware and panel type:

- Analog output panels for PXI DAQ

Layouts

Added the following improvements for the **Edit Layout** function for devices with multiple subsystems:

- The **Edit Layout** window displays each device only once.
- You can create multiple counter input panels for the same device.
- You can assign each subsystem of the same device to different panels. For example, you can assign analog input measurements to one panel and analog output measurements to a different panel for the same device.
- Searching for a specific instrument, such as `Analog Input`, returns all devices that support that instrument type.

> **Note** Devices with multiple subsystems are, for example, PXI DAQ devices. These devices have multiple subsystems, because they handle analog inputs and analog outputs.

RF Signal Generator

- Added functionality to configure script triggers.
- Added functionality to configure markers 1-3.

Network Analyzer

- Introduced the automatic port extension functionality.

Related concepts:

- Editing an Existing Layout in InstrumentStudio

Related reference:

- NI-DAQmx Hardware Support in InstrumentStudio
- RF Signal Generator Settings in InstrumentStudio
- Network Analyzer Settings in InstrumentStudio

InstrumentStudio 2026 Q1 Changes

Learn about new features, behavior changes, and other updates in InstrumentStudio 2026 Q1.

New Hardware Support

This version of InstrumentStudio adds support for the following hardware and panel type:

- Analog input panels for PXI DAQ

RF Signal Analyzer

- Added autolevel capability for the Spectrum Analyzer RFmx personality.
- Added support for the Demod RFmx personality.

RF Signal Generator

- When you save an **RF Signal Generator** panel, InstrumentStudio saves the content of the scripts as well.

Network Analyzer

- InstrumentStudio now reports the correction level for each trace individually.

Related reference:

- NI-DAQmx Hardware Support in InstrumentStudio
- RF Signal Analyzer Settings in InstrumentStudio
- RF Signal Generator Settings in InstrumentStudio
- Network Analyzer Settings in InstrumentStudio

2025 Q4

October 2025

Oscilloscopes

- Added support for specifying the measurement units (in volts or amperes) of a connected probe.

RF Signal Generator

- Added support for additional file types for arbitrary waveform generation.
- Added support for editing script content when operating in Script mode.
- The use of Script mode is no longer limited to debugging an external session.

Professional License Only

The following new features are only available with an InstrumentStudio Professional license.

- Sequencer Tool
  - Added support for the Network Analyzer panel type.
  - Added support for parameter sweeps in RF Signal Analyzer steps.
  - Added support for parameter sweeps in RF Signal Generator steps.

Related concepts:

- Sequencer Tool

Related reference:

- RF Signal Generator Settings in InstrumentStudio
- Oscilloscope Settings in InstrumentStudio

Related information:

- Measurement Plug-Ins User Manual

2025 Q3

July 2025

Added support for in-product calibration without Measurement and Automation Explorer (MAX).

Network Analyzer

- Added support for CW time sweep.
- Added group delay trace.

RF Signal Analyzer (RFmx)

- Added support for importing `.s2p` files to facilitate the configuration of the Noise Figure measurement.

SMU/Power Supply

- The PXIe-4051 now supports power and resistance channel operating modes.

Professional License Only

The following new features are only available with an InstrumentStudio Professional license.

- Measurement Plug-Ins
  - Added support for the "1D array of XY data" data type for measurement plug-ins.
- Sequencer Tool
  - Added support for the RF Signal Analyzer panel type.
  - Added support for the RF Signal Generator panel type.
  - Added support for single-channel sweep and two-channel sweep modes to sequences on DC power instrument panels.

Related concepts:

- Sequencer Tool

Related reference:

- Network Analyzer Settings in InstrumentStudio
- RF Signal Analyzer Settings in InstrumentStudio

Related information:

- Measurement Plug-Ins User Manual

Updates and Changes for InstrumentStudio Extended Support Versions

Browse updates and changes made in InstrumentStudio versions on extended support.

> **Note** If you cannot find changes for your version, it might be a more recent version, documented as a new feature. Or, your version might not have included user-facing updates. You can find more information about non-visible changes, such as bug fixes, compatibility updates, and stability adjustments or maintenance adjustments, in the product *Release Notes*, available on ni.com.

Related information:

- Release Notes

2025 Q2

April 2025

- All instrument panels can be upgraded without upgrading the corresponding driver of the instrument panel. InstrumentStudio 2025 Q2 supports the same driver versions that were supported by InstrumentStudio 2025 Q1.

Network Analyzer

- Added support for trace math.
- Added marker enhancements, including target search and continuous marker mode.
- Added auto-detection of orientation for electronic calibration.

Professional License Only

The following new features are only available with an InstrumentStudio Professional license.

- Sequencer Tool
  - Added support for parameter sweeps in instrument steps.
  - Added support for SCPI instruments (third-party instruments) in sequences.

Related concepts:

- Sequencer Tool

2025 Q1

January 2025

Network Analyzer

- Added support for markers.
- Added support for memory traces.

RF Signal Analyzer (RFmx)

- **Spectrum Analyzer**—Added frequency counter measurement.
- **LTE**—Added support for downlink NB-IoT.
- **Bluetooth**—Added new trace and results for HDT.

RF Signal Generator

- Added the ability to change the active script while debugging test code in script mode.
- Added the ability to switch to CW (continuous waveform) or Arb (arbitrary waveform) mode while debugging test code in script mode.

Oscilloscope

- Enhanced monitoring support: vertical scaling updates automatically based on

test program settings.
- Added support for all PXIe-5172 onboard timebase rates.

Professional License Only

The following new features are only available with an InstrumentStudio Professional license.

- Sequencer Tool
  - Added support for the following native instrument types:
    - FGEN
    - SCOPE
    - SMU/VPS

    You can now add these panels to sequences.
  - Added support for parameter sweeps in measurement plug-in steps.
- Measurement Plug-Ins
  - Added support for parameter sweeps for measurement plug-ins.

Updates and Changes for InstrumentStudio Unsupported Versions

Browse updates and changes made in InstrumentStudio versions that are no longer supported.

> **Note** If you cannot find changes for your version, it might be a more recent version, documented as a new feature. Or, your version might not have included user-facing updates. You can find more information about non-visible changes, such as bug fixes, compatibility updates, and stability adjustments or maintenance adjustments, in the product *Release Notes*, available on *ni.com*.

Related information:

- Release Notes

2024 Q4

October 2024

- Added segmented sweeps to the Network Analyzer instrument.
- Improved pulsed S-parameters, data visualizations, and calibration for Network Analyzer instruments.
- Added channel merging capability to the SMU/Power Supply instrument. This capability allows you to combine multiple channels to achieve a higher output current.

Professional License Only

The following new features are only available with an InstrumentStudio Professional license.

- Added support for SCPI-capable oscilloscopes.

    - Fetch data and screenshot captures from SCPI instruments.
    - Fetch configurations from SCPI instruments.
    - Apply configurations to SCPI instruments.

  Refer to *SCPI Instrument Panels* for more information on using SCPI-capable devices with InstrumentStudio.
- Sequencer Tool
    - Added support for digital multimeters in the Sequencer Tool. You can now create steps to read measurements from supported digital multimeters.
    - Added support for wait steps. You can now create wait steps in your sequences.
    - UI improvements to the sequencer tool. You can now view steps associated with measurement panels.
- Measurement Plug-Ins
    - Added support for measurement plug-in versioning. Refer to the *Measurement Plug-In User Manual* for more information measurement plug-in versioning.
    - Added support for displaying a link to measurement plug-in documentation in InstrumentStudio and TestStand

Related concepts:

- SCPI Instrument Panels in InstrumentStudio

Related information:

- Measurement Plug-Ins User Manual

2024 Q3

July 2024

- Pin maps are no longer required to use measurement plug-ins.
- Added several enhancements to the Network Analyzer instrument. For example, the ability to save S-parameter measurement results to Touchstone format (`.snp`) files.
- Added band power markers to the Spectrum measurement in the RF Signal Analyzer instrument.

Professional License Only

The following new features are only available with an InstrumentStudio Professional license.

- Added support for measurement plug-ins. Refer to the *Measurement Plug-Ins* user manual for more information on developing measurements.
- Added the **Sequencer Tool** for sequencing measurements using measurement plug-ins.
- Added support for connecting to plug-in libraries.

Related concepts:

- Sequencer Tool

Related tasks:

- Connecting to a Plug-In Library in InstrumentStudio

Related information:

- Measurement Plug-Ins User Manual

2024 Q2

April 2024

- Added manual calibration, pulse mode, triggering, and improved calset support to the Network Analyzer instrument.
- Reference waveform and de-embedding file paths are now stored with the RF Signal Analyzer configuration, allowing for automatic reload.
- Added several enhancements for Channel Sounding packets using the RF Signal Analyzer Bluetooth personality.

2024 Q1

January 2024

- Added support for Fixed Markers using RF Signal Analyzer instruments.
- Added custom bitfile support for RF Signal Analyzer and RF Signal Generator instruments.

2023 Q4

October 2023

- You can now undo and redo changes made to panel properties.
- Improved searching and filtering of instruments and measurements in the **Edit Layout** window.
- Use the Network Analyzer instrument to control supported vector network analyzers.
- Added support for capturing IQ data using the RF Signal Analyzer instrument.
- Added support for Channel Sounding packets using the RF Signal Analyzer Bluetooth personality.
- Added several enhancements to the RF Signal Analyzer Pulse personality. For example, support for multi-burst analysis and phase and frequency traces.

2023 Q3

July 2023

- Added full support for the following personalities for the remote RF Signal Analyzer panel:
  - SpecAn
  - LTE
  - NR
  - WLAN
  - Bluetooth

  These personalities are no longer preview features.
- Added new remote connections section in the preferences menu.
- Added support for pulse measurements in the RF Signal Analyzer panel.

Related information:

- Remote RF Signal Analyzer Panels in InstrumentStudio

- Managing Remote Connection Preferences in InstrumentStudio

2023 Q2

April 2023

- Added LTE, NR, and WLAN personalities to remote RF Signal Analyzer panel preview feature.

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

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.

2022 Q4

October 2022

- Added oscilloscope **Manual Override** mode. This mode enables the simultaneous manual configuration of record length and sample size in oscilloscope settings.

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.

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.

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.

2021

April 2021

- Added RFSG Signal Generator panel that supports interactive use and debugging with the following instruments:
  - PXIe-5820
  - PXIe-5830
  - PXIe-5831

    - PXIe-5832
    - PXIe-5840
    - PXIe-5841
- Added support for the PXIe-4135 (40W) and the PXIe-4137 (40W) source measure units. Supported in NI-DCPOWER 20.7.

2020

November 2020

- Added support for the PXIe-4139(40W) source measure unit. Supported in NI-DCPower 20.5.
- Added support for multichannel session in the SMU/Power Supply panel. 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. You can also configure multiple channels of multiple instruments independent of one another within the same session.
- Added support for monitoring merged channels in debug mode on the PXIe-4147. Supported in NI-DCPower 20.6. Merging the channels of an SMU allows you to use multiple channels together. Using multiple channels together increases the current that you can source beyond the normal maximum for a single SMU channel.

Related information:

- Configure 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

- You can synchronize multiple NI-SCOPE devices of the same model in the same oscilloscope panel.
- Added support for selecting trigger sources and events from other NI devices in your system.
- Added the following modes to SMU panels:
  - Waveform graphing
  - Pulse current
  - Pulse voltage
  - Voltage sequence
  - Current sequence

  These modes include 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 the SMU/Power Supply panels to perform the following actions:
  - Hide idle channels
  - Show power calculations
  - Condense channel views
- 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. Such pin maps allow you to group devices and channels by site. Such pin maps also allow you to use pin names for channel aliases.

2018 SP2

January 2019

- Added support for the PXIe-5163 oscilloscope. 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 the following modes to Waveform Generator panels:
  - Arbitrary waveform generation
  - Frequency list
  - Sweep
- Added the following elements to Oscilloscope panels:
  - FFT channels
  - Frequency markers
  - FFT channel measurements
  - Additional two-channel measurements

Related information:

- Capturing Data in InstrumentStudio
- Waveform Modes
- Add FFT Channels and Markers

2018 SP1

September 2018

- Added support for the following oscilloscopes:
  - PXIe-5110
  - PXIe-5111
  - PXIe-5113

  Supported in NI-SCOPE 18.6.

2018

May 2018

- This was the initial release of InstrumentStudio.

InstrumentStudio Supported Hardware

Supported hardware and version availability in InstrumentStudio.

The following sections list supported hardware by driver family and the first version of InstrumentStudio that supports each device.

NI-DCPower Hardware Support in InstrumentStudio

SMU and VPS instruments supported through NI-DCPower and corresponding InstrumentStudio version availability.

Table 1. InstrumentStudio Supported SMU and VPS Instruments

Model Name	First Available
PXIe-4051	2023
PXIe-4112	2018
PXIe-4113	2018
PXIe-4135	2018
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

Model Name	First Available
PXIe-4145	2018
PXIe-4147	2019 SP1
PXIe-4150	2024
PXIe-4151	2023
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 Hardware Support in InstrumentStudio

Digital multimeters supported through NI-DMM and corresponding InstrumentStudio version availability.

Table 2. InstrumentStudio Supported Digital Multimeters

Model Name	First Available
PXIe-4080	2018
PXIe-4081	2018
PXIe-4082	2018
PXI-4065	2018
PXI-4070	2018

Model Name	First Available
PXI-4071	2018
PXI-4072	2018
PCIe-4065	2018
PCI-4065	2018
USB-4065	2018
PCI-4070	2018

NI-FGEN Hardware Support in InstrumentStudio

Waveform generators supported through NI-FGEN and corresponding InstrumentStudio version availability.

Table 3. InstrumentStudio Supported Waveform Generators

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

Model Name	First Available
PXI-5422	2018
PXI-5441	2018
PCI-5402	2018
PCI-5406	2018
PCI-5412	2018
PCI-5421	2018

NI-SCOPE Hardware Support in InstrumentStudio

Oscilloscopes supported through NI-SCOPE and corresponding InstrumentStudio version availability.

Table 4. InstrumentStudio Supported Oscilloscopes

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

Model Name	First Available
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
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

Model Name	First Available
PCI-5922	2018
PCIe-5155	2018
USB-5132	2018
USB-5133	2018

NI-DAQmx Hardware Support in InstrumentStudio

DAQ instruments supported through NI-DAQmx and corresponding InstrumentStudio version availability.

Introduced in InstrumentStudio 2026 Q1.

Note Debug mode is not supported for all NI-DAQmx devices. Support varies by device family. For information about Debug mode support and limitations, see *Limitations of Debug Mode in InstrumentStudio*.

Table 5. InstrumentStudio Supported DAQ Instruments

Model Name	First Available
PXI-4461	2026 Q1 (AI), 2026 Q2 (AO)
PXI-4462	2026 Q1 (AI)
PXI-4472	2026 Q1 (AI)
PXI-4495	2026 Q1 (AI)
PXI-4496	2026 Q1 (AI)
PXI-4498	2026 Q1 (AI)
PXI-6115	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6120	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6122	2026 Q1 (AI)
PXI-6123	2026 Q1 (AI)
PXI-6132	2026 Q1 (AI)

Model Name	First Available
PXI-6133	2026 Q1 (AI)
PXI-6143	2026 Q1 (AI)
PXI-6220	2026 Q1 (AI)
PXI-6221	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6224	2026 Q1 (AI)
PXI-6225	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6229	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6230	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6232	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6233	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6236	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6238	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6239	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6250	2026 Q1 (AI)
PXI-6251	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6254	2026 Q1 (AI)
PXI-6255	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6259	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6280	2026 Q1 (AI)
PXI-6281	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6284	2026 Q1 (AI)
PXI-6289	2026 Q1 (AI), 2026 Q2 (AO)
PXI-6704	2026 Q2 (AO)
PXI-6711	2026 Q2 (AO)
PXI-6713	2026 Q2 (AO)
PXI-6722	2026 Q2 (AO)
PXI-6723	2026 Q2 (AO)

Model Name	First Available
PXI-6733	2026 Q2 (AO)
PXIe-4300	2026 Q1 (AI)
PXIe-4302	2026 Q1 (AI)
PXIe-4303	2026 Q1 (AI)
PXIe-4304	2026 Q1 (AI)
PXIe-4305	2026 Q1 (AI)
PXIe-4309	2026 Q1 (AI)
PXIe-4310	2026 Q1 (AI)
PXIe-4311[1]	2026 Q3 (AI)
PXIe-4322	2026 Q2 (AO)
PXIe-4330	2026 Q1 (AI)
PXIe-4331	2026 Q1 (AI)
PXIe-4339	2026 Q1 (AI)
PXIe-4340	2026 Q1 (AI)
PXIe-4353	2026 Q1 (AI)
PXIe-4357	2026 Q1 (AI)
PXIe-4463	2026 Q2 (AO)
PXIe-4464	2026 Q1 (AI)
PXIe-4466	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-4467	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-4468	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-4480	2026 Q1 (AI)
PXIe-4481	2026 Q1 (AI)
PXIe-4492	2026 Q1 (AI)
PXIe-4496	2026 Q1 (AI)
PXIe-4497	2026 Q1 (AI)

1. InstrumentStudio partially supports the PXIe-4311. The DAQ Analog Input panel displays standard voltage and current channels but does not display calculated power measurement channels.

Model Name	First Available
PXIe-4498	2026 Q1 (AI)
PXIe-4499	2026 Q1 (AI)
PXIe-6124	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6251	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6259	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6321	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6323	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6341	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6343	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6345	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6349	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6351	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6353	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6355	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6356	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6357	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6358	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6361	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6363	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6365	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6366	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6368	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6375	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6376	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6378	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6381	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6383	2026 Q1 (AI), 2026 Q2 (AO)

Model Name	First Available
PXIe-6386	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6396	2026 Q1 (AI), 2026 Q2 (AO)
PXIe-6738	2026 Q2 (AO)
PXIe-6739	2026 Q2 (AO)

Related concepts:

- DAQ Analog Input and DAQ Analog Output Debug Mode Limitations in InstrumentStudio

NI-RFSA Hardware Support in InstrumentStudio

RF signal analyzers supported through NI-RFSA and corresponding InstrumentStudio version availability.

Table 6. InstrumentStudio Supported RF Signal Analyzers

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

Model Name	First Available
PXIe-5841	2022 Q3
PXIe-5842	2022
PXIe-5860	2024 Q3

NI-RFSG Hardware Support in InstrumentStudio

RF signal generators supported through NI-RFSG and corresponding InstrumentStudio version availability.

Table 7. InstrumentStudio Supported RF Signal Generators

Model Name	First Available
PXIe-5820	2021
PXIe-5830	2021
PXIe-5831	2021
PXIe-5832	2021
PXIe-5840	2021
PXIe-5841	2021
PXIe-5842	2022
PXIe-5860	2024 Q3

NI Nigel AI in InstrumentStudio

Information about using NI Nigel AI to obtain guidance, perform supported actions, and explore measurement data within InstrumentStudio.

Introduced in InstrumentStudio 2026 Q3.

NI Nigel AI is an artificial intelligence assistant that helps you learn, navigate, and use InstrumentStudio more effectively. Nigel combines knowledge of InstrumentStudio workflows, project configuration, connected instruments, and product documentation to provide context-aware assistance directly within the InstrumentStudio environment. Nigel is powered by Microsoft Azure OpenAI.

Unlike traditional documentation or search tools, Nigel can use information from your active InstrumentStudio environment to provide guidance that is relevant to the current project, instruments, and workflows. Nigel can help you learn InstrumentStudio concepts, troubleshoot configuration issues, discover product capabilities, perform supported actions, and explore measurement data.

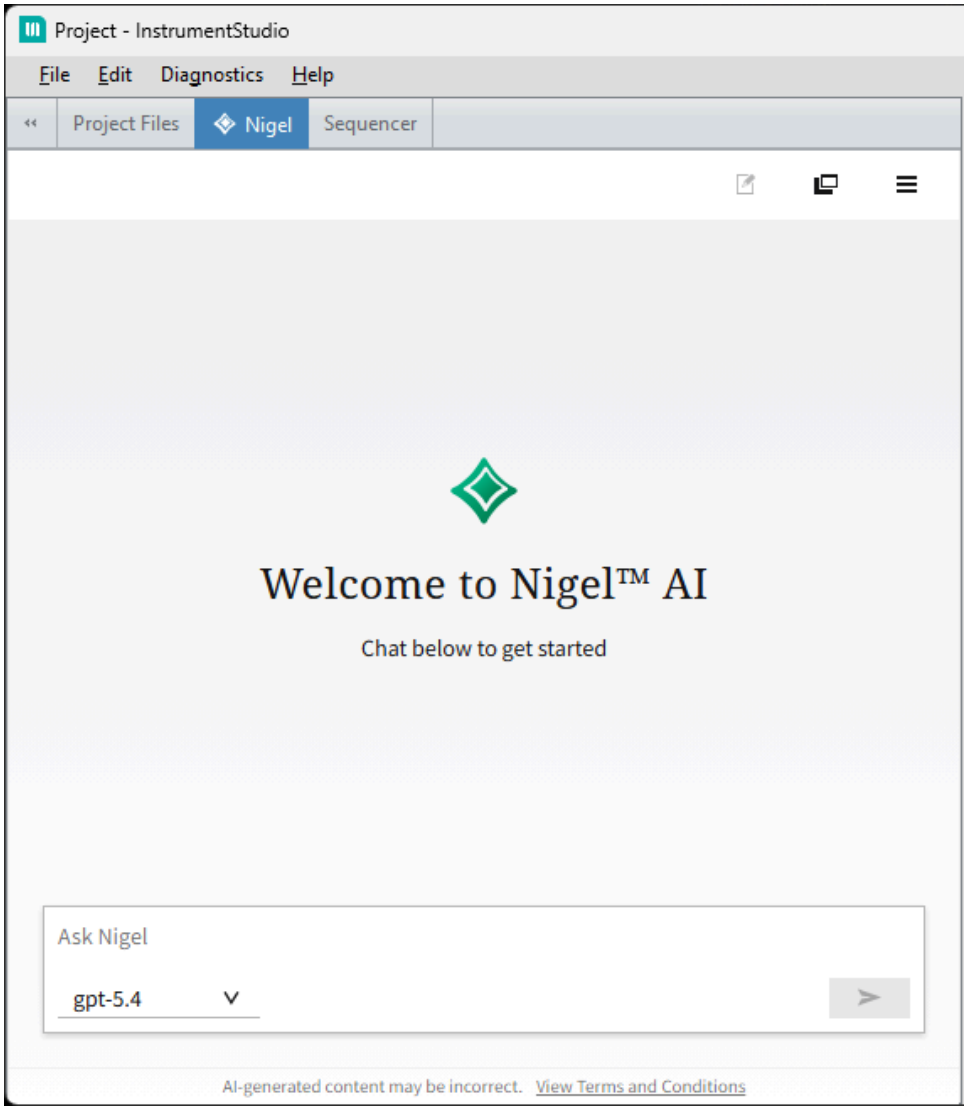
Accessing Nigel

You can access Nigel from the **Nigel** pane in InstrumentStudio.

The **Nigel** pane can be docked within the InstrumentStudio workspace or displayed in a separate floating window.

When you open Nigel for the first time, sign in with your NI user account. Access to Nigel requires an eligible NI account and the appropriate software entitlement. For more information, see *How to Access the Nigel™ AI*.

Figure 1. Nigel Pane in InstrumentStudio



Context-Awareness

Nigel is designed to understand the active InstrumentStudio environment and provide responses based on the current context.

Depending on the available information, Nigel can use context such as:

- Open projects and documents
- Connected instruments
- Instrument configurations
- Panels and layouts
- Project structure

- InstrumentStudio settings

This awareness enables Nigel to provide recommendations, explanations, and troubleshooting guidance that are tailored to the current InstrumentStudio session.

Learning and Troubleshooting with Nigel

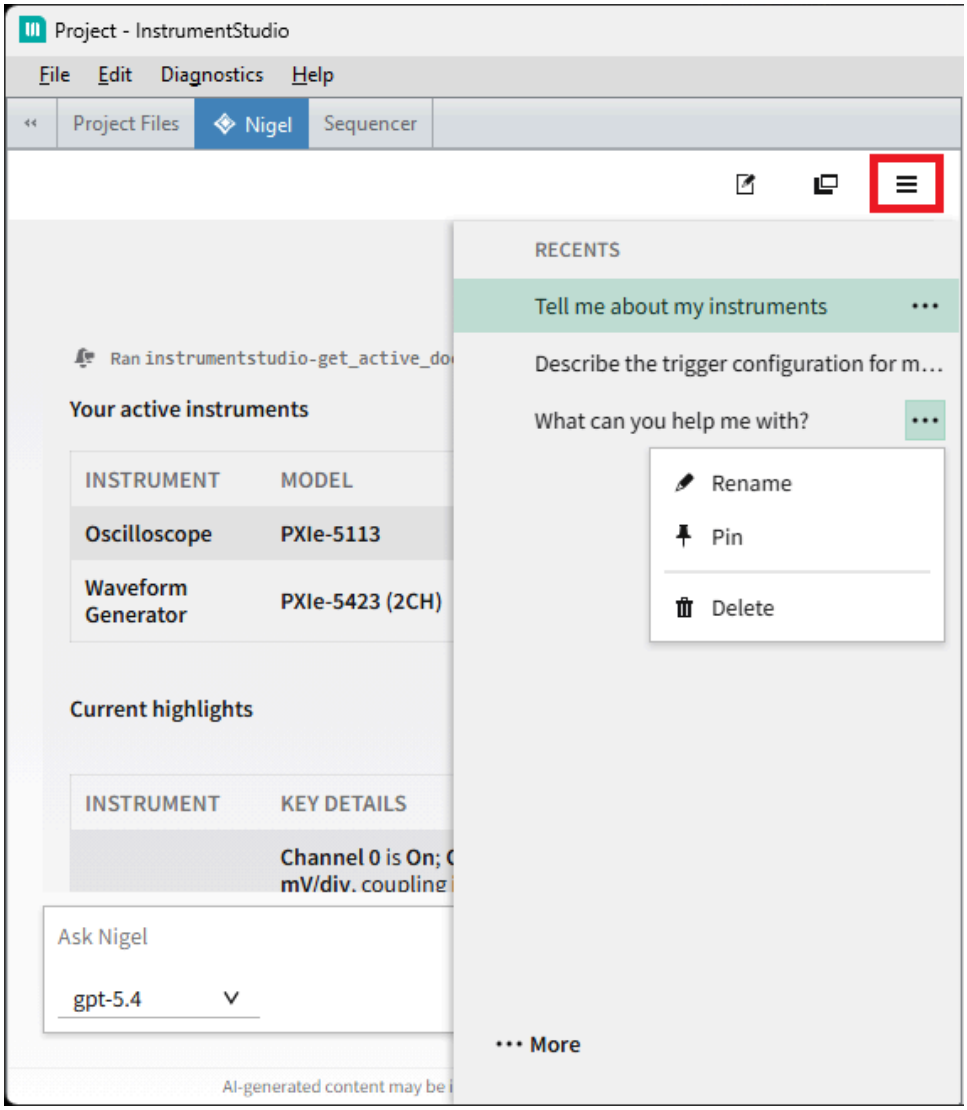
You can use Nigel while working in InstrumentStudio to:

- Learn about InstrumentStudio features
- Receive project-aware help based on active projects and documents
- Understand instrument configuration
- Understand local hardware and software characteristics
- Answer questions about hardware functionality and configuration
- Perform supported actions
- Explore Measurement Data Services data

For example, you can ask Nigel questions such as:

- `Tell me about InstrumentStudio.`
- `What hardware do I have on my system?`
- `How can InstrumentStudio help me with automation?`
- `How do I share an instrument configuration with LabVIEW? And TestStand?`
- `What functionality do you get with the Professional version?`
- `What is the difference between InstrumentStudio plug-ins and measurement plug-ins?`
- `Why is my scope not showing data?`
- `Explain scope triggering like I'm a beginner.`
- `What Measurement Data Services data has been captured on this machine?`

Figure 2. Nigel Providing Information about Active Instruments in InstrumentStudio



Performing Actions with Nigel

In addition to answering questions, Nigel can perform supported actions within InstrumentStudio.

Available actions depend on the installed software, active project, connected instruments, and enabled InstrumentStudio features.

Examples of supported actions include:

- Creating and arranging instrument panels
- Opening instrument panels and project documents

- Debugging systems
- Displaying instrument pinouts
- Exporting configurations
- Capturing data
- Opening preferences

For example, you can ask Nigel:

- `What commands can you execute?`
- `Create a panel showing my devices.`
- `Change the layout for my current document.`
- `Debug my system.`
- `Open my scope.`
- `Show pinouts for my scope.`
- `Export the configuration to TestStand.`
- `Capture data for my document.`

Some actions might require confirmation before Nigel performs the requested operation.

Accessing Nigel Chat History

Nigel stores your chat history.

To access previous conversations, use the **Chat History** button next to the **Create New Chat** button.

You can rename, pin, and delete chats to organize conversations and quickly return to important discussions.

Exploring Measurement Data with Nigel

InstrumentStudio can publish measurement data and metadata to Measurement Data Services. Nigel can help you discover, understand, and interact with data that has been captured and stored in Measurement Data Services.

Depending on available data and services, Nigel can help you:

- Discover captured measurements
- Locate datasets
- Identify measurement metadata
- Compare captured datasets
- Visualize measurement results
- Gain insights from measurement data

For example, you can ask Nigel questions such as:

- `What measurement data has been captured on this machine?`
- `Show my captured waveform data.`
- `Plot measurements for the device with serial number <serial number>.`
- `Plot measurements for the device with product number <product number>.`
- `Compare captured datasets.`

Limitations

Nigel responses are generated by an AI model and can contain inaccuracies.

Before applying recommendations to production systems, verify important technical information and configuration changes.

Nigel capabilities depend on:

- Available project information
- Connected hardware
- Installed software
- Enabled InstrumentStudio features
- Available InstrumentStudio licenses

Some capabilities and command execution features might not be available in all InstrumentStudio editions or configurations.

For AI-assisted workflows that perform actions or use tools within InstrumentStudio, GPT-5.4 generally provides the most consistent results. Other supported models can answer questions and provide guidance, but tool execution and action-based

workflows might be less reliable or produce inconsistent results.

If a requested action does not complete successfully, try the request again or switch to the recommended model.

Disabling Nigel

You can disable the **Discuss with Nigel** feature by setting the `Computer\HKEY_LOCAL_MACHINE\SOFTWARE\National Instruments\Nigel AI Advisor\DisableNigelSharing` registry key (`DWORD (32-bit), 0 or 1`) to `1`.

Related information:

- How to Access the Nigel™ AI
- Measurement Data Services Overview

Navigating InstrumentStudio

Navigation features, interface elements, and project file structure in InstrumentStudio.

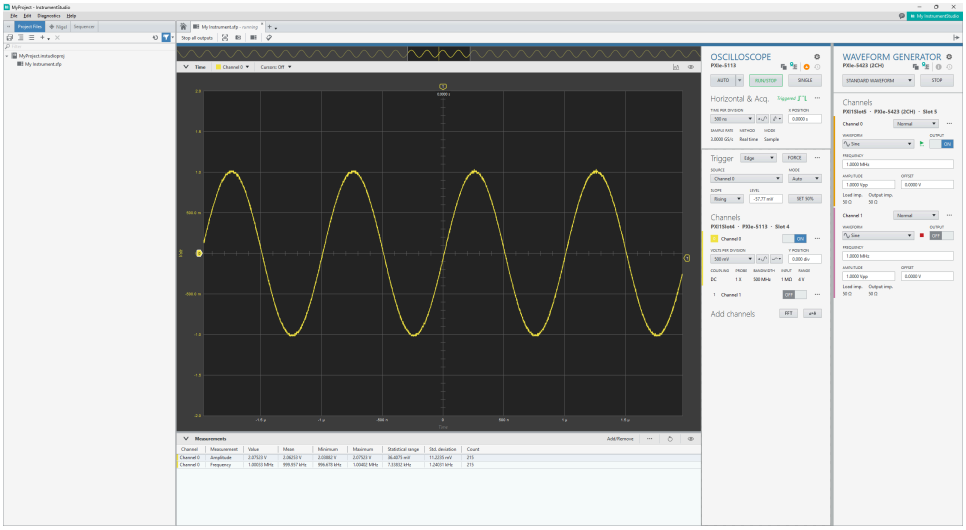
Overview

InstrumentStudio organizes instruments, measurements, project files, and related configuration data within a project.

The InstrumentStudio user interface includes the following elements:

- Navigation panes, such as **Project Files**, **Nigel**, and **Sequencer**
- The **Measurements** table
- Instrument configuration panels
- The Instrument Settings menu
- The Instrument Header menu
- Measurement graphs

Figure 3. Example InstrumentStudio Workspace with a Scope Instrument



Project Files Pane

The **Project Files** pane displays the files associated with the current project, such as:

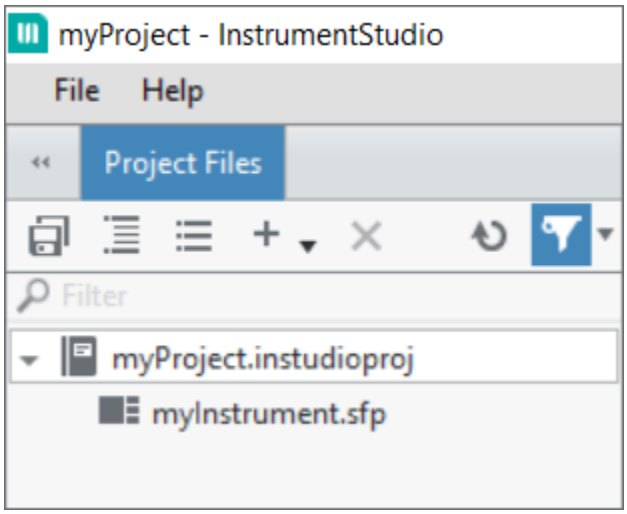
- Data files
- Saved configurations

- Scripts
- Pin maps
- Other supporting files

The **Project Files** pane also supports the following file management actions:

- Create
- Rename
- Delete
- Import
- Export
- Search

Figure 4. InstrumentStudio Project Files Pane



Use the **Project Files** pane to manage the files associated with an InstrumentStudio project, including device configurations, pin maps, captured data, and supporting files.

Table 8. InstrumentStudio Project Documents

File Type	Description
InstrumentStudio project (`.instudioproj`)	Contains the instrument panels added to your layout, device configurations, pin maps, captured data, and other project files.
Soft front panel (`.sfp`)	Contains the settings selected for each instrument in your layout.

File Type	Description
Pin map (`.pinmap`)	Defines how instruments and hardware connect to device under test (DUT) pins and sites.
Technical Data Management Stream (`.tdms`)	Stores data created by taking a snapshot of a graph. InstrumentStudio saves these files by default in the folder that contains your project.
Portable Network Graphic (`.png`)	An image of a graph created by taking a snapshot. InstrumentStudio saves these files by default in the folder that contains your project.

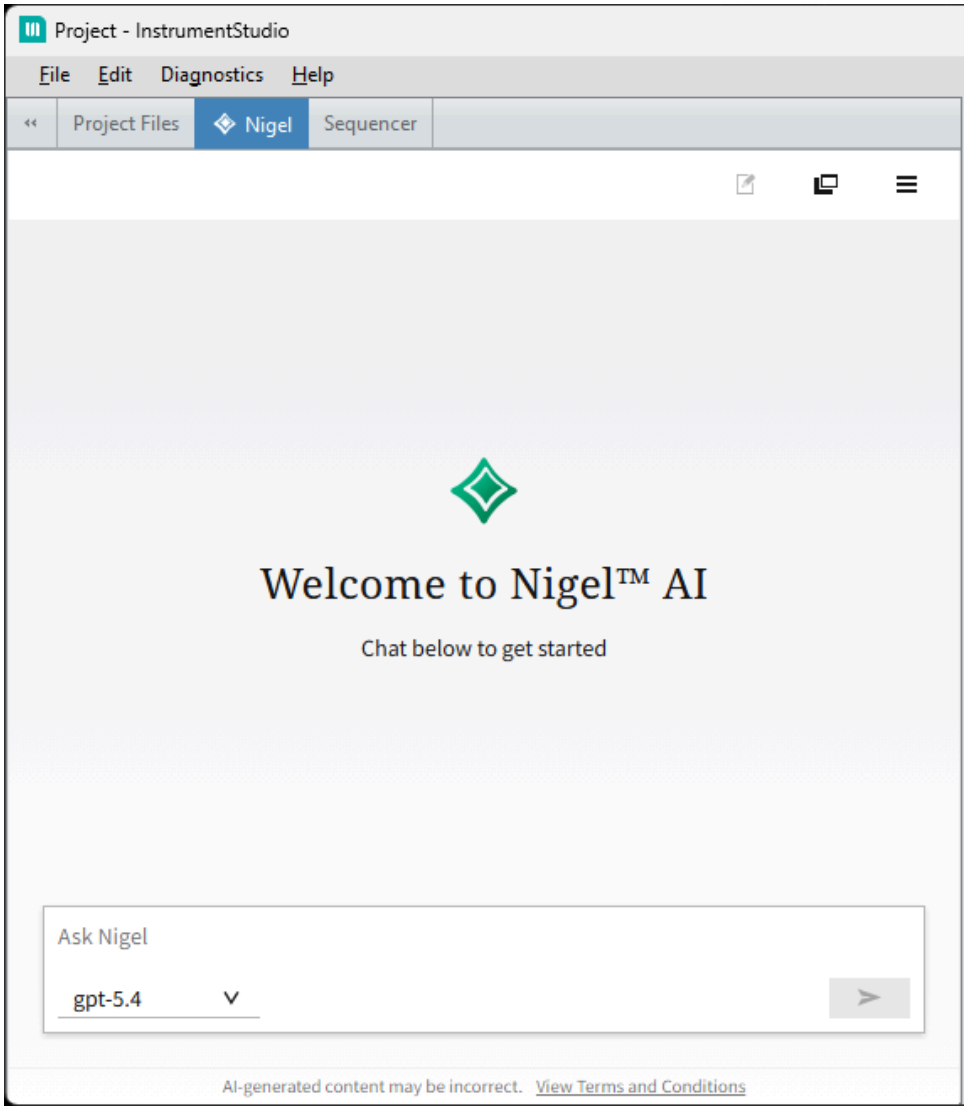
Nigel Pane

The Nigel pane provides access to NI Nigel AI within InstrumentStudio.

You can use Nigel to:

- Ask questions about InstrumentStudio features and workflows.
- Receive context-aware assistance based on the active project and connected devices.
- Perform supported actions.
- Explore Measurement Data Services data.

Figure 5. Nigel Pane in InstrumentStudio



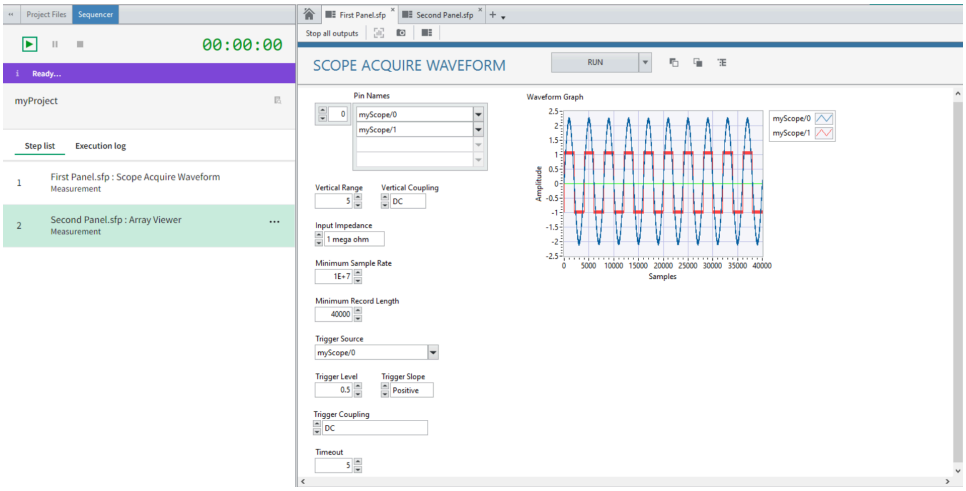
Sequencer Pane

Note This feature is only available with an InstrumentStudio Professional license.

The **Sequencer** pane allows you to do the following tasks:

- Create and modify measurement sequences
- View measurement panels associated with sequence steps
- View execution results in a sequence log

Figure 6. InstrumentStudio Sequencer Tool Pane

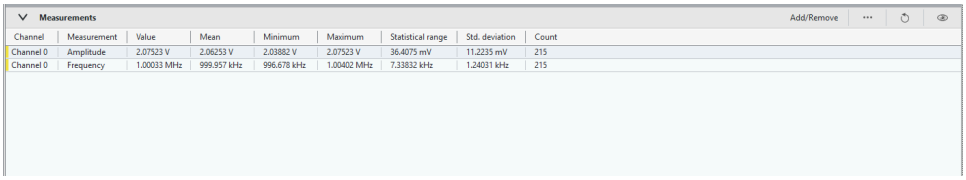


Measurements Table

The **Measurements** table allows you to configure and perform common measurement tasks, such as:

- Measuring signal properties
- Analyzing signal characteristics
- Performing stimulus response tests

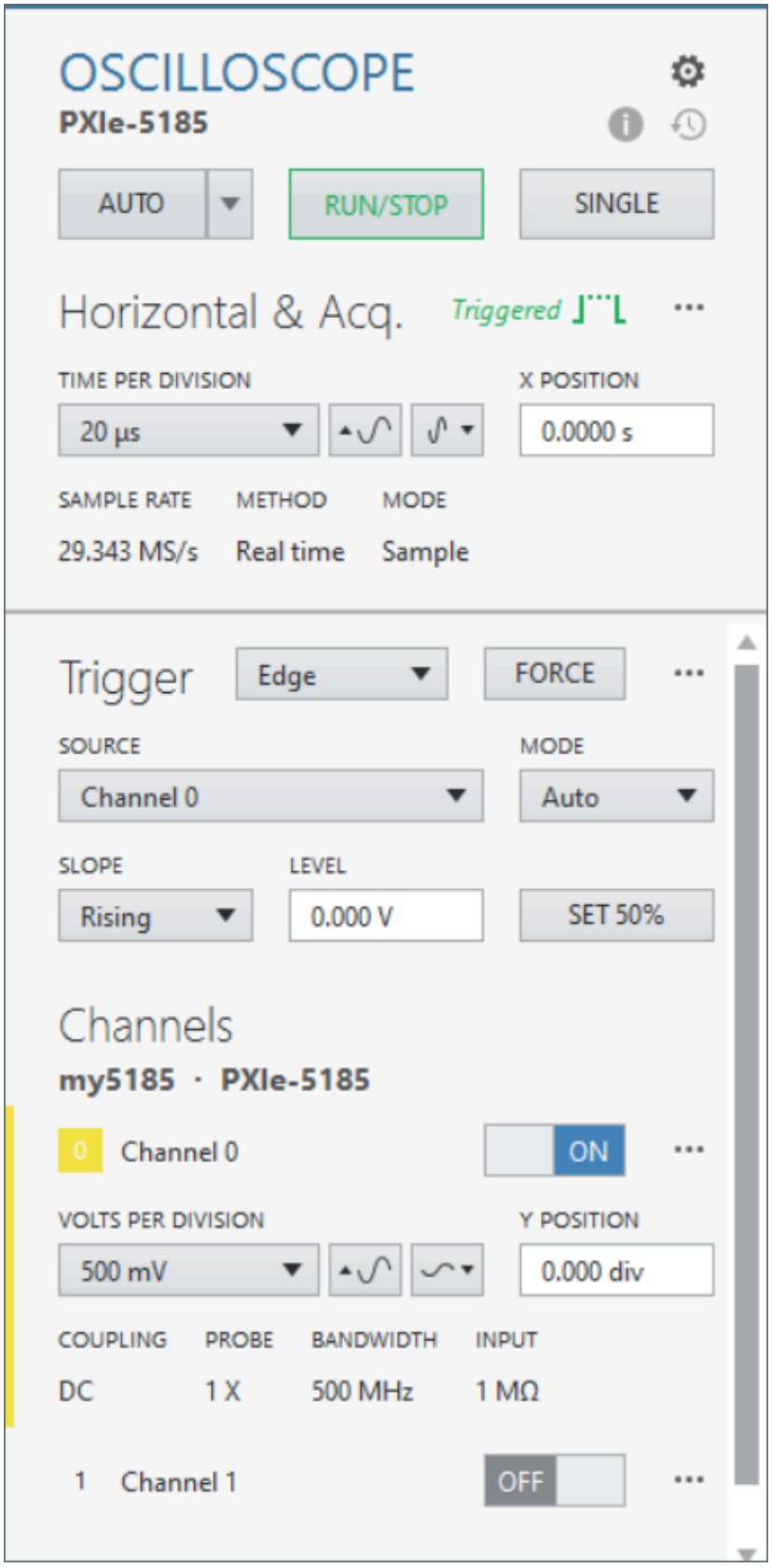
Figure 7. InstrumentStudio Measurements Table



Instrument Configuration Panel

The Instrument Configuration panel allows you to configure instrument settings and monitor instrument status.

Figure 8. InstrumentStudio Instrument Configuration Panel



Instrument Settings Menu

You can access the Instrument Settings menu (⚙) from the upper-right corner of the Instrument Configuration panel.

The available actions depend on the panel type.

The instrument settings menu allows you to:

- Export and import configurations
- Capture data
- Access device-specific actions and information
- Add or remove devices
- Configure debug sessions
- Delete panels
- Configure data sources
- Configure panel-specific settings, such as power line frequency

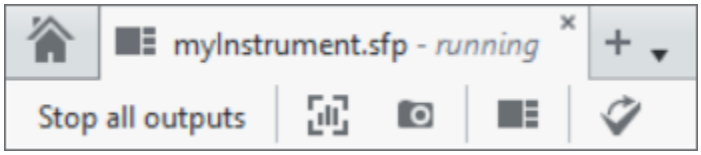
Use this menu to access panel management, data capture, and debugging features.

Instrument Header Menu

The Instrument Header menu allows you to perform the following actions:

- Stop all outputs
- Create new instrument panels (+)
- Capture data ( ) and screenshots ( )
- Edit layouts ( )
- Export to TestStand ( )

Figure 9. InstrumentStudio Instrument Header Menu



Measurement Graphs

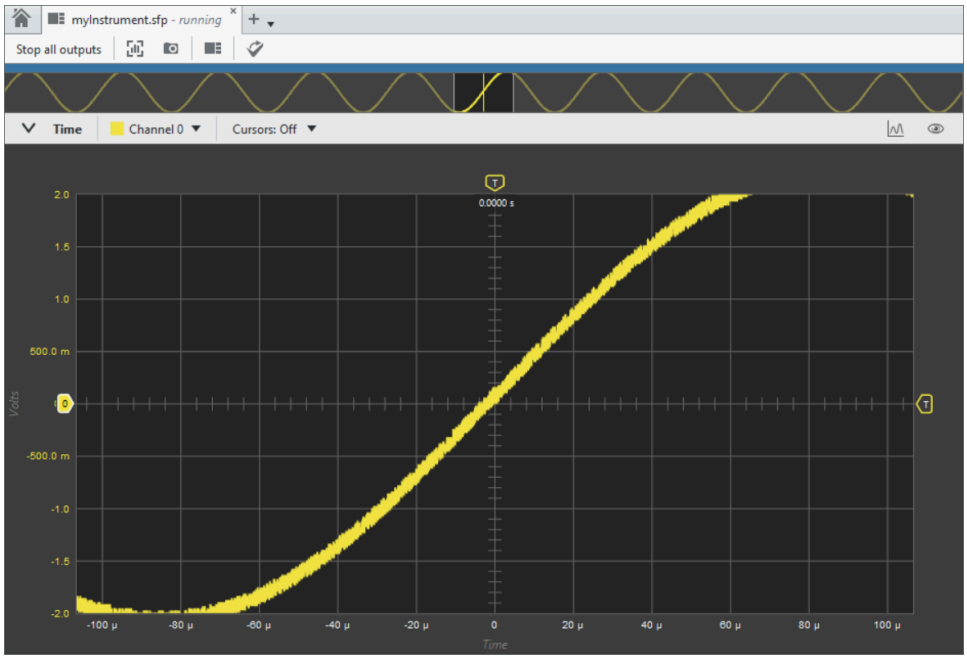
Measurement graphs allow you to visualize data from connected instruments and

interact with measurement results.

You can:

- Zoom
- Pan
- Create annotations

Figure 10. InstrumentStudio Measurement Graph Example



Related concepts:

- NI Nigel AI in InstrumentStudio
- Sequencer Tool

Related information:

- NI TDMS File Format - What is a TDMS File?

Enable Preview Features

Preview features are new features in InstrumentStudio that support specific workflows. Preview features are not release-quality features, and might have functionality gaps. You must enable preview features in the **Preferences** menu.

1. Go to **File** » **Preferences** » **Preview Features**.

Note This menu is only visible if a preview feature is available.

2. Click the toggle to enable the preview features that you want to use.
3. Close the **Preferences** panel.
4. Restart InstrumentStudio.

Install InstrumentStudio

You can install InstrumentStudio and its required dependencies through Package Manager or with an .iso file offline.

Note When you install certain instrument drivers, such as NI-SCOPE or NI-DMM, InstrumentStudio gets automatically installed as well.

Install InstrumentStudio Using Package Manager

To install InstrumentStudio through Package Manager, complete the following steps:

1. Go to the InstrumentStudio Download Page.
2. Log in to *ni.com* with your NI user account.
3. Click the **Download** button to download the `ni-instrumentstudio_<version.number>_online.exe` executable.
4. Run the executable that you downloaded.
5. Follow the instructions that Package Manager provides.

   Note During installation, Package Manager prompts you to install additional software. For information on recommended additional software, see Recommended Additional Software.

6. Restart your device when the installation process finishes.

For information on installing, removing, and upgrading NI software with Package Manager, see *Package Manager User Manual* in the *Related Information* section.

Note Package Manager might install a number of dependencies along with InstrumentStudio. InstrumentStudio requires these dependencies to function. The download and installation of these dependencies might increase total installation time.

Install InstrumentStudio Offline

When you install InstrumentStudio offline, the `.iso` file contains all required

dependencies. Package Manager does not connect to the Internet to download and install InstrumentStudio and its dependencies.

To install InstrumentStudio offline, complete the following steps:

1. Go to the InstrumentStudio Download Page.
2. Log in to *ni.com* with your NI user account.
3. Click the **Install Offline** button to download the `ni-instrumentstudio_<version.number>_offline.iso` file.

   Note The `.iso` file is over 2.5 GB in size.

4. Mount the `.iso` image or burn the `.iso` image to a disk.
5. Run `Install.exe`.
6. Follow the instructions that Package Manager provides.
7. Restart your device when the installation process finishes.

For information on installing, removing, and upgrading NI software with Package Manager, see *Package Manager User Manual* in the *Related Information* section.

Install Additional Drivers

To use the panels in InstrumentStudio, you must install the appropriate drivers for the different instrument types. InstrumentStudio shows which version of which driver you must install for each panel.

If you choose offline installation, the `.iso` image contains the required drivers for all panel types. You can find the installer packages (`.nipkg`) for the drivers in the `pool` directory.

Install Additional Software

When you install InstrumentStudio using Package Manager, you can install additional optional software. Check the corresponding box for all additional software that you want to install. Package Manager checks some boxes in this section by default.

NI recommends that you install the following additional software when you install InstrumentStudio:

Table 9. Recommended Additional Software

Software Name	Default Installation Status
Modular Instruments Support for TestStand	Checked by default
NI Certificates Installer	Checked by default
Plug-in Library CLI	Checked by default
Plug-in Library	Not checked by default

You can install InstrumentStudio using Package Manager on a machine without any other NI software. In this case, Package Manager automatically installs the following additional software:

- LabVIEW Runtime (64-bit)
- LabWindows/CVI Shared Runtime
- License Manager
- Update Service
- TDM Excel Add-In
- .NET Desktop Runtime 8.0 (32-bit)
- .NET Desktop Runtime 8.0 (64-bit)
- ASP.NET Core Runtime 8.0 (32-bit)
- ASP.NET Core Runtime 8.0 (64-bit)
- DataSocket
- LabWindows/CVI .NET Library Runtime
- LabWindows/CVI Analysis Library Runtime
- LabWindows/CVI Network Streams Library Runtime
- LabWindows/CVI Shared Runtime Base
- LabWindows/CVI SQL Toolkit Runtime
- LabWindows/CVI TDM Streaming Library Runtime
- gRPC Device Server
- The most recent version of Measurement Plug-In Support for TestStand
- System Configuration .NET Runtime
- System Configuration Runtime
- Variable Engine
- TestStand Runtime

Related information:

- Package Manager User Manual

Uninstall InstrumentStudio

You can uninstall InstrumentStudio using Package Manager or through the **Programs and Features** functionality of Windows.

> **Note**
>
> When you uninstall InstrumentStudio, no additional software gets removed. Some files and data also remain on the system. For more information, see *Leftover Elements After Uninstallation*.

Uninstall InstrumentStudio Using Package Manager

To uninstall InstrumentStudio and its components using Package Manager, complete the following steps:

1. Open Package Manager.
2. In the **INSTALLED** tab, select InstrumentStudio.
3. Click **REMOVE**.
4. In the **Select** tab of the **NI Package Manager** dialog box, click Allow Removal.
5. In the **Review** tab, click Next.

> **Note** The **Review** tab lists all InstrumentStudio components that the uninstaller removes.

The uninstaller removes InstrumentStudio and the InstrumentStudio components listed in the **Review** tab from your system.

Uninstall InstrumentStudio Manually

To uninstall InstrumentStudio and its components manually, complete the following steps:

1. Navigate to **Programs and Features** in Windows.
2. Locate **NI Software**.
3. Click **Uninstall**.
4. Select InstrumentStudio from the list of NI software.
5. Follow the instructions of the uninstallation wizard.

The uninstaller removes InstrumentStudio and its components from your system.

Related reference:

- Leftover Elements After InstrumentStudio Uninstallation

Leftover Elements After InstrumentStudio Uninstallation

Identify which installed software, files, and configuration data remain after uninstalling InstrumentStudio. Locate leftover local application data, projects, exported data, and plug-ins on your system.

When you uninstall InstrumentStudio, some software and files might remain on your system. Some configuration data might also remain.

Installed Software

All software installed with InstrumentStudio remains on your system after you uninstall InstrumentStudio.

Other NI products might also share these software components. Applications that you install separately might also use these components.

For a list of installed packages and software components, see *Install InstrumentStudio*.

Local Application Data

InstrumentStudio stores local application data in the following directory:

- `C:\Users\<UserName>\AppData\Local\National Instruments\InstrumentStudio <version.number>\`

Note Some folders in this directory are hidden.

The following folders and files remain on your system after you uninstall InstrumentStudio:

- Folders:
  - `ErrorReports`
  - `PluginCache`
- Files:
  - `LauncherProjectStatus.xml`
  - `Preferences.xml`

Projects and Documents

InstrumentStudio stores projects and documents in one of the following locations:

- `C:\Users\<UserName>\Documents\InstrumentStudio Projects\` (default)
- A user-defined location

Projects and documents remain on your system after you uninstall InstrumentStudio.

Exported Data

The following table describes data that InstrumentStudio exports and the corresponding file locations.

Table 10. InstrumentStudio Exported Data and File Locations

Data Type	File Location
Captured data	`C:\Users\<UserName>\Documents\InstrumentStudio Projects\`
Captured Measurement Data Services data	`C:\Users\<UserName>\National Instruments\Measurement Data\`
Data exported to TestStand	`C:\Users\Public\Documents\National Instruments\TestStand\Shared\IO Configurations\`

Data Type	File Location
Exported driver configurations	User-defined

All exported data remains on your system after you uninstall InstrumentStudio.

InstrumentStudio Plug-Ins

InstrumentStudio plug-ins that you added to the `\<installation path>\InstrumentStudio\Addons\` directory remain on your system after you uninstall InstrumentStudio.

Measurement Plug-Ins

All measurement plug-ins remain on your system after you uninstall InstrumentStudio, including plug-ins that you statically registered in the following directory:

- `C:\ProgramData\NationalInstruments\Plug-Ins\Measurements\`

Plug-In Library Data

The following information applies only if you installed Plug-In Library or Plug-In Library CLI.

All plug-in libraries and all plug-ins that you downloaded remain on your system after you uninstall InstrumentStudio.

SCPI Configurations

By default, InstrumentStudio stores `.json` files for SCPI-capable instruments in the following directory:

- `C:\ProgramData\NationalInstruments\Services\Scpi\ScriptDefinitions\`

These configuration files remain on your system after you uninstall InstrumentStudio.

Related concepts:

- Install InstrumentStudio

Panels and Layouts in InstrumentStudio

Panel types, layout structure, and layout requirements in InstrumentStudio.

An InstrumentStudio layout consists of one large panel and up to four small panels.

- The large panel is located in the center of the screen. The large panel includes a measurement graph.
- Small panels are located on the right sidebar. The small panels include channel settings.

Note You can combine multiple devices of the same type into a single panel, except for oscilloscopes. You can only add oscilloscopes of the same model to the same panel.

Click the **Edit layout** () button in the instrument header menu to adjust the arrangement of the soft front panels and their associated devices.

Layouts in InstrumentStudio must meet the following requirements:

- The layout must have exactly one large panel.
- The layout can have up to four small panels. Small panels are not required.
- Place oscilloscopes in a large panel.
- Only add oscilloscopes of the same model to the same panel.

Note You can only create panels for devices if you have the corresponding device driver installed on your system. When you create a panel without the appropriate driver, the panel displays the minimum expected version for that driver.

The following sections provide specific information panel and layout functionalities in InstrumentStudio.

Layout Management in InstrumentStudio

Layout management workflows available in InstrumentStudio.

Before you can take measurements and monitor devices in InstrumentStudio, you must create a layout. You can create a layout from the **Home** screen, from the instrument header menu, or from any instrument settings menu (⚙).

InstrumentStudio provides multiple layout workflows for different use cases. Choose the workflow that best matches your needs.

Creating a Layout Automatically in InstrumentStudio

Create panels automatically for compatible devices detected in the system.

When you create a layout automatically, InstrumentStudio might not show all available devices. For more control over the devices that appear in the soft front panel, see *Creating a Layout Manually*.

To create a layout automatically, complete the following steps:

1. Install the devices that you want to use. Ensure that the installed devices appear in Hardware Manager or Measurement & Automation Explorer (MAX).
2. Open InstrumentStudio.
3. Select the **Auto create layout** option from the **Home** screen or from **File** » **New** » **Auto create layout**.

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

Related concepts:

- Panels and Layouts in InstrumentStudio
- Layout Management in InstrumentStudio

Creating a Layout Manually in InstrumentStudio

Select the devices to include in a layout and control how InstrumentStudio organizes

those devices into panels.

For more information on layout rules and restrictions, see *Panels and Layouts*.

You can use the **Edit Layout** window to create a manual layout. In a manual layout, you can customize which devices the panel displays. You can also group devices of the same type into a single panel, except for oscilloscopes. You can only add oscilloscopes of the same model to the same panel.

To create a layout manually, complete the following steps.

1. Access the **Edit Layout** window in one of the following ways:
   - If an instrument panel is already open, click the **Edit layout** ( ) button in the instrument header menu.
   - From the **Home** screen, select **Manual layout...**.
   - From the toolbar, select **File** » **New** » **Manual layout...**.
   - From an instrument settings menu ( ), select **Add/Remove devices**.

   > **Note** If you access the **Edit Layout** window through an instrument settings menu, InstrumentStudio automatically applies a filter on the device list. This filter is based on which panel you selected the **Add/Remove devices** option from. You can remove or change this filter by clicking the filter icon ( ) next to the search field.

2. Search for the devices that you want to include in the layout. You can find a list of devices in the left panel of the **Edit Layout** window.
   You can filter by device type by clicking the filter icon ( ) next to the search field.
3. Add the device to a panel by selecting **Create large panel** or **Create small panel** from the dropdown menu of the device.
   The device appears in the panel that you select.

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

4. **Optional:** Group devices of the same type together in a single panel. You can select a group name from the dropdown menu of a device.
5. Select **OK** to create the layout that you specified.

Related concepts:

- Panels and Layouts in InstrumentStudio
- Layout Management in InstrumentStudio

Creating a Single-Panel Layout in InstrumentStudio

Create a layout that contains only one large panel and the selected devices.

For more information on layout rules and restrictions, see *Panels and Layouts*.

The single-panel layout option allows you to create a manual layout with a single large panel. You can customize which devices the panel displays. You can also group devices of the same type into a single panel, except for oscilloscopes. You can only add oscilloscopes of the same model to the same panel.

To create or edit a single-panel layout, complete the following steps.

1. From the toolbar, select **File** » **New** » **Single panel layout...**.
   The **Select Panel** window opens.
2. Search for the devices that you want to include in the layout.
   You can filter by device type by clicking the filter icon ( ) next to the search field.
3. Select the devices that you want to include in the layout.
4. Select **OK** to create the layout that you specified.

Related concepts:

- Panels and Layouts in InstrumentStudio
- Layout Management in InstrumentStudio

Editing an Existing Layout in InstrumentStudio

Add devices, remove devices, or reorganize panels in an existing layout.

You can edit a layout through the **Edit Layout** window.

You can access the **Edit Layout** window in one of the following ways:

- Click the **Edit layout** (▣) icon in the instrument header menu.
- Select the **Add/Remove devices** option from an instrument settings menu (⚙).

> **Note** If you select the **Add/Remove devices** option from an instrument settings menu, InstrumentStudio applies a filter on the device list. This filter is based on which panel you selected the **Add/Remove devices** option from. You can remove or change this filter by clicking the filter icon (▼) next to the search field.

Related concepts:

- Panels and Layouts in InstrumentStudio
- Layout Management in InstrumentStudio

Instrument Settings Menu Options in InstrumentStudio

Panel management, device configuration, data capture, and debugging options in InstrumentStudio.

The instrument settings menu (⚙) provides access to panel management, device configuration, data capture, and debugging options. The available options depend on the panel type.

- **Export**—Saves model-specific configuration files to a specified folder. You can import a configuration file to LabVIEW to apply the configuration of the device to a different device session. For more information, see *Export a Device Configuration*.
- **Capture data**—Captures measurement data and screenshots. For more information, see *Capturing Data in InstrumentStudio*.
- **[Device name]**—Opens options for the specified device. From this menu, you can remove the device from the panel, relink devices, and access documentation for the device.
- **Add/Remove devices**—Opens the **Edit Layout** window. For more information, see *Creating and Editing Layouts in InstrumentStudio*.
- **Configure debug session**—Opens the **Configure Debug Session** window, which you can use to monitor or control supported devices that are used by external applications. For more information, see *Debugging Measurements from External Applications in InstrumentStudio*.

- **Delete panel**—Removes the panel from the layout.
- **Probe compensation**—Allows you to enable or disable probe compensation.

Related concepts:

- Capturing Data in InstrumentStudio
- Layout Management in InstrumentStudio
- Debugging Measurements from External Applications in InstrumentStudio
- Missing Devices in InstrumentStudio

Related tasks:

- Export a Device Configuration
- Enable Debugging with C, C++, .NET, and Python Applications in InstrumentStudio

Missing Devices in InstrumentStudio

Missing device indicators and recovery options in InstrumentStudio.

InstrumentStudio considers a device missing when a project references a device that is no longer available. For example, InstrumentStudio reports a device missing if it is disconnected, removed from the system, renamed, or unavailable when you reopen a project.

When InstrumentStudio detects a missing device, it displays the following indicators:

- The title bar of the panel changes from blue to gray.
- The panel displays a **`Missing hardware`** message.
- A caution icon (⚠) appears next to the instrument header menu of the affected panel.
- A caution icon (⚠) appears next to the missing device in the instrument settings menu (⚙).
- A notification appears in the **Edit Layout** window.

You can resolve a missing device in one of the following ways:

Table 11. Relink Versus Replace in InstrumentStudio

Action	Result
Relink a device	Associates the panel with another device of the same model and preserves the existing device configuration.
Replace a device	Associates the panel with a replacement device and creates a new device configuration.

If InstrumentStudio is open, changes made in Hardware Manager or Measurement & Automation Explorer (MAX) automatically apply to InstrumentStudio panels.

If a device is missing because the device name changed while InstrumentStudio was closed, you can relink the device by restoring the original device name in Hardware Manager or MAX.

The following sections provide guidance on managing missing devices.

Related concepts:

- Panels and Layouts in InstrumentStudio
- Layout Management in InstrumentStudio
- Instrument Settings Menu Options in InstrumentStudio

Relinking a Missing Device in InstrumentStudio

Associate a panel with another device of the same model while preserving the existing device configuration in InstrumentStudio.

Use relinking to associate a panel with another device of the same model. InstrumentStudio preserves the existing device configuration and applies that configuration to the relinked device.

To relink a missing device, complete the following steps:

1. Open the instrument settings menu (⚙) for the panel that contains the missing device.
2. Select the missing device.

The missing device has a caution icon ( ) next to it.
3. Select **Relink**.
4. Select the device that you want to associate with the panel.
5. Click **Run** on the relinked panel.

InstrumentStudio associates the panel with the selected device and applies the existing device configuration to that device.

Replacing a Missing Device in InstrumentStudio

Remove a missing device from a layout and replace it with another device in InstrumentStudio.

Use replacement to remove a missing device from a layout and add a replacement device. When you replace a device, InstrumentStudio removes the existing device configuration and creates a new configuration for the replacement device.

To replace a missing device, complete the following steps:

1. Open the Edit Layout window in one of the following ways:
   - Select the **Edit layout** icon ( ) in the instrument header menu.
   - Select **Add/Remove Devices** from an instrument settings menu ( ).
2. Locate the missing device.
   The missing device has a caution icon ( ) next to it.
3. Click the × icon next to the missing device.
   The system removes the device from the layout.
4. Add a replacement device.
   - If the missing device belongs to a group, add a replacement device of the same type to the group.
   - Alternatively, create a new panel for the replacement device.
5. Click **OK**.
   The system saves the layout changes.
6. Click **Run** on the updated panel.

InstrumentStudio removes the missing device from the layout and creates a new configuration for the replacement device.

Device Synchronization in InstrumentStudio

Synchronization types and panel synchronization capabilities in InstrumentStudio.

InstrumentStudio supports software synchronization and hardware synchronization. The synchronization capabilities available to you depend on the panel type and device configuration.

Synchronization Types

The following table compares the synchronization types available in InstrumentStudio.

Table 12. Synchronization Types in InstrumentStudio

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 Support

Synchronization capabilities and limitations for supported panel types.

Digital Multimeters (DMMs)

- All channels are software-synchronized.

Waveform Generators

- All channels are software-synchronized by default.
- Channels within a 2-channel PXIe-5450 or PXIe-5451 waveform generator are hardware-synchronized.

Oscilloscopes

- All channels on a single device are hardware-synchronized.
- All channels in a panel are hardware-synchronized if all devices are the same model and installed in the same chassis.
- Channels across different devices are synchronized with NI-TClk technology.
- You can trigger an oscilloscope from another instrument in the system. To do so, configure a digital trigger and select the source according to the device name.

SMU and VPS Devices

The following synchronization capabilities apply to source measure unit (SMU) devices and variable power supply (VPS) devices.

- Channels configured for sequencing or pulse generation within a single soft front panel are automatically hardware-synchronized (). You can configure routing options by clicking the hardware synchronization icon.
- Channels are software-synchronized () in all other cases.
- You can trigger an SMU from another instrument in the system. To do so, configure a digital trigger and select the source according to the device name.

RF Instruments and Network Analyzers

The following synchronization capabilities apply to RF signal generator (RFSG), RF signal analyzer (RFSA), and network analyzer panels.

- A panel can contain only one device.
- Device synchronization is not supported.

Remote RF Signal Analyzer Panels in InstrumentStudio

Remote RFmx panel connectivity, configuration, and management in InstrumentStudio.

Consider the following limitations when you use a remote RF signal analyzer panel:

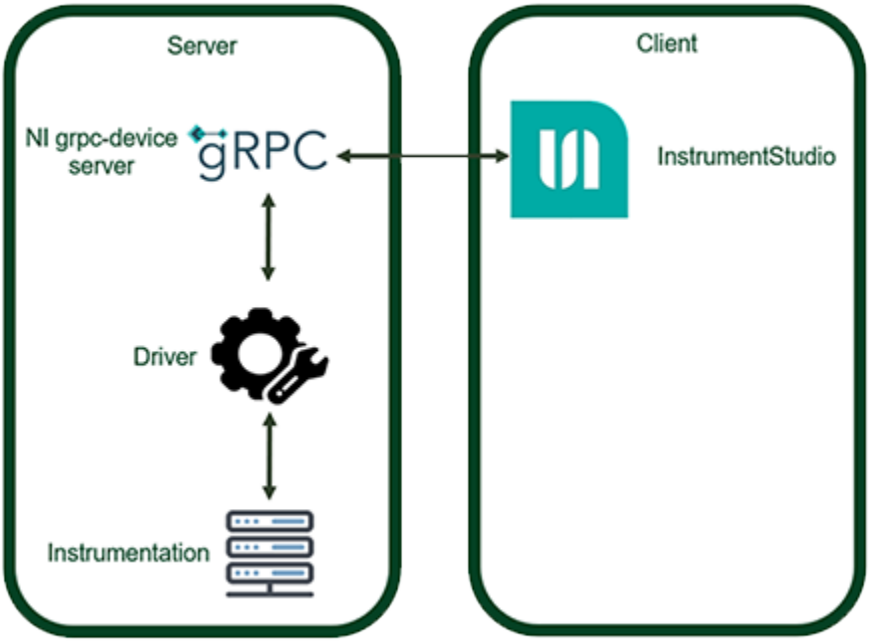
- You can only use the following personalities for remote a remote RF signal analyzer

panel:
  - Spectrum Analyzer
  - LTE
  - WLAN
  - Bluetooth
  - NR
- The remote hardware must be present on the same network in order to be remotely connected.
- The remote panels perform best when the remote hardware is within your local facility topology.
- You cannot create a remote panel for a device that is in use by another remote client.
- You cannot remotely monitor a remote session that is hosted on another client.

Overview of Remote Panel Connections

The NI gRPC-device server facilitates remote panel connections. You do not need to install NI drivers on the client that you use to remotely access a remote RF signal analyzer panel. You must install the required drivers on the server that hosts the hardware.

Figure 11. gRPC Server-Client Architecture



The following sections provide guidance on managing remote RFmx panels.

Configuring a gRPC Server for Remote RF Signal Analyzer Panels in InstrumentStudio

Configure a gRPC server to enable remote connections to RF signal analyzer hardware from InstrumentStudio.

InstrumentStudio connects to RF signal analyzer hardware through a gRPC server. Configure the server on the system that hosts the hardware. The server configuration defines network access and connection behavior.

To configure your system for remote RF signal analyzer panels, complete the following steps.

1. Go to the *NI grpc-device repository* on *github.com*.
2. Install the NI gRPC server on the computer that is connected to the hardware. Follow the instructions provided in the repository.
3. Go to the install location.
4. Open `server_config.json` in an editor.
5. Add the following parameter to the configuration file.

   ```
   "code_readiness": "RestrictedRelease"
   ```

6. Modify the value of `"address"` from `"[::1]"` to `"[::]"`. This setting allows connections from all network interfaces.
7. **Optional:** Modify the value of `"port"` in the configuration file.

   Note The default value for `"port"` is `31763`.

   - Set a specific number to use that port. You must specify this port when you connect from InstrumentStudio.
   - Set `"port"` to `0` to allow the server to assign a port automatically. The server displays the assigned port in the console output when the server starts. Specify the assigned port when you connect from InstrumentStudio.
8. Run the gRPC server. Follow the instructions provided in the repository.

   The gRPC server starts and listens on the configured port. The server now accepts

connections from InstrumentStudio.

InstrumentStudio can now connect to and control RF signal analyzer hardware through the configured gRPC server.

Valid gRPC Server Configuration File

The following example shows a valid configuration file.

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

For more detailed information about server configuration and usage, see *NI grpc-device Repository*.

Related information:

- NI grpc-device Repository

Adding a Remote RFmx Panel to Your Layout in InstrumentStudio

Connect to remote RFmx hardware and add remote panels to a layout in InstrumentStudio.

You must configure your system before you begin this procedure. For more information, see *Configuring a gRPC Server for Remote RF Signal Analyzer Panels in InstrumentStudio*.

To view RFmx hardware measurements remotely using an InstrumentStudio panel, complete the following steps.

1. Enable the **Remote RFmx panel** feature in the InstrumentStudio settings.
2. Click **Add remote host connection**.
3. Select Remote Instrument as the **Host type**.
4. Enter the IP address (or hostname) and the port number of the server that you want to connect to.
5. Click **OK** to add a panel for the remote RFmx device to your layout.
6. Continue setting up your layout as needed.

After you add the panel to your layout, the device name appears as follows:

***Device Name-Model Number*(Remote)**

Note You must take over a device session to interact with a device in control mode. You can only view panels in debug mode when the session runs locally on the device.

Related tasks:

- Configuring a gRPC Server for Remote RF Signal Analyzer Panels in InstrumentStudio

Managing Remote Connection Preferences in InstrumentStudio

You can remove a remote connection or select a default remote device.

1. Go to **File** » **Preferences** » **Remote connections**.
2. Choose one of the following options to manage remote connections:

Option	Description
Remove a Connection	Right-click **Remove device**.
Select a Default Device	Right-click a device from the list of available devices and select **Make default device**. InstrumentStudio uses this device when you create layouts automatically.

SCPI Instrument Panels in InstrumentStudio

SCPI instrument panel setup, configuration, and support in InstrumentStudio.

SCPI instrument panels allow you to interact with SCPI-capable devices in InstrumentStudio. You can use SCPI instrument panels to acquire data, manage configuration settings, and send commands to supported devices. SCPI instrument panels do not include all capabilities available for native NI hardware and are intended to complement third-party instrument workflows.

The following sections describe how to connect, configure, and use SCPI-capable devices in InstrumentStudio.

Related concepts:

- Connecting SCPI-Capable Devices in InstrumentStudio
- Using SCPI Instruments in InstrumentStudio

Related tasks:

- Configuring an SCPI-Capable Device in InstrumentStudio

Related information:

- Measurement & Automation Explorer (MAX) Help
- Hardware Manager User Manual

Connecting SCPI-Capable Devices in InstrumentStudio

Device connection requirements and panel creation workflows for SCPI-capable devices in InstrumentStudio.

InstrumentStudio can detect SCPI-capable devices only after you add the devices to the system through Hardware Manager or Measurement & Automation Explorer (MAX).

After you connect a device, you can add an SCPI instrument panel through the Manual Layout workflow. SCPI-capable devices appear in the device list under **Instruments** » **VISA Instruments**.

Some devices require additional configuration before InstrumentStudio can fully identify and use the device. In these cases, the device can appear as **`Unknown Instrument`** until you complete the required setup.

Configuring an SCPI-Capable Device in InstrumentStudio

Configure support files for SCPI-capable devices that require additional setup in InstrumentStudio.

Some SCPI-capable devices require additional configuration before InstrumentStudio can fully identify and support the device. When additional setup is required, the device can appear as **Unknown Instrument** in the **Manual Layout** window.

Use this procedure to configure SCPI-capable devices that require customization of the generated SCPI support file before InstrumentStudio can fully support the device.

1. Add your instrument to your system using Hardware Manager or MAX.
2. From the InstrumentStudio **Home** screen, go to **Manual layout**.
3. Find your SCPI instrument and add it to a panel.
4. When you create an SCPI instrument panel that requires additional setup, InstrumentStudio prompts you to set up the panel. Click the appropriate button to proceed.
   A file explorer window opens at the location of the new `.json` configuration file that is associated with the selected SCPI-capable device.
5. Open the new `.json` file by navigating to `%PROGRAMDATA%\National Instruments\Services\Scpi\ScriptDefinitions`.

Note You can also access the `.json` file from an SCPI panel. Open the instrument settings menu (⚙) and select **Open SCPI Support File Location**.

6. Read through the comments included in the `.json` file.
   The comments describe what each section of the template does. The comments also explain how to modify the sections for your instrument.
   For example, take the following section of the `.json` file:

```
// List of all channels available on the instrument.
  "Channels": [
```

```
    {
      "DisplayName": "Channel 1",
      "ScpiName": "CHAN1"
    },
    {
      "DisplayName": "Channel 2",
      "ScpiName": "CHAN2"
    },
    {
      "DisplayName": "Channel 3",
      "ScpiName": "CHAN4"
    },
    {
      "DisplayName": "Channel 4",
      "ScpiName": "CHAN4"
    },
  ],
```

You must ensure that the `Channels` section matches the number of available channels on your device. If you set up a two-channel instrument, you must remove the lines that pertain to channels 3 and 4.

7. Save and close the file.

> **Notice** Do not change the name of the `.json` file or move the file from the folder that it was created in.

8. Open the instrument settings menu (⚙) and select **Refresh panel**.

For information on fetching data and configuration settings from SCPI instruments, see *Using SCPI Instruments*.

SCPI Instrument Support in InstrumentStudio

Supported devices, vendors, configuration files, and resources for SCPI instrument panels in InstrumentStudio.

InstrumentStudio supports SCPI instrument panels for a variety of device types and vendors. Support capabilities depend on the device and the associated SCPI configuration file.

Supported Device Types

InstrumentStudio supports the use of the following SCPI-capable instruments:

- Oscilloscopes
- Source Measure Units (SMUs)
- Digital Multimeters (DMMs)

Generic support is available for other SCPI-capable device types.

Supported Vendors

InstrumentStudio includes configuration files for tested devices from the following vendors:

- Agilent
- Keysight
- Tektronix
- Fluke
- Keithley

Configuration files are installed with InstrumentStudio for each of the vendors listed.

Note Although exhaustive testing of third-party SCPI-capable devices is not possible, the included configuration files typically work for all devices from a vendor.

Devices from Other Vendors

InstrumentStudio includes a generic configuration file for SCPI-capable devices from other vendors. When you add one of these devices to a panel, InstrumentStudio generates a new configuration file. Depending on the device, you might need to modify the generated file.

Note NI cannot guarantee that all SCPI-capable devices for other vendors work as expected in InstrumentStudio. Refer to the documentation for your device for more details on its SCPI command set.

Examples and Templates

InstrumentStudio installs example configuration files and templates in `%PROGRAMFILES%\National Instruments\Shared\Services\Scpi\ScriptDefinitions\Examples`.

These examples provide additional guidance for supported script types and device commands.

Note Other product names and company names listed are trademarks or trade names of their respective companies.

Using SCPI Instruments in InstrumentStudio

Data acquisition, configuration management, and command execution with SCPI instruments in InstrumentStudio.

SCPI instrument panels provide tools for acquiring measurement data, managing configuration settings, and communicating directly with connected devices.

The following topics describe common SCPI instrument workflows.

Related tasks:

- Configuring an SCPI-Capable Device in InstrumentStudio

Fetching Data from a SCPI Instrument in InstrumentStudio

Retrieve measurement data and screenshots from a SCPI device in InstrumentStudio.

1. Select the data that you want to capture using the checkboxes in the instrument panel.
   2. **Channels**—Select specific channels to fetch data from. This option sends a SetChannel SCPI command to the device.

      Important You must select at least one channel to fetch data.

3. **Include screen image**—Captures an image of your SCPI instrument screen. The captured image is either in a `.png` or a `.bmp` format, depending on the device.
4. Click **Fetch Data** to fetch the data from the selected sources.

> **Note** You can also fetch this data using **Capture Data** and **Capture Screenshot**.

To fetch new data, trigger data collection on your SCPI instrument and repeat this process.

Configuration Management for SCPI Instruments in InstrumentStudio

Configuration retrieval and application workflows for SCPI instruments in InstrumentStudio.

When you make changes to your device, such as turning a setting knob or pressing a button, the configuration settings for the device change. The device stores these changes. If you have a preferred configuration for a specific task, you can fetch and store the configuration settings in an SCPI instrument panel.

Fetching Existing Configuration Settings

To fetch existing configuration settings, complete the following steps.

1. Open the instrument configuration utility by clicking the instrument configuration utility icon (🔧).
2. Select the **Fetch Configuration** tab.
3. Click **Fetch**.
4. Save the InstrumentStudio project to store the fetched configuration settings in the SCPI instrument panel.

The fetched configuration settings appear in the configuration utility window.

Applying Configuration Settings

To apply configuration settings, complete the following steps.

1. Open the instrument configuration utility by clicking the

instrument configuration utility icon (🔧).
2. Select the **Fetch Configuration** tab.
3. Click **Apply**.

The device uses the configuration settings stored in the SCPI instrument panel.

Sending Commands to a SCPI Device in InstrumentStudio

Send custom SCPI commands to a connected SCPI device in InstrumentStudio.

To send SCPI commands to your SCPI-capable devices using a SCPI instrument panel, complete the following steps.

1. Click the 🔧 icon to open the instrument configuration utility.
2. Click the **Send Command** tab.
3. Click **Send Command** to open the command editor.
4. Enter the SCPI command that you want to send to your SCPI-capable device in the textbox.

> **Note** Refer to the user manual of your device for more information on supported SCPI commands.

5. Click **Send** to send the SCPI command to your device.

Device Settings in InstrumentStudio

Instrument configuration settings and configuration export options for supported InstrumentStudio devices.

The following sections contain information about configuring instruments and exporting configurations.

Digital Multimeter Settings in InstrumentStudio

Measurement configuration and acquisition settings for Digital Multimeter (DMM) instruments.

Auto Range Modes

Auto range modes automatically detect the signal range when the range for a signal is unknown.

Auto range modes allow you to measure a signal when you do not know the range of the signal. Instead of specifying a range for a voltage measurement or a current measurement, you can enable one of the following auto range modes:

- **Auto** ( )—Takes a reading before each measurement to select the smallest appropriate range.
- **Auto once** ( )—The first point that the digital multimeter (DMM) measures determines the range for the rest of the measurements.

Auto range modes allow the DMM to take a series of measurements. Then, the DMM adjusts the range until the measurement falls within the smallest range appropriate for that measurement.

Note Model-specific considerations might apply when you use auto range modes. For more information, see the *NI-DMM User Manual* or the *User Manual* for your specific DMM device.

Related information:

- NI Digital Multimeters

Enable Automatic Level Range

Complete the following steps to enable automatic level ranging:

1. Click ··· to open the **Channel Settings** window.
2. Enable one of the auto range modes in the **Range** section of the window.
   - Click the icon to enable Auto mode.
   - Click the icon to enable Auto once mode.

The icon that you clicked turns blue (or) and the corresponding auto range mode is enabled.
To disable an auto range mode, click the blue icon or select the other auto range mode.
Related reference:

- Managing Device Configurations in InstrumentStudio

Perform Cable Compensation

When you connect a DMM to a DUT with switches, fixtures, or cables, these instruments might introduce errors into the measurement. Open and short cable compensation minimizes these measurement errors when you take capacitance or inductance measurements.

Compensation consists of the following steps:

- Measuring the error
- Applying the measured error to the actual measurement

This process allows you to correct and minimize the errors introduced by the test system. You must renew the compensation values before you take a measurement at a specified function and a specified range. If the range or function changes, the compensation type defaults none. Then, you must perform compensation again for accurate measurements.

Perform Open Compensation for Capacitance Measurements 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. 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. InstrumentStudio applied the compensation if the symbol appears.

Perform Short Compensation for Capacitance Measurements 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 with a low-impedance connection between the **HI** terminal and the **LO** terminal of the DMM. Use cables and switches with low capacitance and low path resistance.
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. InstrumentStudio applied the compensation if the symbol appears.

LCR Device Settings in InstrumentStudio

Measurement, calibration, and compensation settings for LCR devices.

Calibrate an LCR Device

The accuracy of a device naturally drifts over time. You can perform self-calibration to compensate for this drift. Depending on your version of InstrumentStudio, you can perform self-calibration in Measurement & Automation Explorer (MAX) or in InstrumentStudio.

Self-calibration ensures accurate measurements by doing the following:

- Compensating for board-level temperature variations
- Compensating for device degradation
- Ensuring that the device operates in the ranges that you define in the test specifications

To calibrate an LCR device, complete the following steps:

1. In the LCR device instrument panel, open the instrument settings menu (⚙).
2. Click **Device Slot** » **Calibration** » **Self Calibrate...**.
   InstrumentStudio 2025 Q2 and earlier versions perform self-calibration in MAX. Later versions of InstrumentStudio perform self-calibration without MAX.

When to Calibrate an LCR Device

Performing self-calibration regularly ensures accuracy in your test measurements.

Performing self-calibration is recommended in the following cases:

- You install 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 changed significantly.
- The onboard temperature of the LCR device drifted by at least ±5 °C since the last self-calibration.
- 24 hours passed since the previous self-calibration.

Perform Compensation on an LCR Device

When you connect an LCR device to a DUT with switches, fixtures, or cables, these instruments might introduce errors into the measurement. To prevent the measurement errors that these instruments introduce, apply compensation before you take the measurement.

Perform cable and LCR device compensation after, for example, the following actions:

- You make changes to the test system setup.
- You perform an LCR device calibration on a new LCR device.

For more information on when to perform new compensations, see When to Generate New Data for Custom Cable Compensation or LCR Compensation.

For more information on how to perform compensation on an LCR device, see *Compensation of LCR Measurements* in the *Related Information* section.

To perform compensation on an LCR device, complete the following steps:

1. Calibrate the LCR device.
2. Determine the type of cable compensation to perform based on the following factors:

Option	Recommended When
Standard Cable Compensation	You use a standard NI cable that has a **Cable Length** profile in the **Instrument Settings** dialog (⋯) and a simple test setup. That is, the test setup excludes complex circuitry, such as switches, that might affect LCR measurements.
Custom Cable Compensation	You use non-NI cabling and a complex test setup. That is, the test setup includes complex circuitry, such as switches, that might affect LCR measurements.

Note Apply cable compensation for low-level LCR measurements where

the effects of cabling might be significant relative to the quantity that you measure.

Note Apply Short LCR Compensation first when you use custom non-NI cabling to ensure accurate measurements.

3. Determine the type of LCR compensation to perform based on the following factors:

Option	Recommended When
Open LCR Compensation	Stray admittance in your test setup influences the LCR measurements. **Note** Such stray admittance is present in most test setups.
Short LCR Compensation	The impedance of your DUT is <100 Ω. The residual impedance of cabling and fixtures might be a significant percentage of your DUT impedance.
Load LCR Compensation	The reference load value is similar to your DUT. Your test setup also includes complex circuitry, such as switches, that might affect LCR measurements.

Note To perform Load LCR Compensation, you must first perform Open LCR Compensation and Short LCR Compensation.

Related information:

- Compensation of LCR Measurements

Perform Standard Cable Compensation

Use the **Instrument Settings** window to select the appropriate cable length for your instrument. Use standard compensation if you use standard cables for your

instruments.

1. Click the ··· icon in your LCR device instrument panel to open the **Instrument Settings** window.
2. Select your cable length from the **Cable type** dropdown menu.

Perform Custom Cable Compensation

Use the **Instrument Settings** window to select custom compensation. Use custom compensation if you use custom cables for your instruments.

Disconnect the 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 might be a switch, fixture, or any other circuit that is used to connect the LCR device cabling to the DUT. When you perform custom cable compensation, only connect the custom cables 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 performing the following steps:

1. Click the ··· icon in your LCR device instrument panel to open the **Instrument Settings** window.
2. Select the Custom option from the **Cable type** dropdown.

Verify that InstrumentStudio applied compensation by performing the following steps:

1. Click **Custom Compensation** (⇆).
2. Check if **Apply compensation** is enabled.

For more information on how to perform compensation on an LCR device, see *Compensation of LCR Measurements* in the *Related Information* section.

Related information:

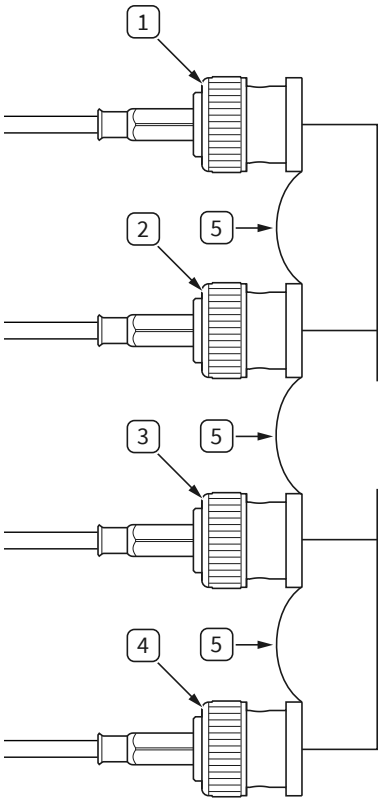
- Compensation of LCR Measurements

Perform Open LCR Compensation

Apply open compensation with the **Open Compensation** (☞) option in the instrument panel.

Disconnect the DUT from the LCR device 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. The following diagram shows how to set up an open configuration for open LCR compensation:

Figure 12. Open Configuration Diagram



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

To perform Open LCR Compensation, do the following steps:

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**.

Verify that InstrumentStudio applied compensation by performing the following steps:

1. Click **Open Compensation** ( ).
2. Check if **Apply compensation** is enabled.

For more information on how to perform compensation on an LCR device, see *Compensation of LCR Measurements* in the *Related Information* section.

Related information:

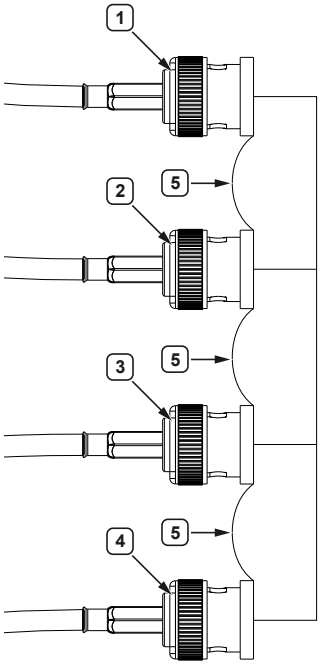
- Compensation of LCR Measurements

Perform Short LCR Compensation

Apply **Short Compensation** ( ) option in the instrument panel. You must perform Short LCR Compensation when you use custom non-NI cabling.

Disconnect the DUT from the LCR device at the connection point on the DUT. Leave the cabling, switches, and DUT fixtures connected to the LCR device. The following diagram shows how to set up a short configuration for short LCR compensation:

Figure 13. Short Configuration Diagram



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

Note You must perform Short LCR Compensation when you use custom non-NI cabling. Complete Custom Cable Compensation before you perform Short LCR Compensation.

To perform Short LCR Compensation, do the following steps:

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**.
5. Click **OK**.

Verify that InstrumentStudio applied compensation by performing the following steps:

1. Click **Short Compensation** ().
2. Check if **Apply compensation** is enabled.

For more information on how to perform compensation on an LCR device, see *Compensation of LCR Measurements* in the *Related Information* section.

Related information:

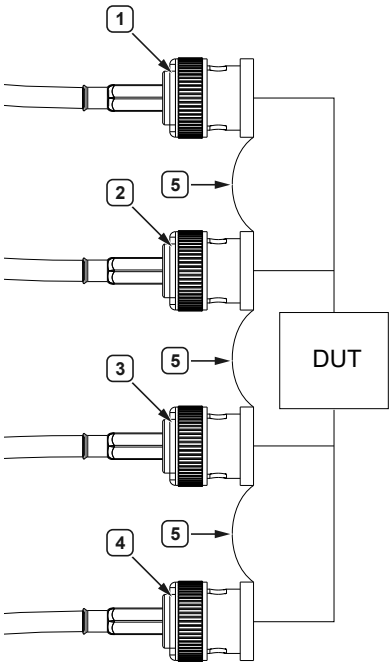
- Compensation of LCR Measurements

Perform Load LCR Compensation

Apply open compensation with the **Load Compensation** () option in the instrument panel.

Connect a reference load to the fixture that will contain your DUT. The following diagram shows how to set up a load configuration for load LCR compensation:

Figure 14. Load Configuration Diagram



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

To perform Load LCR Compensation, do the following steps:

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**.

Verify that InstrumentStudio applied compensation by performing the following steps:

1. Click **Load Compensation** ().
2. Check if **Apply compensation** is enabled.

For more information on how to perform compensation on an LCR device, see *Compensation of LCR Measurements* in the *Related Information* section.

Related information:

- Compensation of LCR Measurements

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

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

Generate new custom cable compensation data or LCR compensation data in the following cases:

- 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 that contains 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 with a different part number from the DUT for which you originally generated the compensation data.

Note For Load LCR Compensation, choose a new reference load that is appropriate for the new DUT.

Oscilloscope Settings in InstrumentStudio

Acquisition, triggering, and display settings for Oscilloscope instruments.

Oscilloscope Setup Actions

You can configure the setup of an oscilloscope with the Auto setup or the Default setup option.

To change setup mode, click the arrow next to the button that displays the currently selected setup mode.

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

Settings Changed by Auto Setup

Setting an oscilloscope to Auto setup changes the following settings to the following values:

General Settings	Value
Acquisition Mode	Normal
Reference Clock	Internal

Note The *Changed by Auto setup* value means that Auto setup changes the corresponding setting to different values depending on the signal.

Vertical Settings	Value
Vertical Coupling	Unchanged by Auto setup
Vertical Bandwidth	Full

Vertical Settings	Value
Vertical Range	Changed by Auto setup
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
Min Record Length	Changed by Auto setup
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

Set the Sample Rate and the Record Length Manually

By default, InstrumentStudio optimizes the record length or the sample rate of an oscilloscope when taking measurements.

Configure the Record Length

You can configure the record length by setting **Data Display** to one of the following

options:

- **Live**—InstrumentStudio sets the maximum record length to 1 million samples.
- **On Demand**—InstrumentStudio sets the maximum record length to 10 million samples.

Manual Override Mode

To configure larger records, use Manual Override mode. Manual Override mode allows for the configuration of both the sample rate and the record length of an oscilloscope.

> **Note** NI recommends that you only enable Manual Override mode for test parameters that require specific sample rate and record length values. Otherwise, to get the best interactive experience with an oscilloscope, do not enable this setting.

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

1. Click the ··· icon for the instrument to open the **Instrument Settings** window.
2. On the **Acquisition** tab, click the **Manual Override** mode toggle.
   The toggle is in the **On** position.
3. Enter the desired values for **Sample Rate** and **Record Length**.

> **Note** If the sample rate or the record length is configured improperly, enabling **Manual Override** mode might lead to unexpected graph behavior. For example, if your device does not have enough onboard memory to accommodate measurement settings, your graphs might not render properly. You might also get an error message in InstrumentStudio. Other settings might also cause data to be unreadable or take longer than expected to appear. Take, for example, entering a **Sample rate** of 1 kS/s with a **Record length** of 3.5 MS. In this case, it takes around an hour for the measurement data to appear in your oscilloscope panel in InstrumentStudio.

Acquisition Status

You can determine the current state of an acquisition by looking at the acquisition status icon. The acquisition status icon is in the **Horizontal & Acq.** header of an

oscilloscope panel.

InstrumentStudio differentiates between the following acquisition statuses:

- **Triggered**—The device met the trigger condition and performed an acquisition according to the specified trigger settings.
- **Auto**—After a certain amount of time passes without a trigger firing, the device automatically triggers an acquisition. The Auto status displays briefly when automatic acquisition occurs.

  > **Note** You must set the trigger mode to **Auto** to enable automatic triggering.

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

Interpolation Methods

You can set the interpolation method of an oscilloscope acquisition from the **Instrument Settings** window.

Click the dropdown menu for the **Interpolation method** setting in the **Acquisition** tab of the **Instrument Settings** window.

You can set interpolation to one of the following settings:

- **No interpolation**—The oscilloscope does not interpolate data.
- **Auto**—The oscilloscope interpolates data when the visible time for each division is too small for acquisitions that are performed at the maximum sample rate.

  > **Note** When you set the sample rate of the oscilloscope to Manual, the oscilloscope treats the sample rate that you enter as the maximum sample rate. The Auto interpolation method considers headroom when determining whether to perform interpolation. If the headroom is low between the bandwidth and the sample rate of the oscilloscope, the oscilloscope does not interpolate the data.

- **On**—The oscilloscope interpolates data as often as possible and does not take

headroom into account. If you set the interpolation method to On, the oscilloscope interpolates data in all cases, except for the following:

- The Random Interleaved Sampling (RIS) sampling method is set for the oscilloscope.
- 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.

Click the dropdown menu for the **Sampling method** setting in the **Acquisition** tab of the **Instrument Settings** window. You can set sampling to one of the following methods:

- **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 Not all oscilloscopes support RIS. See the user manual of your device to determine whether it supports RIS.

Sample Modes

You can select different sample modes from the **Instrument Settings** window.

Click the dropdown menu for the **Sample mode** setting in the **Acquisition** tab of the **Instrument Settings** window. You can set sample mode to one of the following settings:

- **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 the oscilloscope saves the maximum value and the minimum value of the input signal within the configured sample interval. Use Peak detect when you want to acquire measurements for a longer time but you still want to detect transients.

- **Averaging**—The oscilloscope averages the waveform data of continuous acquisitions that are based on the averaging count. The oscilloscope performs an exponential moving average and shows the number of averages. The number of averages shows how many waveforms are read and averaged.

Add FFT Channels and Markers

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

Create FFT Channels

1. From a large oscilloscope panel, click the **FFT** button in the **Add Channels** section. A frequency graph appears.
2. Select a **Source** for the FFT channel. You can select any single channel that already exists.
3. Select a **Window** function for the FFT channel.
4. (Optional) Configure additional settings for the FFT channel by clicking the ··· icon next to the FFT channel. For more information, see Search for and Compute Peaks.

Create Markers

A marker fetches the amplitude (y-location) of a particular sample at the specified frequency (x-location) of a particular sample.

Note You can only create markers after you create an FFT channel.

Complete the following steps to create markers:

1. Select the **Markers** dropdown menu in the **Frequency Graph** header.
2. Select **Markers: On** to enable markers. The **Markers** table appears below the frequency graph.
3. From the **Markers** table, select the marker that you want to configure. By default, only Marker 0 is enabled.

Note You can enable markers by selecting a marker from the **Markers**

table. Then, change the marker mode from Off to Normal. You can enable up to 12 markers.

4. Select the FFT channel that you want to measure with the selected marker.
5. 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. The **Markers** table also displays the current level, mode, and source channel of the markers.

Identify and Compute Peaks

To use the peak search functions to locate peaks, you must first enable at least one FFT channel and one marker.

The controls for the peak search functions () are on the left side of the **Frequency** tab.

Peaks are samples for which the amplitude rises and falls around a threshold. You can move a selected marker to the following locations:

- The highest peak
- The next highest peak
- A peak that is directly adjacent to the marker (both left or right)

Compute Peaks Using Peak Threshold and Peak Excursion

You can configure peak threshold and peak excursion settings by selecting the ⚙ icon in the **Frequency** tab. The ⚙ icon is directly to the right of the peak search functions ().

Peak threshold defines the minimum amplitude level that a sample must rise above to be considered a peak. You can set a peak threshold value without setting a peak excursion value. Without peak excursion, InstrumentStudio considers a peak eligible for every amplitude measurement that is above the peak threshold value.

Peak excursion defines the minimum amplitude variation that is required in a signal to be considered a peak. Peak excursion is always specified in relation to a peak threshold value.

When you define peak excursion, InstrumentStudio considers a peak eligible if the peak meets all of the following conditions:

- The signal rises above the threshold level
- The rise of the signal above the peak threshold matches or exceeds the peak excursion value
- The fall of the signal from above the peak threshold matches or exceeds the peak excursion value
- The signal falls below the threshold level

The following figure and the following table show how to identify eligible peaks:

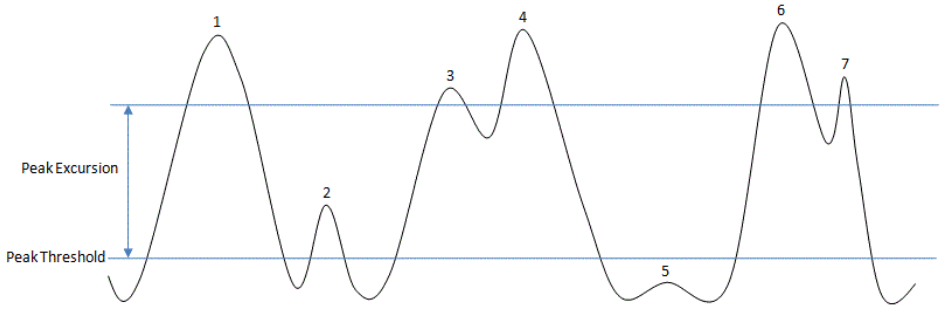


Table 13. Peak Eligibility Examples

Peak	Peak Eligibility	Peak Characteristics
Peak 1	✓ Peak 1 meets the excursion condition and is an eligible peak.	• Rises above the peak threshold • The rise above the peak threshold matches or exceeds the peak excursion value • The fall from above the peak threshold matches or exceeds the peak excursion value • Falls below the threshold level

Peak	Peak Eligibility	Peak Characteristics
Peak 2	⦸ Peak 2 does not meet the excursion condition and is not an eligible peak.	• Rises above the peak threshold • ⦸ Does not rise above the peak threshold by at least the peak excursion value
Peak 3	⦸ Peak 3 does not meet the excursion condition and is not an eligible peak.	• Rises above the peak threshold • The rise above the peak threshold matches or exceeds the peak excursion value • ⦸ The fall from above the peak threshold does not exceed the peak excursion value • ⦸ The signal rises above the amplitude level of Peak 3
Peak 4	✓ Peak 4 meets the excursion condition and is an eligible peak.	• Rises above the peak threshold • During the rise, Peak 4 slightly falls, which happens after Peak 3 • However, the rise above the peak threshold matches or exceeds the peak excursion value • The fall from above the peak threshold matches or exceeds the peak excursion value • Falls below the threshold level
Peak 5	⦸ Peak 5 is not detected as a peak.	⦸ Does not rise above the threshold level

Peak	Peak Eligibility	Peak Characteristics
Peak 6	✓ Peak 6 meets the excursion condition and is an eligible peak.	• Rises above the peak threshold • The rise above the peak threshold matches or exceeds the peak excursion value • During the fall, the signal slightly rises, which appears as Peak 7 • However, the signal does not rise above the amplitude level of Peak 6 before the signal falls again • The total fall, which starts at the amplitude level of Peak 6, exceeds the peak excursion value • Falls below the peak threshold
Peak 7	⦸ Peak 7 does not meet the excursion condition and is not an eligible peak.	• Falls below the peak threshold • The fall from above the peak threshold matches or exceeds the peak excursion value • ⦸ The rise above the peak threshold does not match or exceed the peak excursion value

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.

To configure FFT averaging modes, click the ··· icon for the FFT channel, which opens

the **Software Channel Settings** window.

You can choose one of the following FFT averaging modes:

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

Measure Power Spectral Density

Power spectral density measures the power content compared to the frequency of a signal. You can view the power spectral density of a signal when you analyze data in the frequency domain. Power spectral density helps you determine which frequency ranges have strong or weak variations in power.

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 below the time chart of the large panel.
3. Configure the **FFT Axis** settings in the following way:
   a. In the header of the frequency chart, select the **Chart Options** button.
   b. In the **Y-axis** section, set **Units** to V/√(Hz) or dBm/Hz.
   c. In the **X-axis** section, set **Scaling** to Logarithmic.
4. Run the panel if it does not already run.
   InstrumentStudio plots the power spectral density data on the **Frequency** chart.

For better frequency resolution, you can configure averaging through the FFT channel settings. You can also modify the resolution bandwidth through the axis settings in **Chart Options**.

Measurement Types

InstrumentStudio features the following measurement types when you create oscilloscope channels:

Table 1. Oscilloscope Measurement Types

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

Amplitude

The High measurement minus the Low measurement.

Peak-to-Peak

The Maximum measurement minus the Minimum measurement.

High

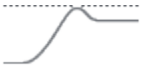
The High measurement is calculated using the selected High-Low Method. If you select the Histogram High-Low method, the High measurement is calculated using the most common value found in the upper 40% of the waveform. If you select the Peak High-

Low method, the High measurement is calculated using the Maximum measurement value of the waveform.

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 Minimum measurement value of the waveform.

Maximum



The maximum value found in the waveform.

Minimum

The minimum value found in the waveform.

High Ref Level

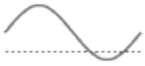
The level of the signal at the high reference level. If you set the reference level unit to Percentage, the High Ref Level measurement is calculated with the selected High-Low method. If you set the reference level unit to Absolute, the value that you set as the High reference level is used for the measurement.

Mid Ref Level

The level of the signal at the mid reference level. If you set the reference level unit to

Percentage, the Mid Ref Level measurement is calculated with the selected High-Low method. If you set the reference level unit to Absolute, the value that you set as the Mid reference level is used for the measurement.

Low Ref Level



The level of the signal at the low reference level. If you set the reference level unit to Percentage, the Low Ref Level measurement is calculated with the selected High-Low method. If you set the reference level unit to Absolute, the value that you set as the Low reference level is used for the 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:

$$\text{RMS} = \sqrt{\left[\sum\left(\frac{\text{Waveform}_i}{\text{Number of points}}\right)^2\right]}$$

Mean

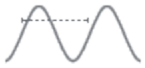
The mean over the entire waveform.

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	$\text{Waveform}\left(\frac{(n-1)}{2}\right)$
Even	$\frac{\left(\text{Waveform}\left(\frac{m}{2}\right) + \text{Waveform}\left(\frac{n}{2+1}\right)\right)}{2}$

Cycle RMS



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

$$\text{Cycle RMS} = \sqrt{\left[\sum\left(\frac{\text{Waveform}_i}{\text{Number of points}}\right)^2\right]}$$

, 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. The High or Low Reference Levels are used to apply a hysteresis.

Frequency

1.0 divided by the Period measurement, in Hz.

Positive Duty Cycle

The Positive Pulse Width divided by the Period times 100.

$$\text{Positive Duty Cycle} = \left(\frac{\text{Positive Pulse Width}}{\text{Period}}\right) x100$$

Negative Duty Cycle

The Negative Pulse Width divided by the Period times 100.

$$\text{Negative Duty Cycle} = \left(\frac{\text{Negative Pulse Width}}{\text{Period}}\right) x100$$

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.

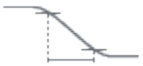
Negative Pulse Width

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

A time span that begins when the first rising edge of the waveform crosses the Low Reference Level. The time span lasts until the same rising edge crosses the High Reference Level. The time span starts from the Low Reference Level crossing that immediately precedes the High Reference Level crossing.

Fall Time



A time span that begins when the first falling edge of the waveform crosses the High Reference Level. The time span lasts until 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.

$$\text{Rise Rate} = \frac{\text{Rise Time}}{(\text{High Reference Level} - \text{Low Reference Level})}$$

Fall Rate

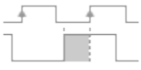
The Fall Time divided by the High Reference Level minus the Low Reference Level.

$$\text{Fall Rate} = \frac{\text{Fall Time}}{(\text{High Reference Level} - \text{Low Reference Level})}$$

Delta Time

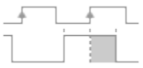
A time span that begins when the first edge of the waveform crosses the Mid Reference Level. The time span lasts until the second edge of the waveform crosses the Mid Reference Level. The second edge does not have to be on the same channel as the first edge.

Setup Time



A time span that begins when the waveform crosses the Mid Reference Level. The time span lasts until the configured clock channel crosses the Mid Reference Level. The Setup Time measurement uses the crossing of the clock channel that is the closest to the middle of the graph.

Hold Time



A time span that begins when the configured clock channel crosses the Mid Reference Level. The time span lasts until the waveform crosses the Mid Reference Level. The Hold Time measurement uses the crossing of the clock channel that is the closest to the middle of the graph.

Crosspoint Level

The level at which two waveforms intersect. The second waveform does not have to be on the same channel as the first waveform. The Crosspoint Level measurement uses the intersection that is the closest to the middle of the graph.

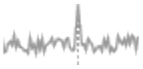
Crosspoint Time

The time at which two waveforms intersect. The second waveform does not have to be on the same channel as the first waveform. The crosspoint time measurement uses the intersection that is the 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 that are close to the DC bin, but not in the DC bin, are still 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 that are close to the DC bin, but not in the DC bin, are still detected.

Customize Oscilloscope Measurements

You can use Reference Levels and Gating to customize the measurements displayed in the measurement table.

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

Gating

Use gating options to select the region of captured data that is used to perform measurements. This region of data appears in the measurement table.

- **Screen**—Performs measurements only on the data that is visible in the time domain graph of the large panel.
- **Record**—Performs measurements on the full data record.
- **Custom**—Performs measurements only on the data between set positions. You can configure custom gating positions with time values that are relative to the trigger of the data capture starting point.
  - **Position 1**—Sets the first position of the gated data relative to the trigger to calculate measurements.
  - **Position 2**—Sets the second position of the gated data relative to the trigger to calculate measurements.

Reference Levels

You can customize the oscilloscope measurements that appear on the measurement table. Adjust 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 a specified amplitude. You can use reference levels to do the following tasks:

- Ascertain rise and fall times
- Calculate waveform cycles
- Customize certain oscilloscope measurement types

In InstrumentStudio, you must set reference levels by absolute levels or by the percentage of the waveform that you want to acquire. The following oscilloscope measurement types are calculated using reference levels:

Table 1. Oscilloscope Measurement Types Calculated with Reference Levels

- High Ref Level
- Mid Ref Level
- Low Ref Level
- 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 are interpreted as percentages of the waveform or as absolute levels.

- **Percentage**—Interprets the reference levels as a percentage using the selected High-Low method.
- **Absolute**—Interprets the reference levels as absolute levels, such as volts.

High-Low Method

If you set the **Reference Level Unit** setting to **Percentage**, you can configure how the state levels are computed by selecting the High-Low method. The following configuration options are available:

- **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 one of the following waves:
  - A square wave, by ignoring the overshoot and the undershoot

  - 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.
- **Peak**—Searches the waveform for its maximum and minimum levels.

Related reference:

- Managing Device Configurations in InstrumentStudio

Network Analyzer Settings in InstrumentStudio

Measurement and calibration settings for Network Analyzer instruments.

Note InstrumentStudio supports the RFmx VNA personality for network analyzer instruments.

Characterize RF Devices

Use the network analyzer panel to perform measurements on and to characterize RF devices.

1. Create a network analyzer panel by clicking **New (** + ▾ **)** » **Network Analyzer**.
2. Configure your instrument. Click the ··· icon to open the **Network Analyzer Settings** window.

Note

If you did not do so previously, calibrate your network analyzer instrument. Uncalibrated instruments might generate unexpected and inaccurate results when you take measurements. To calibrate your instrument, complete the following steps:

a. Open the **Network Analyzer Settings** window.
b. Select the **Calibration** tab.
c. Choose a calibration method.
d. Apply the corrections. For more information, see Apply Corrections to Network Analyzers.

3. Connect your DUT if it is not already connected.
4. Run the measurement by selecting one of the following options:
   - Click **RUN** to measure continuously.
   - Click **SINGLE** to capture a single measurement.

Apply Corrections to Network Analyzers

To ensure accurate measurements, you must apply the corrections that the calibration process generates.

Before you begin, ensure the following:

- You already calibrated your device.
- You saved a calset (`.ncst`) file.

1. Click the **+** button next to the **Active Calset** setting.
2. Select the calibration data file that corresponds to your current configuration.
3. Click **Open**.
4. Ensure that the **Correction** setting is set to ON.

Adjustments are now automatically applied when you use this instrument.
Related information:

- PXIe-5842 Self-Calibration

Port Extension

When you add a switch, a fixture, or a cable to your Network Analyzer instrument after calibration, you introduce loss and additional electrical delay. The loss and the delay shift the measurement reference plane away from the DUT. Port extension compensates for these factors by restoring the reference plane to the correct location without requiring a new calibration. InstrumentStudio supports manual and automatic port extension.

To configure port extension, complete the following steps:

1. Click the ··· button to open the **Network Analyzer Settings** window.
2. Go to the **Correction** tab of the window.

3. In the **Port Extension** section, check the **Enabled** checkbox.

Automatic Port Extension

The automatic port extension feature automatically performs calculations related to port extension, such as delay, electrical length, and loss. You can use automatic port extension even when the added components have unknown electrical delay or electrical characteristics. To determine the appropriate delay correction, InstrumentStudio analyzes the measured response of a known open or short termination.

Perform Measurements with Automatic Port Extension

Complete the following steps to perform an open or short measurement with the automatic port extension feature:

Ensure that you have InstrumentStudio 2026 Q2 or later installed.

1. Click the ··· button to open the **Network Analyzer Settings** window.
2. Go to the **Correction** tab of the window.
3. In the **Port Extension** section, check the **Enabled** checkbox.
4. Click **Auto Port Extension** to open the **Auto Port Extension** window.
5. In the **Select Ports** dropdown menu, select the ports that you want to measure.
6. **Optional:** Configure the following settings:
   - Enable or disable loss measurements
   - Apply regularization to loss measurements
   - Change frequency modes
7. Click **Measure Open** or **Measure Short**.
   Another **Auto Port Extension** window appears.
8. Follow the instructions of the **Auto Port Extension** window.

InstrumentStudio measures the open and short standards for the selected ports. The results appear in the corresponding fields for each port in the **Network Analyzer Settings** window.

RF Signal Analyzer Settings in InstrumentStudio

Signal analysis, measurement, and calibration settings for RF Signal Analyzer

instruments.

Analyzing RF Signals with an RF Signal Analyzer

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

1. Create an RF signal analyzer panel by clicking **New (** + ▾ **)** » **RF Signal Analyzer**.
   The default spectrum analyzer view appears in the instrument panel.
2. Click the **Personalities** button at the top of the panel to select a personality for the measurement.

   > **Note** If a personality does not appear in the menu, ensure that InstrumentStudio supports the personality. Also ensure that the driver for that personality is installed.

   The instrument panel and highlighted settings in the instrument configuration panel change based on selected personality.
3. Select the desired measurements from the settings panel using the **ADD/REMOVE** button.
4. Configure the signal directly through the panel or through the ***<Personality>*** **Settings** window.
   Click the ··· icon on the panel to access the ***<Personality>*** **Settings** window.
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

- Pulse
- Demod

De-Embedding RFSA Devices

De-embedding is 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.

To access de-embedding properties, complete the following steps:

1. Click the ··· icon to open the ***<Personality>* Settings** window.
2. On the **Instrument** tab, scroll down to the **De-embedding** section.

You can configure the following options to apply de-embedding to your RF device:

- **Enabled**—Check the box to enable de-embedding.
- **Status**—Displays whether an `.s2p` table is loaded into the device session.
- **Filename**—Loads an `.s2p` file into the device session to use for de-embedding.

Perform 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. Self-calibration also ensures that the device operates within the ranges that appear in your test specifications.

Perform Self-Calibration

Complete the following steps to perform self-calibration:

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

> **Note** InstrumentStudio warns you that the process might take several minutes to finish. InstrumentStudio also locks the instrument until the

process completes.

3. Click **Continue** to proceed.

Perform 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.

Complete the following steps to perform self-calibration with user-defined ranges:

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

Check Calibration History

Complete the following steps to check the calibration history:

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

Import Bitfile Data

Bitfiles define FPGA configurations, which allow you to customize your RF device for specific testing applications. Importing a bitfile to an InstrumentStudio RFSG or RFSA instrument loads the specified configuration on the device. The loading process also sets the file as the default file to associate with the device model in InstrumentStudio.

Note Your device must have a user-accessible FPGA to load custom bitfiles. For information on whether your device has user-accessible FPGA, see the user manual, getting started guide, or specifications document for your device.

Complete the following steps to import an existing bitfile for your RF device:

1. In the device panel, open the instrument settings menu (⚙).
2. Click **Bitfile**.

> **Note** The bitfile must be in the following folder:
>
> `User\Public\Documents\National Instruments\FPGA Extensions Bitfiles\target name`

3. Select the bitfile that you want to use and click **OK**.
   InstrumentStudio prompts you to restart the application. You must restart InstrumentStudio for the changes to take effect.

InstrumentStudio now uses the selected bitfile as the default file for both RFSG and RFSA instruments for this device model.

Related information:

- NI-RFSA User Manual
- NI-RFSG User Manual

RF Signal Generator Settings in InstrumentStudio

Signal generation and calibration settings for RF Signal Generator instruments.

Generate 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 by clicking **New (** + ▾ **)** » **RF Signal Generator**.
2. Select a mode of operation with the dropdown menu on the device panel.
   - **Continuous Waveform (CW)**—Configures the RF signal generator to output a continuous waveform at the specified frequency and the specified level.
   - **Arbitrary Waveform (Arb)**—Configures the RF signal generator to output a

complex signal according to the waveform and the waveform settings that are specified in an external file.
    - **Script**—Configures an RF signal generator to output a sequence of multiple waveforms according to the instructions specified in the script text. For more information on scripting, see *NI-RFSG User Manual* in the *Related Information* section.
3. Specify the type of signal to generate.
    - For a CW signal, specify the signal settings by opening the **Instrument Settings** (⋯) window.
    - For an Arb signal, click the **+** button in the **Selected Waveform** section. Then browse for and select a waveform file. You can then specify the signal settings. Open the **Instrument Settings** (⋯) window and go to the **Arb waveform** section.
4. Select **Run** and set the output to ON. The signal generates with the settings that you specify.

> **Note**
>
> - In CW mode, the panel starts running immediately. To output a generated waveform, set the output switch on the device panel to **ON**.
> - In Arb mode, you must first select a waveform, then press **Run**.
> - In Arb mode, the panel continues to run and generate the waveform unless you set the **Waveform repetition** mode to Finite. You can configure this setting in the **Instrument Settings** (⋯) window. When you set this mode to Finite, the panel outputs the specified number of repetitions and then stop.

Related concepts:

- Layout Management in InstrumentStudio

Related information:

- NI-RFSG User Manual

Arbitrary Waveform Properties

Settings and values associated with waveforms generated in **Arb** mode.

When you change a waveform, the waveform property values update to reflect the settings for that waveform. When you download a waveform from a file, the property values automatically populate with the values that are in the file. Changes that you make to the waveform properties only apply to the selected waveform.

Debug Test Programs

The NI-RFSG 20.7 driver introduces waveform properties. Prior to the 20.7 release, not all waveform properties were available. These properties also apply globally to all waveforms. When you debug a test program that uses global properties, the global values appear for the selected waveform. However, changes that you make to the waveform properties in InstrumentStudio only apply to the selected waveform.

The following table shows legacy global property names and the updated waveform-specific 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

Waveform Properties in Script Mode

Settings and values associated with waveforms generated in **Script** mode.

When operating in **Script** mode, the following sets of properties are present:

- The properties under the **Script** section correspond to the global properties that are used for the selected script.
- The properties under the **Waveforms** section correspond to the waveform properties and show the values for the selected waveform.

Waveform properties take precedence over the global properties.

Perform 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. Self-calibration also ensures that the device operates within the ranges that appear in your test specifications.

Perform Self-Calibration

Complete the following steps to perform self-calibration:

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

   > **Note** InstrumentStudio warns you that the process might take several minutes to finish. InstrumentStudio also locks the instrument until the process completes.

3. Click **Continue** to proceed.

Perform 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.

Complete the following steps to perform self-calibration with user-defined ranges:

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

Check Calibration History

Complete the following steps to check the calibration history:

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

RF Signal Generator Status

You can see the current generation status of the RF signal generator with the generation status icon. The generation status icon is at the top of the panel.

Table 14. Generation Status of the RF Signal Generator

Icon	Status	Description
▶	Generating output	Signal generation is running on this channel and the output appears in the device panel. This status might also mean that the channel is waiting for a trigger.
■	Not generating output	Signal generation is stopped on this channel.
🔗	Unknown status	The signal generation status is unknown. Run the panel to refresh the output status.

Note The RF signal generator only outputs the waveform if the following conditions apply:

- Signal generation is running
- Output is set to ON

Output generation stops when you close an RF signal generator panel, except when you monitor an external session.

De-Embedding RFSG Devices

De-embedding is 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.

To access de-embedding properties, complete the following steps:

1. Click the ··· icon to open the **Instrument Settings** window.
2. On the **RF** tab, scroll down to the **De-embedding** section.

You can configure the following options to apply de-embedding to your RF device:

- **Type**—Specifies the type of de-embedding you want to use. The following de-embedding methods are available:
  - **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.
- **Filename**—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 you use multi-port modules.

SMU and VPS Device Settings in InstrumentStudio

Source and measurement settings for SMU and VPS devices.

Modes of Operation

When you use an SMU or a VPS device, you can use different device operating modes to acquire measurements.

Device Operating Modes

The following table describes the behaviors and uses of these device operating modes:

Operating Mode	Sourcing Behavior	Large Panel Plotting Behavior	Pause Behavior	Potential Uses	Notes
Charting ()	Allows one or more channels to source a supported signal.	Appends samples to the chart as the samples are measured.	Channels continue to source at the last configured level when you pause measurements.	Observe signals over an infinite duration.	–
Waveform ()	Allows one or more channels to source a supported signal.	Plots samples to the graph all at once. Capture time determines the length of the graph.	Channels continue to source at the last configured level when you pause measurements.	Observe signals over a shorter, finite duration, similar to an oscilloscope.	InstrumentStudio automatically configures the device for a faster sample rate. InstrumentStudio also runs the output continuously in a loop. To change this behavior, click **SINGLE**.
Single-Channel Sweep () Two-Chanel	Sweeps through configured steps for a single channel (Single-Channel Sweep) or two	Plots all sample step results on the graph as the step results are measured.	Both measurement and output channels stop running when you pause measurements.	Allows you to plot the relationship between current and voltage in an I-V curve.	When you use Two-Channel Sweep, the sweep acts as a nested loop. In this nested loop, the inner channel executes all the steps for each

Operating Mode	Sourcing Behavior	Large Panel Plotting Behavior	Pause Behavior	Potential Uses	Notes
Sweep ( )	channels (Two-Channel Sweep).				step that is configured in the outer channel.
Sweep (LCR mode)	Sweeps through configured frequency or DC bias steps for a single channel (Single-Channel Sweep).	Plots all sample step results on the graph as the step results are measured.	Both measurement and output channels stop running when you pause measurements.	Allows you to plot the relationship between frequency or DC bias and enables LCR readbacks.	–

Note The Waveform and Sweep modes are not available on the following devices:

- PXI-4110
- PXIe-4112
- PXIe-4113
- PXI-4130
- PXI-4132

Note In debug mode, the Waveform mode is only available for single-channel sessions. You cannot use Waveform mode when a session has two channels.

Note **Sweep** modes are available on the PXIe-4190 device when operating in LCR mode. To change to LCR mode, complete the following steps:

1. Click the ··· icon in the device panel to open the **Channel Settings**

window.
2. Click the **Mode** dropdown menu.
3. Select the **LCR meter** option.

Channel Operating Modes

In addition to device operating modes, you can also specify an operating mode for each channel.

To change the operating mode of a channel, use the dropdown menu in the **Channels** section of the panel. A channel operating mode only appears in the dropdown menu if your current device supports that operating mode.

When you change the operating mode of a channel, InstrumentStudio pauses all panel measurements. All enabled outputs continue to generate signal at the last configured level. The updated operating mode of the channel does not take effect until the panel starts running again.

You can configure the following channel operation modes:

- **Idle**—The channel is not programmed. You can change a channel to **Idle** from another mode. In this case, the channel continues to generate a 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.
- **Power or Resistance**—Sources a constant power or a constant resistance at a single configured level, up to a current limit, while the output is enabled. Applies to electronic loads.

  Note For electronic loads, references in documentation and software to *sourcing* refer to *sinking*. While references to *output* refer to *input*. InstrumentStudio classifies an electronic load as an SMU/power supply.

- **Pulse voltage or Pulse current**—Generates a single pulse at the pulse level, then returns to the bias level.

> **Note** 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 sequences or current sequences, see Configure Sequences.

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

Automatic Level Range

Automatic level range automatically determines the signal range when the range for a signal is unknown.

When a measured signal exceeds the manually specified range in InstrumentStudio, InstrumentStudio cannot accurately measure that signal. InstrumentStudio warns you that the measurement is overrange.

To avoid overranging, set the **Voltage level range** or the **Current level range** option to Auto. This setting allows InstrumentStudio to use incoming signal data to automatically set level ranges, which increases the accuracy of measurements.

Enable Automatic Level Range

1. Click ··· to open the **Channel Settings** window.

2. Select Auto in the dropdown menu of the **Voltage/Current level range** option.

To disable autorange, select Manual in the dropdown menu of the **Voltage/Current level range** option.

> **Note** Depending on the operating mode of the channel, the name of the option to set the level range changes accordingly. The name of the option in Voltage operating mode is **Voltage level range**. The name of the option in Current operating mode is **Current level range**.

Configure Automatic Level Range

To configure how automatic level ranging responds to incoming signals, click ··· to open the **Channel Settings** dialog box. Then, you can configure the following settings:

- **Aperture time mode**—Specifies how InstrumentStudio determines aperture time for the automatic level range. You can set **Aperture time mode** to Auto or Manual.
- **Aperture time unit**—Specifies the units of the **Aperture time** setting. You can change this setting to Seconds or Power Line Cycles (PLCs). You can only change this setting if you set **Aperture time mode** to Manual.
- **Aperture time**—Specifies the aperture time that the automatic level range algorithm uses. You can only configure this value if you set **Aperture time mode** to Manual.

  > **Note** Aperture time is a minimum value. InstrumentStudio scales up this value to optimize the algorithm for different ranges. For smaller ranges, InstrumentStudio scales up the value to account for signal noise.

- **Autorange behavior**—Specifies the algorithm that the hardware uses for measurement autoranging. You can choose from the following behaviors:
  - **Range up to limit then down**—Go to range limit then range down until the measured value is within the threshold.
  - **Range up**—Go up one range when the signal reaches the upper threshold.
  - **Range up and down**—Go up or down one range when the signal reaches the upper threshold or the lower threshold.
- **Minimum current range**—Specifies the lowest range that InstrumentStudio uses when you enable automatic level range.

  > **Note** Limiting the lowest range might improve the speed of autoranging.

> Limiting the lowest range also minimizes thrashing between ranges when you measure a noisy signal.

Configure Channel Sessions

You can change the session type to Multichannel or Single channel (compatibility). The default setting is Multichannel. InstrumentStudio version 202Q1 and newer have different default behavior.

NI-DCPower Independent Channel Sessions

InstrumentStudio versions 2020 Q1 and newer export a singular session configuration for all channels by default. However, using NI-DCPower independent channel sessions allows you to configure multiple channels within the same session independently. The channels within the session can belong to the same instrument or to multiple instruments.

Considerations When Configuring Sessions with SMU Devices

By default, InstrumentStudio creates a singular multichannel session for all devices in a single SMU device. To change session types, open the instrument settings menu (⚙) of the device in your panel. Then, hover your cursor over **Session Type** and select the session type. You can set the session type to Multichannel or Single channel (compatibility). The default setting is Multichannel.

In certain cases, you must change session types. You must change the **Session type** to Single channel (compatibility) in the following cases:

- Exporting a session to a platform that does not support NI-DCPower independent channel sessions. These platforms include, for example, the following:
  - Python
  - LabWindows/CVI
  - LabVIEW NXG
  - Linux desktop
- Exporting a session to NI-DCPower version 20.5 and earlier.
- Exporting the session to end-user code that uses deprecated NI-DCPower Initialize functions in C/LabVIEW.

- Exporting the session to end-user code that uses deprecated constructors in .NET.

Note Exported sessions that are created in InstrumentStudio with independent channels cannot be imported into sessions that are created with single channels. To verify your session channel configuration, select the instrument settings menu (⚙) and hover your cursor over **Session type**.

SMU/VPS Output Status

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

Table 15. Output Status of the SMU/Power Supply Channel

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 to start an operation until the trigger condition occurs.
	Unknown status	The output status of the channel in the panel is unknown. Run the panel to refresh the output status.

Note Output continues when closing a panel.

Configure 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 SourceAdapt technology, see *NI SourceAdapt Next-Generation SMU Technology* in the *Related Information* section.

Note Not all SMUs support SourceAdapt technology. Refer to the documentation for your SMU to see whether the SMU supports SourceAdapt.

1. Place a SourceAdapt-compatible SMU in the large panel.
2. Set the device operating mode of the SMU panel to Waveform ( ).
3. Click ··· to open the **Channel Settings** window.
4. Scroll down to the **SourceAdapt tuning parameters** section of the **Channel Settings** window.
5. Select one of the following SourceAdapt presets:
   - 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 by configuring the fields in the **SourceAdapt tuning parameters** section.

Related information:

- NI SourceAdapt Next-Generation SMU Technology

Configure Sequences

Power sequencing on an SMU/Power Supply allows you to output the desired power-up sequence for a DUT with multiple power sources.

Power sequencing is only available if you set the operating mode of the channel to Voltage Sequence or Current Sequence. You can configure the power-up sequence by defining the following characteristics of each power supply channel:

- Power-up order
- Power-up timing
- Voltage or current level

Note

- Within the same panel, channels that operate in Voltage Sequence, Current Sequence, Pulse Voltage, or Pulse Current mode are automatically hardware-synchronized. For more information on hardware and software synchronization within panels, see *Synchronize Devices*.
- Sequence channel operating modes are only available on power supplies that support advanced sequencing. For more information on advanced sequencing and supported devices, see *Advanced Sequencing* in the *Related Information* section.

1. Place an SMU or a power supply in a panel.
   If you place the device in a large panel, the graph shows a preview of the output sequence that the channel will generate.
2. Set the device operating mode to Waveform ( ) or Charting ( ).
   Setting the device operating mode to Waveform ( ) allows for fast sampling. Waveform mode also 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 and the **Channel Settings** dialog box.
4. Edit the steps in the step table.
   You can edit the steps in the following ways:
   - Enter values manually
   - Import data from a file
   - Paste data from the clipboard
5. **Optional:** Tune the output of the sequence with SourceAdapt parameters in the **Channel Settings** window.
6. Enable the channel output and run the panel to begin the sequence.

Related concepts:

- Device Synchronization in InstrumentStudio

Related reference:

- Managing Device Configurations in InstrumentStudio

Related information:

- Advanced Sequencing

Waveform Generator Settings in InstrumentStudio

Waveform generation and signal configuration settings for Waveform Generator instruments.

Use the Waveform Generator panel to generate waveforms and configure your Waveform Generator instruments.

Generate 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 one of the following waveform modes from the dropdown menu of the device panel:
   - Standard waveform—Generates a waveform using the channel settings that you specify.
   - Arbitrary waveform—Generates a waveform using settings specified in a waveform file and any additional channel settings that you specify.

   > **Note** For more information on waveform modes, see Waveform Modes.

3. Specify the type of the waveform.
   - For a standard waveform, you can select the waveform type with the **Waveform** dropdown menu.

   > **Note** If you select a User-defined waveform, you must load a waveform file. User-defined waveform files must be in the `.csv` format, with the exact number of samples that the *Specifications* document of your device defines. For example, the PXIe-5433 requires 8192 samples for a user-defined waveform. The waveform samples must be in a single row or a single column.

- For an arbitrary waveform, you must load a waveform file. Click the **...** button next to the **Filename** field to browse for the waveform file. The waveform file specifies the type of waveform to generate.

  Note Arbitrary waveform files can be in the `.csv` format or a binary format:
  - **`.csv`**—For single-channel instruments, waveform samples must be in a single row or a single column. For two-channel instruments, the waveform samples for each channel must be in a separate column.
  - **Binary**—For single-channel instruments, waveform samples can be 8-byte double-prevision or 2-byte half-precision floating point. The byte order can be little-endian or big-endian. For two-channel instruments, the samples of both channels must be interleaved.

4. Configure the waveform channel settings directly on the large panel or in the **Channel Settings** window. To open the **Channel Settings** window, click on the ··· icon next to the channel. The settings that you can configure change based on the waveform mode and waveform type that you select.

   Note If you generate an arbitrary waveform, you must select a trigger mode for the waveform. To select a trigger mode, complete the following steps:
   a. Click on the ··· icon to open the **Channel Settings** window.
   b. Scroll down to the **Triggers** section.
   c. Use the dropdown menu of the **Mode** setting.

   For more information on arbitrary waveform trigger modes, see *Advanced Waveform Sequencing and Triggering on Arbitrary Waveform Generators* in the *Related Information* section.

5. Select **Run**.

The waveform generates with the specified settings.

Related information:

- Advanced Waveform Sequencing and Triggering on Arbitrary Waveform Generators

Waveform Modes

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

- **Standard waveform**—A standard waveform has the following characteristics:
  - Outputs waveforms according to the channel settings that you specify in InstrumentStudio or an external application.
  - Allows you to generate different waveform types (for example, sine, square, or triangle) at precise frequencies.
  - Allows you to change the output waveform frequency of a standard waveform with a short response time during waveform generation.
  - The Sweep operation mode allows you to configure a range of frequencies to generate over a specified number of steps and a specified time duration.
  - The List operation mode allows you to configure a sequence of frequencies to generate for specified durations.

  > **Note** The User-defined standard waveform type allows you to generate a periodic waveform with a finite, specified number of points. The waveform generation uses Direct Digital Synthesis (DDS). To generate a user-defined waveform of any size, use Arbitrary Waveform mode.

- **Arbitrary waveform**—Outputs waveforms according to settings that are specified in an external file. An arbitrary waveform allows you to define large, complex waveforms with 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.

Configure Waveform Channels and Triggers

You can configure channels, triggers, and other parameters of the waveform generator from the **Channel Settings** window.

Click the ··· icon to open the **Channel Settings** window. You can select the channel that you want to configure on the left side of the window.

Waveform Channel Settings

You can configure the following settings in the *devicename/channel_number* section

of the **Channel Settings** window:

- **Output Impedance**—Sets the output impedance value to 50 Ω or 75 Ω.
- **Load Impedance**—Sets load impedance to one of the following values:
  - **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. InstrumentStudio applies the analog filter after the interpolated signal. When combined with a digital filter, the combination 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 (either Differential or Single-ended).

Waveform General Settings

The settings in the **Waveform** section of the **Channel Settings** window correspond to the controls on the panel. The settings that appear in this section depend on the waveform mode and the type of the standard waveform.

Waveform Trigger Settings

You can configure the following settings in the **Triggers** section of the **Channel Settings** window:

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

> **Note** For more information on arbitrary waveform trigger modes, see *Advanced Waveform Sequencing and Triggering on Arbitrary Waveform Generators* in the *Related Information* section.

- **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

The settings in the **Hardware event output terminals** section of the **Channel Settings** window determine the chassis terminal to which InstrumentStudio exports hardware events.

Related information:

- Advanced Waveform Sequencing and Triggering on Arbitrary Waveform Generators

Waveform Generator Status

You can see the current generation status of non-idle channels in the **Channel Settings** window or through the generation status icon. The generation status icon is next to the **Waveform** dropdown menu of a channel.

Table 16. Generation Status of the Waveform Generator

Icon	Status	Description
	Generating output	Waveform generation is running on this channel.
	Not generating output	Waveform generation is stopped on this channel.
	Waiting for trigger	The channel is waiting to start an operation until the trigger condition occurs.
	Unknown status	The output status of the channel is unknown. Run the panel to refresh the output status.

Note Waveform generation stops when you close the waveform generator

panel.

DAQ Analog Input Device Settings in InstrumentStudio

Acquisition, timing, triggering, and channel settings for DAQ Analog Input instruments.

Introduced in InstrumentStudio 2026 Q1.

Timing Settings

By default, InstrumentStudio manages timing settings automatically based on the trigger settings and the channel settings. You can also configure timing settings manually.

To configure timing settings manually, click the blue icon to the right of the **Mode** dropdown menu. The following sections describe the effect of configuring each setting.

Timing Mode

You can set the **Mode** setting to one of the following options:

- **Continuous**—Continuously acquires data with hardware timing and graphs the most recently acquired waveform.
- **Finite**—Performs a single data acquisition with hardware timing and graphs the acquired waveform.
- **Software**—Continuously acquires data with software timing and charts the most recently acquired sample.

Note

In Continuous mode, InstrumentStudio might skip samples to match the rate of the hardware acquisition. The channel statistics in the bottom pane do not consider skipped samples.

Sample Rate

You can only configure the **Sample rate** setting in the Continuous and the Finite timing modes.

The **Sample rate** setting defines the desired sample clock rate, which sets the time interval between samples.

The sample rate that you set must match one of the supported rates for your DAQ device. If you set the sample rate to an unsupported value, NI-DAQmx automatically changes the sample rate to a supported value. The **Actual sample rate** field shows the changed sample rate value.

Samples to Read

You can only configure the **Samples to read** setting in the Continuous and the Finite timing modes.

The **Samples to read** setting defines the following sampling characteristics:

- In Continuous timing mode, the number of samples that appear on the graph at a time. The combination of the **Sample rate** setting and the **Samples to read** setting controls how often the graph updates.
- In Finite timing mode, the number of samples that InstrumentStudio acquires and graphs. After the acquisition completes, you can zoom in to view parts of the data in more detail.

Trigger Settings

You can configure the mode, the type, and the source of the trigger on the instrument panel.

Trigger Mode

To define which triggers an acquisition responds to, set the **Mode** setting to one of the following options:

- **None**—The default setting. Acquisition starts and stops when you press the **Run** or the **Stop** button. The duration of the acquisition is based on the timing mode.
- **Start**—When you click **Run**, the acquisition waits for a hardware trigger. When the trigger condition is met, the acquisition begins.
- **Reference**—When you click **Run**, InstrumentStudio acquires data continuously and waits for a hardware trigger. When the trigger condition is met, the acquisition completes. The resulting graph displays a finite number of samples before and after the trigger. The trigger establishes a reference point in the data, which appears as a vertical dashed line in the graph. Data acquired up to the reference point is pretrigger data. Data acquired after the reference point is posttrigger data.
- **Start and Reference**—When you click **Run**, the acquisition waits for a start trigger before beginning acquisition. After acquisition begins, the acquisition continues until a reference trigger occurs. The resulting graph displays a finite number of samples before and after the reference trigger.

Trigger Type

To define the type of hardware event for a trigger, set the **Type** setting to one of the following options:

- **Digital Edge**—The trigger occurs when the trigger source transitions according to the requested edge (rising or falling). For more information on configuring digital edge triggers, see Digital Edge Trigger Settings.
- **Analog Edge**—The trigger occurs when the trigger source crosses the requested level with the requested slope (rising or falling). For more information on configuring analog edge triggers, see Analog Edge Trigger Settings.
- **Analog Window**—The trigger occurs when the trigger source enters or leaves a window that is defined by two voltage levels. For more information on configuring analog window triggers, see Analog Window Trigger Settings.

Note Not all DAQ devices support all combinations of trigger mode and trigger type. For information on which combinations your DAQ device supports, see the *Specifications* document for your DAQ device on *ni.com/docs*.

Trigger Source

To specify a terminal as the source terminal of the trigger, use the **Source** setting in

one of the following ways:

- For digital triggers, you can specify a digital terminal on the DAQ device that you are acquiring from. You can also specify a digital terminal that you can route over the PXI backplane.
- For analog triggers, you can specify a channel or an analog terminal on the DAQ device that you are acquiring from.

Trigger Status

Use the trigger status icon to determine the current state of a triggered acquisition. The trigger status icon is located to the right of the **Run** or the **Stop** button. The icon only appears when you set the trigger mode to Start, Reference, or Start and Reference.

Table 17. Trigger Status of the DAQmx Device

Icon	Status	Description
	Triggered	The device met the trigger condition and began an acquisition that is based on the specified trigger settings.
	Stopped	Acquisition is stopped on the device.
	Waiting	The device is waiting to start an acquisition until the trigger condition occurs.

Note In Monitor mode, when a trigger is not configured, the trigger status indicator displays acquisition status rather than trigger status. The indicator displays Stopped or Acquiring instead of Stopped, Waiting, or Triggered.

Reference Trigger Settings

When a reference-triggered acquisition ends, the graph displays a finite number of samples before and after the trigger. You can change the number of displayed samples by setting the values of the **Pretrigger samples** and the **Samples to read** fields.

The **Pretrigger samples** field determines the number of acquired samples that InstrumentStudio displays before the trigger occurs. The *Samples to Read−Pretrigger Samples* formula determines the number of acquired samples that InstrumentStudio displays after the trigger occurs.

Digital Edge Trigger Settings

You can set the **Edge** setting to one of the following options:

- **Rising**—The trigger occurs when the source signal transitions from a low to a high logic level.
- **Falling**—The trigger occurs when the source signal transitions from a high to a low logic level.

Analog Edge Trigger Settings

You can configure the level, slope, and hysteresis values of analog edge triggers.

Level

You can configure the value of the **Level** field to specify the signal level at which the trigger occurs.

When the trigger source is a channel, the trigger level has the same unit of measurement as the channel.

When the trigger source is an analog terminal, the trigger level always has volts as the unit of measurement.

Slope

You can set the **Slope** setting to one of the following options:

- **Rising**—The trigger occurs when the source signal crosses from below the level to above the level.
- **Falling**—The trigger occurs when the source signal crosses from above the level to below the level.

Hysteresis

You can configure the value of the **Hysteresis** field to ensure that signal noise does not cause unwanted triggers.

A rising slope trigger is valid when the source signal passes below `Level-Hysteresis`, then passes above the **Level** value.

A falling slope trigger is valid when the source signal passes above `Level+Hysteresis`, then passes below the **Level** value.

Analog Window Trigger Settings

You can configure the window mode and the window limits of analog window triggers.

Window Mode

You can set the **Window mode** setting to one of the following options:

- **Entering**—The trigger occurs when the source signal passes from outside the window to inside the window.
- **Leaving**—The trigger occurs when the source signal passes from inside the window to outside the window.

Window Limits

The values of the **Bottom** and the **Top** fields specify the two signal levels that define the window.

Channel Settings

You can configure various settings, such as signal types and input limits for DAQ Analog Input devices.

Channel Names

The **Name** setting specifies a display name or alias for the channel. To modify the channel name, click the ··· button to open the **Channel Settings** window.

The physical channel name (for example, `PXI1Slot3/ai0`) appears at the top of the

Channel Settings window.

Signal Types and Sensor Types

The **Signal type** setting determines the type of signal to measure. In certain cases, the **Signal type** setting also defines the type of sensor to measure the signal with. You can modify the signal type in one of the following ways:

- Click the ··· button to open the **Channel Settings** window.
- Select the dropdown menu that is on the right of the channel name.

The dropdown menu groups the signal types around the following measurement categories:

- Electrical
- Temperature
- Bridge
- Integrated Electronics Piezoelectric (IEPE) and Charge
- Position

Note If the channel is set to **Idle**, the channel is disabled and not included in the acquisition.

The following tables describe what each setting measures and, if applicable, the type of sensor to measure the signal with:

Table 18. Electrical

Signal Type	Description
Charge	Measures electrical charge.
Current	Measures electrical current.
Custom voltage with excitation	Measures voltage or bridge ratios with DC voltage excitation.
Resistance	Measures electrical resistance.
Voltage	Measures electrical potential.

Table 19. Temperature

Signal Type	Description
Temperature (RTD)	Measures temperature with a resistance temperature detector (RTD).
Temperature (thermistor)	Measures temperature with a thermistor.
Temperature (thermocouple)	Measures temperature with a thermocouple.

Table 20. Bridge

Signal Type	Description
Force (bridge)	Measures force with a load cell.
Pressure (bridge)	Measures pressure with a bridge-based pressure sensor.
Strain	Measures strain with a strain gauge.
Torque (bridge)	Measures torque with a bridge-based torque sensor.

Table 21. Integrated Electronics Piezoelectric (IEPE) and Charge

Signal Type	Description
Acceleration (charge)	Measures acceleration with a piezeoelectric accelerometer in charge mode.
Acceleration (IEPE)	Measures acceleration with an IEPE accelerometer.
Acceleration (4 wire DC voltage)	Measures acceleration with an accelerometer that uses DC voltage excitation.
Force (IEPE)	Measures dynamic force with an IEPE load cell or impact hammer.
Sound pressure (IEPE)	Measures sound pressure with an IEPE accelerometer.
Velocity (IEPE)	Measures dynamic velocity or vibration with an IEPE velocity transducer.

Table 22. Position

Signal Type	Description
Angular position (RVDT)	Measures angular position with a Rotary Variable Differential Transformer (RVDT).
Linear position (LVDT)	Measures linear position with a Linear Variable Differential Transformer (LVDT).
Proximity (eddy current)	Measures relative proximity with an eddy current proximity probe.

By default, the signal type of the first channel is set to a device-specific default type. The signal type of the remaining channels is set to Idle.

To enable more channels, change their **Signal type** setting to the desired signal type. To disable channels, change their **Signal type** setting to Idle.

To show or hide idle channels, click the button that is in the upper-right corner of the graph. Then check or clear the checkbox under **Show » Idle channels**.

Input Limits, Units, and Hardware Range

You can specify the expected signal range for a channel by setting the values of the **Minimum** and the **Maximum** fields. These values are specified in units that are appropriate for the selected signal type.

For certain signal types, such as acceleration or temperature, you can select different units of measurement through the following steps:

1. Click the ··· button to open the **Channel Settings** window.
2. Select a unit of measurement from the **Units** dropdown menu.

NI-DAQmx chooses the hardware range that best matches the input limits and other sensor settings. The **Hardware range** field displays the chosen range in electrical units such as volts, V/V, or ohms.

Signal-Specific Settings

To configure signal-specific settings, click the ··· button to open the **Channel Settings**

window.

For certain signal types, the front panel displays signal-specific read-only indicators. These indicators show the configuration of the channel.

You can hide input limits, hardware range, and signal-specific indicators through the following steps:

1. Click the button on the right side of the channel pane.
2. Set **View mode** to Compact.

To unhide these characteristics, set **View mode** to Default.

Readback Display

Below the input limits and signal-specific indicators, each channel displays a readback of the sample that the channel most recently acquired.

Configure Multiple Channels

To configure multiple channels with the same settings, complete the following steps:

1. Click the button to open the **Channel Settings** window.

   Note You can configure multiple channels even if you open the **Channel Settings** window by clicking the ··· button. Use `ctrl+click`, `shift+click`, or `ctrl+A` to select multiple channels.

2. Configure the settings that the right side of the **Channel Settings** window displays. The window only displays the settings that the selected channels share.

If a setting has different, non-conflicting values on different channels, the control for that setting displays Mixed values.

If a setting has conflicting values on different channels, the control for that setting displays No common option. Conflicting values are, for example, a **Units** dropdown menu that displays Volt (V) for one channel and Ampere (A) for another channel.

Advanced and Device-Specific Settings

The **Channel Settings** window displays advanced settings, such as **Coupling** and **Terminal configuration**.

For certain devices, the **Channel Settings** window also displays device-specific settings, such as **ADC timing mode** or **Filter frequency**.

DAQ Analog Output Device Settings in InstrumentStudio

Signal generation, timing, triggering, and channel settings for DAQ Analog Output instruments.

Introduced in InstrumentStudio 2026 Q2.

Timing Settings

By default, InstrumentStudio manages timing settings automatically based on the trigger settings and the channel settings. You can also configure timing settings manually.

To configure timing settings manually, click the blue icon to the right of the **Mode** dropdown menu. The following sections describe the effect of configuring each setting.

Timing Mode

You can set the **Mode** setting to one of the following options:

- **Continuous**—Continuously generates a waveform with hardware timing.
- **Finite**—Performs a single waveform generation with hardware timing.
- **Software**—Outputs the specified constant DC voltage or current offsets and charts the current and previous channel outputs.

Sample Rate

You can only configure the **Sample rate** setting in the Continuous and the Finite timing modes.

The **Sample rate** setting defines the desired sample clock rate, which sets the time interval between samples.

The sample rate that you set must match one of the supported rates for your DAQ device. If you set the sample rate to an unsupported value, NI-DAQmx automatically changes the sample rate to a supported value. The **Actual sample rate** field shows the changed sample rate value.

Samples per Channel

You can only configure the **Samples per channel** setting in the Continuous and the Finite timing modes.

The **Samples per channel** setting defines the following sampling characteristics:

- **Continuous**—The number of samples in the output buffer of the DAQ device.
- **Finite**—The total number of samples to generate.

Trigger Settings

You can configure the mode, the type, and the source of the trigger on the instrument panel.

Trigger Mode

To define which triggers the waveform generation responds to, set the **Mode** setting to one of the following options:

- **None**—The default setting. Generation starts and stops when you press the Run or the Stop button. The duration of the generation is based on the timing mode.
- **Start**—When you click Run, the generation waits for a hardware trigger. When the trigger condition is met, the generation begins.

Trigger Type

To define the type of hardware event for a trigger, set the **Type** setting to one of the following options:

- **Digital Edge**—The trigger occurs when the trigger source transitions according to the requested edge (rising or falling). For more information on configuring digital edge triggers, see Digital Edge Trigger Settings.
- **Analog Edge**—The trigger occurs when the trigger source crosses the requested

level with the requested slope (rising or falling). For more information on configuring analog edge triggers, see Analog Edge Trigger Settings.

- **Analog Window**—The trigger occurs when the trigger source enters or leaves a window that is defined by two voltage levels. For more information on configuring analog window triggers, see Analog Window Trigger Settings.

Note Not all DAQ devices support all combinations of trigger mode and trigger type. For information on which combinations your DAQ device supports, see the *Specifications* document for your DAQ device on *ni.com/docs*.

Trigger Source

To specify a terminal as the source terminal of the trigger, use the **Source** setting in one of the following ways:

- For digital triggers, you can specify a digital terminal on the DAQ device that you are acquiring from. You can also specify a digital terminal that you can route over the PXI backplane.
- For analog triggers, you can specify a channel or an analog terminal on the DAQ device that you are acquiring from. DAQ analog output panels do not support triggering on an analog input channel.

Trigger Status

Use the trigger status icon to determine the current state of a triggered generation. The trigger status icon is located to the right of the Run or the Stop button. The icon only appears when you set the trigger mode to Start.

Table 23. Trigger Status of the DAQmx Device

Icon	Status	Description
	Generating	The device met the trigger condition and began a generation that is based on the specified trigger settings.
	Stopped	Generation is stopped on the device.

Icon	Status	Description
	Waiting	The device is waiting to start a generation until the trigger condition occurs.

Digital Edge Trigger Settings

You can set the **Edge** setting to one of the following options:

- **Rising**—The trigger occurs when the source signal transitions from a low to a high logic level.
- **Falling**—The trigger occurs when the source signal transitions from a high to a low logic level.

Analog Edge Trigger Settings

You can configure the level, slope, and hysteresis values of analog edge triggers.

Level

You can configure the value of the **Level** field to specify the signal level at which the trigger occurs.

Slope

You can set the **Slope** setting to one of the following options:

- **Rising**—The trigger occurs when the source signal crosses from below the level to above the level.
- **Falling**—The trigger occurs when the source signal crosses from above the level to below the level.

Hysteresis

You can configure the value of the **Hysteresis** field to ensure that signal noise does not cause unwanted triggers.

A rising slope trigger is valid when the source signal passes below `Level-Hysteresis`, then passes above the **Level** value.

A falling slope trigger is valid when the source signal passes above `Level+Hysteresis`, then passes below the **Level** value.

Analog Window Trigger Settings

You can configure the window mode and the window limits of analog window triggers.

Window Mode

You can set the **Window mode** setting to one of the following options:

- **Entering**—The trigger occurs when the source signal passes from outside the window to inside the window.
- **Leaving**—The trigger occurs when the source signal passes from inside the window to outside the window.

Window Limits

The values of the **Bottom** and the **Top** fields specify the two signal levels that define the window.

Channel Settings

You can configure various settings, such as signal types and output limits for DAQ Analog Output devices.

Channel Names

The **Name** setting specifies a display name or alias for the channel. To modify the channel name, click the ··· button to open the **Channel Settings** window.

The physical channel name (for example, `PXI1Slot3/ao0`) appears at the top of the **Channel Settings** window.

Signal Types

The **Signal type** setting determines the type of signal to generate. You can modify the signal type in one of the following ways:

- Click the ··· button to open the **Channel Settings** window.

- Select the dropdown menu that is on the right of the channel name.

The dropdown menu contains the following signal types:

- **Current** (A)—Outputs electrical current.
- **Voltage** (V)—Outputs electrical potential.

Note If the channel is set to **Idle**, the channel is disabled and not included in the generation.

By default, the signal type of the first channel is set to a device-specific default type. The signal type of the remaining channels is set to Idle.

To enable more channels, change their **Signal type** setting to the desired signal type. To disable channels, change their **Signal type** setting to Idle.

To show or hide idle channels, click the button that is in the upper-right corner of the graph. Then check or clear the checkbox under **Show » Idle channels**.

Output Limits, Units, and Hardware Range

You can specify the expected signal range for a channel by setting the values of the **Minimum** and the **Maximum** fields. These values are specified in units that are appropriate for the selected signal type.

NI-DAQmx chooses the hardware range that best matches the output limits. The **Hardware range** field displays the chosen range in volts or amperes.

Note

All DAQ Analog Output devices keep outputs enabled while the device generates data. As a result, the DAQ Analog Output panel does not include an output enable control.

Some DAQ Analog Output devices support configurable output enable behavior through the NI-DAQmx AO.IdleOutputBehavior property, which controls whether outputs remain enabled after generation stops. The DAQ

Analog Output panel does not support configuring the AO.IdleOutputBehavior property.

Waveform Settings

The **Waveform type** setting determines how the generated signal changes over time. You can set the **Waveform type** to one of the following settings:

- Constant DC (⎓)

  Note Voltage or current, depending on the signal type.

- Sine (∿)
- Square (⎍)
- Triangle (∿)
- Sawtooth (⩘)
- Inverted Sawtooth (⩘)

By default, the waveform type of each channel is set to Constant DC (⎓).

Waveform-Specific Settings

You can configure waveform-specific settings for the different waveform types.

To configure waveform-specific settings, click the ··· button to open the **Channel Settings** window.

For certain waveform types, the front panel displays signal-specific read-only indicators. These indicators show the configuration of the channel.

You can hide output limits, hardware range, and waveform-specific indicators through the following steps:

1. Click the button on the right side of the channel pane.
2. Set **View mode** to Compact.

To unhide these characteristics, set **View mode** to Default.

Constant DC Waveform Settings

You can configure the **Offset** setting to specify the voltage or current that the device outputs.

Standard Waveform Settings

Standard waveforms include the following signal types:

- Sine (∿)
- Square (⎍)
- Triangle (∿)
- Sawtooth (⩘)
- Inverted Sawtooth (⩙)

You can configure the following waveform-specific settings for standard waveforms:

- **Amplitude**—Specifies the peak-to-peak voltage or current of the waveform.
- **Frequency**—Specifies the frequency of the waveform.

> **Note**
>
> InstrumentStudio configures the DAQ device to continuously regenerate data in the output buffer.
>
> Certain timing settings and output buffer sizes might prevent InstrumentStudio from generating the desired frequency. In this case, InstrumentStudio automatically selects the closest possible frequency. You can see this frequency in the **Actual frequency** indicator. If the difference between the desired frequency and the actual frequency exceeds 10%, InstrumentStudio displays an error.

- **Offset**—Specifies the midpoint of the waveform on the Y-axis.
- **Phase**—Specifies the starting phase of the waveform in degrees.

Square Waveform Settings

For Square waveforms (⎍), you can open the **Channel Settings** window (⋯) to configure the **Duty cycle** setting. This setting specifies the percentage of the waveform

period where the square wave outputs high.

Waveform Preview

The **Waveform preview** graph shows a preview of the waveforms that InstrumentStudio generates with the configured timing settings and channel settings. When you change a setting, the preview graph updates. This update occurs whether or not the panel is generating data.

Channel Outputs Chart

When you configure Software timing, the panel displays the **Channel outputs** chart. The chart displays the current and previous channel outputs.

Unlike the **Waveform preview** graph, the **Channel outputs** chart only updates while the panel is running.

When you configure Continuous or Finite timing, the panel displays the **Waveform preview** graph instead.

Configure Multiple Channels

To configure multiple channels with the same settings, complete the following steps:

1. Click the button to open the **Channel Settings** window.

   > **Note** You can configure multiple channels even if you open the **Channel Settings** window by clicking the ··· button. Use `ctrl+click`, `shift+click`, or `ctrl+A` to select multiple channels.

2. Configure the settings that the right side of the **Channel Settings** window displays. The window only displays the settings that the selected channels share.

If a setting has different, non-conflicting values on different channels, the control for that setting displays Mixed values.

If a setting has conflicting values on different channels, the control for that setting displays No common option. Conflicting values are, for example, a **Signal type** dropdown menu that displays Voltage (V) for one channel and Current (A) for another

channel.

Managing Device Configurations in InstrumentStudio

Export device configurations from InstrumentStudio for use in applications such as LabVIEW, TestStand, Python, and C#.

Limitations of Exporting a Device Configuration

You cannot export a configuration that includes a waveform generator with the following waveform characteristics:

- Standard waveforms with the List or the Sweep operation mode.
- Standard waveforms of the User-defined waveform mode.
- Arbitrary waveforms that you create from a `.csv` waveform file.

Export a Device Configuration

After you configure a device in InstrumentStudio, you can export that configuration to a device configuration file. You can then import that configuration file to LabVIEW or another application to apply the device configuration to a different session.

1. Configure the device.
2. Open the instrument settings menu (⚙).
3. Select **Export** » **Driver configuration...**
4. Select the location to save the configuration file.

Import a Device Configuration

You cannot import a device configuration file into InstrumentStudio. If you want to save a layout or a device configuration for future use in InstrumentStudio, complete one of the following actions:

- Save the entire project by clicking **File** » **Save all**.
- Save the soft front panel of the device by clicking **File** » **Save [device name].sfp**.

To apply device settings from a device configuration file to a device session in LabVIEW,

you must import the device configuration file to LabVIEW. To import the configuration file, use the Import Attribute Configuration LabVIEW VI. The implementation of the Import Attribute Configuration VI is driver-specific. For more information, see the appropriate driver documentation in the *Related Information* section.

Related information:

- NI-SCOPE Import Attribute Configuration File
- NI-DCPower Import Attribute Configuration File
- NI-DMM Import Attribute Configuration File
- NI-FGEN Import Attribute Configuration File

Export Configurations to TestStand

You can export configurations to save time when you automate test sequences in TestStand.

After you configure your hardware in InstrumentStudio, you can export your configuration for use in TestStand sequences. You can export device and panel configurations to recent and supported 64-bit versions of TestStand. You can then use these configurations with LabVIEW, C#, C, or CVI automation code in a TestStand sequence.

Export an InstrumentStudio Configuration

You can use InstrumentStudio to create an IO configuration file. This IO file contains the driver configuration and the panel layout from InstrumentStudio.

Before you export an InstrumentStudio configuration to TestStand, ensure that the following conditions are met:

- Use a recent and supported 64-bit version of TestStand.
- Instrument channels that you want to include in the export are not in the Idle operation mode. InstrumentStudio does not export the configuration of instrument channels in the Idle operation mode.
- Select a specific site in the site filter, or select the **System pins** option to only export system pins. You cannot export an InstrumentStudio configuration if you select the **All sites** option in the site filter.

You can configure the default format of the exported file name by selecting **File** » **Preferences** and navigating to the **Export to TestStand** section.

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

   > **Note** If the **Export to TestStand** ( ) button does not appear, ensure that you have a recent and supported version of TestStand installed.

2. Specify a name for the exported configuration.

InstrumentStudio now creates an IO configuration file. The IO file contains the driver configuration and the panel layout from InstrumentStudio.

Insert an InstrumentStudio Configuration in a TestStand Sequence

You can insert your IO configuration file as a step within a TestStand sequence.

When you insert an IO configuration file into a TestStand sequence, TestStand creates a copy of the configuration file. You can specify the step in which TestStand copies the configuration file. When you run the sequence, this step creates sessions for the InstrumentStudio-configured instruments. Then, TestStand applies the configuration to the new sessions.

TestStand automatically stores the new sessions in local variables. You can do the following actions with the local variables:

- Pass a session from the local variable to automation code in Action steps
- Apply a new configuration to an existing session

You can also use the new session in a sweep loop to configure the sequence of values for different attributes in each instrument.

For more information on using TestStand, see *TestStand User Manual* in the *Related Information* section.

Related information:

- TestStand User Manual

Edit an Exported Configuration

You can edit an exported configuration in a specific TestStand step. The changes to the configuration commit back to the TestStand step or the IO configuration file that you edited.

1. Import an InstrumentStudio configuration to TestStand.
2. Navigate to the TestStand step that you want to edit.
3. Select **Edit**.
   InstrumentStudio opens in **Edit** mode and displays the instrument panels in the same layout as the document you exported the configuration from.
4. Edit the configuration in InstrumentStudio.
5. Click **OK**.

Capturing Data in InstrumentStudio

Capture measurement data and screenshots from a layout or panel and optionally publish captured data to Measurement Data Services using InstrumentStudio.

You can capture measurement data from supported layouts and panels. You can also capture a screenshot without capturing measurement data.

By default, InstrumentStudio:

- Saves screenshots as `.png` files.
- Saves measurement data as `.tdms` files.
- Does not publish captured data to Measurement Data Services.

To change data capture preferences, select **File** » **Preferences** » **Capture data**.

You can configure the following settings:

- File name
- File format
- Destination directory
- Measurement Data Services publishing

For more information on TDMS files, see *What is a TDMS File?*.

Capturing Data from a Layout

Use this option to capture all panels in the current layout.

1. Locate the instrument header menu.
2. Select the **Capture data** icon ( ).

InstrumentStudio captures content according to the configured data capture preferences.

The captured screenshot contains the visible layout at the time of capture.

The captured measurement data contains measurement results and device information for devices in the layout.

Capturing Data from a Panel

Use this option to capture data from a single instrument panel.

1. Open the instrument settings menu (⚙) for the panel.
2. Select **Capture data**.

InstrumentStudio captures content according to the configured data capture preferences.

The captured screenshot contains the visible panel at the time of capture.

The captured measurement data contains measurement results and device information for the device represented by the panel.

Capturing a Screenshot Only

Use this option to capture an image file without saving measurement data.

1. Locate the instrument header menu.
2. Select the **Capture screenshot** icon (📷).

InstrumentStudio:

- Copies the image to the clipboard
- Does not create a data file

Using Captured Data

You can open TDMS files using NI software or supported third-party tools, including:

- LabVIEW
- LabWindows/CVI
- DIAdem
- VeriStand

CSV exports can contain one or more CSV files and associated metadata files, depending on the captured instruments and measurements.

CSV files contain:

- **Columns**—Channel, device, and measurement type
- **Rows**—Measurement results (for example, mean and range) and calculated statistics (for example, standard deviation)

Publishing Data to Measurement Data Services

Introduced in InstrumentStudio 2026 Q3.

You can configure InstrumentStudio to automatically publish captured data to Measurement Data Services (MDS).

When publishing is enabled, InstrumentStudio saves captured data locally and publishes the data to Measurement Data Services.

Measurement Data Services MDS provides a standardized data store for measurement data. You can query and retrieve stored data for downstream analysis in other tools. Measurement Data Services does not perform data analysis.

To enable publishing:

1. Select **File** » **Preferences** » **Capture data**.
2. Enable data publishing in the **Measurement Data Services** section.

When enabled, InstrumentStudio publishes captured data automatically after each capture.

Before publishing data, InstrumentStudio prompts you to provide unit-under-test (UUT) information, including the product part number and serial number. InstrumentStudio associates this information with the published data.

After you confirm the publishing operation, InstrumentStudio publishes the captured data to Measurement Data Services.

Measurement Data Services components are installed with InstrumentStudio. No additional installation steps are required.

When publishing is enabled, InstrumentStudio publishes:

- Measurement data from the capture
- Metadata about the layout, panel, and devices

You can retrieve and use published data with:

- NI Nigel AI
- OData REST APIs
- Measurement Data Services APIs for LabVIEW
- Measurement Data Services APIs for Python
- Measurement Data Services APIs for C#

These tools enable you to:

- Query measurement results
- Retrieve data for analysis workflows
- Integrate measurement data into automated systems

Use Measurement Data Services when you need to:

- Store measurement data centrally
- Query and retrieve measurement data from the data store
- Enable automated data processing or analysis workflows

If you only need local files for manual inspection, you can continue to use captured data files without publishing.

Related information:

- What is a TDMS File?
- Measurement Data Services Overview

Debugging Measurements from External Applications in InstrumentStudio

Use the debug mode in InstrumentStudio to monitor and control devices that are in use by an external application.

Overview

You can use debug mode to monitor and troubleshoot devices that are in use by an external application, such as LabVIEW, TestStand, or a C, C++, .NET, or Python application.

In debug mode, InstrumentStudio does not communicate directly with hardware. Instead, it connects to device sessions that are owned by the external application.

This behavior allows you to inspect and modify device configuration while the external application is running.

For information about how measurement plug-ins execute and return data, see *Understanding Measurement Plug-In Execution*.

How Debug Mode Works

When you use debug mode, InstrumentStudio connects to an existing device session created by an external application.

What InstrumentStudio does:

- Reads data from the session
- Displays configuration and measurement results
- Optionally allows limited control of the session

What InstrumentStudio does not do:

- Start or own the session

- Control when measurements execute
- Override all device operations

The external application remains the owner of the session and controls execution.

Using Debug Mode

A device opened in Debug mode starts in Monitor mode. In Monitor mode, you can observe device behavior and measurement activity without modifying the device configuration.

To modify supported device settings, switch to Control mode by clicking **Control**. To return control to the external application, click **Monitor** or resume execution in the external application.

To debug a device used by an external application, complete the following steps.

1. Start the external application and open the device session.
2. Open InstrumentStudio.
3. Create or open a panel for the device.

   You can use the **Debug** command on the Home screen to locate devices that are available for debugging.

When InstrumentStudio detects that an external application owns the device session, the panel enters Monitor mode.

In Monitor mode, you can:

- Observe device behavior.
- View configuration values.
- View measurement results when available.

To modify supported device settings, complete the following steps.

1. Pause execution in the external application if necessary.
2. Click **Control**.
3. Inspect or adjust the device configuration.

InstrumentStudio transitions to Control mode.

To return control to the external application, click **Monitor** or resume execution in the external application.

What Data You Can See

The data available in debug mode depends on how the external application interacts with the device.

InstrumentStudio can display:

- Device configuration values
- Session state
- Measurement results that the external application reads or fetches

In most cases, measurement values update only when the external application calls a read, fetch, or measure operation.

For some supported devices, InstrumentStudio may display live updates. See *Live Measurement View*.

When Debug Mode Behaves Differently

Debug mode behavior depends on the external application, device configuration, and operation mode.

You may observe the following behavior:

- Measurement values do not update continuously
- Some controls are unavailable or read-only
- Changing settings may affect execution in the external application
- Control is limited when the external application actively manages the device

These behaviors occur because the external application owns the device session.

Limitations and Unsupported Scenarios

Debug mode does not support all device types, configurations, or operation modes.

Examples include:

- Sweep mode for some SMU panels
- Script or sequence modes for certain waveform generators
- Streaming or list configurations for RF signal generators

For a complete list of device-specific limitations, see *Limitations of Debug Mode*.

Performance Considerations

Enabling monitoring or control may affect application performance.

- Monitoring may increase communication between InstrumentStudio and the session
- Control mode may interrupt or alter the execution flow
- Debugging active measurements may reduce throughput

Use debug mode only when you need to inspect or troubleshoot behavior.

Troubleshooting

If debug mode does not behave as expected, check the following:

Table 24. Troubleshooting Debug Mode

Possible Cause	Action
No active session	Ensure the external application has opened the device session.
Session is not accessible	Verify that the device session supports shared access.
Data does not update	Ensure the external application is calling read, fetch, or measure operations.
Limited control or read-only settings	Verify that the external application is not actively

Possible Cause	Action
	controlling the same settings.

The following sections provide additional information about debug mode workflows, configuration options, troubleshooting, and device-specific limitations.

Debug Mode Limitations in InstrumentStudio

General limitations of debug mode and links to device-specific limitation topics.

Overview

Debug mode does not support all device types, configurations, and operating modes.

The following limitations may apply across multiple device families:

- Some operation modes are not supported in debug mode.
- Some controls are unavailable or read-only.
- The external application may restrict access to device settings.
- Changes in debug mode may not be allowed while the external application actively controls the device.

For device-specific limitations, see the following sections.

DAQ Analog Input and DAQ Analog Output Debug Mode Limitations in InstrumentStudio

Requirements, unsupported configurations, and debug mode limitations for DAQ Analog Input panels and DAQ Analog Output panels in InstrumentStudio.

Introduced in InstrumentStudio 2026 Q3.

Overview

The following requirements and limitations apply when you use debug mode with

DAQ Analog Input panels and DAQ Analog Output panels.

These limitations apply in addition to the general debug behavior described in *Debug Measurements from External Applications*.

Version Requirements

- Debug mode requires NI-DAQmx 2024 Q3 (24.5) or later.
- Earlier NI-DAQmx versions support standalone mode only.
- The *Using Breakpoints in C/C++/C#/Python Applications* workflow requires NI-DAQmx 2024 Q4 (24.8) or later.
- With NI-DAQmx 24.5, InstrumentStudio excludes devices configured for breakpoint-assisted debugging. To debug these devices, open **Configure Debug Session** and clear the breakpoint option.

Unsupported Device Families

The following NI-DAQmx device families support standalone operation in InstrumentStudio but do not support Monitor mode or Control mode.

M Series devices:

- PXI-62xx
- PXIe-62xx

DSA devices:

- PXI-4461
- PXI-4462
- PXI-4472
- PXI-449x
- PXIe-449x

Analog Output devices:

- PXI-6704
- PXI-6711
- PXI-6713

- PXI-6722
- PXI-6723
- PXI-6733

Unsupported Physical Channels

- DAQ Analog Input panels and DAQ Analog Output panels support external physical channels only. Internal physical channels are not supported.
- Tasks that contain both external and internal physical channels are partially supported. InstrumentStudio monitors only supported external channels and generates a warning.

Unsupported Signal Types

- Calculated power channels are not supported. If you monitor a PXIe-4311 power channel session, the panel displays a warning.
- PXIe-4467 and PXIe-4468 Pure Tone Generator channels are not supported.

Unsupported Scale Types

- InstrumentStudio only supports global linear custom scales. Polynomial custom scales, map range scales, and table custom scales are not supported.
- Force (bridge), Pressure, and Torque channels only support two-point linear scaling. Polynomial scaling and table scaling are not supported.
- TEDS Force (bridge), Pressure, and Torque channels are not supported.

Unsupported Timing Types

- Sessions that use single-point hardware timing are not supported.

Monitor Mode Limitations

- Monitor mode only displays samples acquired while InstrumentStudio is actively monitoring the DAQmx task. DAQmx does not cache samples for DAQmx Read or DAQmx Write calls that occur while InstrumentStudio is not monitoring the task.
- Monitor mode waits for DAQmx API calls to complete. Blocking calls such as DAQmx Read and DAQmx Wait Until Done can significantly reduce Monitor mode responsiveness.

The following approaches can improve Monitor mode responsiveness when the application uses blocking NI-DAQmx calls:

1. Add a delay to the application loop.
2. Poll Read.AvailSampPerChan and call DAQmx Read only when sufficient samples are available.
3. Use the `Every N Samples Acquired Into Buffer` event to invoke DAQmx Read.
4. Step through the application in a debugger.
5. In LabVIEW, enable execution highlighting.

Chart and Graph Limitations

- Multiple software-timed samples returned in a single DAQmx Read call share the same timestamp. InstrumentStudio displays only the final sample.
- Reading zero samples clears the graph for sample-clock timing but does not clear the graph for software timing.
- Unscaled DAQmx Read variants are only displayed when engineering units match hardware units.
- InstrumentStudio does not support monitoring raw variants of DAQmx Read or DAQmx Write. When the application uses a raw variant of DAQmx Read or DAQmx Write, InstrumentStudio displays a warning and does not display the data.

Trigger Status Limitations

- Without a configured trigger, the trigger status indicator displays acquisition status rather than trigger status.
- Monitor mode cannot update trigger status while a blocking DAQmx Read is running.
- For reference-triggered acquisitions, the trigger status indicator is based on when samples become available to read rather than when the reference trigger occurs.
- The trigger status indicator can be inaccurate for retriggered and multi-record acquisitions.

Task Ownership Limitations

- Tasks with TDMS logging enabled cannot be controlled.
- Taking control of a task temporarily pauses DAQmx event notifications.

DAQ Analog Input Limitations

- TEDS is supported in Monitor mode and Control mode, but not in standalone mode.
- Diagnostic fault reporting is unavailable in Monitor mode and Control mode.

DAQ Analog Output Control Mode Limitations

- Regeneration requires the task to remain committed or running.
- Hardware range changes do not automatically scale application-written data.
- Changing waveform type sets other channels to Constant DC.

Related concepts:

- DAQmx Task Selection in InstrumentStudio Debug Mode

Related tasks:

- Monitor and Control Devices Used in an External Session

Related reference:

- NI-DAQmx Hardware Support in InstrumentStudio

RF Signal Generator Debug Mode Limitations in InstrumentStudio

Device-specific limitations and control restrictions for RF signal generators when you use debug mode in InstrumentStudio.

The following limitations apply when you use debug mode with an RF signal generator.

Table 25. InstrumentStudio RF Signal Generator Debug Mode Limitations

Limitation	Description
Control restrictions	You cannot take control of an RF signal generator when streaming is enabled or when an active configuration list is present.

Limitation	Description
Script mode behavior	Prior to NI-RFSG 2025 Q4, you cannot return to Script mode after selecting **Preset**, and waveform-specific controls are read-only in Script mode.

Waveform Generator Debug Mode Limitations in InstrumentStudio

Device-specific limitations and unsupported configurations for waveform generators when you use debug mode in InstrumentStudio.

When you use debug mode with a waveform generator, consider the following limitations.

Table 26. InstrumentStudio Debug Mode Waveform Generator Limitations

Limitation	Description
Unsupported modes	Debug mode does not support arbitrary sequence mode or script mode. You cannot control sessions that use these modes.
List mode restrictions	List mode control is disabled when InstrumentStudio is in Monitor mode or Control mode.
Mode switching restrictions	In Control mode, you cannot switch to List mode or sweep mode from another operation mode.
Mixed channel configurations	In Monitor mode, you cannot take control of a panel that contains both standard and arbitrary waveform channels.

For more information about Monitor mode and Control mode, see *Monitor and Control Devices Used in an External Session*.

Enter Debug Mode

InstrumentStudio has debug mode enabled by default. You can enter debug mode

from the **Home** screen by selecting **Debug**.

When you enter debug mode, InstrumentStudio searches for open device driver sessions. Then, based on the result of this search, InstrumentStudio creates a new layout. The devices in the newly created layout automatically enter debug mode.

You can also enter debug mode while an InstrumentStudio project is already open. With this method, a device that is in use by an external session automatically enters debug mode in the following cases:

- The device is placed in a layout
- The device is already part of an active layout

Take, for example, a scenario where you run an oscilloscope in an InstrumentStudio panel alongside a LabVIEW application that acquires measurements from that oscilloscope. In this case, the InstrumentStudio panel enters debug mode until the LabVIEW application (the external session using the oscilloscope) closes.

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

Note If a device cannot enter debug mode, debug mode might be disabled. You can enable debug mode in the following way:

1. Select **Configure debug session** from the instrument settings menu ( ). The **Configure Debug Session** window opens.
2. Fill the **Configure Debug Session** checkbox next to the device that you want to debug.
3. Click **OK**.

If you change settings in the **Configure Debug Session** window while an external session is open, you must restart the external session to apply the changes.

Enable Debugging with C, C++, .NET, and Python Applications in InstrumentStudio

Enable debugging with C, C++, .NET, and Python applications in the **Configure Debug Session** window.

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

To enable debugging in C, C++, .NET, and Python environments, complete the following steps.

1. Locate the instrument settings menu (⚙).
2. Select **Configure Debug Session**.
   The **Configure Debug Session** window opens.
3. Select the checkbox in the **Using Breakpoints in C/C++/.NET/Python Applications** column for the device that you want to debug.
4. Click **OK**.
   InstrumentStudio enables breakpoints in C, C++, .NET, and Python external applications.
5. Debug the application or device.
6. Reopen the **Configure Debug Session** window.
7. Clear all checkboxes in the **Using Breakpoints in C/C++/.NET/Python Applications** column.
   InstrumentStudio disables breakpoint support for the selected devices.
8. Restart the external application.
   InstrumentStudio applies the changes.

Monitor and Control Devices Used in an External Session

You must enable debug mode to access the **Monitor** button and **Control** button.

1. Select **Monitor**.
   InstrumentStudio obtains read-only access to the external session. InstrumentStudio also disables the controls of the device. 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**, InstrumentStudio takes ownership of the device in the following cases:
   - You are not monitoring any external application sessions.
   - The external application is paused.

   > **Note**
   >
   > When you select **Control**, InstrumentStudio takes control of the device only until the external application calls the device again. Afterwards, the external application takes control of the device and InstrumentStudio returns to monitoring the session.
   >
   > You can use breakpoints to pause the external application. Pausing the external application allows InstrumentStudio to retain control of the device until you made your changes.
3. Edit the device settings.
4. Click **Monitor** to return to monitoring the session and return control of the device to the external session.
   InstrumentStudio applies the updated device settings to the external session.

Related concepts:

- DAQmx Task Selection in InstrumentStudio Debug Mode

DAQmx Task Selection in InstrumentStudio Debug Mode

Task selection behavior for DAQmx tasks when multiple debuggable tasks are available in Monitor mode or Control mode.

Introduced in InstrumentStudio 2026 Q3.

When multiple DAQmx tasks are available for debugging, the DAQ panel can automatically select a task or allow manual task selection.

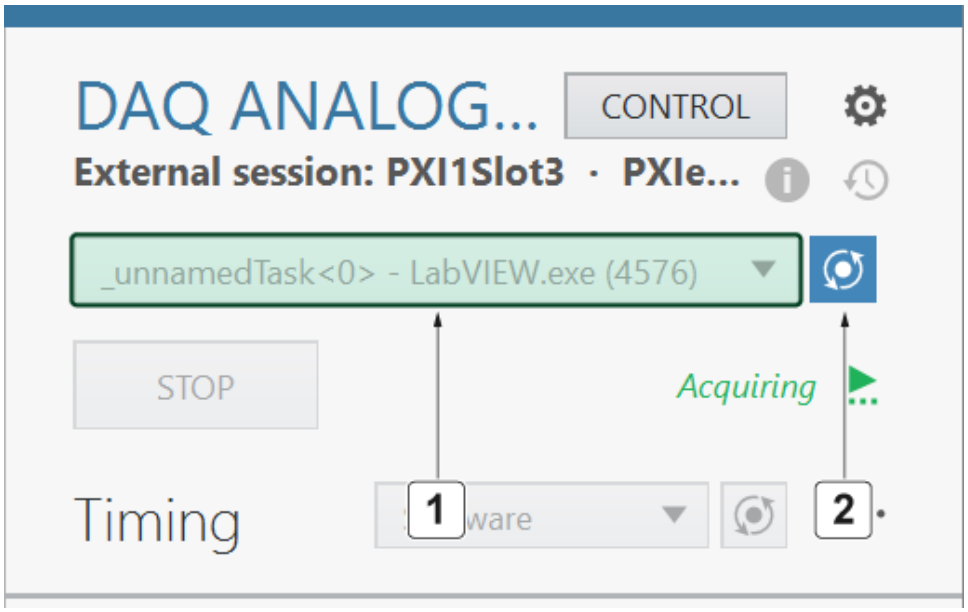
InstrumentStudio can debug a DAQmx task regardless of the current task state.

When Automatic Task Selection is enabled, the panel automatically selects the DAQmx task with the highest task state. For example, a running task is selected instead of a reserved or configured task.

To debug a specific task, disable Automatic Task Selection and select the task from the task selector. When Automatic Task Selection is disabled, the panel debugs only the selected task.

Note DAQmx allows multiple tasks that use the same resources to exist simultaneously, but only one task can be in the Reserved state at a time.

Figure 15. InstrumentStudio DAQmx Task Selection Controls in Debug Mode



- **1. Task selector**—Selects the DAQmx task to debug when automatic task selection is disabled.
- **2. Automatic Task Selection**—Enables or disables automatic selection of a DAQmx task for debugging.

Related concepts:

- DAQ Analog Input and DAQ Analog Output Debug Mode Limitations in InstrumentStudio

Related tasks:

- Monitor and Control Devices Used in an External Session

Live Measurement View in InstrumentStudio

Learn how InstrumentStudio displays measurement data for supported SMU/VPS devices while you monitor devices in debug mode.

Overview

Live Measurement View displays channel measurements for supported SMU/VPS devices while you monitor devices in debug mode.

For supported devices, InstrumentStudio can display measurement data even when an external application is not actively retrieving measurement data.

For general debug behavior, see *Debug Measurements from External Applications*.

Unsupported Devices

Live Measurement View is not supported for the following SMU/VPS devices:

- PXI-4110
- PXI-4130
- PXI-4132
- PXIe-4112
- PXIe-4113
- PXIe-4154
- PXIe-4162
- PXIe-4163
- PXIe-4190

For these devices, the external application must call Measure or Fetch to make measurement data available to InstrumentStudio.

Considerations

Live Measurement View is device dependent.

If a device does not support Live Measurement View, InstrumentStudio can display only measurement data made available by the external application.

Disable Debugging

You can disable debugging for individual devices in the **Configure Debug Session** window.

1. Select **Configure debug session** from the instrument settings menu (⚙).
   The **Configure Debug Session** window opens.
2. Clear 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.

Pin Maps

Pin maps define the following elements of a test site:

- Test instruments and hardware
- Sites and pins connected to a DUT
- How the test instruments and hardware connect to DUT pins

InstrumentStudio preserves the mappings that are created in TestStand Semiconductor Module (TSM) pin maps.

The following sections provide specific information on how to use pin maps in InstrumentStudio:

Set 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. Choose one of the following options 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 that 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 appears next to the pin map file in the project files pane if InstrumentStudio detects a problem. For more information, see *Pin Map Errors and Warnings*.
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. For more information on site filtering, see *Using the Site Filter*.
6. Apply pin filtering to view selected pins within a site in your panels. For more information on pin filtering, see *Apply a Pin Filter*.

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

Note When you launch InstrumentStudio from the TestStand Semiconductor Module (TSM), InstrumentStudio automatically uses the pin map that is configured in TSM.

Related tasks:

- Create a New Pin Map
- Select a Default Pin Map
- Add Pin Map Files to a Project
- Using the Site Filter
- Apply a Pin Filter

Related reference:

- Pin Map Errors and Warnings

Create a New Pin Map

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

1. To create a new pin map file (`.pinmap`), complete one of the following actions:
   - Select **File** » **New** » **Pin Map**.

- Click **Add** in the project files pane toolbar.

2. Use the **Pin Map Editor** to configure your pin mappings.
   The **Pin Map** tab displays mappable items. Use the left pane to browse and configure the mappings that you want to include in your pin map. To manually edit the pin map file text, click the **XML** tab.

   > **Note** You can only create a new pin map in a saved project. InstrumentStudio prompts 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.

Select a Default Pin Map

You can select a default pin map to add that pin map to all new projects that you create in InstrumentStudio.

1. Go to **File** » **Preferences** » **Pin and site mapping**.
2. Click the **Browse** button to open the **Default pin map file** dialog.
3. Select a pin map file.

InstrumentStudio adds the default pin map to the project files when you create a new project. InstrumentStudio also automatically uses the default pin map as the active project pin map.

Remove a Default Pin Map

Complete the following steps to remove an existing default pin map preference:

1. Go to **File** » **Preferences** » **Pin and site mapping**.
2. Go to the **Pin and site mapping** section.
3. Click the **Clear** button to remove the default pin map.

Add Pin Map Files to a Project

You can use pin maps that you create outside of InstrumentStudio by adding the

existing pin map file to your project.

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 automatically makes 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**.

Edit 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.
3. Click **OK** to close the **Pin Map Editor**.

Using the Site Filter

After you set an active pin map in InstrumentStudio, the devices in your panel appear as pins. Use the site filter in the document toolbar to display pins from a selected site.

InstrumentStudio filters your pins by the following characteristics:

- Sites
- System Pins
- Unmapped channels

1. In your instrument panel, click the **Site filter** drop-down menu 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 you can map each device channel to multiple pins on multiple sites. You must select sites and pins in the RF signal generator panel settings (⋯).
- You cannot export an InstrumentStudio configuration to TestStand when 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 reference:

- Managing Device Configurations in InstrumentStudio

Apply a Pin Filter

Use a pin filter to select which sites and system pins to view in an instrument panel. Pin filtering improves navigation in a panel.

Before you apply pin filtering, you must select an active project pin map. For more information, see *Set an Active Project Pin Map*.

1. Click the **Pin Filter** button (⊟) in your device or the instrument panel.
2. Enable the pins that you want to view in your panel.
3. Click **OK**.

Your panels now display configuration settings for pins that you select in the pin filter. The measurement panels also display measurements for those pins only. Information about hidden and non-idle pins appears under the **Pins** section of the panel.

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

Note InstrumentStudio saves the pin filter state in the instrument panel when you close the panel. When you re-open the panel, InstrumentStudio automatically restores the pin filter state.

Related tasks:

- Set an Active Project Pin Map

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 the following:

- Improperly formatted pin map files
- Pin map settings that do not correspond to configurations in your test system

To determine the cause of the error, double-click the pin map file to open the **Pin Map Editor**. The dialog displays a message with specific information about the cause of the error or warning. For information on pin map errors and warnings, see the following table:

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

For detailed information on pin map files, see *Pin Map File XML Structure (TSM)* in the *Related Information* section.

Related information:

- Pin Map File XML Structure (TSM)

Plug-Ins

With InstrumentStudio, you can host plug-in libraries, build application-specific plug-ins, and use measurement plug-ins that work across different products.

> **Note** This feature is only available with an InstrumentStudio Professional license.

The following sections provide information on how to use plug-ins in InstrumentStudio:

InstrumentStudio Plug-Ins

You can use InstrumentStudio to create and host application-specific plug-ins written in LabVIEW or C#. These plug-ins run together with the InstrumentStudio panels that are used to configure PXI instruments.

> **Note** This feature is only available with an InstrumentStudio Professional license.

Plug-ins have a runtime configuration and an edit-time configuration. You can export a plug-in configuration to TestStand to use it in a measurement step. In this case, the runtime configuration is provided to the sequence as a variable. Both of these configurations persist when you save the project that contains the configurations.

You cannot create plug-ins from the **File** » **New** menu. For information on how to create InstrumentStudio plug-ins, see *InstrumentStudio Plug-Ins on GitHub* in the *Related Information* section.

When you host a plug-in 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 plug-ins

For examples of InstrumentStudio plug-ins, see *InstrumentStudio Plug-Ins on GitHub* in the *Related Information* section.

Limitations of Plug-In Support

The following limitations apply when you host plug-ins in InstrumentStudio:

- Hosting plug-ins that are created in C# requires Visual Studio 2019 or newer and .NET 6.0.
- Plug-ins built in the 2022 Q3 or earlier versions of InstrumentStudio are not compatible. You must rebuild the plug-ins in a newer version of InstrumentStudio.
- You cannot use *Measurement Studio* controls when you host plug-ins that are created in C#.

Related information:

- InstrumentStudio Plug-Ins on GitHub

Measurement Plug-Ins in InstrumentStudio

Use measurement plug-ins in InstrumentStudio to run reusable measurements across supported devices and workflows.

> **Note** This feature is only available with an InstrumentStudio Professional license.

Overview

A measurement plug-in is a reusable measurement that runs as a service and can be used across multiple NI products, including InstrumentStudio and TestStand.

A measurement plug-in includes:

- Measurement logic
- Metadata
- An optional user interface

Measurement plug-ins generate the measurement logic and the metadata in Python or LabVIEW.

> **Note** For Python-based measurement plug-ins, you must install additional development tools and dependencies before you can create or run measurements. For details, see *Python Measurement Development Dependencies*.

For instructions on running a measurement plug-in in InstrumentStudio, see *Running a Measurement Plug-In*.

How Measurement Plug-Ins Work

InstrumentStudio does not run measurement logic directly. Instead, it connects to measurement services that run in the background.

When a measurement plug-in is available, InstrumentStudio detects the service and makes the plug-in available in the user interface.

To appear in InstrumentStudio, a measurement plug-in must be:

- Installed on the system or available through a plug-in library
- Running or registered as a service

Measurement Plug-Ins compared to InstrumentStudio Plug-Ins

- Measurement plug-ins provide executable measurements that run as services.
- InstrumentStudio plug-ins extend the user interface and panel behavior.

Additional Resources

For more information about developing and using measurement plug-ins, see *Measurement Plug-Ins User Manual*.

Related tasks:

- Running a Measurement Plug-In

Related information:

- Python Measurement Development Dependencies
- Measurement Plug-Ins User Manual
- Measurement Plug-Ins SDK for LabVIEW
- Measurement Plug-Ins SDK for Python

Running a Measurement Plug-In

1. On the **Home** screen, select **Manual layout**.
   The **Edit layout** screen appears.
2. Locate your measurement plug-in in the list.

> **Note** If your measurement plug-in does not appear in the **Edit layout** window, ensure that the plug-in is started. You can configure the plug-in to start automatically or you can start the plug-in manually. For more information, see *Statically Register a Measurement Service* in the *Developing a Measurement Plug-In with LabVIEW* section or the *Developing a Measurement Plug-In with Python* section of the *Measurement Plug-Ins User Manual*.

3. Select **Create large panel** in the drop-down list for your measurement plug-in.
   The large panel updates to list the selected measurement plug-in.
4. Click **OK**.
   InstrumentStudio opens a new project and displays the measurement UI.
5. **Optional:** If the measurement plug-in requires a pin map, ensure that the project contains a referenced pin map. Also ensure that the pin map is marked as active.
6. Click **Run** in the measurement UI.

> **Tip** You can change the behavior of the **Run** button so that the measurement runs continuously. Click the arrow at the side of the button and select **Run Continuously** to change the button behavior. Once the measurement is running, the button text changes. Click **Stop** to stop the

measurement.

Related information:

- Developing a Measurement Plug-In with LabVIEW
- Developing a Measurement Plug-In with Python

Plug-In Versioning

You can manage, select, and synchronize plug-in versions in InstrumentStudio.

Versioned Plug-Ins

For more information on versioning measurement plug-ins with LabVIEW or Python, see *Plug-In Versioning* in the *Measurement Plug-Ins User Manual*.

Selecting A Measurement Plug-In Version

You can add a measurement plug-in to your project layout in the **Edit layout** or the **Manual layout** windows. Use the version selector drop-down list to select the version of the measurement plug-in that you want to use. If only one version of a plug-in exists, InstrumentStudio displays the version number next to the plug-in.

Note Measurement plug-in versions are automatically synchronized to match the version that is stored in your project.

Synchronizing Plug-In Versions within a Project

You can also synchronize plug-in versions manually. To synchronize the plug-in version that a measurement panel uses, open the instrument settings menu (⚙) and click **Synchronize Version**. This process updates the panel to use the latest version of your measurement plug-in.

Note InstrumentStudio only applies version synchronization to the selected measurement panel. If multiple panels use the same measurement plug-in, you must repeat this process for each panel that you want to synchronize.

Related information:

- Plug-In Versioning

Connecting to a Plug-In Library in InstrumentStudio

Connect to a *plug-in library* in InstrumentStudio to discover and add measurement plug-ins from a shared repository.

> **Note** This feature is only available with an InstrumentStudio Professional license.

A plug-in library is an HTTP service that hosts measurement plug-ins. You can use a plug-in library to discover, download, and share plug-ins across systems and teams.

When you connect to a plug-in library, InstrumentStudio can display available measurement plug-ins in **Edit Layout**.

For information about installing, starting, and configuring plug-in libraries, see *Plug-In Libraries for Measurement Plug-Ins*.

Before you begin this procedure, verify the following:

- You have the address and port of the plug-in library
- The plug-in library is accessible from your system

1. Select **Edit Layout** » **Add remote host connections**.

   > **Note** You can add a plug-in library to your project with the **Edit Layout** mode.

2. Click **Add**.
3. Select **Plug-In Library** as the **Host type**.
4. Enter the IP address (or hostname) and the port number of the plug-in library that you want to connect to.
5. Click **OK**.

You are now connected to a plug-in library.

- Measurement plug-ins from the library have become available in **Edit Layout**.
- You can add available plug-ins to panels.
- InstrumentStudio retrieves plug-ins from the library when needed.

If InstrumentStudio cannot connect to the plug-in library or discover available measurement plug-ins, check the following items.

Possible Cause	Action
Incorrect connection settings	Verify the connection settings.
plug-in library is not available	Ensure the library is running and reachable.
Network or firewall restrictions	Check network access and firewall settings.

To run a measurement plug-in after adding it to a panel, see *Running a Measurement Plug-In*.

Related tasks:

- Running a Measurement Plug-In

Related information:

- Plug-In Libraries

Sequencer Tool

The sequencer tool allows you to perform a series of ordered tasks using instrument panels and measurement plug-ins. Automation with the sequencer tool allows you to efficiently perform multi-step measurements, saving time and ensuring consistency.

Note This feature is only available with an InstrumentStudio Professional license.

The following sections provide specific information on how to use the sequencer tool in InstrumentStudio:

Configure Steps

Each sequence step is associated with an InstrumentStudio panel. The panel contains the step configuration. InstrumentStudio immediately applies any changes to the step.

Supported Panel Types

The sequencer tool supports specific panel types only.

The sequencer tool supports the following panel types and the following operations for each panel type:

Panel Type	Start	Read	Stop
Digital Multimeter	✓	✓	✓
Measurement Plug-Ins		✓	
Network Analyzer	✓	✓	
Oscilloscope	✓	✓	
RF Signal Analyzer	✓	✓	
RF Signal Generator	✓		✓
SCPI Instrument		✓	

Panel Type	Start	Read	Stop
SMU/Power Supply	✓	✓	✓
Waveform Generator	✓		✓

Accessing the Sequencer Tool

The sequencer tool requires an InstrumentStudio Professional license.

Click the **Sequencer** pane at the top of the project window.

Build a Sequence

You can add panels and steps to a sequence in a project that you already saved.

Note Save you current project before you begin building a sequence. You cannot use the sequencer tool in an unsaved project.

1. Navigate to the panel that you want to add to a sequence.
2. Click the **Add to Sequence** ( ) button.

Tip You can hover over the **Add to Sequence** ( ) button to see a list of all associated steps that contain the selected panel.

To ensure that you added the correct steps to the sequence, open the **Sequencer** pane. Then, check the list of steps that appear in the **Step list** view.

Sweeps

Sweeps allow you to repeat selected steps in a sequence. Sweeps also allow you to increment the values of parameters across the individual measurement steps within the sweep.

Add a Sweep Loop to a Sequence

To add a sweep loop to a sequence, click the add step button (**+**) for the steps that you want to sweep. Then, select the **Sweep loop** option.

Sweep Behavior

When you run a sequence that contains a sweep, the sequence runs the steps within the sweep loop. You can configure the parameter settings that the sequence uses when you configure the sweep. In general, the sweep executes once for each new parameter value specified in the sweep settings.

For example, you want to run a test sequence. This test sequence sends a series of three different voltages to a device using an SMU panel. Then, the sequence reads data back from the device using a DMM panel. In this case, the sequence executes in the following order:

- Sweep 1, Step 1, Parameter Value 1: `SMU[Start]` sends 1.15 V to device.
- Sweep 1, Step 2: DMM reads output from device.
- Sweep 1, Step 3: `SMU[Stop]`
- Sweep 2, Step 1, Parameter Value 2: SMU sends 1.35 V to device.
- Sweep 2, Step 2: DMM reads output from device.
- Sweep 2, Step 3: `SMU[Stop]`
- Sweep 3, Step 1, Parameter Value 3: SMU sends 1.5 V to device.
- Sweep 3, Step 2: DMM reads output from device.
- Sweep 3, Step 3: `SMU[Stop]`

Once a sweep loop is completed, the next steps of the sequence execute.

Sweep Strategies

You can configure sweep loop behavior by opening the step settings panel (⚙). The following sweep strategies are available in the sequencer tool:

- **Start, Stop, Step**—Specify a start value and a stop value, as well as the step size to use.
- **Start, Stop, Count**—Specify a start value and a stop value, as well as the amount of steps to use within that range.

- **Array**— Manually specify a list of values to sweep through.

Sweep Step Settings

The following settings are available in the parameter settings panel:

Note InstrumentStudio shows a preview of values for the selected settings.

- **Sweep Name**—The name that you want to use for the parameter to be swept.
- **Data Type**—The data type of the parameter.
- **Strategy**—The sweep behavior that you want to use. Refer to *Sweep Strategies* for more information on sweep types.
- **Distribution**—How to distribute steps between the **Start** values and the **Stop** values that are specified for a sweep. Can be linear or geometric.
- **Start**—The initial parameter value to use when performing a sweep.
- **Stop**—The final parameter value to use when performing a sweep.
- **Step**—The step size to use when selecting a **Start, Stop, Step** sweep strategy.
- **Count**—The number of steps to create when selecting a **Start, Stop, Count** sweep strategy.
- **Values**—When you use an **Array** sweep strategy, use this table to enter the values that you want to sweep through. When you select other strategies, this table displays the values that the selected sweep settings generated.

Configuring Sweep Parameters for Sequence Steps

After you configure the sweep step settings, you can also configure how each step within the sweep handles the sweep parameter. You can configure sweep parameters in the step settings for each step within a sweep. Step settings allow you to use the sweep parameter for any appropriate step input. You can select the same sweep parameter on multiple step inputs.

When you correctly configure a step to a sweep, InstrumentStudio highlights the panel item that is associated with the sweep with a purple box.

Supported Panel Types

InstrumentStudio supports sweeps for the steps of the following features and panel types:

- Measurement plug-in steps
- All supported instrument panels, including SCPI instrument panels

Modify a Sequence

You can modify existing sequences and panels that are part of a sequence.

Changing Sequencer Panel Configurations

You can change the configuration of a panel that is used in multiple steps. In this case, InstrumentStudio applies the changes to all instances of that panel in the sequence.

Note Changes made to panel configurations apply immediately.

Reuse a Panel

If you want to use different configurations with the same panel, you must create a new panel. Click **Duplicate Panel** ( ) to quickly make a duplicate of the selected panel.

Reorder a Sequence

Drag and drop a step to the desired position in the sequence to change the order of the steps within a sequence.

View Steps

Double-click a step to quickly access and edit the panel associated with it.

Execute a Sequence

Executing a sequence performs all the steps that are defined in the sequence. When the sequence ends, you can view the data that InstrumentStudio measured during the sequence. You can also view the sequence report to ensure that the sequence executed all defined steps successfully.

Before you execute a sequence, ensure that all required devices in your system are

present, connected, and powered on.
Click **Run** to execute the sequence.
The timer in the sequencer pane starts, which indicates that the sequence is actively executing.

> **Note** When you execute a sequence, InstrumentStudio creates an execution report and saves that report in the `Sequence Results` folder. The `Sequence Results` folder is in the project folder.

View a Sequence Report

A sequence report is a `.pdf` file that details the results of a sequence. InstrumentStudio generates a sequence report every time you execute a sequence.

InstrumentStudio adds the sequence execution report files to the **Project Files** pane. You can also find the sequence report file in the `Sequence Results` folder. The `Sequence Results` folder is located inside the project folder.

1. Wait for the sequence to execute.
2. Click the **View Report** button. The generated report opens in the default `.pdf` viewer on your system.

View a Sequence Execution Log

When you execute a sequence, InstrumentStudio creates an execution log in the sequencer pane. You can use the execution log to check the status and the execution time of each step.

The execution log updates in real time while the sequence runs. The execution log shows the current step and the steps that InstrumentStudio already executed.

1. Open the **Sequencer** pane.
2. Click **Execution log**.

© 2026 National Instruments Corporation.