UHF User Manual

Zurich
Instruments

UHF User Manual

Zurich Instruments AG

Publication date Revision 38200

Revision History

Revision 38200, 14-July-2016:

The entire document was updated to comply with the changes of the 16.04 LabOne release.

Highlights of the changes and additions to the UHFLI product are:

- NEW option UHF-AWG Arbitrary Waveform Generator: 1.8 GSa/s, 14 bit, 2 channels, 128 MSa waveform memory, modulation mode, high-level LabOne Sequence Editor integrating waveform generation and editing tools, run-time variables
- NEW option UHF-CNT Pulse Counter: 4 counter modules, 225 MHz maximum count rate, pulse tagging mode, background subtraction
- Software Trigger: new grid mode for 2D data capture and frame averaging for imaging applications and for advanced data acquisition synchronized with the UHF-AWG signal generation
- Sweeper: new index sweep and node sweep modes for advanced measurement modes synchronized with the UHF-AWG signal generation
- Sweeper: support for negative frequencies for use in external sideband generation
- Scope: new resample feature to eliminate trigger jitter when using averaging in combination with reduced sampling rate
- Ref inputs: new duty cycle reconstruction feature for improved locking to TTL reference signals with low duty cycle
- Config tab: update reminder for the LabOne software
- Specifications: added Signal Output random jitter specification 4.5 ps (RMS) at 100 MHz, 6 dBm sine output
- Specifications: removed environment policy

A more detailed list of all technical changes can be found in the LabOne release notes.

Revision 34390, 22-Dec-2015:

The entire document was updated to comply with the changes of the 15.11 product release.

Highlights of the changes and additions to the UHFLI product are:

- Digital differential input mode using Signal Inputs 1 and 2
- Network discovery: automatic search of Data Server instances and instruments from all Zurich Instruments series in the local network
- Plotter: hardware trigger inputs can be displayed individually
- Sweeper: improved documentation (history, averaging, settling time)
- Boxcar: fixed repetitive restarting when ExtRef/PLL is used, interrupt data stream when changing parameters
- Specification change: maximum initial accuracy of the internal clock (ovenized crystal) is now ±1 ppm (previously ±0.5 ppm)

- Discontinued option UHF-10G Optical Ethernet

A more detailed list of all technical changes can be found in the LabOne release notes.

Revision 31421, 8-Jul-2015:

Two new chapters were added: Troubleshooting is now detailed in Chapter 7 and Chapter 6 was added to provide a quick introduction to the Signal Processing Basics. Moreover, the entire document was updated to comply with the changes of the 15.05 product release.

Highlights of the changes and additions to the UHFLI product are:

- Lock-in Tab: Added functional block diagrams for every demodulator to display lock-in functionality and signal routing dynamically
- Scope, Demodulators and PID: full support of input signal scaling
- File Manager: new tab for direct access of measurements, settings, and log files
- Instrument Presets: flexible choice of start-up configuration for stand alone operation (PC independent)

A more detailed list of all technical changes can be found in the LabOne release notes.

Revision 28900, 18-Mar-2015:

Document update of all chapters to comply with the changes of the 15.01 product release.

Highlights of the changes and additions to the UHFLI product are:

- Specification change: typical input noise at 100 kHz is now 4 nV/√Hz (previously 5 nV/√Hz)
- Sweeper: Indicator for estimated sweep time
- PID: PID Advisor with auto tune
- AU: Support of multiplication
- AU: Support of boxcar data
- Scope: Spectral Density for FFT of Scope Data
- Scope: Support of different FFT window functions
- NEW Option UHF-DIG Digitizer: Scope enhancement with continuous scope streaming, Scope trigger output on Trigger 1/2, Gated triggering, Hold-off specified as number of trigger events, Support of boxcar, demodulator, and PID data recording; Cross domain triggering for scope based on boxcar, demodulator, and PID data
- Boxcar: Reporting of the current data streaming rate

A more detailed list of all technical changes can be found in the LabOne release notes.

Revision 26210, 30-Sep-2014:

Document update of all chapters to comply with the changes of the 14.08 product release.

Highlights of the changes and additions to the UHFLI product are:

- Arithmetic Unit: a new tab that allows the control of 4 arithmetic units
- Sinc filter for Sweeper: increases speed of sweeps at low frequencies
- Scope: trigger performance, functionality and display have been further improved
- Scope: dual channel support (requires UHF-DIG Digitizer option)
- Scope: improved averaging and persistence refresh handling
- Sweeper: now supports data provided from the PID, boxcar and arithmetic unit
- Sweeper: simultaneous display of multiple traces
- Sweeper: additional application mode to support 3-omega measurements
- UHF-PID PID option: low pass filter for the D part now accessible in the user interface
- Auxiliary outputs can now output also the PID shift, e.g. frequency adjustment in a PLL
- UHF-MOD Modulation option: full access to phase, time constant, and filter order for the individual side bands

- UHF-BOX Boxcar option: averaging replaces the integration, provides better usability and more intuitive behavior
- New Harmonics Analyzer for UHF-BOX option: bar chart display for FFT of periodic waveform analyzer

A more detailed list of all technical changes can be found in the LabOne release notes.

Revision 23144, 22-Apr-2014:

Document overhaul and extension compliant to 14.02 product release. Updates include the getting started chapter, the ordering guide, added new tutorials, and updated the functional description. As of this release, the LabOne software contains installation files for both HF2 Series and UHF Series products. Also, as of this release, programming of the device by one of the supported APIs is described in a separated UHF Programmer's Manual.

Detailed changes and additions to the UHFLI product:

- Full support for UHF-10G Optical Ethernet option
- Boxcar option: support for baseline suppression
- PID option: added phase unwrap feature
- Periodic waveform analyzer (PWA): increased number of bins to 1024
- Periodic waveform analyzer (PWA): higher update rate
- UDP port assignment per device starting from port 8013
- Ethernet: improved reconnect after cable disconnect
- Start-up screen with device and setting selection: added support of multiple devices per server
- Improved Device connect/disconnect without server restart
- User interface: added cursor Math (with copy & paste of values)
- User interface: added relative cursor
- Lock-in: Vpk, Vrms, dBm support
- CSV transfer to other applications (Excel,...) via LiveLink
- Added histogram to oscilloscope
- Sweeper: Unbiased standard deviation
- Sweeper: Speed increase down to 6 ms per sweep point
- Plotter: Support for PID and boxcar streaming data

Detailed changes and additions to the HF2LI/HF2IS products:

- HF2LI-MOD option: fixed calculation of index of modulation
- HF2LI-PID option: fixed calculation of MOD sidebands
- Sweeper: PID setpoint sweeper

Revision 20274, 22-Nov-2013:

Document overhaul and extension compliant to 13.10 product release. Updates include the getting started chapter, the ordering guide, added new tutorials, and updated the functional description. As of this release, the LabOne tooltips inside of the user interface correspond to the description of the functional elements in this user manual.

Detailed changes and additions to the product:

- Instrument back panel: former Trigger 1/2 on the back panel of the instrument have been renamed to Trigger 3/4.
- USB connectivity: USB high-speed 480 Mbit/s fully supported as interface alternative to LAN. Simpler connectivity
- NEW option UHF-BOX Boxcar Averager 1: boxcar and periodic waveform analyzer (PWA, jitter free averaging scope) on signal inputs (requires UHF-BOX option)
- NEW option UHF-BOX Boxcar Averager 2: multi-channel boxcar, periodic waveform analyzer (PWA) on boxcar outputs
- Linux support

- Scope: oscilloscope and FFT spectrum analyzer are now integrated on a single tab
- Scope: sampling rates down to 27 kSa/s
- Scope: dual edge trigger
- General User Interface: improved design and drag & drop functionality for all tabs
- Lock-in: integrated Tandem demodulation (full support demodulation of auxiliary input and auxiliary output signals as demodulator inputs)
- Lock-in: output amplitude setting in V and dBm
- Lock-in: support for edge and level triggers
- Lock-in: phase to zero adjustment
- PID: simultaneous operation of all 4 controllers at a rate of 14 MSa/s
- PLL: center point adjustment
- Plotter: multi-trace support and vertical axis groups
- Plotter: quick add trace feature
- Sweeper: additional sweep parameters
- Sweeper: much higher sweep speed and support for odd configurations
- Spectrum: new name of former ZoomFFT panel
- Spectrum: filter compensation and absolute frequency control
- Spectrum: windowing effect reduction
- Spectrum: calculation of spectral density and power on FFT spectrum
- Numeric: increase font size of numerical values
- SW Trigger: triggering on Ref / Trigger connectors
- SW Trigger: automatic trigger level adjustment
- SW Trigger: triggering on Ref / Trigger connectors
- Auxiliary: automatic adjustment of Preoffset and Offset to zero outputs
- Config: improved data streaming and unified directory to CSV and MATLAB
- API / Programming: LabVIEW 64-bit support
- API / Programming: timestamp support for some data types (API revision 4)

Revision 18265, 30-Jul-2013:

Large revision of the specification chapter compliant to 13.06 product release. Moved many parameters from minimum/maximum to typical when parameter is characterized but not specifically tested during production. Also updated the getting started section.

With 13.06 all tooltips of the user interface have been updated, providing a considerable increase of usability. The functional description chapter is still small. The user manual will be overhauled with much more information with the next release.

Revision 17290, 23-May-2013:

Updated the connecting to the UHFLI section in the getting started chapter to reflect software usability improvements in software release 13.02.

Revision 15874, 11-Feb-2013:

Updated the getting started chapter with more detailed information on setup and several screenshots. Other minor edits in the whole document.

Revision 15785, 1-Feb-2013:

This is the first version of the UHFLI user manual related to software release 13.01. The main available sections are the getting started, the functional overview, a first tutorial of the user interface, and the specifications. Other sections will follow.

Table of Contents

Declaration of Conformity

The manufacturer

Zurich Instruments
Technoparkstrasse 1
8005 Zurich
Switzerland

declares that the product

UHFLI Lock-in Amplifier, 600 MHz, 1.8 GSamples/s

fulfills the requirements of the European guidelines

- 2004/108/EC Electromagnetic Compatibility
- 2006/95/EC Low Voltage
- 2011/65/EU Restriction of Hazardous Substances

The assessment was performed using the directives according to Table 1.

Table 1. Conformity table

EN 61326-1:2006	Emissions for industrial environments, immunity for industrial environments
EN 55011	Group 1, class A and B (the product was tested in typical configuration)
EN 61000-4-2	CD 4 kV, AD 8 kV
EN 61000-4-3	10 V/m 80% AM 80 MHz - 1 GHz
	3 V/m 80% AM 1 MHz - 2 GHz
	1 V/m 80% AM 2 MHz - 2.7 GHz
EN 61000-4-4	2 kV power line
	1 kV USB line
EN 61000-4-5	1 kV line-line, 2 kV line-earth
EN 61000-4-6	3 V 80% AM, power line
EN 61010-1:2001	Safety requirements for electrical equipment for measurement, control and laboratory use

Figure 1. CE Logo

Chapter 1. Getting Started

This first chapter guides you through the initial set-up of your UHF Instrument in order to make your first measurements. This chapter comprises of:

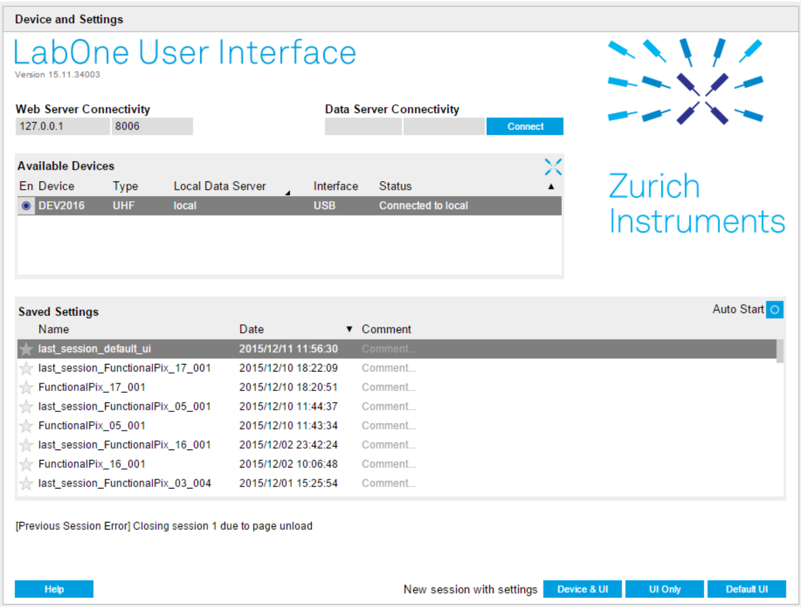
- Quick Start Guide for the impatient
- Inspecting the package content and accessories
- List of essential handling and safety instructions
- Installing LabOne, the UHF Instrument software, on your host computer
- Powering-on the device and connecting the device to a host computer
- Performing basic operation checks on the instrument

This chapter is delivered as a hard copy with the instrument upon delivery. It is also the first part of the UHF User Manual.

1.1. Quick Start Guide

This page addresses all the people who impatiently are awaiting their new gem to arrive and want to see it up and running quickly. Please proceed with the following steps:

1. Check the package content. Besides the Instrument there should be a country-specific power cable, a USB cable, an Ethernet cable and a hard copy of the user manual Chapter 1.
2. Check the Handling and Safety Instructions in Section 1.3.
3. Download and install the latest LabOne software from the **Zurich Instruments Download Center**. Choose the download file that fits your PC (e.g. Windows with 64-bit addressing). For more detailed information see Section 1.4.
4. Connect the Instrument to the power line, turn it on and then connect in with the measurement PC by using the USB cable. The necessary drivers will now be installed automatically. The front panel LED will blink orange at this stage.
5. Start the LabOne User Interface from the Windows Start Menu. The default Web Browser will open and display a start screen as shown below. The front panel LED turns from blinking orange to a steady blue.



6. Click the Default UI button on the lower right the UI. The default configuration is loaded and the first measurements can be taken. In cases the device could not be found or the UI does not start at all, please be referred to Section 1.5.1.
7. The UHFLI User Manual is included in a LabOne installation, it can be accessed in Windows via `Start Menu` → `All programs / All apps` → `Zurich Instruments` → `UHFLI User Manual` .

If any problems are encountered whilst setting up the instrument and software please see the troubleshooting section at the end of this chapter.

Once the Instrument is up and running we recommend to go through some of the tutorials given in Chapter 3. Moreover, Chapter 4 provides a general introduction to the various tools and settings tabs with tables in each section providing a detailed description of every UI element as well. For specific application know-how the **Blog section** of the Zurich Instruments web page will serve as a valuable resource that is constantly updated and expanded.

Note

The responsiveness of web browser user interface can be rather slow and still consuming plenty of CPU power when graphical hardware acceleration is not enabled. On most computers the situation can easily be improved by either

- Go to the NVIDIA control panel. Select graphic processor. Apply.
- Control panel: Control Panel\Appearance and Personalization\Display\Screen Resolution. Advanced settings. Trouble shoot. Change settings. (Does not work with NVIDIA, with NVIDIA you need to use the NVIDIA control panel)

Some computers have two graphic chip sets installed, an Intel and a NVIDIA chip set. Activating the NVIDIA along with the acceleration is strongly recommended to achieve best possible performance. The only drawback changing these settings is a slightly increased power consumption.

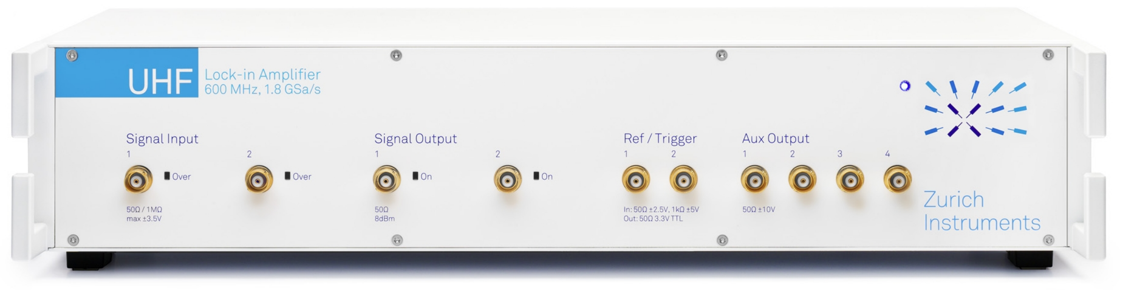
1.2. Inspect the Package Contents

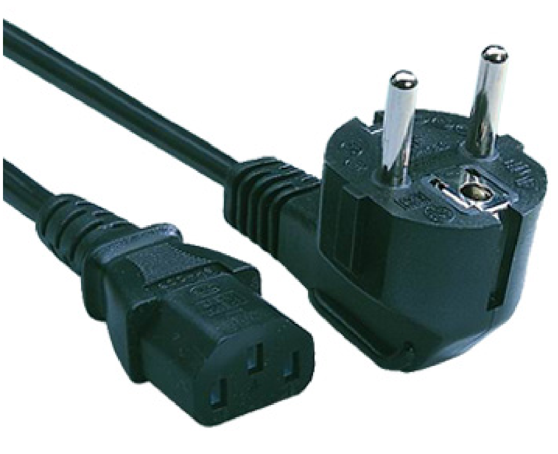
If the shipping container appears to be damaged, keep the container until you have inspected the contents of the shipment and have performed basic functional tests.

Please verify:

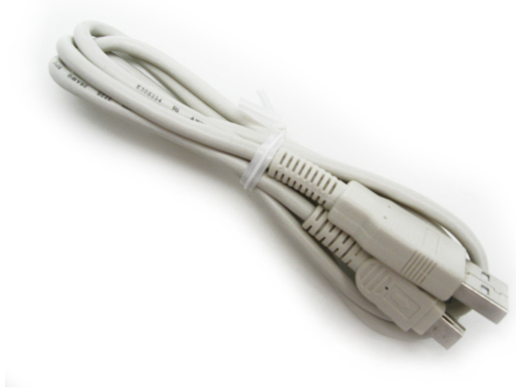
- You have received 1 Zurich Instruments UHF Instrument
- You have received 1 power cord with a power plug suited to your country
- You have received 1 USB cable and/or 1 LAN cable (category 5/6 required)
- A printed version of the "Getting Started" section
- The "Next Calibration" sticker on the rear panel of the Instrument indicates approximately 2 years ahead in time. Zurich Instruments recommends calibration intervals of 2 years
- The MAC address of the instrument is displayed on a sticker on the back panel

Table 1.1. Package contents for the UHF Instrument





the power cord (e.g. EU norm)



the USB cable

 the power inlet, with power switch and fuse holder	 the LAN / Ethernet cable (category 5/6 required)
 the "Next Calibration" sticker on the back panel of your instrument	 the MAC address sticker on the back panel of your instrument

The UHF Instrument is equipped with a multi-mains switched power supply, and therefore can be connected to most power systems in the world. The fuse holder is integrated with the power inlet, and can be extracted by grabbing the holder with two finger nails (or small screwdrivers) at the top and at the bottom at the same time. A spare fuse is contained in the fuse holder. The fuse description is mentioned in the specification chapter.

Carefully inspect your Instrument. If there is mechanical damage or the amplifier does not pass the basic tests, then you should immediately notify the Zurich Instruments support team at `<support@zhinst.com>`.

1.3. Handling and Safety Instructions

The UHFLI is a sensitive electronic instrument, which under no circumstances should be opened, as there are high-voltage parts inside which may be harmful to human beings. There are no serviceable parts inside the instrument. Do not install substitute parts or perform any unauthorized modification to the product. Opening the instrument immediately cancels the warranty provided by Zurich Instruments.

Do not use this product in any manner not specified by the manufacturer. The protective features of this product may be affected if it is used in a way not specified in the operating instructions.

The following general safety instructions must be observed during all phases of operation, service, and handling of the instrument. The disregard of these precautions and all specific warnings elsewhere in this manual may affect correct operation of the equipment and its lifetime.

Zurich Instruments assumes no liability for the user's failure to observe and comply with the instructions in this user manual.

Table 1.2. Safety Instructions

Ground the instrument	The instrument chassis must be correctly connected to earth ground by means of the supplied power cord. The ground pin of the power cord set plug must be firmly connected to the electrical ground (safety ground) terminal at the mains power outlet. Interruption of the protective earth conductor or disconnection of the protective earth terminal will cause a potential shock hazard that could result in personal injury and potential damage to the instrument.
Measurement category	This equipment is of measurement category I (CAT I). Do not use it for CAT II, III, or IV. Do not connect the measurement terminals to mains sockets.
Maximum ratings	The specified electrical ratings for the connectors of the instrument should not be exceeded at any time during operation. Please refer to Chapter 5 for a comprehensive list of ratings.
Do not service or adjust anything yourself	There are no serviceable parts inside the Instrument.
Software updates	Frequent software updates provide the user with many important improvements as well as new features. Only the last released software version is supported by Zurich Instruments.
Warnings	Instructions contained in any warning issued by the instrument, either by the software, the graphical user interface, notes on the instrument or mentioned in this manual must be followed.
Notes	Instructions contained in the notes of this user manual are of essential importance for

	the correct interpretation of the acquired measurement data.
Location and ventilation	This instrument or system is intended for indoor use in an installation category II and pollution degree 2 environment as per IEC 61010-1. Do not operate or store the instrument outside the ambient conditions specified in Chapter 5. Do not block the ventilator opening on the back or the air intake on the side of the chassis and allow a reasonable space for the air to flow.
Cleaning	To prevent electrical shock, disconnect the instrument from AC mains power and disconnect all test leads before cleaning. Clean the outside of the instrument using a soft, lint-free, cloth slightly dampened with water. Do not use detergent or solvents. Do not attempt to clean internally.
AC power connection and mains line fuse	For continued protection against fire, replace the line fuse only with a fuse of the specified type and rating. Use only the power cord specified for this product and certified for the country of use. Always position the device so that its power switch and the power cord are easily accessed during operation.
Main power disconnect	Unplug product from wall outlet and remove power cord before servicing. Only qualified, service-trained personnel should remove the cover from the instrument.
RJ45 plugs	The two RJ45 plugs on the back panel labeled "Peripheral ZCtrl" are not intended for Ethernet LAN connection. Connecting these plugs with an Ethernet device may damage the Instrument and/or the Ethernet device.
Operation and storage	Do not operate or store at the instrument outside the ambient conditions specified in Chapter 5.
Handling	Do not drop the Instrument, handle with due care, do not store liquids on the device as there is a chance of spilling and damage.

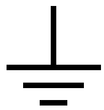
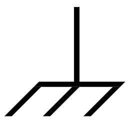
When you notice any of the situations listed below, immediately stop the operation of the Instrument, disconnect the power cord, and contact the support team at Zurich Instruments, either through the website form or by email at <support@zhinst.com>.

Table 1.3. Unusual Conditions

Fan is not working properly or not at all	Switch off the Instrument immediately to prevent overheating of sensitive electronic components.
Power cord or power plug on instrument is damaged	Switch off the Instrument immediately to prevent overheating, electric shock, or fire. Please exchange the power only with a power

	cord specified for this product and certified for the country of use.
Instrument emits abnormal noise, smell, or sparks	Switch off the Instrument immediately to prevent large damage.
Instrument is damaged	Switch off the Instrument immediately and secure it against unintended operation.

Table 1.4. Symbols

	Earth ground
	Chassis ground
	Caution. Refer to accompanying documentation
	DC (direct current)

1.4. Software Installation and Update

The UHFLI Instrument is operated from a host computer with the LabOne software. To install the LabOne software on a PC administrator rights are required. Following installation, to simply run the software, a regular user account is sufficient. Instructions for downloading the correct version of the software packages from the Zurich Instruments website are described below in the platform dependent sections. It is recommended to regularly update to the latest software version provided by Zurich Instrument as described in this section.

1.4.1. Installing LabOne on Windows

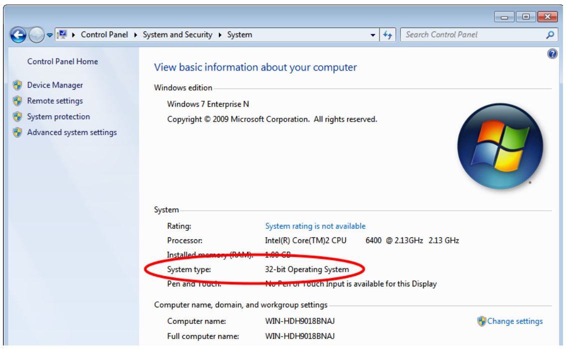
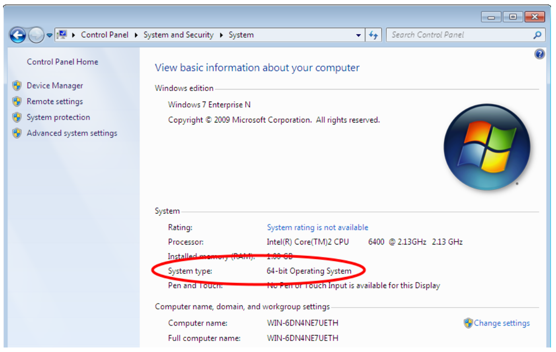
The installation packages for Zurich Instruments LabOne software are available as Windows installer .msi packages. The software is available on the Zurich Instruments download page, www.zhinst.com/downloads. Please ensure that you have administrator rights for the PC where the software is to be installed and that you download the correct software installer for the PC's processor architecture (32-bit or 64-bit), for help see the section called “Determining PC Architecture on Microsoft Windows”. See www.zhinst.com/labone/compatibility for a comprehensive list of supported Windows systems.

Determining PC Architecture on Microsoft Windows

In case you are unsure which Windows architecture you are using, it can be checked as follows:

- Windows 7: `Control panel → System and Security → System/System type`
- Windows 8: `Control panel → System → System/System type`

Table 1.5. Find out the OS addressing architecture (32-bit or 64-bit)

Windows 7 (32-bit: x86)	Windows 7 (64-bit: x64)
	

Windows LabOne Installation

1. The UHFLI Instrument should not be connected to your computer during the LabOne software installation process
2. Start the `LabOne32/64-xx.xx.xxxxxx.msi` LabOne installer program by a double click and follow the instructions. Please note that Windows Administrator rights are required for installation. The installation proceeds as follows:
   - On the welcome screen click the **Next** button.

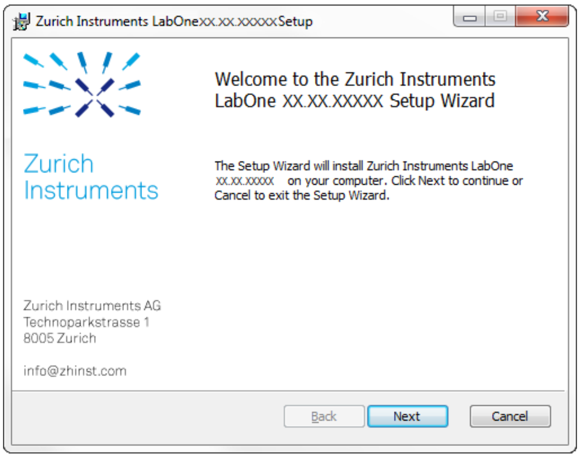


Figure 1.1. Installation welcome screen

- After reading through the Zurich Instruments license agreement, check the "I accept the terms in the License Agreement" check box and click the **Next** button.
- Review the features you want to have installed. For the UHFLI Instrument the `UHFLI Series Device`, `Web Server` and `API` features are required. Please install the features for other device classes as well as required. If you would like to install shortcuts on your desktop area enable the feature `Desktop Shortcuts`. To proceed click the **Next** button.

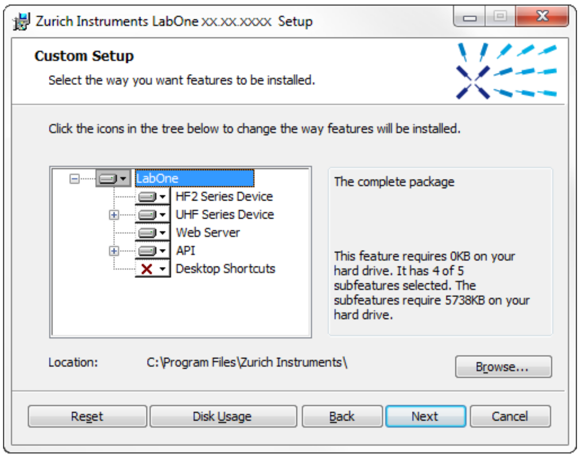


Figure 1.2. Custom setup screen

- Click the **Install** button to start the installation process.
- Windows will ask up to two times to reboot the computer. Make sure you have no unsaved work on your computer. Actually a reboot is practically never required, so that one may safely click **OK**.

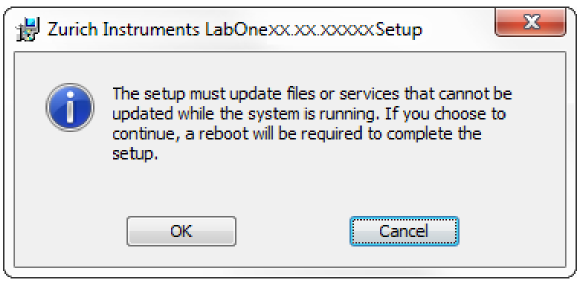


Figure 1.3. Installation reboot request

- On Windows Server 2008 and Windows 7 it is required to confirm the installation of up to 2 drivers from the trusted publisher Zurich Instruments. Click on **Install**.

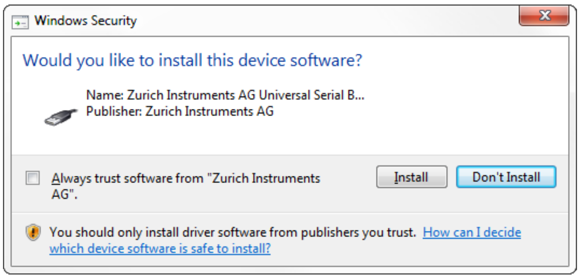


Figure 1.4. Installation driver acceptance

- Click **OK** on the following notification dialog.

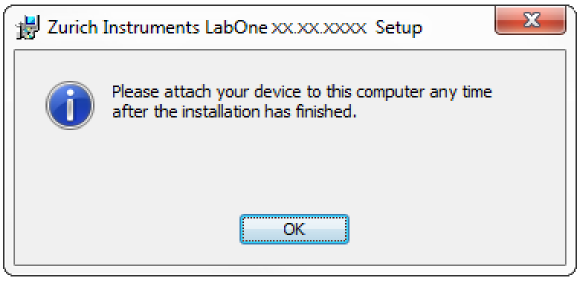


Figure 1.5. Installation completion screen

3. Click **Finish** to close the Zurich Instruments LabOne installer.
4. You can now start the LabOne User Interface as described in Section 1.5.2, LabOne Software Start-up, and choose an instrument to connect to via the Device Settings Dialogue (described in the section called “Device and Settings Dialog”).

Warning

Do not install drivers from another source and therefore not trusted as originating from Zurich Instruments.

1.4.2. Installing LabOne on Linux

Requirements

Ensure that the following requirements are fulfilled before trying to install the LabOne software package:

1. Officially, Ubuntu 12.04 LTS and 14.04 LTS (i386, amd64) are supported although in practice LabOne software may work on other platforms. Please ensure that you are using a Linux distribution that is compatible with Ubuntu/Debian, but preferably Ubuntu 12.04 LTS or 14.04 LTS.
2. You have administrator rights for the system.
3. The correct version of the LabOne installation package for your operating system and platform have been downloaded from the Zurich Instruments downloads page:
   - LabOneLinux<arch>-<release>.<revision>.tar.gz, for example:

   ```
   LabOneLinux32/64-xx.xx.xxxxx.tar.gz
   ```

Please ensure you download the correct architecture (32-bit/64-bit) of the LabOne installer. The `uname` command can be used in order to determine which architecture you are using, by running:

```
uname -m
```

in a command line terminal. If the command outputs " `x686`" the 32-bit version of the LabOne package is required, if it displays " `x86_64`" the 64-bit version is required.

Linux LabOne Installation

Proceed with the installation in a command line shell as follows:

1. Extract the LabOne tarball in a temporary directory:

   ```
   tar xzvf LabOneLinux<arch>-<release>-<revision>.tar.gz
   ```

2. Navigate into the extracted directory.

   ```
   cd LabOneLinux<arch>-<release>-<revision>
   ```

3. Run the install script with administrator rights and proceed through the guided installation, using the default installation path if possible:

   ```
   sudo bash install.sh
   ```

   The install script lets you choose between the following three modes:

   - Type "a" to install the Data Server program, the Web Server program, documentation and APIs.
   - Type "u" to install `udev` support (only necessary if HF2 Instruments will be used with this LabOne installation and not relevant for other instrument classes).
   - Type "ENTER" to install both options "a" and "u".

4. Test your installation by running the software as described in the next section.

Running the Software on Linux

The following steps describe how to start the LabOne software in order to access and use your instrument in the User Interface.

1. Start the UHFLI Data Server program at a command prompt:

   ```
   $ ziDataServer
   ```

2. Start the Web Server program at a command prompt:

   ```
   $ startWebServer
   ```

3. Start an up-to-date web browser and enter the `127.0.0.1:8006` in the browser's address bar to access the Web Server program and start the LabOne User Interface. The LabOne Web Server installed on the PC listens by default on port number 8006 instead of 80 to minimize the probability of conflicts.

4. You can now start the LabOne User Interface as described in Section 1.5.2 and choose an instrument to connect to via the Device Settings Dialogue as described in the section called “Device and Settings Dialog”.

Important

Do not use two Data Server instances running in parallel, only one instance may run at a time.

Uninstalling LabOne on Linux

The LabOne software package copies an uninstall script to the base installation path (the default installation directory is `/opt/zi/`). To uninstall the LabOne package please perform the following steps in a command line shell:

1. Navigate to the path where LabOne is installed, for example, if LabOne is installed in the default installation path:

   ```
   $ cd /opt/zi/
   ```

2. Run the uninstall script with administrator rights and proceed through the guided steps:

   ```
   $ sudo bash uninstall_LabOne<arch>-<release>-<revision>.sh
   ```

1.5. Connecting to the Instrument

After the LabOne software has been installed, the UHFLI instrument can be connected to a PC by using either the USB cable or the 1 Gbit/s (1GbE) Ethernet LAN cable supplied with the instrument. Using the LAN connection is particularly straightforward when DHCP IP address allocation is activated in the network. Direct point-to-point connection can also be used.

1.5.1. LabOne Software Architecture

The Zurich Instruments LabOne software gives quick and easy access to the instrument from a host PC. LabOne also supports advanced configurations with simultaneous access by multiple software clients (i.e., LabOne User Interface clients and/or API clients), and even simultaneous access by several users working on different computers. Here we give a brief overview of the architecture of the LabOne software. This will help to better understand the following chapters.

The software of Zurich Instruments lock-in amplifiers is server based. The servers and other software components are organized in layers as shown in Figure 1.6. The lowest layer running on the PC is the LabOne Data Server which is the interface to the connected lock-in amplifier. The middle layer contains the LabOne Web Server which is the server for the browser-based LabOne User Interface. This graphical user interface, together with the programming user interfaces, are contained in the top layer. The architecture with one central Data Server allows multiple clients to access a device with synchronized settings. The following sections explain the different layers and their functionality in more detail.

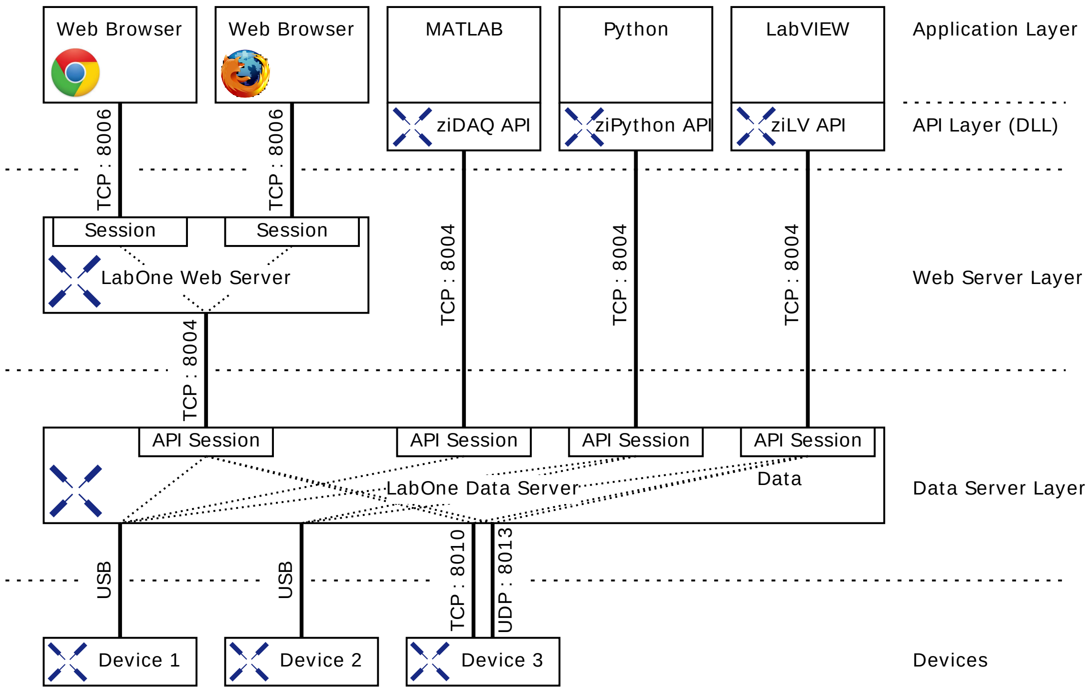


Figure 1.6. Software architecture

LabOne Data Server

The **LabOne Data Server** program is a dedicated server that is in charge of all communication to and from the device. The Data Server can control a single or also multiple lock-in amplifiers. It will distribute the measurement data from the instrument to all the clients that subscribe to it. It also ensures that settings changed by one client are communicated to other clients. The device settings are therefore synchronized on all clients. On a PC only a single instance of a LabOne Data Server should be running.

LabOne Web Server

The LabOne Web Server is an application dedicated to serving up the web pages that constitute the LabOne user interface. The user interface can be opened with any device with a web browser. Since it is touch enabled, it is possible to work with the LabOne User Interface on a mobile device like a tablet. The LabOne Web Server supports multiple clients simultaneously. That is to say that more than one session can be used to view data and to manipulate the instrument. A session could be running in a browser on the PC on which the LabOne software is installed. It could equally well be running in a browser on a remote machine.

With a LabOne Web Server running and accessing an instrument, a new session can be opened by typing in a network address and port number in a browser address bar. In case the Web Server runs on the **same** computer, the address is the localhost address (both are equivalent):

- `127.0.0.1:8006`
- `localhost:8006`

In case the Web Server runs on a **remote** computer, the address is the IP address or network name of the remote computer:

- `192.168.x.y:8006`
- `myPC.company.com:8006`

The most recent versions of the most popular browsers are supported: Chrome, Firefox, Edge, Safari and Opera.

LabOne API Layer

The lock-in amplifier can also be controlled via the application program interfaces (APIs) provided by Zurich Instruments. APIs are provided in the form of DLLs for the following programming environments:

- MATLAB
- Python
- LabVIEW
- C

The instrument can therefore be controlled by an external program and the resulting data can be processed there. The device can be concurrently accessed via one or more of the APIs and via the user interface. This enables easy integration into larger laboratory setups. See the LabOne Programming Manual for further information. Using the APIs, the user has access to the same functionality that is available in the LabOne User Interface.

1.5.2. LabOne Software Start-up

This section describes the LabOne User Interface start-up. If the LabOne Software is not yet installed on the PC please follow the instructions in Section 1.4 Software Installation. If the device is not yet connected please find more information in Section 1.5.3 Device Connectivity.

The most straightforward method to control and obtain data from the instrument is to use the LabOne User Interface, which can be found under the Windows Start Menu (see Figure 1.7 and Figure 1.8): Click and select `Start Menu → All programs / All apps → Zurich Instruments → LabOne User Interface`. This will open the User Interface in a new tab in your default web browser and start the LabOne Data Server and LabOne Web Server programs in the background. A detailed description of the software structure is found in the Section 1.5.1 LabOne Software Architecture.

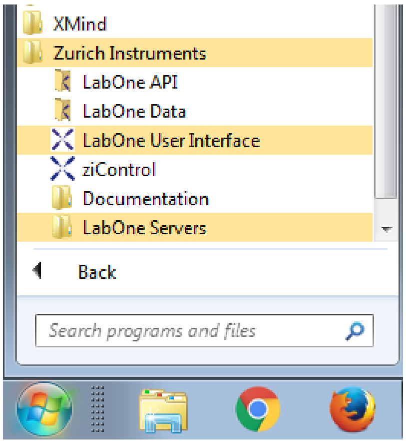


Figure 1.7. Link to the LabOne User Interface in the Windows 7 Start Menu (All programs)

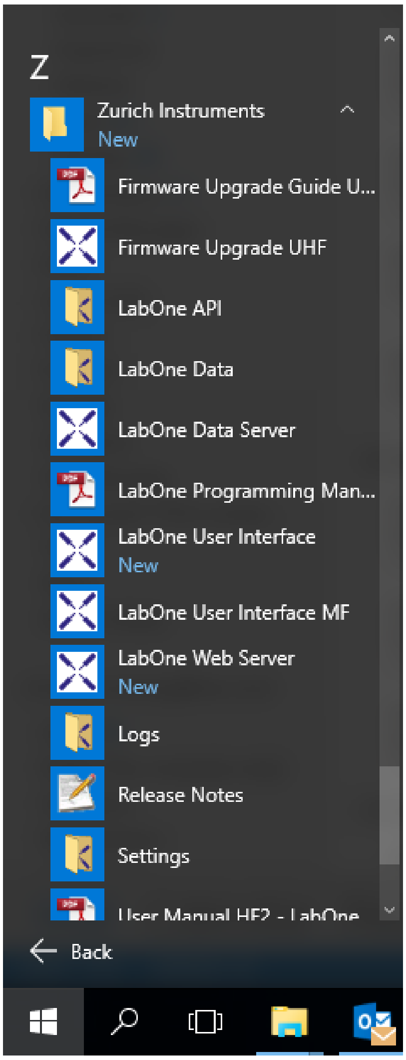


Figure 1.8. Link to the LabOne User Interface in the Windows 10 Start Menu (All apps)

The LabOne User Interface is an HTML5 browser-based program. This simply means that the user interface runs in a web browser and that a connection using a mobile device is also possible; simply specify the IP address (and port 8006) of the PC running the user interface.

Note

The user interface requires the LabOne Web Server (that runs in combination with the LabOne Data Server). Instead of starting the User Interface directly in your default browser as described above, it's possible to start the LabOne Data Server andLabOne Web Server programs independently and then connect via a browser of your choice:

1. Start the LabOne Data Server by selecting `Start Menu → Programs/All Apps → Zurich Instruments → LabOne Servers → LabOne Data Server`.
2. Start the LabOne Web Server by selecting `Start Menu → Programs/All Apps → Zurich Instruments → LabOne Servers → LabOne Web Server`.
3. In a web browser of your choice start the LabOne User Interface (the graphical user interface) by entering the localhost address with port 8006 to connect to the LabOne Web Server: `127.0.0.1:8006`

Note

By creating a shortcut to Google Chrome on your desktop with the Target `path\to\chrome.exe -app=http://127.0.0.1:8006` set in Properties you run the LabOne User Interface in Chrome in application mode which improves the user experience by removing the unnecessary browser controls.

Device and Settings Dialog

After starting the LabOne user interface software, a dialog is shown to select the device and settings for the session. The term session is used for an active connection between the user interface and the device. Such a session is defined by device settings and user interface settings. Several sessions can be started in parallel. The sessions run on a shared LabOne Web Server. A detailed description of the software architecture can be found in Section 1.5.1 Software Architecture.

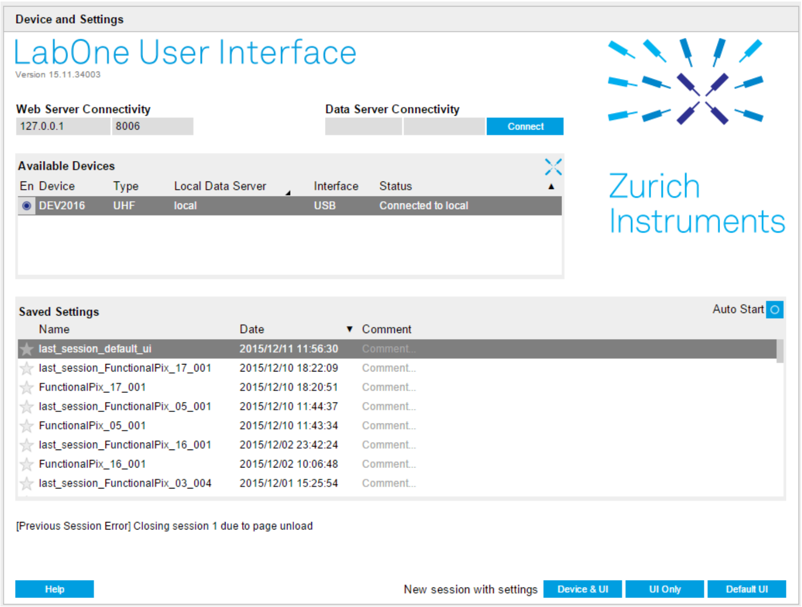


Figure 1.9. Dialog Device and Settings

The Device and Settings dialog consists of four sections: Web Server Connectivity, Data Server Connectivity, Available Devices, and Saved Settings. By default, the dialog is set to Local Data Server mode in the Available Devices section. In that case, the list of Available Devices will contain all instruments directly connected to the host PC via USB or to the local network via 1GbE. . Once your instrument appears in the Available Devices section, perform the following steps to start a new session:

1. Select an instrument in the Available Devices list.
2. Select a setting file in the Saved Settings list unless the **Default UI** is used.
3. Start the session by clicking **Device & UI**, **UI Only**, or **Default UI**.

If there are no setting files listed, starting the LabOne User Interface by clicking the button **Default UI** will start a session using factory defaults.

Note

Opening a new session with the **Device & UI** button can affect existing sessions since the device settings are shared between them. In that case, consider using the **UI Only** button to open a new session.

Note

In case devices from other Zurich Instruments series (UHF, HF2, MF) are used in parallel, the list of Available Devices section can contain those as well.

The following sections describe the functionality of the Device and Settings dialog in detail.

Data Server Connectivity

The Device and Settings dialog represents a Web Server shown under Web Server Connectivity. However, on startup the Web Server is not yet connected to a LabOne Data Server, which is why the fields under Data Server Connectivity are empty. With the **Connect/Disconnect** button the connection to a Data Server can be opened and closed.

This functionality can usually be ignored when working with a single UHFLI Instrument and a single host computer. Data Server Connectivity is important for users operating their instruments from a remote PC, i.e., from a PC different to the PC where the Data Server is running or for users working with multiple instruments. The Data Server Connectivity function then gives the freedom to connect the Web Server to one of several accessible Data Servers. This includes Data Servers running on remote computers controlling UHF or HF2 Instruments, and also Data Servers running on an MF instrument.

In order to work with either a UHF or HF2 Instrument remotely, proceed as follows. On the computer directly connected to the UHFLI (Computer 1) open a User Interface session and change the Connectivity setting in the Config tab to "From Everywhere", cf. Section 4.14.

On the remote computer (Computer 2), open the Device and Settings dialog by starting up the LabOne User Interface. Change the dialog mode from **Local Data Server** to **All Data Servers** by opening the drop-down menu in the header row of the Available Devices table. This will make the Instrument connected to Computer 1 visible in the list. Select the device and connect to the remote Data Server by clicking on Connect. Then start the User Interface as described above.

Note

When using All Data Servers mode, take great care to connect to the right instrument especially in larger local networks. Always identify your instrument based on its device serial of the form DEV-xxxx which can be found on the instrument back panel.

Available Devices

The Available Devices section gives an overview of the visible devices. A device is ready for use if either marked free or connected. The first column of the list holds the **Enable** button controlling the connection between the device and a Data Server. This button is greyed out until a Data Server is connected to the LabOne Web Server using the **Connect** button. If the button is enabled the device is connected by the LabOne Data Server. In this case no other LabOne Data Server running on another PC can access the device. Only one interface and LabOne Data Server can access the device.

The second column indicates the device serial and the third column shows the instrument type (HF2, UHF, MFLI, or MFIA). The fourth column indicates shows the IP address of the LabOne Data Server controlling the device, if it is not a local one. The next column shows the interface type. For UHF Instruments the interfaces USB or 1GbE are available. The interface is listed if physically connected. For MF series instruments the interface is always indicated as PCIe (this corresponds to the interface between the embedded PC and the measurement unit inside the MF instrument) regardless of the connection to the host computer (which can be USB or 1GbE). The LabOne Data Server will scan for the available devices and interfaces every second. If a device has just been switched on or physically connected it may take up to 20 s before it becomes visible to the LabOne Data Server. If an interface is physically connected but not visible please read Section 1.5.3 Device Connectivity.

The last column indicates the status of the device. Table 1.6 explains the meaning of the possible device status information.

Table 1.6. Device Status Information

Connected	The device is connected to a LabOne Data Server, either on the same PC (indicated as local) or on a remote PC (indicated by its IP address). The user can start a session to work with that device.
Free	The device is not in use by any LabOne Data Server and can be connected by clicking the **Enable** button. Alternatively, a session can also be started directly by clicking on **Device & UI**, **UI Only**, **Default UI** without prior connecting.
In Use	The device is in use by a LabOne Data Server. As a consequence the device cannot be accessed by the specified interface. To access the device, a disconnect is needed.
Device needs FW upgrade	The firmware of the device is out of date. Please first upgrade the firmware. See Section 1.6 Upgrading the Lock-In Amplifier Firmware.
Device not yet ready	The device is visible and starting up.

Saved Settings

Settings files can contain both UI and device settings. UI settings control the structure of the LabOne User Interface, e.g. the position and ordering of opened tabs. Device settings specify the set-up of a device. The device settings persist on the device until the next power cycle or until overwritten by loading another settings file.

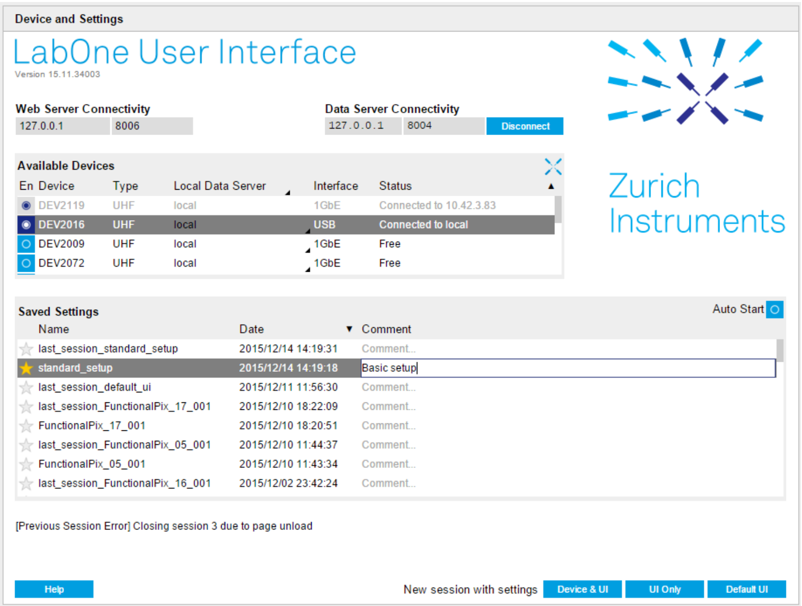
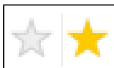


Figure 1.10. Dialog Device and Settings

The columns are described in Table 1.7. The table rows can be sorted by clicking on the column header that should be sorted. The default sorting is by time. Therefore, the most recent settings are found on top. Sorting by the favorite marker or setting file name may be useful as well.

Table 1.7. Column Descriptions

	Allows favorite settings files to be grouped together. By activating the stars adjacent to a settings file and clicking on the column heading, the chosen files will be grouped together at the top or bottom of the list accordingly. The favorite marker is saved to the settings file. When the LabOne user interface is started next time, the row will be marked as favorite again.

Name	The name of the settings file. In the file system, the file name has the extension .xml.
Date	The date and time the settings file was last written.
Comment	Allows a comment to be stored in the settings file. By clicking on the comment field a text can be typed in which is subsequently stored in the settings file. This comment is very useful to describe the specific conditions of a measurement.

Special Settings Files

Certain file names have the prefix " `last_session_`". Such files are created automatically by the LabOne Web Server when a session is terminated either explicitly by the user, or under critical error conditions, and save the current UI and device settings. The prefix is prepended to the name of the most recently used settings file. This allows any unsaved changes to be recovered upon starting a new session.

If a user loads such a last session settings file the " `last_session_u`" prefix will be cut away from the file name. Otherwise, there is a risk that an auto-save will overwrite a setting which was saved explicitly by the user.

The settings file with the name " `default_ui`" also has special meaning. As the name suggests this file contains the default UI settings. See button description in Table 1.8.

Table 1.8. Button Descriptions

Device & UI	The Device and UI settings contained in the selected settings file will be loaded.
UI Only	Only the UI settings contained in the selected settings file will be loaded. The device settings remain unchanged.
Default UI	Loads the default LabOne UI settings. The device settings remain unchanged.
Auto Start	Skips the session dialog at start-up if selected device is available. The default UI settings will be loaded with unchanged device settings.

Note

The factory default UI settings can be customized by saving a file with the name " `default_ui`" in the Config tab once the LabOne session has been started and the desired UI setup has been established. To use factory defaults again, the " `default_ui`" file must be removed from the user setting directory.

Note

The user setting files are saved to an application-specific folder in the user directory structure. On Windows, the folder can be opened in a file explorer by following the link in the Windows Start Menu: Click and select `Start Menu → Programs → Zurich Instruments → LabOne Servers → Settings`.

Note

Double clicking on a device row in the Available Devices block is a quick way of starting the default LabOne UI. This action is equivalent to selecting the desired device and clicking the **Default UI** button.

Double clicking on a row in the Saved Settings block is a quick way of loading the LabOne UI with the those device and UI settings. This action is equivalent to selecting the desired settings file and clicking the **Device & UI** button.

Messages

The LabOne Web Server will show additional messages in case of a missing component or a failure condition. These messages display information about the failure condition. The following paragraphs list these messages and give more information on the user actions needed to resolve the problem.

Lost Connection to the LabOne Web Server

In this case the browser is no longer able to connect to the LabOne Web Server. This can happen if the Web Server and Data Server run on different PCs and a network connection is interrupted. As long as the Web Server is running and the session did not yet time out, it is possible to just attach to the existing session and continue. Thus, within about 15 seconds it is possible with **Retry** to recover the old session connection. The **Reload** button opens the dialog Device and Settings shown in Figure 1.9. The figure below shows an example of this dialog.

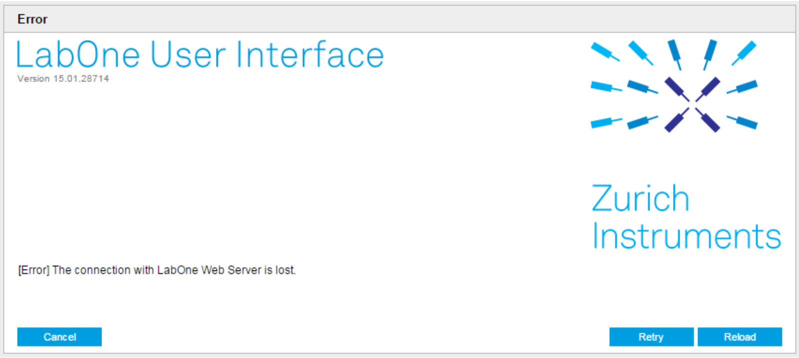


Figure 1.11. Dialog: Connection Lost

Reloading...

If a session error cannot be handled the LabOne Web Server will restart to show a new Dialog Device and Settings as shown in the section called “Device and Settings Dialog”. During the restart a window is displayed indicating that the LabOne User Interface will reload. If reloading does not happen the same effect can be triggered by pressing F5 on the keyboard. The figure below shows an example of this dialog.

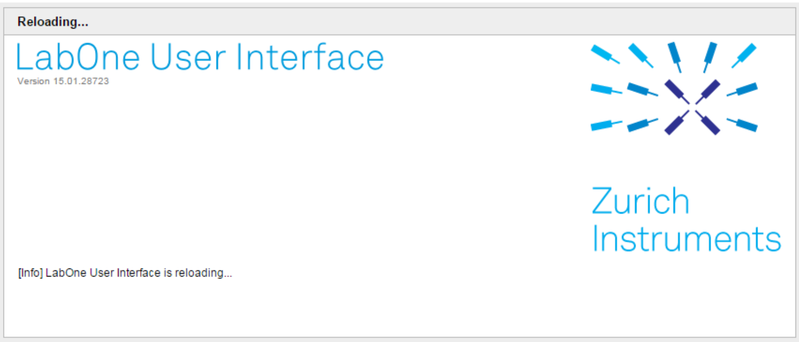


Figure 1.12. Dialog: Reloading

No Device Discovered

An empty "Available Devices" list means that no devices were discovered. This can mean that no LabOne Data Server is running, or that it is running but failed to detect any devices. The device may be switched off or the interface connection fails. For more information on the interface between device and PC see Section 1.5.3 Device Connectivity. The figure below shows an example of this dialog.

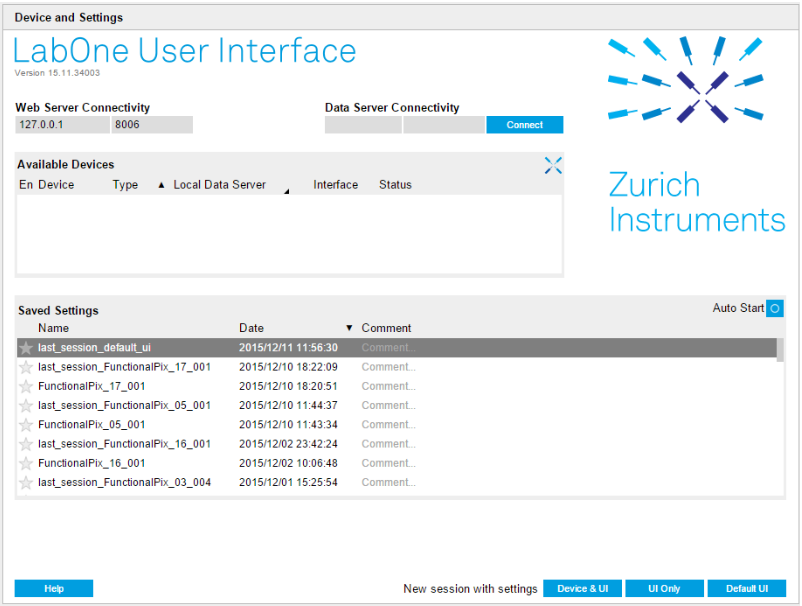


Figure 1.13. No Device Discovered

No Device Available

If all the devices in the "Available Devices" list are shown grayed, this indicates that they are either in use by another Data Server, or need a firmware upgrade. For firmware upgrade see Section 1.6. If all the devices are in use, access is not possible until a connection is relinquished by the another Data Server.

Device firmware upgrade needed

If a device listed in the Available Devices section needs a firmware upgrade, see Section 1.6 Upgrading the Lock-In Amplifier Firmware.

1.5.3. Device Connectivity

There are several ways to connect to the Zurich Instruments lock-in amplifier from a host computer. The device can either be connected by Universal Serial Bus (USB) or by Ethernet. The USB connection is a point to point connection between the device and the PC on which the Data Server runs. The Ethernet connection can be a point to point connection or an integration of the device into the local network (LAN). Depending on the network configuration and the installed network card, one or the other connectivity is better suited. This section gives a brief introduction to different methods.

If a device is connected to a network multiple PCs can access the same device. However, there is no shared device access possible at the same time. To control the access to a device two different connectivity states are needed: visible and connected.

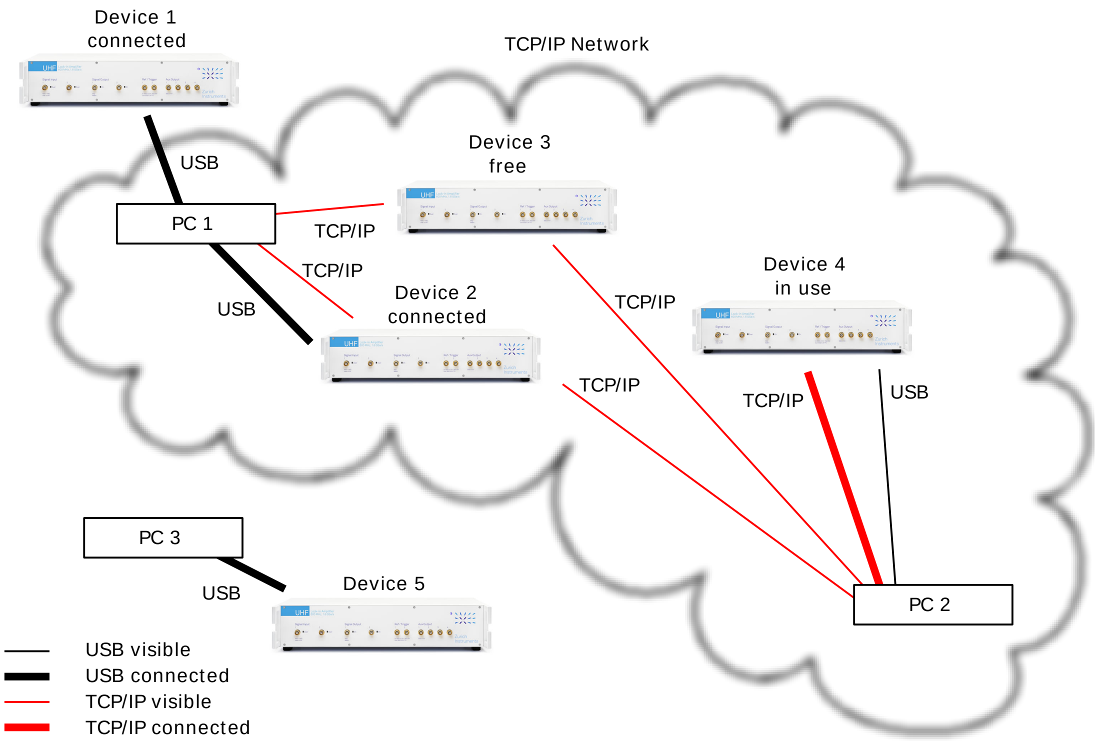


Figure 1.14. Connectivity

Figure 1.14 shows some examples of possible configurations of PC-to-device connectivity.

- Server on PC 1 is connected to device 1 (USB) and device 2 (USB).
- Server on PC 2 is connected to device 4 (TCP/IP).
- Server on PC 3 is connected to device 5.
- The device 3 is free and visible to PC 1 and PC 2 over TCP/IP.
- Both device 2 and device 4 are accessible by TCP/IP and USB interface. Only one interface is logically connected to the server.

It is important to distinguish if a device is just physically connected over USB or Ethernet or actively controlled be the LabOne Data Server. In the first case the device is visible to the LabOne Data Server. In the second case the device is connected (logically).

Visible Devices

A device is visible if the Data Server can identify it. On a TCP/IP network several PCs running a Data Server will detect the same device as visible. If a device is once discovered, the LabOne Data Server might initiate a connection to access the device and stream measurement data. Only a single Data Server can be connected to a device at a time.

Connected Device

Once connected to a device, the Data Server has exclusive access to data of that device. If another Data Server from another PC already has an active connection to the device, the device is still visible but cannot be connected by a second PC.

Although a Data Server has exclusive access to a connected device, the Data Server can have multiple clients. Like this, multiple browser and API sessions can access the device simultaneously.

Universal Serial Bus (USB) Connection

To control the device over USB, connect the instrument with the supplied USB cable to the PC on which the LabOne Software is installed. The USB driver needed for controlling the device is included in the LabOne Installer package. Ensure that the device uses the latest firmware. The software will automatically use the USB interface for controlling the device if available. If the USB connection is not available, the Ethernet connection may be selected. It is possible to enforce or exclude a specific interface connection.

Note

To use the device exclusively over the USB interface modify the shortcut of the LabOne User Interface and LabOne Data Server in the Windows Start menu. Right-click and go to Properties, then add the following command line argument to the Target LabOne User Interface: `--interface-usb true --interface-ip false`

Device Discovery USB

Devices connected over USB can be automatically connected by the Data Server as there is only a single host PC to which the device interface is physically connected.

auto-connect = on

This is the default behavior. If a device is attached via a USB cable, a connection will be established automatically.

auto-connect = off

To disable automatic connection via USB, add the following command line argument when starting the Data Server: `--auto-connect=off`

This is achieved by right clicking the LabOne Data Server shortcut in the Start menu, selecting "Properties" and adding the text to the Target field as shown in Figure 1.15.

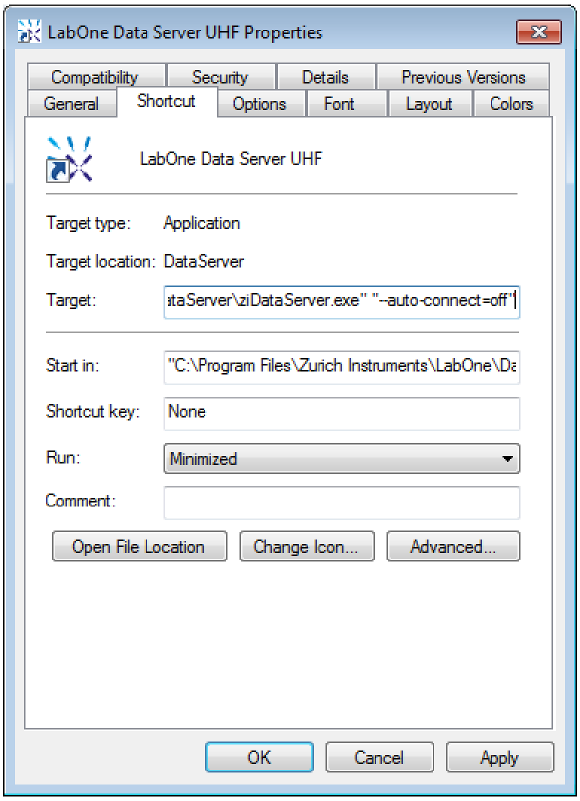


Figure 1.15. auto-connect

Device Discovery TCP/IP 1GbE

There are primarily two methods for connecting to the device via TCP/IP.

- Multicast DHCP
- Multicast point-to-point (P2P)
- Static Device IP

Multicast DHCP is the simplest and preferred connection method. Other connection methods can become necessary when using network configurations that are in contradiction with the local policies. This particularly concerns the enabling of Jumbo frames, which is an essential setting for good performance when using high data transfer rates.

Note

To use the device exclusively over the Ethernet interface, modify the shortcut of the LabOne User Interface UHF and LabOne Data Server UHF in the Windows Start menu. Right-click and go to Properties, then add the following command line argument to the Target field: `--interface-usb false --interface-ip true`

Multicast DHCP

The most straightforward TCP/IP connection method is to rely on a LAN configuration to recognize the UHF Instrument. When connecting the instrument to a LAN, the DHCP server will assign an IP address to the UHFLI like to any PC in the network. In case of restricted networks, the network administrator may be required to register the device on the network by means of the MAC address.

The MAC address is indicated on the back panel of the instrument. The LabOne Data Server will detect the device in the network by means of a multicast.

If the network configuration does not allow or does not support multicast, or the host computer has other network cards installed, it is necessary to use a static IP setup as described below. The UHF Instrument is configured to accept the IP address from the DHCP server, or to fall back into IP address `192.168.1.10` if it does not get the address from the DHCP server.

Requirements

- Network supports multicast

Multicast Point-to-Point (P2P)

Setting up a point-to-point network consisting only of the host computer and the UHFLI avoids problems related to special network policies. Since it is nonetheless necessary to stay connected to the internet, it is recommended to install two network cards in your computer, one of which is used for network connectivity (e.g. internet), the other can be used for connecting to the UHF Instrument. Notebooks can generally profit from wireless LAN for internet connection.

In such a P2P network the IP address of the host computer needs to be set to a static value.

1. Use one of the network cards and set it to static IP in TCP/IPv4 using the IP address `192.168.1.n`, where n=[2..9] and the mask `255.255.255.0`, see Figure 1.16 (go to `Control Panel → Internet Options → Network and Internet → Network and Sharing Center → Local Area Connection → Properties`).
2. Start up the LabOne User Interface normally. If your instrument does not show in the list of Available Devices, the reason may be that your network card does not support multicast. In that case use a static device IP as described below.

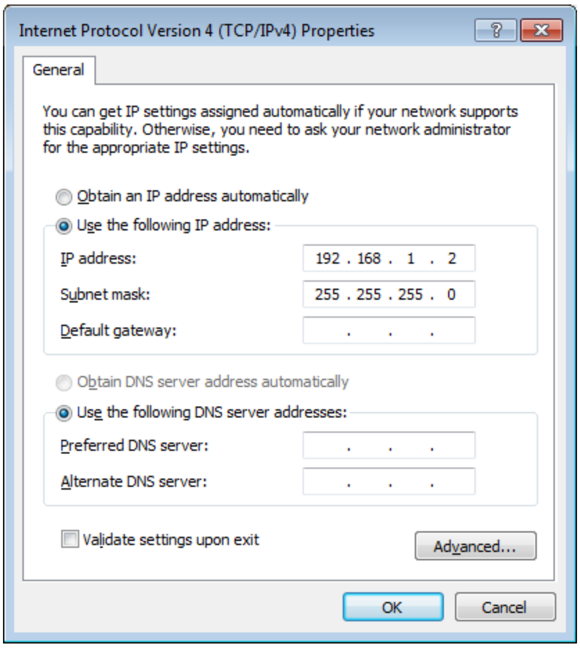


Figure 1.16. Static IP configuration for the host computer

Requirements

- Two networks cards needed for additional connection to internet
- Network adapter (NIC) of PC supports multicast

- Network adapter connected to the device must be in static IP4 configuration

Note

A power cycle of the UHF Instrument is required if it was previously connected to a network that provided a IP address to the instrument and then the user decides to run in static IP configuration.

Note

Only IP v4 is currently supported. There is no support for IP v6.

Warning

Changing the IP settings of your network adapters manually can interfere with its later use, as it cannot be used anymore for network connectivity until it is set again for dynamic IP.

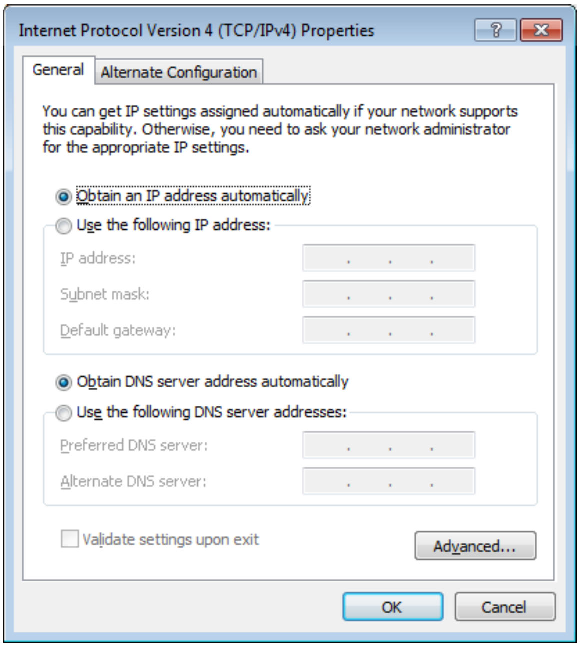


Figure 1.17. Dynamic IP configuration for the host computer

Static Device IP

Using a static IP address for the host computer is necessary to set up a point-to-point network. On top of that, a static device IP configuration can be necessary in the rare cases in which the network card does not support multicast.

1. Connect the Ethernet port of the static IP configured network card to the 1GbE port on the back panel of the UHF Instrument
2. Modify the shortcut of the LabOne User Interface UHF and LabOne Data Server UHF in the Windows Start menu. Right-click and go to Properties, then add the following command line argument to the Target field: `--device-ip 192.168.1.10`.

   The LabOne User Interface UHF shortcut Target field should look like this:

```
"C:\Program Files\Zurich Instruments\LabOne\WebServer\ziWebServer.exe"
--auto-start=1 --server-port=8004 --resource-path "C:\Program Files
\Zurich Instruments\LabOne\WebServer\html\\" --device-ip 192.168.1.10
```

The LabOne Data Server UHF shortcut Target field should look like this:

```
"C:\Program Files\Zurich Instruments\LabOne\DataServer
\ziDataServer.exe" --device-ip 192.168.1.10
```

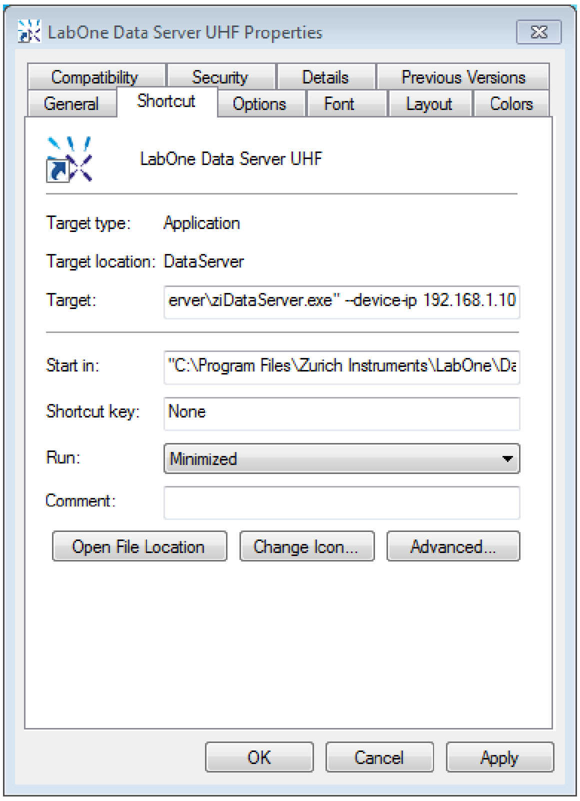


Figure 1.18. Static IP shortcut modification

3. (Optional) To verify the connection between the host computer and the UHF Instrument, open a DOS command window and ping the IP address entered above

Requirements

- Device IP must be known
- Needs network administrator support on networks with dynamic IP configuration

1.6. Upgrading the Lock-In Amplifier Firmware

The LabOne software consists of both software that runs on your PC and software that runs on the UHF Lock-in Amplifier itself. In order to distinguish between the two, the later will be called firmware for the rest of this document. When upgrading to a new software release, it's also necessary to upgrade the UHF firmware. If the device firmware is out of date and needs an upgrade, this is indicated in the Device and Settings Dialog of the LabOne user interface. See the section called “Device and Settings Dialog”.

1.6.1. Preparation

In order to upgrade the UHF firmware, you must first take the following steps:

1. Download and install the appropriate version (32-bit/64-bit) of the LabOne software on your PC. Administrator rights are necessary for the software installation. Please see Section 1.4 Software Installation.
2. Either start the UHF Lock-in Amplifier or, if the UHF was already running, switch off and restart the UHF Lock-in Amplifier.
3. Connect the UHF to the PC with the LabOne installation via USB cable.

1.6.2. Starting the UHF Firmware Upgrade Utility

The UHF Firmware Upgrade Utility is the program used to perform a UHF firmware upgrade, it is a GUI (Graphical User Interface) included in the standard LabOne installation.

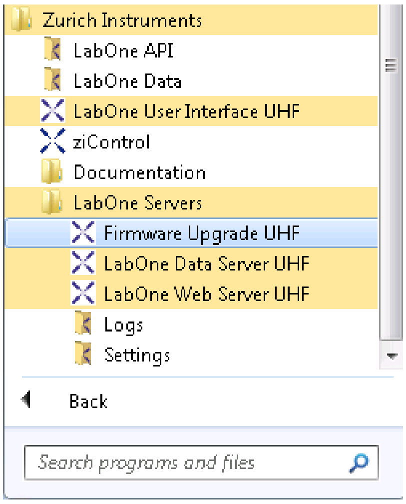


Figure 1.19. Starting the Firmware Upgrade Utility via the Windows Start Menu

To start the Firmware Upgrade Utility:

Click and select `Start Menu` → `All Programs` → `Zurich Instruments` → `LabOne Servers` → `Firmware Upgrade UHF`.

Note

It's not necessary to have administrator rights in order to start or use the UHF Firmware Upgrade Utility.

Important

Do not disconnect the USB cable to the UHF or power-cycle the UHF whilst performing any of the following steps.

Upon starting the Firmware Upgrade Utility it should detect the device that is connected to the PC via USB. The device ID is displayed next to "Device:".

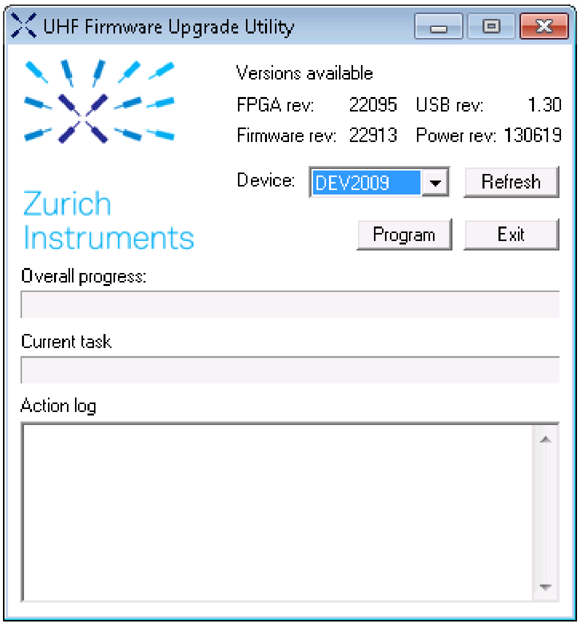


Figure 1.20. The UHF Firmware Upgrade Utility upon start-up

Select the device you would like to upgrade

Select which device you would like to upgrade via the pull-down menu. If no device is listed, please try the following steps:

1. Ensure that the USB cable is properly connected.
2. Try power-cycling the device.
3. Click the **Refresh** button.

Program the firmware of the connected device

Click the **Program** button to check the version of the current firmware and install the new firmware on the device.

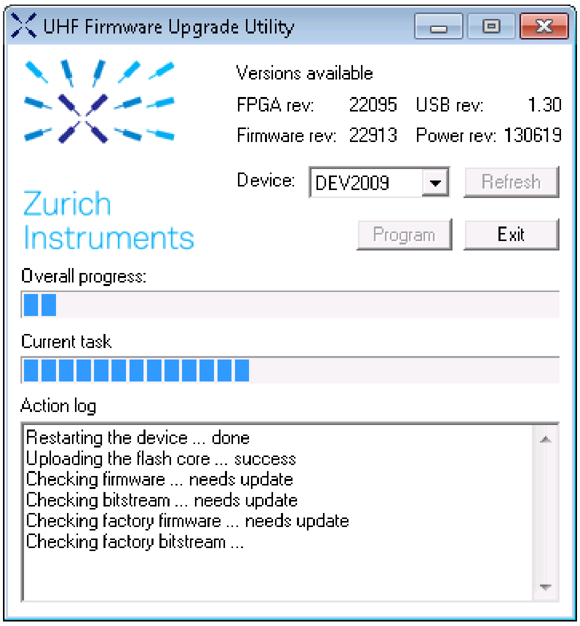


Figure 1.21. Verifying the UHF firmware version

Important

After clicking Program and the upgrade is finished it is always necessary to power-cycle the UHF to resume normal operation, even if the firmware was previously up-to-date.

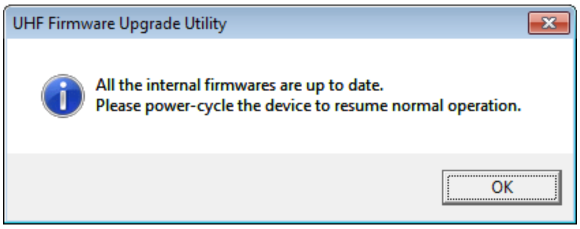


Figure 1.22. Pop-up Box indicating successful installation of the new firmware

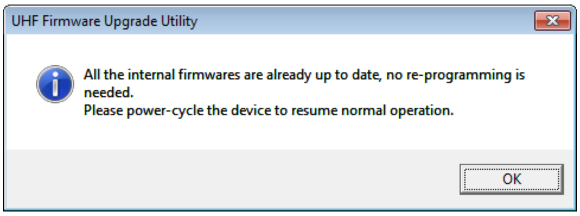


Figure 1.23. Pop-up Box indicating that the firmware was already up-to-date

Close the UHF Firmware Upgrade Utility

Click the **Exit** button to close the UHF Firmware Upgrade Utility.

If you encounter any issues whilst upgrading the UHF firmware, please contact Zurich Instruments at support@zhinst.com.

1.7. Troubleshooting

This section aims to help the user solve and avoid problems whilst using the software and operating the instrument.

1.7.1. Common Problems

Your UHFLI Instrument is an advanced piece of laboratory equipment with many more functionalities than a traditional lock-in amplifier. In order to benefit from these, the user needs access to a large number of settings in the LabOne User Interface. The complexity of the settings might overwhelm a first-time user, and even expert users can get surprised by certain combinations of settings. To avoid problems, it's good to use the possibility to save and load settings in the Config Tab. This allows one to keep an overview by operating the instrument based on known configurations. This section provides an easy-to-follow checklist to solve the most common mishaps.

The software cannot be installed or uninstalled: please verify you have Windows administrator rights. Windows systems: if prompted or required install the .NET Framework, see Section 1.7.3.

The Instrument does not turn on: please verify the power supply connection and inspect the fuse. The fuse holder is integrated in the power connector on the back panel of the instrument.

The Instrument has a high input noise floor (when connected to host computer by USB): the USB cable connects the Instrument ground to computer ground, which might inject some unwanted noise to the measurements results. In this case it is recommended to use the Ethernet connection which is galvanically isolated using a UTP Cat 5 or 6 cable (UTP stands for “unshielded twisted pair”).

The Instrument performs poorly at low frequencies (below 160 kHz with 50Ω or below 100 Hz with 1 MΩ coupling) : the signal inputs of the instrument might be set to AC operation. Please verify to turn off the AC switch in the Lock-in or In / Out tab.

The Instrument performs poorly during operation: the demodulator filters might be set too wide (too much noise) or too narrow (slow response) for your application. Please verify if the demodulator filter settings match your frequency versus noise plan.

The Instrument performs poorly during operation: clipping of the input signal may be occurring. This is detectable by monitoring the red LEDs on the front panel of the instrument or the OVI flags on the status tab of the user interface. It can be avoided by adding enough margin on the input range setting (for instance 50% to 70% of the maximum signal peak).

The Instrument performs strangely when working with the UHF-MF Multi-frequency option: it is easily possible to turn on more signal generators than intended. Check the generated Signal Output with the integrated oscilloscope and check the number of simultaneously activated oscillator voltages.

The Instrument performs close to specification, but higher performance is expected: after 2 years since the last calibration, a few analog parameters are subject to drift. This may cause inaccurate measurements. Zurich Instruments recommends re-calibration of the Instrument every 2 years.

The Instrument measurements are unpredictable: please check the Status tab to see if any of the warning is occurring (red flag) or has occurred in the past (yellow flag).

The Instrument does not generate any output signal: verify that signal output switch has been activated in the Lock-in tab or In / out tab.

The Instrument locks poorly using the digital I/O as reference: make sure that the digital input signal has a high slew rate and clean level crossings.

The Instrument locks poorly using the auxiliary analog inputs as reference: the input signal amplitude might be too small. Use proper gain setting of the input channel.

The sample stream from the Instrument to the host computer is not continuous: check the communication (COM) flags in the status bar. The three indicate occasional sample loss, packet loss, or stall. Sample loss occurs when a sampling rate is set too high (the instruments sends more samples than the interface and the host computer can absorb). The packet loss indicates an important failure of the communications to the host computer and compromises the behavior of the instrument. Both problems are prevented by reducing the sample rate settings. The stall flag indicates that a setting was actively changed by the system to prevent UI crash.

The user interface does not start or starts but remains idle: verify that the LabOne Data Server and LabOne Web Server have been started and are running on your host computer.

The user interface is slow and the web browser process consumes a lot of CPU power: make sure that the hardware acceleration is enabled for the web browser that is used for LabOne. For the Windows operating system, the hardware acceleration can be enabled in Control Panel\Display \Screen Resolution. Go to Advanced Settings and then Trouble Shoot. In case you use a NVIDIA graphics card, you have to use the NVIDIA control panel. Go to Manage 3D Settings, then Program Settings and select the program that you want to customize.

1.7.2. Location of the log files

On Windows, the log files can be accessed through the start menu (`All apps/all programs → Zurich Instruments → Logs`).

For Windows 7, 8, and 10 the log files are located in the following directories:

- LabOne Data Server: `C:\Users\[USER]\AppData\Local\Temp\Zurich Instruments\LabOne\ziDataServerLog`
- LabOne Web Server: `C:\Users\[USER]\AppData\Local\Temp\Zurich Instruments\LabOne\ziWebServerLog`

On Windows XP:

- LabOne Data Server: `C:\Documents and Settings\[USER]\Local Settings\Temp\Zurich Instruments\LabOne\ziDataServerLog`
- LabOne Web Server: `C:\Documents and Settings\[USER]\Local Settings\Temp\Zurich Instruments\LabOne\ziWebServerLog`

1.7.3. Windows .NET Framework Requirement

The Zurich Instruments LabOne software installer requires the Microsoft .NET Framework to be installed on Windows systems. This is normally already installed on most Windows systems but may need to be additionally installed on some computers running Windows XP and Vista. If the .NET Framework is not available a message will be shown that this requirement is missing when the LabOne installer is started.

It is possible to check whether and which version of the Microsoft .NET Framework is installed on your system under Windows Start -> Control panel -> Add and Remove Programs. The minimum requirement is Microsoft .NET Framework 3.5 Service Pack 1. In case the required version is not installed, it can be installed through Windows Update tool (Windows Start -> Control panel -> Windows Update).

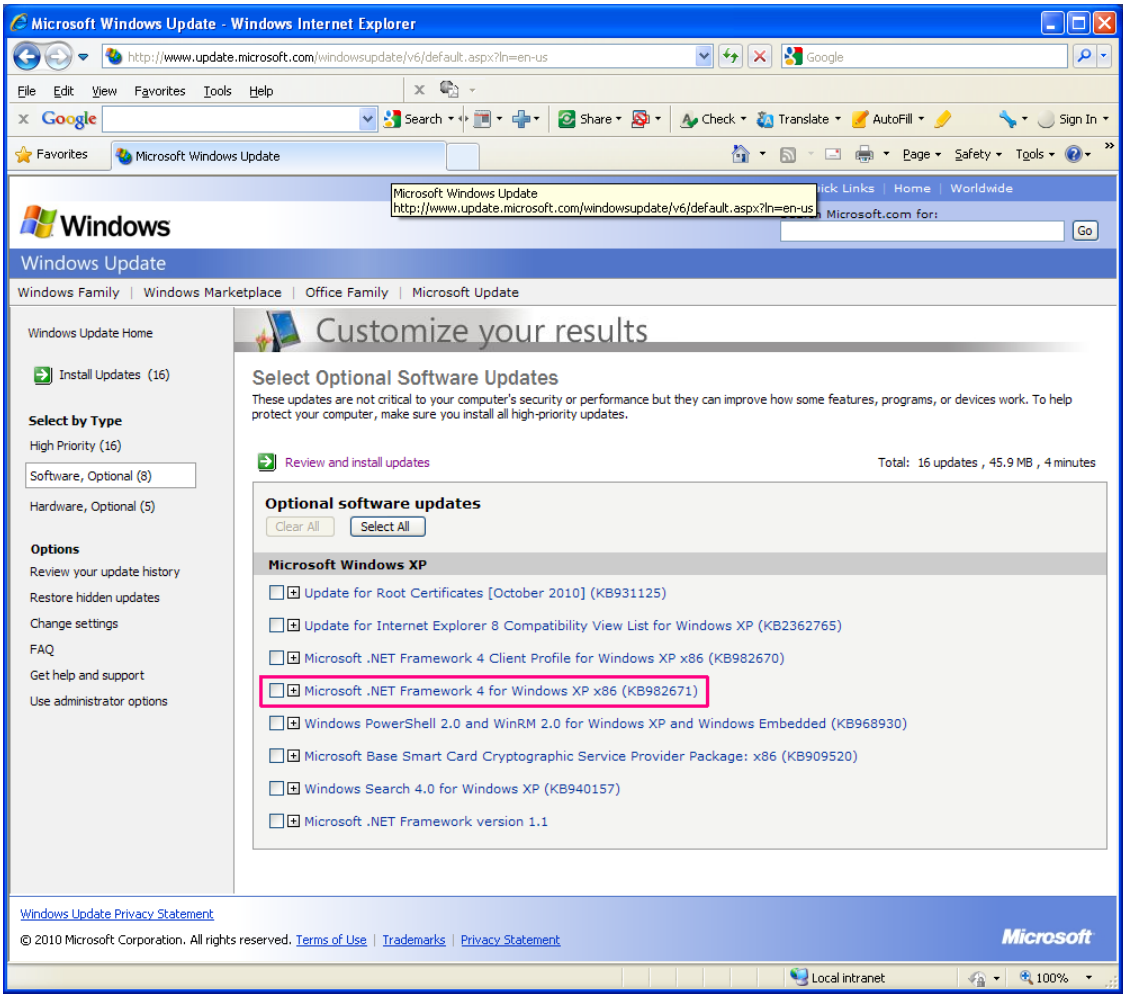


Figure 1.24. Installation of the .NET Framework.

Chapter 2. Functional Overview

This chapter provides the overview of the features provided by the UHF Instrument. The first section contains the description of the graphical overview and the hardware and software feature list. The next section details the front panel and the back panel of the measurement instrument. The following section provides product selection and ordering support.

2.1. Features

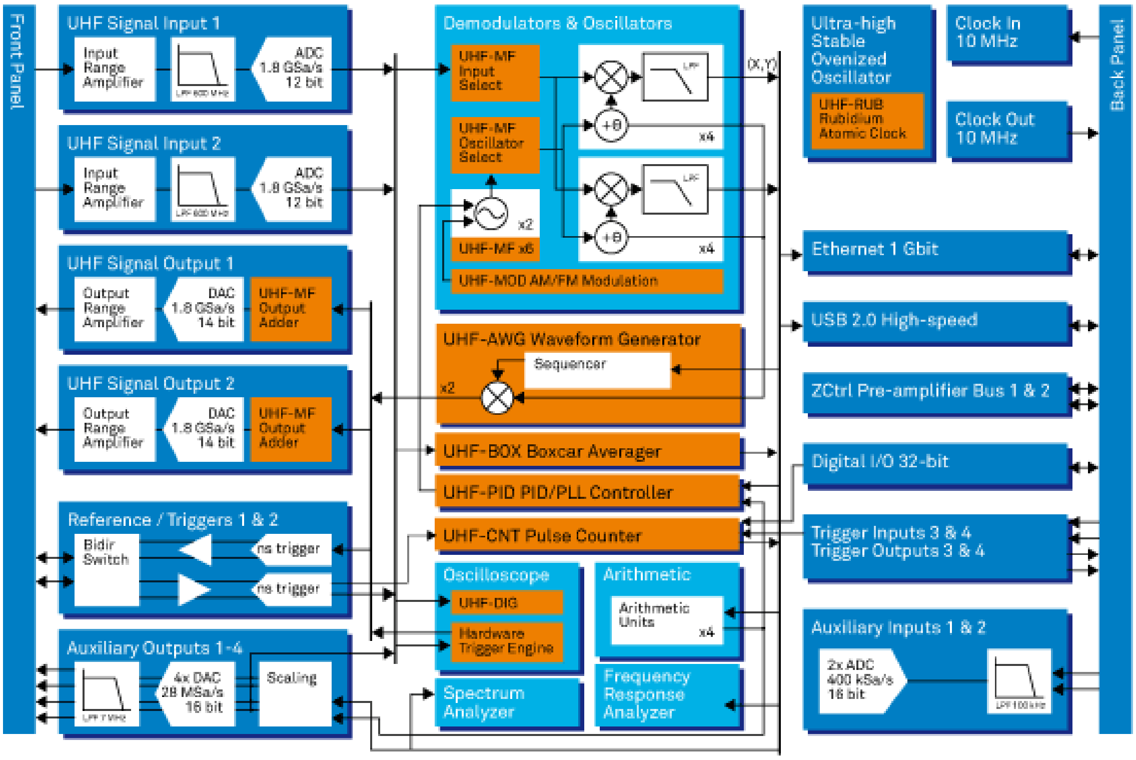


Figure 2.1. UHF Instrument overview

The UHF Instrument according to Figure 2.1 consists of several internal units (light blue color) surrounded by several interface units (dark blue color) and the front panel on the left-hand side and the back panel on the right-hand side. The orange blocks are optional units that can be either ordered at the beginning or upgraded later in the field (exceptions are mentioned in Section 2.5). The arrows between the panels and the interface units indicates the physical connections and the data direction flow. Only a very small subset of internal connections is depicted.

The signal of interest to be measured is often connected to one of the two UHF signal inputs where it is amplified to a defined range and digitized at very high speed. The resulting samples are fed into the digital signal processor consisting of up to 8 dual-phase demodulators. The output samples of the demodulators flow into one digital interface to be transferred to a host computer (LAN and USB interfaces) or are available on the auxiliary outputs on the front panel of the UHF Instrument.

The numerical oscillators generate sine and cosine signal pairs that are used for the demodulation of the input samples and also for the generation of the UHF output signals. For this purpose, the Output Adder can generate a linear combination of the oscillator outputs to generate a multi-frequency output signal: digital to analog conversion and signal scaling (range) are supported.

Hardware trigger and reference signals are used for various purposes inside the instrument, such as triggering demodulation, triggering oscilloscope data acquisition, or to generate external reference clocks or triggering signals to other equipment.

Lock-in Operating Modes

- Internal reference mode
- External reference mode

- Auto reference mode
- Dual-lock-in operation (two independent lock-in amplifiers in the same box)
- Triple-harmonic mode (simultaneous measurement at three harmonic frequencies)
- Arbitrary frequency mode (optional, simultaneous measurement at six arbitrary frequencies)

Ultra-high-frequency Signal Inputs

- 2 low-noise UHF inputs, single-ended, 600 MHz bandwidth
- Variable input range
- Switchable input impedance
- Selectable AC/DC coupling

Ultra-high-frequency Signal Outputs

- 2 low-distortion UHF outputs, single-ended, 600 MHz bandwidth
- Variable output range

Demodulators & Reference

- Up to 8 dual-phase demodulators
- Up to 8 programmable numerical oscillators
- Up to 2 external reference signals
- Up to 4 input and up to 4 output trigger signals
- Individually programmable demodulator filters
- 128-bit internal processing
- 64-bit resolution demodulator sample
- 48-bit internal reference resolution

Auxiliary Input and Outputs

- 4 auxiliary outputs, user defined signals
- 2 auxiliary inputs, general purpose

High-speed Connectivity

- USB 2.0 high-speed 480 Mbit/s host interface
- LAN 1 Gbit/s controller interface
- DIO: 32-bit digital input-output port
- ZCtrl: 2 ports peripheral control
- Clock input connector (10 MHz)
- Clock output connector (10 MHz)

Extensive Time and Frequency Domain Analysis Tools

- Numeric tool
- Oscilloscope
- Frequency response analyzer
- FFT spectrum analyzer
- ZoomFFT spectrum analyzer

- Spectroscope
- SW trigger

Software Features

- Web-based, high-speed user interface with multi-instrument control
- Data server with multi-client support
- API for C, LabVIEW, MATLAB, Python based instrument programming

2.2. Front Panel Tour

The front panel BNC connectors and control LEDs are arranged as shown in Figure 2.2 and listed in Table 2.1.

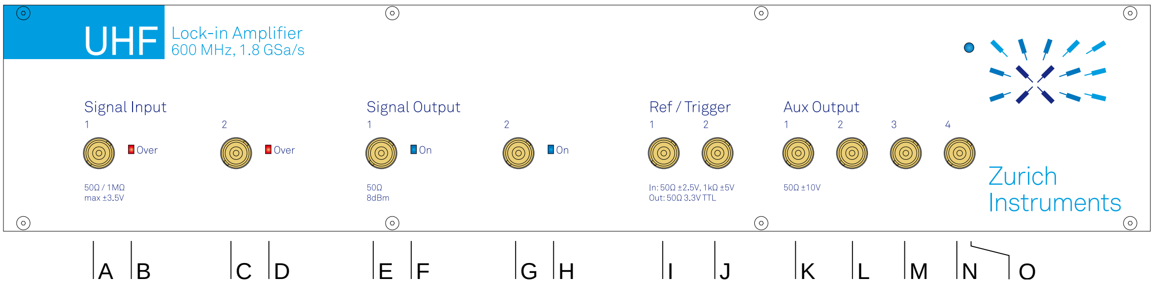


Figure 2.2. UHF Instrument front panel

Table 2.1. UHF Instrument front panel description

Position	Label / Name	Description
A	Signal Input 1	single-ended UHF input
B	Signal Input 1 Over	this red LED indicates that the input signal saturates the A/D converter and therefore the input range must be increased or the signal must be attenuated
C	Signal Input 2	single-ended UHF input
D	Signal Input 2 Over	this red LED indicates that the input signal saturates the A/D converter and therefore the input range must be increased or the signal must be attenuated
E	Signal Output 1	single-ended UHF output
F	Signal Output 1 ON	this blue LED indicates that the signal output is actively driven by the instrument
G	Signal Output 2	single-ended UHF output
H	Signal Output 2 ON	this blue LED indicates that the signal output is actively driven by the instrument
I	Ref / Trigger 1	analog reference input, TTL reference output, or bidirectional digital TTL trigger
J	Ref / Trigger 2	analog reference input, TTL reference output, or bidirectional digital TTL trigger
K	Aux Output 1	this connector provides an user defined signal, often used to output demodulated samples (X,Y) or (R,Θ)
L	Aux Output 2	this connector provides an user defined signal, often used to output demodulated samples (X,Y) or (R,Θ)
M	Aux Output 3	this connector provides an user defined signal, often used to output demodulated samples (X,Y) or (R,Θ)
N	Aux Output 4	this connector provides an user defined signal, often used to output demodulated samples (X,Y) or (R,Θ)
O	Power	this LED indicates that the instrument is powered
		color blue: the device has an active connection over USB or Ethernet

Position	Label / Name	Description
		color orange: indicates ready to connect. The device is ready for connection over USB or Ethernet. The internal auto calibration process is also indicated by an orange LED
		color orange blinking: device is in start-up mode and waiting for an IP address. As long as the device does not have a dynamic IP address or does use its static default address a connection attempt over Ethernet will fail

2.3. Back Panel Tour

The back panel is the main interface for power, control, service and connectivity to other ZI instruments. Please refer to Figure 2.3 and Table 2.2 for the detailed description of the items.

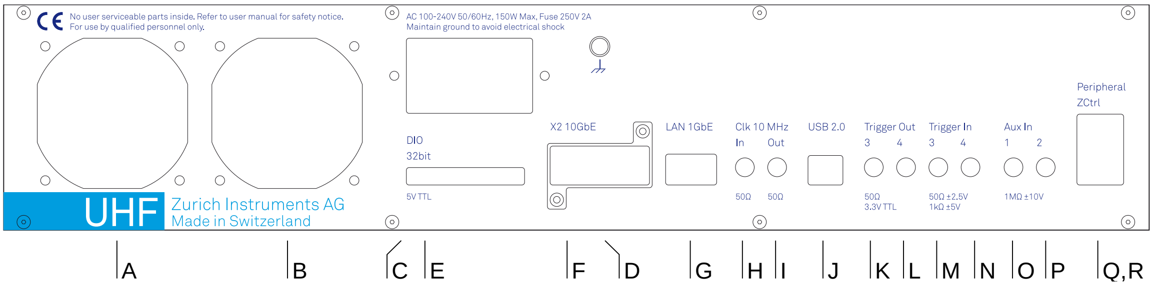


Figure 2.3. UHF Instrument back panel

Table 2.2. UHF Instrument back panel description

Position	Label / Name	Description
A	-	ventilator (important: keep clear from obstruction)
B	-	ventilator (important: keep clear from obstruction)
C	Power inlet	power inlet with ON/OFF switch
D	Earth ground	4 mm banana jack connector for earth ground, electrically connected to the chassis and the earth pin of the power inlet
E	DIO	32-bit digital input/output connector
F	X2 10GbE	10 Gbit LAN connector
G	LAN 1GbE	1 Gbit LAN connector
H	Clk 10 MHz In	clock input (10 MHz) to be used for synchronization from external instruments
I	Clk 10 MHz Out	clock output (10 MHz) to be used for synchronization of external instruments
J	USB	universal serial bus host computer connection
K	Trigger Out 3	digital TTL trigger output - note: some UHF Instruments indicate Trigger 1 on the back panel instead of Trigger 3
L	Trigger Out 4	digital TTL trigger output - note: some UHF Instruments indicate Trigger 2 on the back panel instead of Trigger 4
M	Trigger In 3	digital trigger input - note: some UHF Instruments indicate Trigger 1 on the back panel instead of Trigger 3
N	Trigger In 4	digital trigger input - note: some UHF Instruments indicate Trigger 2 on the back panel instead of Trigger 4
O	Aux In 1	auxiliary input
P	Aux In 2	auxiliary input
Q	ZCtrl 1	peripheral pre-amplifier power & control bus - attention: this is not an Ethernet plug, connection to an Ethernet network might damage the instrument
R	ZCtrl 2	peripheral pre-amplifier power & control bus - attention: this is not an Ethernet plug, connection to an Ethernet network might damage the instrument

2.4. Signalling pathways diagram

The following diagram illustrates the UHF's various signal inputs, signal outputs, functional blocks along with the multitude of signalling pathways inside the instrument and towards the host computer.

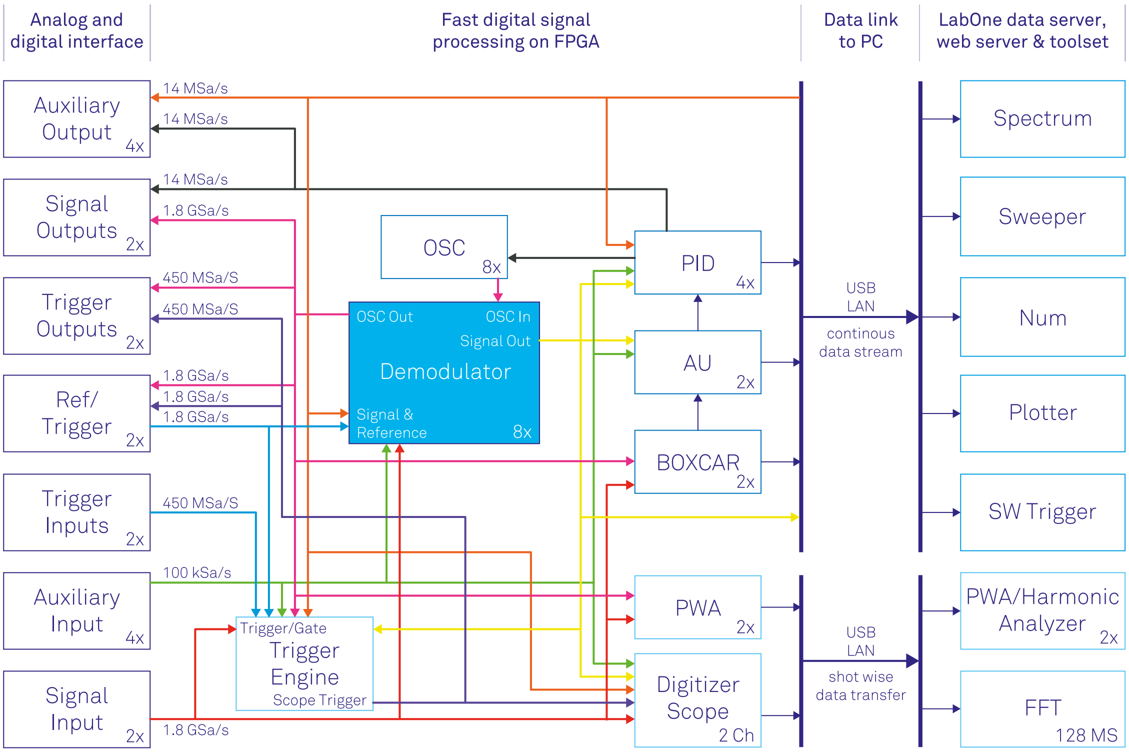


Figure 2.4. UHF Instrument main functional blocks and associated signal pathways

The main goal is to illustrate how much complexity can be absorbed by a single instrument and to inspire users finding our new uses cases by combining the different entities in new ways. The colors of the signal paths are arbitrary and meant to increase contrast but have no technical meaning. Also the plot neither aims for completeness or ultimate accuracy.

2.5. Ordering Guide

Table 2.3 provides an overview of the available UHF products. Upgradeable features are options that can be purchased anytime without need to send the Instrument to Zurich Instruments.

Table 2.3. UHF Instrument product codes for ordering

Product code	Product name	Description	Upgrade in the field possible
UHFLI	UHFLI Lock-in Amplifier	base product	-
UHF-PID	UHF-PID Quad PID/PLL Controller	option	yes
UHF-DIG	UHF-DIG Digitizer	option	yes
UHF-MF	UHF-MF Multi-frequency	option	yes
UHF-MOD	UHF-MOD AM/FM Modulation	option	yes
UHF-BOX	UHF-BOX Boxcar Averager	option	yes
UHF-RUB	UHF-RUB Rubidium Atomic Clock	option	no

Table 2.4. Product selector

Feature	UHFLI	UHFLI + UHF-MF	UHFLI + UHF-PID	UHFLI + UHF-MF + UHF-PID
Internal reference mode	yes	yes	yes	yes
External reference mode	yes	yes	yes	yes
Auto reference mode	yes	yes	yes	yes
Dual-channel operation (2 independent measurement units)	yes	yes	yes	yes
Signal generators	2	2	2	2
Superposed output sinusoidals per generator	1	up to 8	1	up to 8
Quad-harmonic mode	yes	yes	yes	yes
Multi-frequency mode	-	yes	-	yes
Arbitrary frequency mode	-	yes	-	yes
Number of demodulators	8	8	8	8
Simultaneous frequencies	2	8	2	8
Simultaneous harmonics	4+4	-	4+4	-
External references	2	2	2	2
PID controllers	-	-	4	4
600 MHz, 1.8 GSa/s	yes	yes	yes	yes
Dynamic reserve	100 dB	100 dB	100 dB	100 dB
Lock-in range	600 MHz	600 MHz	600 MHz	600 MHz
USB 2.0 480 Mbit/s	yes	yes	yes	yes
LAN 1 Gbit/s	yes	yes	yes	yes

Chapter 3. Tutorials

The tutorials in this chapter have been created to allow users to become more familiar with the basic technique of lock-in amplification, the operation of host-based lock-in amplifiers, the LabOne web browser based user interface, as well as some more advanced lock-in measurement techniques. In order to successfully carry out the tutorials, users are required to have certain laboratory equipment and basic equipment handling knowledge. The equipment list is given below.

Note

For all tutorials, you must have LabOne installed as described in the Getting Started Chapter.

- 1 USB 2.0 cable, 1 LAN cable (supplied with your UHFLI Instrument)
- 3 BNC cables
- SMA cable and adaptors
- 1 male BNC shorting cap (optional)
- 1 oscilloscope (optional)
- 1 BNC T-piece (optional)
- 1 resonator (for the PLL tutorial)

3.1. Simple Loop

Note

This tutorial is applicable to all UHF Instruments. No specific options are required. If the UHF-MF Multi-frequency option is installed then some of the required settings will differ from those indicated below.

3.1.1. Goals and Requirements

This tutorial is for people with no or little prior experience with Zurich Instruments lock-in amplifiers. By using a very basic measurement setup, this tutorial shows the most fundamental working principles of an UHF instrument and the LabOne UI in a step-by-step hands on approach.

There are no special requirements for this tutorial.

3.1.2. Preparation

In this tutorial, you are asked to generate a signal with the UHFLI Instrument and measure that generated signal with the same instrument. This is done by connecting Signal Output 1 to Signal Input 1 with a short BNC cable (ideally < 30 cm). Alternatively, it is possible to connect the generated signal at Signal Output 1 to an oscilloscope by using a T-piece and an additional BNC cable. Figure 3.1 displays a sketch of the hardware setup.

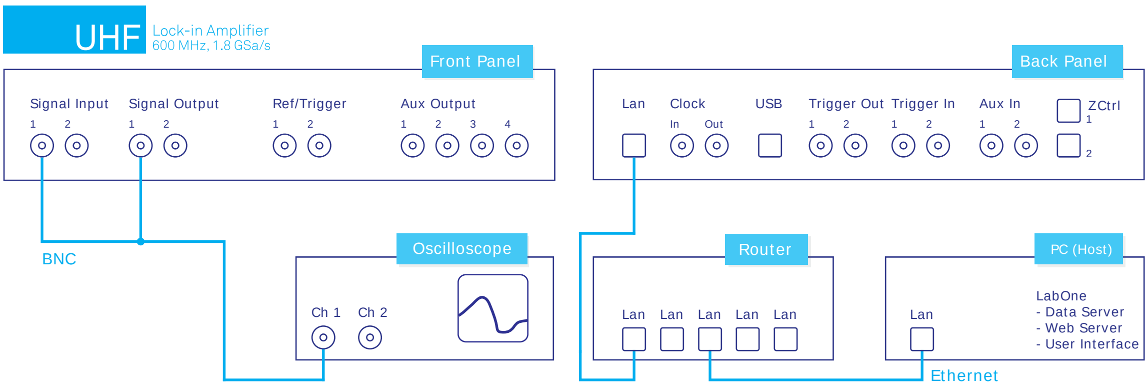


Figure 3.1. Tutorial simple loop setup (LAN connection shown)

Note

This tutorial is for all UHF units with lock-in capability irrespective of which particular option set is installed. (Note that if the UHF-MF Multi-frequency option is installed there is slight difference in the test signal generation procedure, section 3.1.3).

Connect the cables as described above. Make sure that the UHF unit is powered on and then connect the UHF directly by USB to your host computer or by Ethernet to your local area network (LAN) where the host computer resides. Start the LabOne User Interface UHF from the Windows start menu. The LabOne Data Server UHF and the LabOne Web Server are automatically started and run in the background.

3.1.3. Generate the Test Signal

Perform the following steps in order to generate a 30 MHz signal of 0.5 V peak amplitude on Signal Output 1.

1. Change the frequency value of oscillator 1 (Lock-in tab, Oscillators section) to 30 MHz: click on the field, enter 30000000 or 30 M in short and press either <TAB> or <ENTER> on your keyboard to activate the setting.
2. In the Signal Outputs section (right hand side on the Lock-in tab), set the Range pull-down to 1.5 V, the Offset to 0 V and the amplitude to 500 mV for Output 1.
3. By default all physical outputs of the UHF are inactive to prevent damage to connected circuits. Now it is time to turn on the main output switch by clicking on the button labeled "On". The switch turns to blue indicates now "On"
4. If you have an oscilloscope connected to the setup, you should now be able to see the generated signal.

Table 3.1 quickly summarizes the instrument settings to be made.

Table 3.1. Settings: generate the reference signal

Tab	Section	#	Label	Setting / Value / State
Lock-in	Oscillator	1	Frequency	30 MHz
Lock-in	Output	1	Amplitude	500 mV
Lock-in	Output	1	Offset	0 V
Lock-in	Output	1	On	On

3.1.4. Check the Test Input Signal

Next, you adjust the input parameters range, impedance and coupling to match the following values:

Table 3.2. Settings: generate the reference signal

Tab	Section	#	Label	Setting / Value / State
Lock-in	Signal Inputs	1	Range	1 V
Lock-in	Signal Inputs	1	Scaling	1 V / V
Lock-in	Signal Inputs	1	AC	On
Lock-in	Signal Inputs	1	50 Ω	On

The range setting ensures that the analog amplification on the Signal Input 1 is set such that the dynamic range of the input high-speed digitizer is optimal without clipping the signal. The graphical range indicator next to the numerical range setting shows about 50% usage of the possible dynamic range.

The incoming signal can now be observed over time by using the Scope Tab. A Scope view can be placed in the web browser by clicking on the icon in the left sidebar or by dragging the Scope Icon to one of the open Tab Rows. Choose the following settings on the Scope Tab to display the signal entering Signal Input 1:

Table 3.3. Settings: generate the reference signal

Tab	Section	#	Label	Setting / Value / State
Scope	Horizontal		Sampling Rate	1.8 GHz

Tab	Section	#	Label	Setting / Value / State
Scope	Horizontal		Length	2560 pts
Scope	Vertical		Channel 1	Signal Input 1
Scope	Trigger		Enable	On
Scope	Trigger		Level	0 V

The Scope tool now displays single shots of Signal Input 1 with a temporal distance given by the Hold off Time. The scales on top and on the right of the graphs indicate the zoom level for orientation. The icons on the left and below the figure give access to the main scaling properties and allow to store the measurement data as a SVG image file or plain data text file. Moreover, panning can be achieved by clicking and holding the left mouse button inside the graph while moving the mouse.

Note

Zooming in and out along the horizontal dimension can by achieved with the mouse wheel, for the vertical zoom the shift key needs to be pressed and again the mouse wheel can by used for adjustments.

Having set the Input Range to 1 V ensures that no signal clipping occurs. If you set the Input Range to 0.2 V, clipping can be seen immediately on the scope window accompanied by a red error flag on the status bar in the lower right corner of the LabOne User Interface. At the same time, the LED next to the Signal Input 1 BNC connector on the instrument's front panel will turn red. The error flag can be cleared by pressing the clear button marked with the letter C on the right side of the status bar after setting the Input Range back to 1 V.

The Scope is a very handy tool for checking quickly the quality of the input signal. Users can either use Scope to adjust the optimal input range setting or to check if the software trigger level is set correctly. The Scope window can display up to 64 k points/samples on the web browser. For the full description of the Scope tool please refer to the functional description.

3.1.5. Measure the Test Input Signal

Now, you are ready to use UHFLI to demodulate the input signal and measure its amplitude and phase. You will use two tools of the LabOne User Interface: Numerical and the Plotter.

First, adjust the following parameters on the Lock-in Tab for demodulator 1 (or choose another demodulator if desired):

Table 3.4. Settings: generate the reference signal

Tab	Section	#	Label	Setting / Value / State
Lock-in	Demodulators	1	Harm	1
Lock-in	Demodulators	1	Phase	0
Lock-in	Demodulators	1	Input	Sig In 1
Lock-in	Demodulators	1	Sinc	OFF
Lock-in	Demodulators	1	Order	3 (18 dB/Oct)
Lock-in	Demodulators	1	TC / BW 3dB	9.3 ms / 8.7 Hz
Lock-in	Demodulators	1	Rate	100 Sample/s (automatically adjusted to 107 Sample/s)

Tab	Section	#	Label	Setting / Value / State
Lock-in	Demodulators	1	Trigger	Continuous
Lock-in	Demodulators	1	Enable	ON

These above settings configure the demodulation filter to the third-order low-pass operation with a 9 ms integration time constant. Alternatively, the corresponding bandwidths BW NEP or BW 3 dB can be displayed and entered. The output of the demodulator filter is read out at a rate of 107 Hz, implying that 107 data samples are sent to the host PC per second with equidistant spacing. These samples can be viewed in the Numerical and the Plotter tool which we will examine now.

The Numerical tool provides the space for 16 or more measurement panels. Each of the panels has the option to display the samples in the Cartesian (X,Y) or in the polar format (R,Θ) plus other quantities such as the Demodulation Frequencies and Auxiliary Inputs. The unit of the (X,Y,R) values are by default given in V_{RMS}. The scaling and the displayed unit can be altered in the Signal Input section of the Lock-in Tab. The numerical values are supported by graphical bar scale indicators to achieve better readability, e.g. for alignment procedures. Display zoom is also available by holding the control key pressed while scrolling with the mouse wheel. Certain users may observe rapidly changing digits. This is due to the fact that you are measuring thermal noise that maybe in the μV or even nV range depending on the filter settings. This provides a first glimpse of the level of measurement precision capable with your UHFLI instrument.

If you wish to play around with the settings, you can now change the amplitude of the generated signal, and observe the effect on the demodulator output.

Next, we will have a look at the Plotter tool that allows users to observe the demodulator signals as a function of time. It is possible to adjust the scaling of the graph in both directions, or make detailed measurements with 2 cursors for each direction. Signals of the same signal property are automatically added to the same default y-axis group. This ensures that the axis scaling is identical. Signals can be moved between groups. More information on y-axis groups can be found in the section called “Plot area elements”.

Try zooming in along the time dimension using the mouse wheel or the icons below the graph to display about one second of the data stream. While zooming in, the mode in which the data are displayed will change from a min-max envelope plot to linear point interpolation depending on the density of points along the x axis as compared to the number of pixels available on the screen.

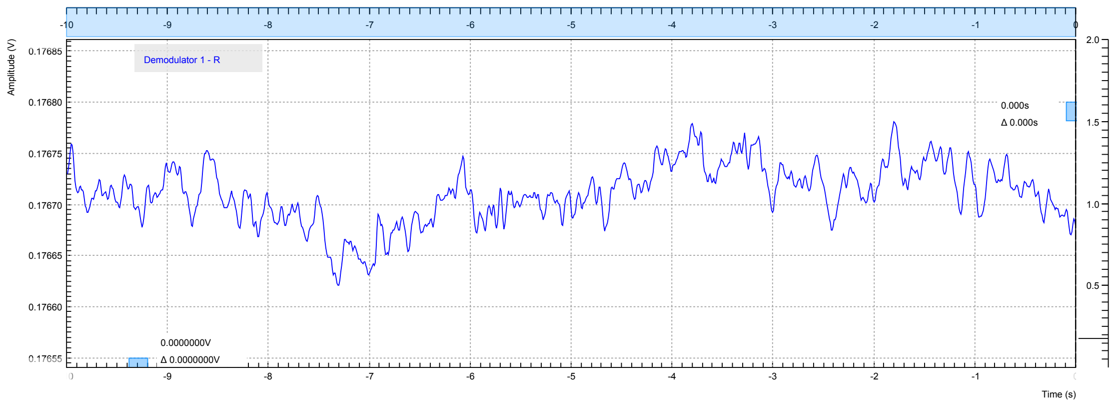


Figure 3.2. LabOne User Interface Plotter displaying demodulator results continuously over time (roll mode)

Data displayed in the Plotter can also be saved continuously to the computer memory. Please have a look at Section 4.1.4 for a detailed description of the data saving and recording functionality. Instrument and user interface settings can be saved and loaded using the Config tab (Settings section).

3.1.6. Different Filter Settings

As next step in this tutorial you will learn to change the filter settings and see their effect on the measurement results. For this exercise, use the second demodulator with the same settings as the first except in changing the time constant of the integration to 1 ms which corresponds to a 3 dB bandwidth of 83 Hz.

Table 3.5. Settings: generate the reference signal

Tab	Section	#	Label	Setting / Value / State
Lock-in	Demodulators	1	Order	3 (18 dB/Oct)
Lock-in	Demodulators	1	TC / BW 3dB	1 ms / 83 Hz

Lowering the time constant reduces the filter integration time of the demodulators. This will in turn "smooth out" the demodulator outputs and hence increases available time resolution. Note that it is recommended to keep the sample rate 7 to 10 times the filter 3 dB bandwidth. The sample rate will be rounded off to the next available sampling frequency. For example, typing 1 k in the Rate field will result in 1.7 kSa/s which is sufficient to not only properly resolve the signal, but also to avoid aliasing effects. Figure 3.3 shows data samples displayed for the two demodulators with different filter settings described above.

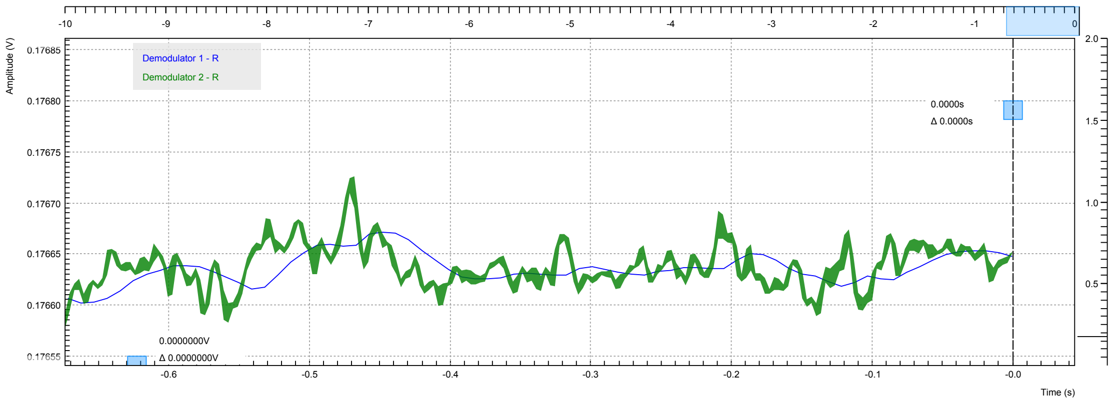


Figure 3.3. LabOne User Interface Plotter: Demodulator 1 (TC = 9.3 ms, blue), Demodulator 2 (TC = 1 ms, green)

Moreover, you may for instance "disturb" the demodulator with a change of test signal amplitude, for example from 0.5 V to 0.7 V and vice-versa. The green plot will go out of the display range which can be re-adjusted by clicking the "Auto Scale" button, cf. Section 4.1.3. With a large time constant, the demodulated data change slower in reaction to the change in the input signal compared to a small time constant. In addition, the number of stable significant digits in the Numerical tool will also be higher with a high time constant.

3.2. External Reference

Note

This tutorial is applicable to all UHF Instruments. No specific options are required. N.B. if the UHF-MF Multi-frequency option is installed then some of the required settings will differ from those indicated below.

3.2.1. Preparation

This tutorial explains how to perform demodulation using an external reference frequency. An external reference will be simulated by using one of the UHFLI internal oscillators. The signal from this internal oscillator will be fed to one of the signal outputs and then fed back in using various connections in order to reference another internal oscillator used for demodulation.

First of all, connect the Signal Output 2 connector to both Signal Input 1 and to the Ref/Trigger Input 1 connector using two BNC cables and a BNC T-junction. The measurement setup is shown in the following figure.

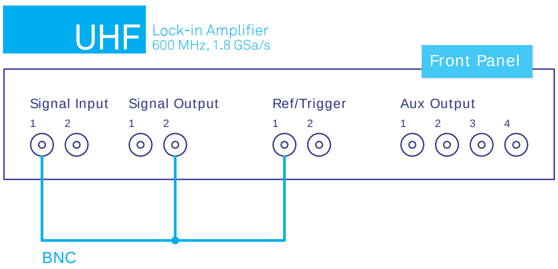


Figure 3.4. External reference on Signal Input 2

Connect the cables as described above. Make sure the UHFLI is powered on, and then connect the UHFLI through the USB to your PC, or to your local area network (LAN) where the host computer resides. After starting LabOne the default web browser opens with the LabOne graphical user interface.

The tutorial can be started with the default instrument configuration (e.g. after a power cycle) and the default user interface settings (i.e. as is after pressing F5 in the browser).

3.2.2. Generate the Test Signal

In this section you generate a 30.0 MHz signal oscillating between 0 V and +/-0.5 V on Output 2 for use as the external reference. The Lock-in settings for generating and analyzing the test signal are shown in the following table.

Table 3.6. Settings: generate the reference signal

Tab	Section	#	Label	Setting / Value / State
Lock-in	Output	2	Range	1.5 V
Lock-in	Output	2	Amplitude	1.0 V
Lock-in	Output	2	Offset	0.0
Lock-in	Output	2	On	On

Tab	Section	#	Label	Setting / Value / State
Lock-in	Oscillators	2	Frequency	30 MHz
Lock-in	Demodulators	5	Enable	On
Lock-in	Input	2	Range	1.5 V
Lock-in	Input	2	AC	ON
Lock-in	Input	2	50 Ω	ON

To quickly visualize the signal, we can reconnect the Signal Output 2 with Signal Input 2 and check the signal shape on the Scope using the following settings.

Table 3.7. Settings: acquire the reference signal

Tab	Section	#	Label	Setting / Value / State
Scope	Vertical		Channel 1	Signal Input 2
Scope	Trigger		Trigger	ON
Scope	Trigger		Signal	Signal Input 2
Scope	Trigger		Level	50 mV
Scope			Run / Stop	ON

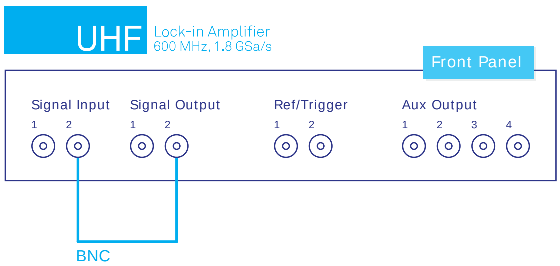


Figure 3.5. External reference on Signal Input 2

The resulting scope trace should look similar as indicated in the following screen capture.

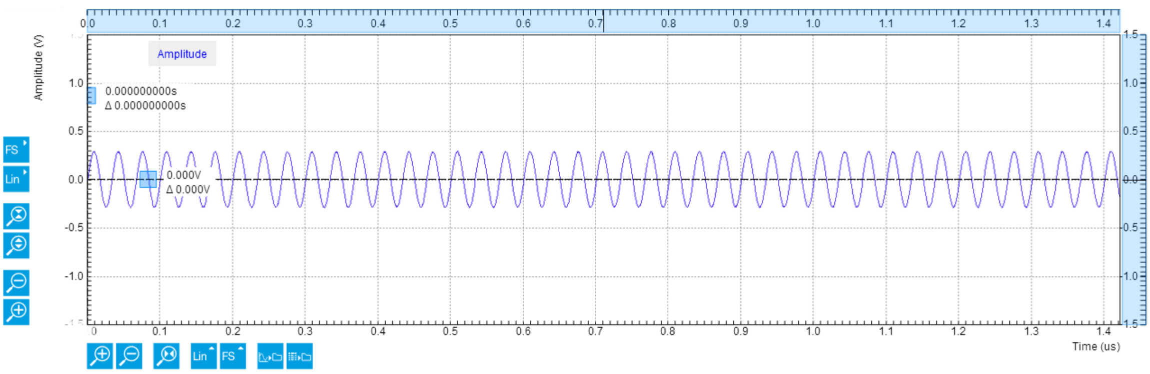


Figure 3.6. Reference signal viewed with the internal scope

Note

Alternatively, the Scope mode Frequency Domain FFT (instead of Time Domain) can be used to check the frequency content of the signal. Set the scale settings automatic for the X axis and logarithmic scale (dB) for the Y axis for convenient viewing. The averaging filter can be set Exp Moving Avg to reduce the noise floor on the display.

3.2.3. Activate the External Reference Mode

After putting back the cable as indicated in Figure 3.4 the external reference mode can be activated and output the regenerated signal of interest. The following additional settings have to be adjusted:

Table 3.8. Settings: acquire the reference signal

Tab	Section	#	Label	Setting / Value / State
Lock-in	Output	2	Range	1.5 V
Lock-in	Output	2	Offset	0 V
Lock-in	Output	2	Amplitude	1 V
Lock-in	Output	2	Enable	ON
Lock-in	Demodulator	1	Enable	ON
Lock-in	Signal Input	1	Range	1.2 V
Lock-in	Signal Input	1	AC	OFF
Lock-in	Signal Input	1	50 Ω	OFF

In general, Demodulator 4 and Demodulator 8 can be set to the external reference mode to track the external reference at Signal Input 1 and Signal Input 2, respectively. The external reference can come from the Sig In 1 and 2,Trig 1 and 2 (in the front), Trig 3 and 4 (in the back), or Aux In 3 and 4 (in the back). The 4 Auxiliary Outputs can also be chosen in the external reference mode although they are not exactly to be considered as an external reference. They are useful in the case of tandem demodulation where the result of a first lock-in operation is fed into a second lock-in, typically at a lower frequency. For this tutorial, Sig In 1 is selected as the external reference for Demodulator 4 (i.e. under the Signal column) and activated by selecting ExtRef in the (Reference) Mode column.

Table 3.9. Settings: choosing trigger source and switch to external reference mode

Tab	Section	#	Label	Setting / Value / State
Lock-in	Demodulators	4	Signal	Sig In 1
Lock-in	Demodulators	4	Mode	ExtRef

As a result the oscillator 1 frequency indicator in the Oscillator section almost immediately changes from 10 MHz to 30 MHz. Once the external reference mode has been enabled, the frequency of oscillator 1 changes continuously, adapting to the frequency of the external reference signal. This can be verified by changing the frequency of oscillator 2 and noting how the frequency of oscillator 1 follows. A green indicator appears besides the reference selection for channel 1 indicating that the instrument has locked to an external reference. Graphically, this can be nicely viewed in the Plotter by displaying the frequency of Demodulator 1 and then changing the frequency of the oscillator 2 in quantities of, say, 1 kHz:

Table 3.10. Settings: displaying demodulator reference frequency over time

Tab	Section	#	Label	Setting / Value / State
Plotter	Tree		Input Signal	/0/sample/Frequency
Plotter			Run / Stop	On

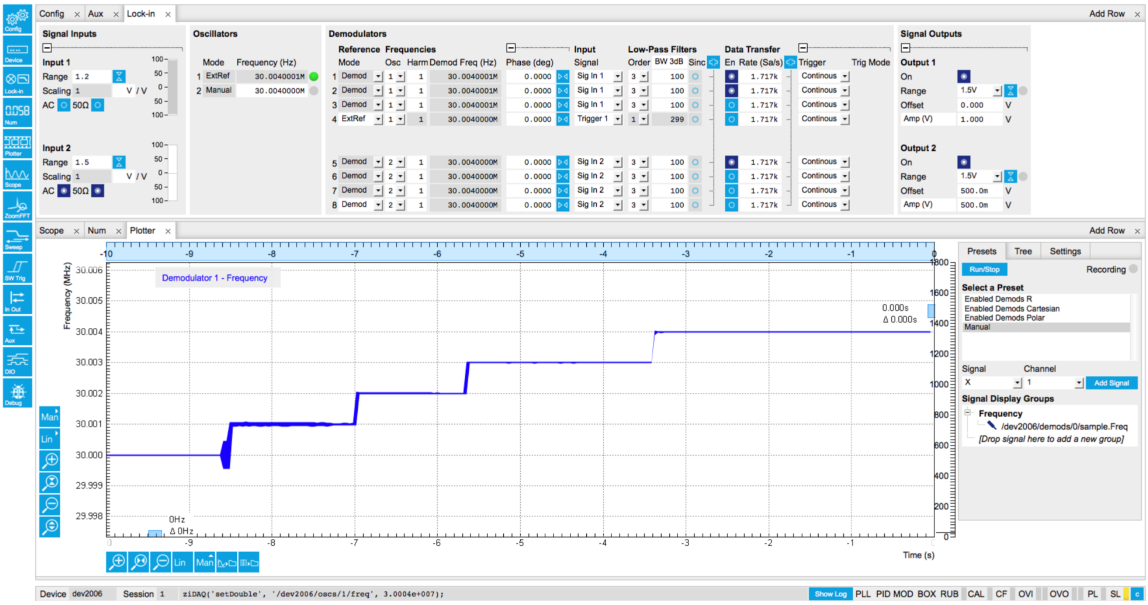


Figure 3.7. LabOne enabling external reference mode

At this point, it is worth noting that the external reference signal is never used directly for demodulation. Instead, the frequency and phase of the external reference signal is mapped to one of the internal oscillators first through an internal phase locked loop. This internal oscillator can then serve as a reference for any of the demodulators. This mapping procedure is implemented with an automatic bandwidth adjustment that assures optimum operation over the whole frequency range for a broad variety of signal qualities in terms of frequency stability as well as the signal-to-noise ratio. Over the course of automatic adjustment, the Low-Pass Filter bandwidth of the associated demodulators 4 or 8 usually ramps down until a final value is reached after a few seconds. The indicated bandwidth also marks an upper limit to the bandwidth of the phase locked loop that does the mapping of the external signal to the internal oscillator. The following figure shows a typical result in the plotter for the frequency tracking immediately after it is turned on.

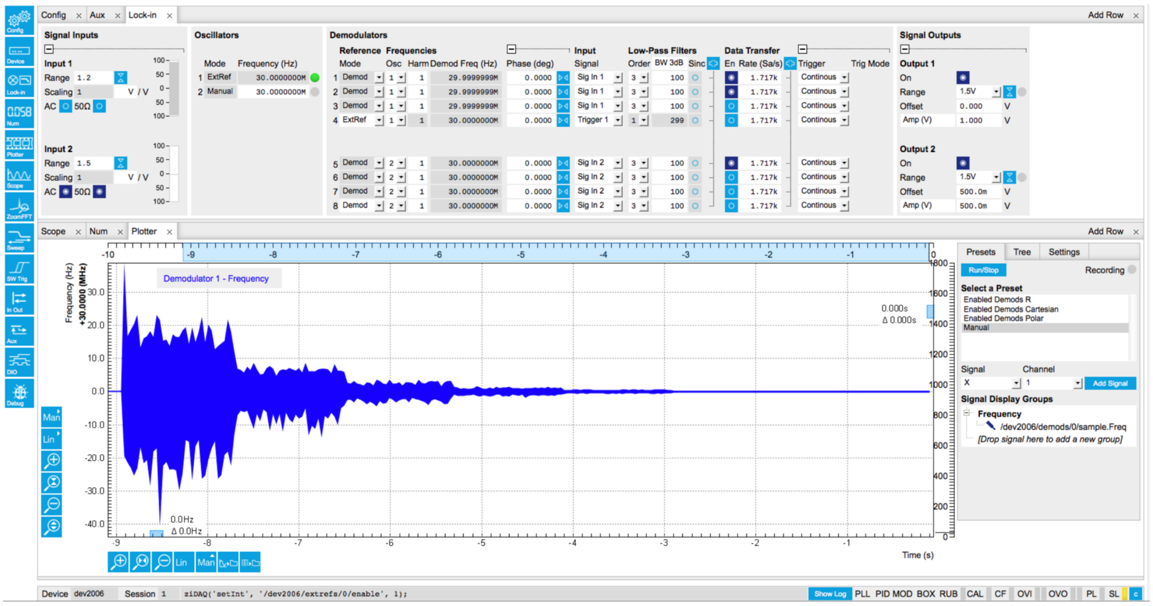


Figure 3.8. Frequency tracking of an external reference signal over time with automatic bandwidth adjustment

3.2.4. Providing the Reference Signal to Ref / Trigger Input

In this section you will slightly modify the setup to use Ref/Trigger Input 1 (instrument front side) as a entry port for the external reference instead of Signal Input 1. A sketch of the modified setup is shown in Figure 3.9.

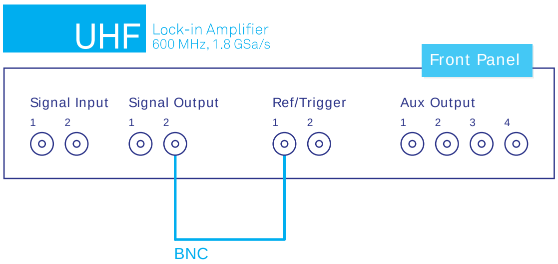


Figure 3.9. External reference using Ref/Trigger Input 1 setup

There are 2 Ref/Trigger inputs on the front side of the instrument and two more on the rear side. By using the dedicated trigger inputs, both Signal Inputs remain available for simultaneous two-input measurement . The drawback is that one cannot observe the external reference signal on the Scope tool when an REF/Trigger inputs are used.

Ref/Trigger Inputs are comparator based digital channels where the input impedance can be set to either 50 Ω or 1 kΩ in the Ref / Trigger section in the DIO tab. Moreover, a suitable Trigger threshold can be defined by adjusting the Input Level definitions.

Note

It is important to know that the trigger to discriminate the two logical states operates on the positive edge with a hysteresis of about 100 mV. Consequently, a peak-to-peak signal amplitude of minimum 200 mV should be provided as a external reference signal to guarantee reliable switching.

Note

For signal frequencies larger than 10 MHz, the 50 Ω input termination is strongly recommended to avoid signal reflections in the cable that can lead to false switching events.

The following DIO settings are used for this example:

Table 3.11. Settings: acquire the reference signal

Tab	Section	#	Label	Setting / Value / State
DIO	Ref / Trigger	1	Input Level	250 mV
DIO	Ref / Trigger	1	Coupling 50 Ω	ON
DIO	Ref / Trigger	1	Drive	OFF

When the signal is applied with a proper discrimination threshold chosen, both control LEDs will turn on to indicate that the channel alternates quickly between high-low logical states. Once this is happening, one can then select Trigger 1 as a Signal Input for demodulator 4 in order to reference oscillator 1.

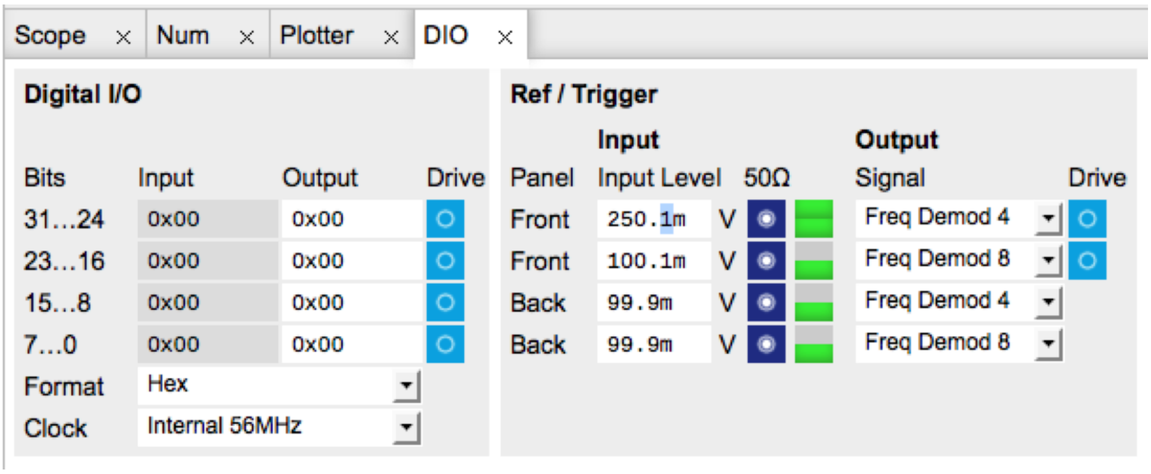


Figure 3.10. Configuring DIO 0 as reference input

The default settings are chosen such that a standard 3.3 V TTL signal can be directly attached without further adjustments. This can be easily tested by connecting a TTL reference signal to the outputs on the back panel. A sketch of the modified setup is shown on Figure 3.11. You should now see as well that the oscillator 1 now tracks the frequency generated from oscillator 2.

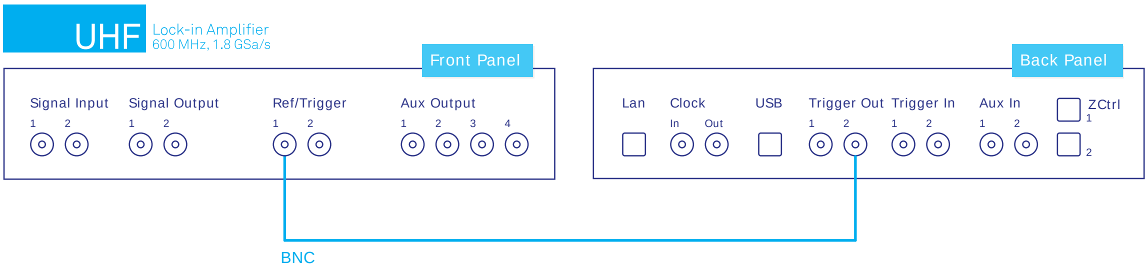


Figure 3.11. Referencing to a TTL signal using Ref/Trigger Input 1

3.2.5. Using the Ref/Trigger Input with TTL signals

In this section you will modify the setup to use Ref/Trigger Input 2 (instrument front side) as a entry port for TTL reference signal provided on Trigger Output 1 (instrument backside). A sketch of the modified setup is shown on Figure 3.12.

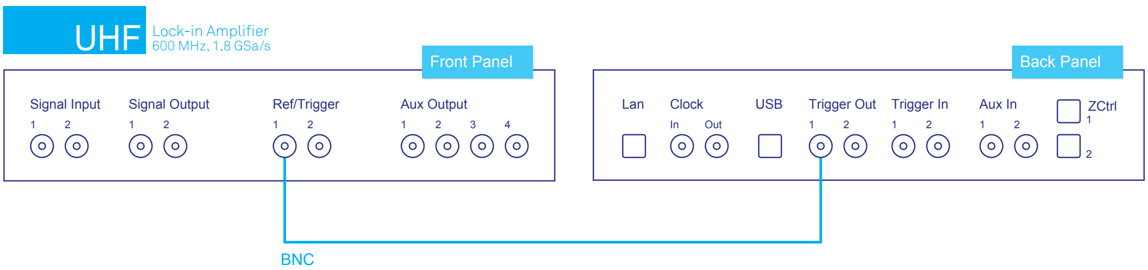


Figure 3.12. Referencing to a TTL signal using Ref/Trigger Input 1

When using the Ref/Trigger Inputs, one needs to be aware that they are comparator based digital channels where the input coupling can be selected to be either 50Ω or 1 kΩ in the Ref / Trigger section in the DIO tab. Moreover, a suitable Trigger threshold can be defined by adjusting the Input Level definitions.

Note

It is important to know that the trigger to discriminate the two logical states operates on positive slopes with a hysteresis of about 100 mV. As a consequence a peak to peak signal amplitude of minimum 200 mV should be provided as a external reference signal to guarantee reliable operation.

Note

For signal frequencies larger than 10 MHz using 50Ω input coupling is strongly recommended to avoid signal reflections in the cable that can lead to false events or measurement artifacts.

The default settings are chosen such that a standard 3.3 V TTL signal can be directly attached without further adjustments. The following DIO settings are used for this example.

Table 3.12. Settings: acquire the reference signal

Tab	Section	#	Label	Setting / Value / State
DIO	Ref / Trigger	1	Input Level	250 mV
DIO	Ref / Trigger	1	Coupling 50 Ω	ON
DIO	Ref / Trigger	1	Drive	ON

When the signal is applied and a proper discrimination threshold chosen both control LEDs are lid to indicate that the channel alternates quickly between both logical states. As soon as this is the case, one can select Trigger 2 as a Signal Input for demodulator 8 in order to reference oscillator 2 to oscillator 1.

3.3. Amplitude Modulation

Note

This tutorial is applicable to UHF Instruments with the UHF-MF Multi-frequency and the UHF-MOD AM/FM Modulation options installed.

3.3.1. Goals and Requirements

This tutorial explains how to generate an amplitude modulated (AM) signal as well as how to demodulate an AM signal by reading out amplitude and phase of the carrier and the two sidebands simultaneously. The tutorial can be done using a simple loop back connection.

3.3.2. Preparation

To perform this tutorial, one simply needs to connect a BNC cable from Signal Output 1 to Signal Input 1 as shown in Figure 3.13. This will allow the user to perform the AM modulation and demodulation in this tutorial without needing an external source.

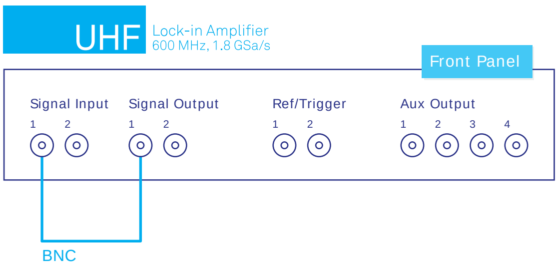


Figure 3.13. Internally generated AM signal measured on Signal Input 1

Note

This tutorial is for all UHF units with lock-in capability as well as with the UHF-MF Multi-frequency and UHF-MOD AF/FM Modulation options installed.

Connect the cables as described above. Make sure the UHFLI is powered on, and then connect the UHFLI through the USB to your PC, or to your local area network (LAN) where the host computer resides. After starting LabOne, the default web browser opens with the LabOne graphical user interface.

The tutorial can be started with the default instrument configuration (e.g. after a power cycle) and the default user interface settings (e.g. as is after pressing F5 in the browser).

3.3.3. Generate the Test Signal

In this section you will learn how to generate an AM signal with a 10.0 MHz, 1.0 V sinusoidal carrier modulated by a second 100 kHz, 500 mV sinusoid. The Lock-in tab and the MOD tab settings are shown in the following table.

Table 3.13. Settings: generate the AM signal

Tab	Section	#	Label	Setting / Value / State
MOD	Oscillators	1	Enable	ON
MOD	Oscillators	1	Carrier	AM / 10.0 M
MOD	Oscillators	1	Sideband 1	100.0 k
MOD	Input	1	Channel	Sig In 1
MOD	Generation	1	Signal Outputs	1
MOD	Generation	1	Carrier (V)	1.0 / ON
MOD	Generation	1	Modulation (V)	200.0 m / ON
Lock-in	Output	1	Range	1.5 V
Lock-in	Output	1	On	ON
Lock-in	Demodulators	1	Enable	ON
Lock-in	Demodulators	2	Enable	ON
Lock-in	Demodulators	3	Enable	ON
Lock-in	Demodulators	5	Enable	OFF
Lock-in	Input	1	Range	1.5 V
Lock-in	Input	1	50 Ω	ON

To quickly verify that the AM signal is generated correctly, we can check the spectrum of the AM signal on Signal Input 1 using the Scope tool with the following settings. The Scope basically displays the FFT spectrum of Signal Input 1. With a sampling rate of 28 MHz, it satisfies sufficiently the Nyquist rate to see the 10 MHz carrier. The 64000 points samples correspond to about 2.3 ms of the sampled duration. This should be enough to capture the frequency spectrum at kHz resolution.

Note

The maximum sample window displayed in the Scope is 64000 points.

Table 3.14. Settings: acquire the reference signal

Tab	Section	#	Label	Setting / Value / State
Scope	Horizontal		Mode	Freq Domain FFT
Scope	Horizontal		Sampling Rate	28 MHz
Scope	Horizontal		Length (pts)	64000
Scope			Run/Stop	ON

You should now observe a spectrum like the one shown in the screen capture below. All amplitudes are measured in peak values. The center carrier frequency and the sideband frequencies should have half of the generated amplitudes i.e. about 0.5 V and 50 mV, respectively. This is due to the voltage divider effect from the combination of the 50 Ω output port impedance and the 50 Ω input termination impedance. The additional 0.5 factor for the two sidebands is due to the fact that the original AM modulation signal power is shared between two sidebands.

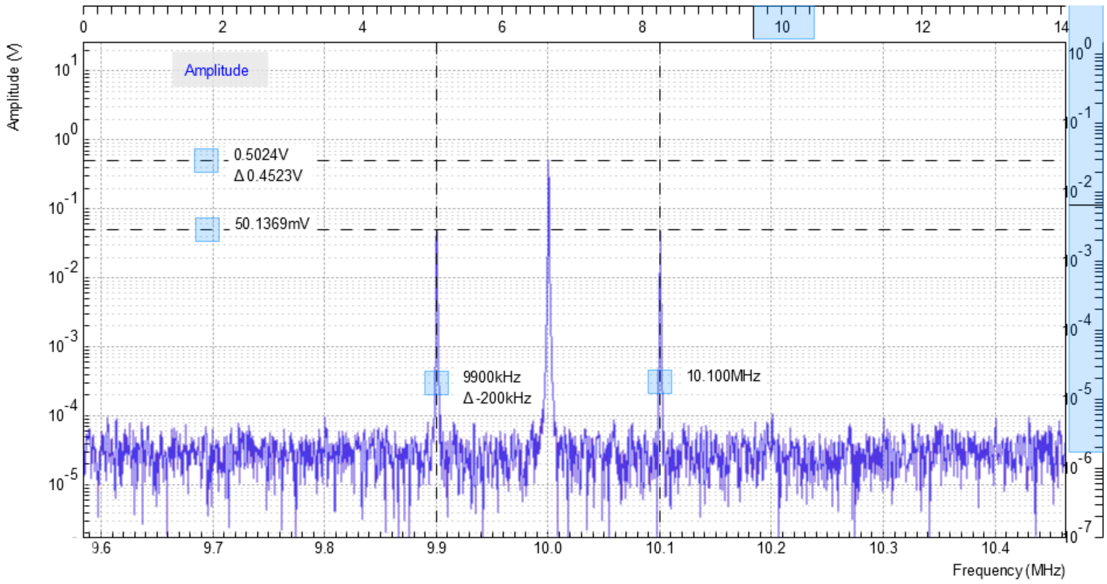


Figure 3.14. Generated AM signal with UHFLI

3.3.4. AM Demodulation Result

If you look at the Demod Freq column under the Lock-in tab, you will see that the demodulation frequencies of all three frequency components are stated clearly: 10 MHz on demodulator 1, 10.1 MHz on demodulator 2 and 9.9 MHz on demodulator 3. You can now read out simultaneously the magnitude and the phase (R,Θ) or (X, Y) of the carrier component on demodulator 1, and the upper and lower sideband components on demodulator 2 and 3, respectively. The measurement result is shown under the Numeric tab as shown in Figure 3.15

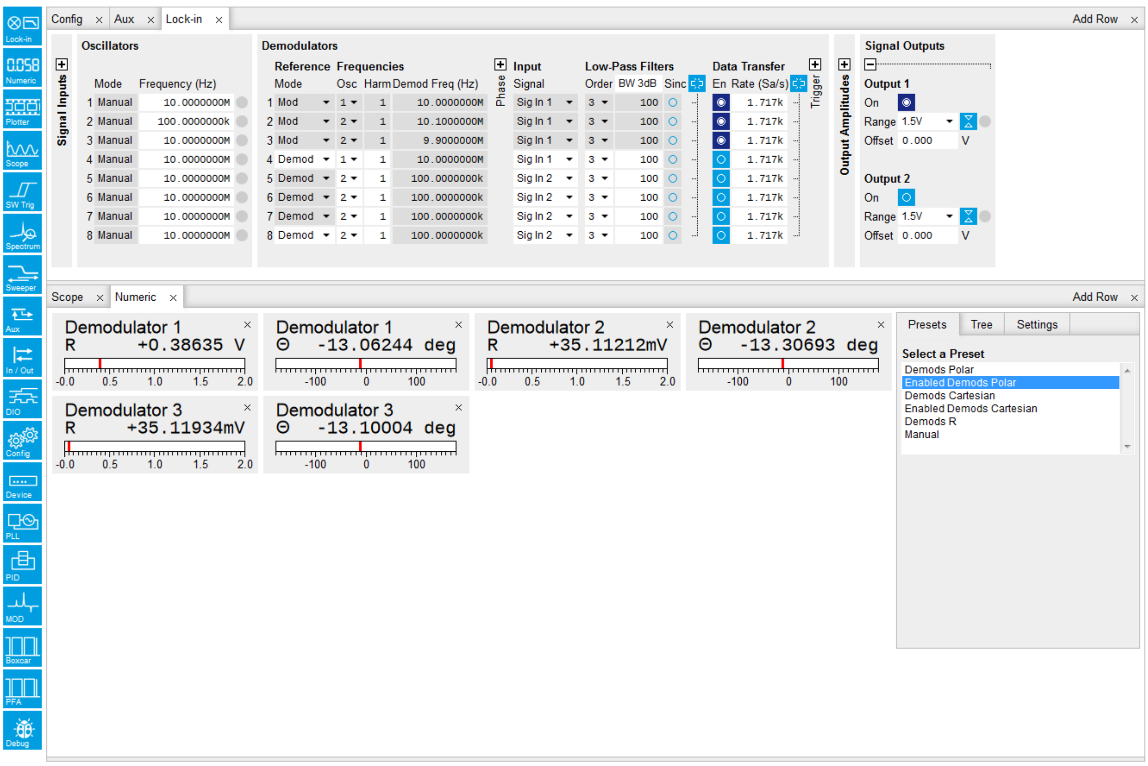


Figure 3.15. Numerical results of AM demodulation under the Numeric tab

Note

By selecting "Enable Demod Polar" in the Numeric tab, only the enabled demodulator outputs will show.

If we take the sum of the double sideband's amplitude (i.e. demodulator 2 and 3) and divide it by the amplitude of the carrier (demodulator 1), we will get an AM modulation index of $h=A_{sideband}/A_{carrier}=0.2$. This is exactly the index we had used to generate the AM signal in the MOD tab.

3.4. Phase-locked Loop

Note

This tutorial is applicable to UHF Instruments with the UHF-PID Quad PID/PLL Controller option installed.

3.4.1. Goals and Requirements

This tutorial explains how to track the resonance frequency shift of a resonator using the PLL. To perform this tutorial, one simply needs to connect a resonator between Signal Output 2 to Signal Input 2.

3.4.2. Preparation

Connect the cables and the resonator as shown in the diagram below. Make sure the UHFLI is powered on, and then connect the UHFLI through the USB to your PC, or to your local area network (LAN) where the host computer resides. After starting LabOne the default web browser opens with the LabOne graphical user interface. .

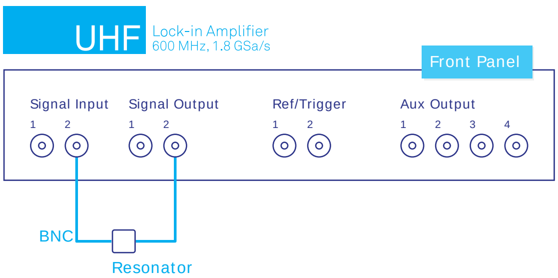


Figure 3.16. PLL connection with UHF

The tutorial can be started with the default instrument configuration (e.g. after a power cycle) and the default user interface settings (e.g. as is after pressing F5 in the browser).

3.4.3. Determine the Resonance of the Resonator

In this section you will learn first how to find the resonance of your resonator by using the frequency sweeper tool under the Sweeper tab. To get started, one could in theory define a frequency sweep range from DC to 600 MHz and slowly narrow down the range using multiple sweeps in order to find the resonance peak of interest. But in practice, it would make more sense to already have a small guess range in the span of a couple of MHz, not more. This will save the overall sweep time especially in cases where your resonator Q is low and therefore the peak would be close to the noise floor. The Sweeper tab and Lock-in tab setup is shown below. The frequency sweeper can be found under the Sweeper tab.

Table 3.15. Settings: acquire the reference signal

Tab	Section	#	Label	Setting / Value / State
Lock-in	Output Amplitudes	8	Amp 2 (V)	100.0 m / ON

Tab	Section	#	Label	Setting / Value / State
Lock-in	Signal Outputs		Output 2	ON
Lock-in	Demodulators	8	Osc	8
Lock-in	Demodulators	8	Input	Sig In 2
Lock-in	Data Transfer	8	Amp 2 (V)	ON
Sweeper	Settings		Sweep Param.	oscs/7/freq
Sweeper	Settings		Input Channel	Demod R / 8
Sweeper	Settings		Start (Hz)	1.0 M
Sweeper	Settings		Stop (Hz)	3.0 M
Sweeper	History		Length	2
Sweeper	Settings		Dual Plot	ON
Sweeper	Settings		Run/Stop	ON

In this exercise, we are using the DEMODULATOR 8 row to generate the sweep signal as well as demodulating the resonator output. The Lock-in settings ensure especially that the oscillator used both for the sweep signal and the demodulation is the same (i.e. the oscillator 2). In addition, the input must be set to Signal Input 2 as shown in the connection diagram.

Once the Sweeper Run/Stop button is clicked, the sweeper will continuously and repeatedly sweep the frequency response of the quartz oscillator. The user can then use the zoom tools to get a higher resolution on the interested resonance peak since one may have several resonance peaks in the frequency spectrum. The history length of 2 allows the user to keep on the screen one previous sweep while adjusting the zoom. To redefine the start and stop frequencies for a finer sweeper range, one needs to deactivate first the Dual Plot mode and then pres the Copy Range button. This will automatically entering the zoomed sweep window range into the Start and Stop of the swept frequency range. Remember to turn off Run/Stop button under the Sweeper tab when done.

Note

The sweep frequency resolution will get finer when zooming in horizontally using the Copy Range button even without changing the number of points.

When a resonance peak has been found, you should get a spectrum similar to two screen shots below. In this example, we have selected the resonance peak at about 2.151 MHz. The phase response of the resonator started at about 90 degrees but decreases abruptly until reaching the value of about 4.7 degrees at the resonance peak.

Note

For most resonators, a phase shift of approximately 90 degrees at resonance can be expected, if the cables are not excessively long.

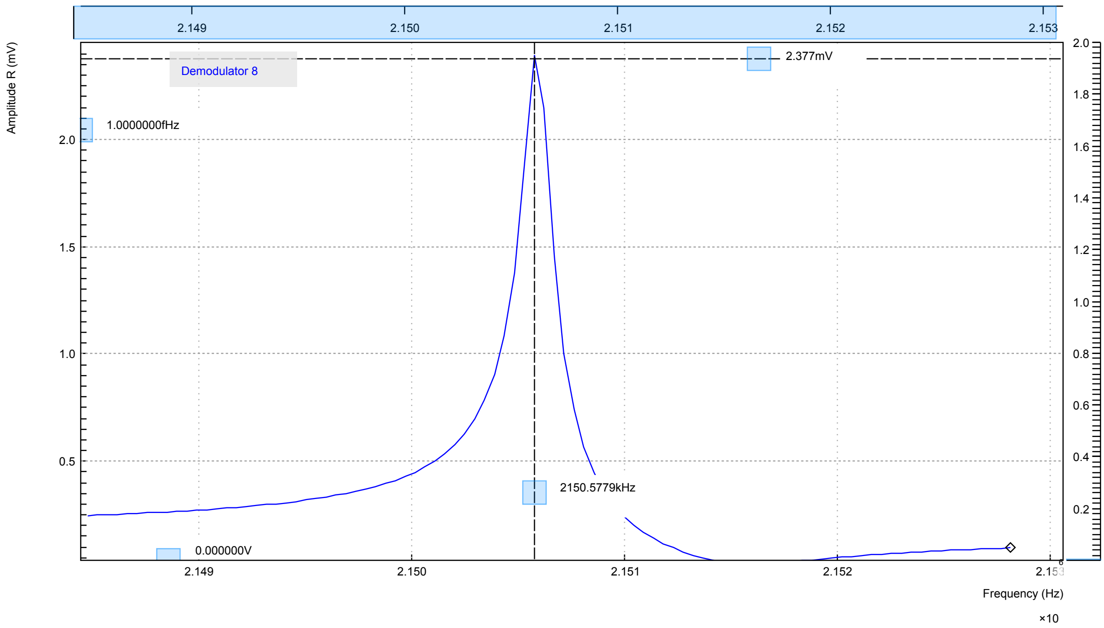


Figure 3.17. frequency sweep amplitude response

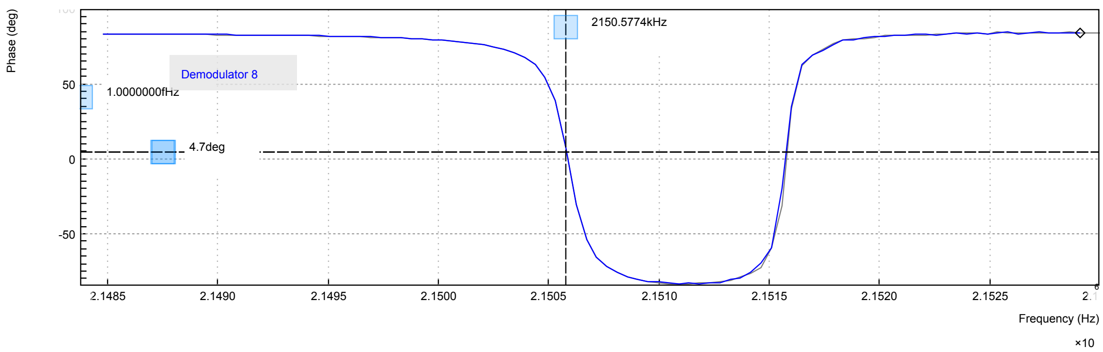


Figure 3.18. frequency sweep phase response

3.4.4. Resonance Tracking with the PLL

Now that we have located the resonance frequency and its phase, we can now track the drift in resonance frequency by locking on to the phase that we just measured using the Sweeper, hence the name phase locked loop. The phase locked loop is available under the PLL tab. There are two PLLs in each UHF unit. For this tutorial, we will use PLL 2. We first set up the basic PLL 2 fields as shown in the table below, using the values from the Sweeper.

Table 3.16. Settings: acquire the reference signal

Tab	Section	#	Label	Setting / Value / State
PLL	PLL 2		Center Freq (Hz)	2.1506 M
PLL	PLL Settings		Oscillator	8
PLL	PLL Settings		Demodulator	8
PLL	PID Settings	1	Setpoint (deg)	+4.7

In this case, we must also select the 8th oscillator and demodulator 8 for the phase locked loop operation. Now, we need to set up the closed loop response of the PLL. One can use the PLL Advisor for such purpose. For this tutorial, we will not use Advanced Mode but rather will just set the Target BW (Hz) to be 1.0 k. One then needs to press on the Advise button to see the simulated open loop response. This will also generate a set of PID parameters as shown in the screen shot below. One can observe that the -3dB point is roughly at 1 kHz as specified. Once you are happy with the response, then simply press on the ToPLL button to copy the PID parameters back to the PLL 2 setting. To start the PLL operation, simply click on the Enable button. This will launch the phase locked loop operation.

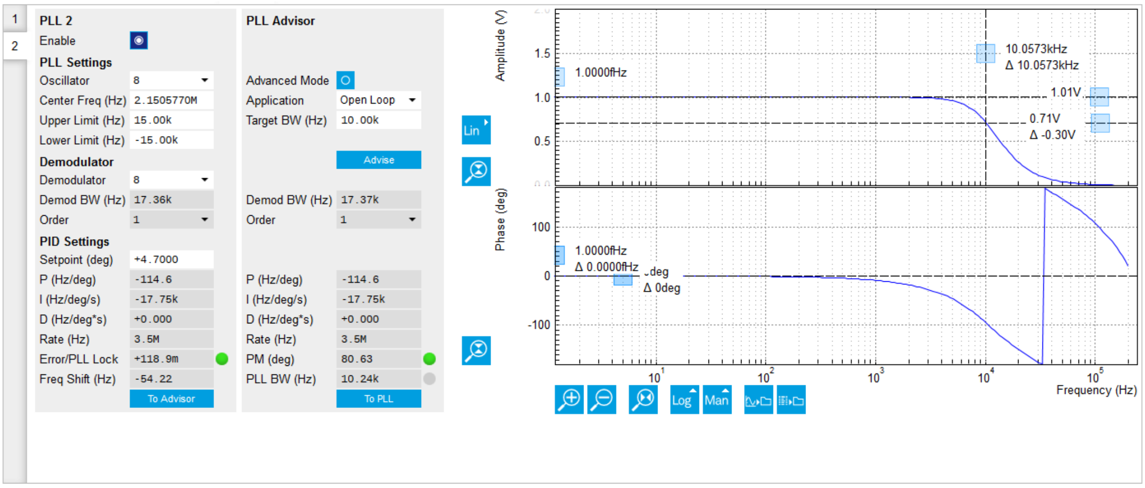


Figure 3.19. PLL settings and simulation in the PLL tab

When the PLL is locked, the green indicator beside the label Error/PLL Lock will be switched on. The actual frequency shift is shown in the field Freq Shift (Hz).

Note

At this point, it is recommended to adjust the signal input range by pressing on the Auto Range button in the Lock-in tab. This will sometimes help the PLL to lock to an input signal with a better signal-to-noise ratio.

The easiest way to visualize the frequency drift is to use the Plotter tool. One simply needs to select Frequency and Channel 8 and then press the button Add Signal. This will add an additional signal in the Plotter window. The frequency short-term drift noise can be further reduced sometimes by decreasing the PLL bandwidth.

3.5. Automatic Gain Control

Note

This tutorial is applicable to UHF Instruments with the UHF-PID Quad PID/PLL Controller option installed.

3.5.1. Goals and Requirements

This tutorial explains how to set up a PID controller for automatic gain control. The tutorial can also be performed as a continuation to the previous PLL tutorial i.e. the PLL can be kept running. Just like the PLL tutorial, an external quartz resonator is used as the device-under-test. To perform this tutorial, one simply needs to connect a resonator between Signal Output 2 to Signal Input 2.

3.5.2. Preparation

Connect the cables as illustrated below. Make sure the UHFLI is powered on, and then connect the UHFLI through the USB to your PC, or to your local area network (LAN) where the host computer resides. After starting LabOne the default web browser opens with the LabOne graphical user interface.

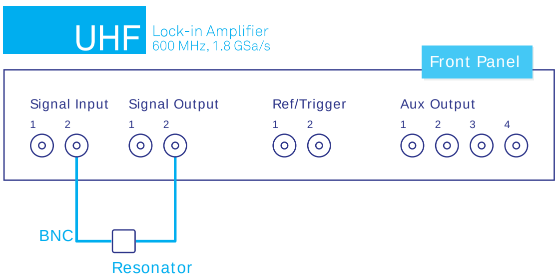


Figure 3.20. PID connection with UHF

The tutorial can be started with the default instrument configuration (e.g. after a power cycle) and the default user interface settings (e.g. as is after pressing F5 in the browser).

3.5.3. Automatic Gain Control

In this section you will learn how to control the output amplitude of your device-under-test. In theory, you can control the amplitude of any devices connected in the feedback configuration through a PID. In this case, we will use a resonator driven at its resonance frequency by one of two UHFLI signal generators and then measured with one of two lock-in channels.

If you are continuing the PLL tutorial, then we can just leave the PLL enabled. Otherwise, you should know how to generate an excitation signal at the modulation that you require and then measure the signal amplitude that you want to control. The device-under-test does not need to be a resonator. As shown in the screen shot below, we are measuring an amplitude of about 2.4 mV at the peak of the resonance. The goal is to control this amplitude to be a programmable value given by the user on-the-fly.

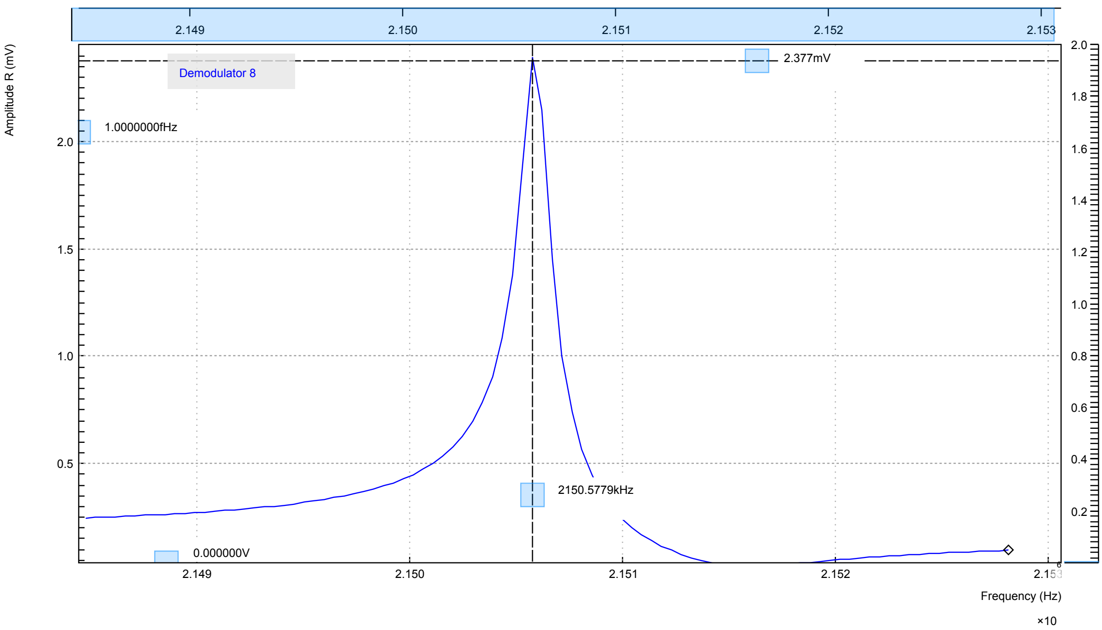


Figure 3.21. resonance amplitude to be controlled

For using the PID for AGC, we need to pull up a PID tab. For this tutorial, let us use PID 3. And then we need to set up the input and output of the PID 3 controller. The settings are shown in the table below.

Note

Please note that PLL 1 and PLL 2 are in fact the same as PID 1 and PID 2, respectively.

Table 3.17. Settings: acquire the reference signal

Tab	Section	#	Label	Setting / Value / State
PID	Input	3		Demodulator: R / 8
PID	Output	3		Output 1 Amplitude / 8
PID	Output	3	Center (V)	0
PID	Output	3	Upper Limit (V)	1.0
PID	Output	3	Lower Limit (V)	0

The most difficult part of PID controller setting is to select the proper P, I and D gain values. In this tutorial, we will use the Good Gain method developed by Finn Haugen of Telemark University College in Norway in 2010 for PID controller tuning. This is, in essence, a procedure to select PID parameters through real time observation of the closed loop step response.

Note

The Good Gain method can be considered to be a closed loop tuning method. Other types of closed loop PID tuning methods include the Ziegler-Nichols method, the Tyreus-Luyben method, and the damped oscillation method. The open loop tuning methods are, for example, the open loop

Ziegler-Nichols method, the C-H-R method, the Cohen and Coon method, the Fertik method, the Ciancone-Marline method, the IMC method, and the minimum error criteria methods.

The Good Gain method has the merit of being easily observable. There are only a few steps to follow using this PID tuning method:

1. Enable the PID. We are, initially, trying to manually adjust the system in open loop such that the controlled signal is close to its final value.
2. Set all P, I and D values to zero. Increase P gradually until you get a slight overshoot in the step response. This is done by manually adjust the set point and observe the controlled signal response. You should now observe the error between the measurement and the set point value getting smaller and smaller as P increases. Note that with the P controller, one can get close but never exactly to the final setpoint value. Make sure that the PID input or output is not unintentionally soft limited in minimum or maximum values (e.g. limited in amplitude, frequency etc).

Note

The Plotter tool is a very good way to observe the step response while adjusting the PID gain parameters as shown below.

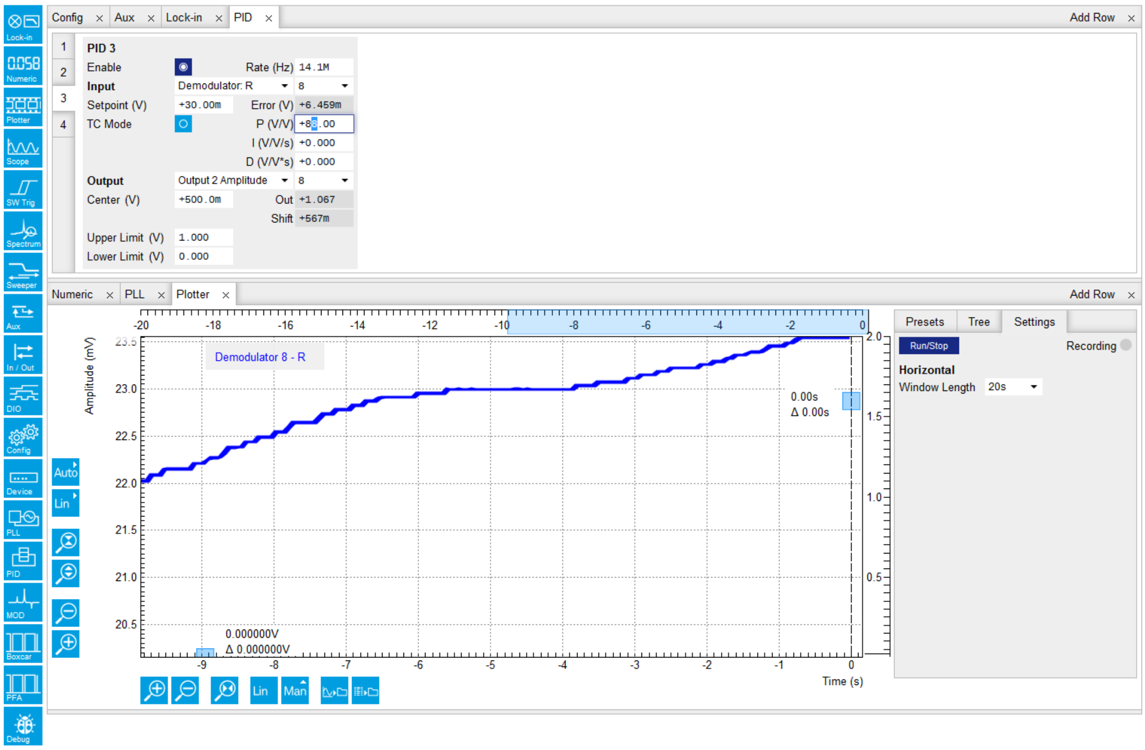


Figure 3.22. PID step response observation using the Plotter

3. Once the above condition is met, then set I to the value of $1.5T_{ou}$. T_{ou} is the delta time between the overshoot and the undershoot of the step response. Increase I gradually until the error value gets very close to 0. One can slightly decrease the P value by 50% to 80% if PID becomes slightly unstable.
4. One can potentially set D to 1/4 of I although it is not necessary and sometimes it might not even bring any improvement.

5. Check loop response again by applying a step response like in Step 2. Adjust mainly the P, I value accordingly for fine tuning.

Note

The set point can be manually toggled to create the step response condition.

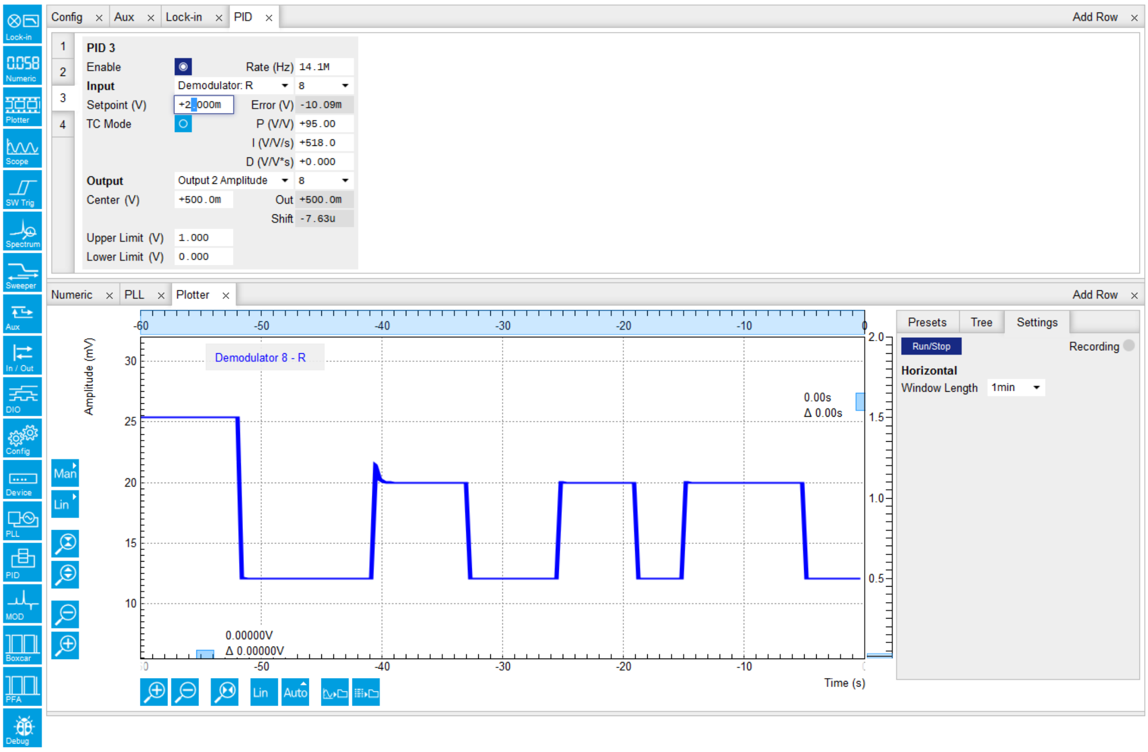


Figure 3.23. PID step response fine tuning by trying out different responses to set points

3.6. PWA and Boxcar Averager

Note

This tutorial is applicable to UHF Instruments with the UHF-BOX Boxcar Averager option installed.

3.6.1. Goals and Requirements

This tutorial explains how to set up a periodic waveform analyzer (PWA) and a boxcar averager for measuring periodic signals with low duty cycles. The advantages of using the PWA and the boxcar averager over a digital scope or a lock-in amplification technique will be explained and demonstrated as follows.

The duty cycle and the signal energy that is available in the fundamental frequency scale almost linearly. For example, a rectangular signal pulse with 50% duty cycle has only 1/3 of the signal amplitude in the fundamental frequency. And if the duty cycle is further halved, then the signal in the fundamental is also halved. Hence, lock-in amplification, which normally references to the fundamental frequency, may not always be the best way to recover a signal if the pulse waveform has a duty cycle smaller than 50%. In this case, boxcar averaging may be the more efficient measurement method. If the signal spreads out over many harmonic components without any prominent peak, a boxcar detection scheme might be the wiser choice to achieve the best possible signal-to-noise ratio.

To perform the measurements in this tutorial, one will require a 3rd-party programmable arbitrary waveform/function generator for narrow pulse generation, or alternatively the UHF-AWG Arbitrary Waveform Generator option installed.

3.6.2. Preparation

Connect the cables as illustrated below. Make sure the UHF is powered on, and then connect the UHF through the USB to your PC, or to your local area network (LAN) where the host computer resides. After starting LabOne the default web browser opens with the LabOne graphical user interface.

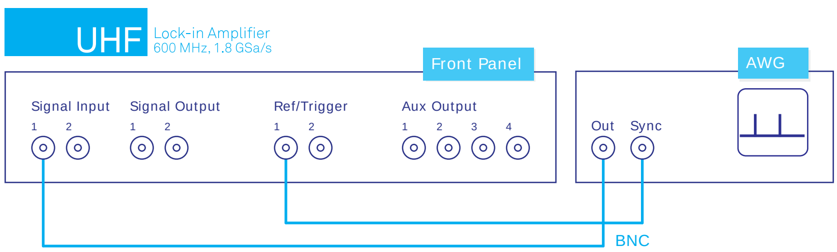


Figure 3.24. UHF connections to an external arbitrary wave generator

The tutorial can be started with the default instrument configuration (e.g. after a power cycle) and the default user interface settings (e.g. as is after pressing F5 in the browser).

3.6.3. Low Duty Cycle Signal Measurement

There are a couple of ways to measure a low duty cycle signal with the UHF. The obvious method is to use the Scope function inside the LabOne interface to observe the sampled signal in the time domain. The other method is to use the PWA and the boxcar averager. Both methods will be shown. The first task is to generate a test signal.

Narrow Pulse Signal Generation

Using the external arbitrary waveform generator, generate a pulse with the following specifications.

Table 3.18. Narrow pulse signal specifications

Pulse Specification	Section
Pulse Type	Square
Amplitude	100 mVpp
Frequency	9.7 MHz
Duty Cycle	< 16%

Note

For this exercise, an Agilent 33500B Truefrom waveform generator is used. The minimum duty cycle for a 9.7 MHz signal is limited to about 16%.

The LabOne Scope can be used to observe the generated pulse waveform. Connect the output of the AWG directly to Signal Input 1 of the UHFLI. The Scope settings in LabOne are given in the table below. Also, the AWG should also be able to provide a TTL synchronization signal to be connected to the Ref / Trigger input. This trigger signal will be used later on for the PWA.

Table 3.19. Settings: observe the pulse waveform

Tab	Section	#	Label	Setting / Value / State
Lock-in	Signal Inputs	1	AC	On
Lock-in	Signal Inputs	1	50Ω	On
Lock-in	Signal Inputs	1	Range	200.0 m
Scope	Display/Vertical		Channel 1	Signal Input 1/On
Scope	Trigger		Signal	Signal Input 1/On
Scope	Trigger		Enable	On
Scope	Trigger		Hysteresis	10.0 m
Scope	Trigger		Run/Stop	On

One should now be able to observe Signal Input 1 similar to the following waveform in the Scope window. The Scope is set to self trigger on the pulse edges. Use the horizontal zoom to focus on a single period. This can be done by rolling the mouse wheel forward to zoom in the horizontal axis. To zoom in on the vertical axis, press down the Shift key and roll the mouse wheel. One can also recenter the waveform by pressing on the left mouse button and dragging the Scope plot area.

One can observe that the shape of the supposedly square pulse does not have sharp edges as one would expect. This is due to the effect of the 600 MHz low pass filter at the input of the UHF. In fact, the signal input bandwidth of 600 MHz corresponds to about 1.5 ns rise time (20% - 80%). Here, the sampled pulse width shown in the Scope is measured to be about 29 ns or 30% duty cycle. The smeared out waveform has a duty cycle bigger than the 16% that was originally set.

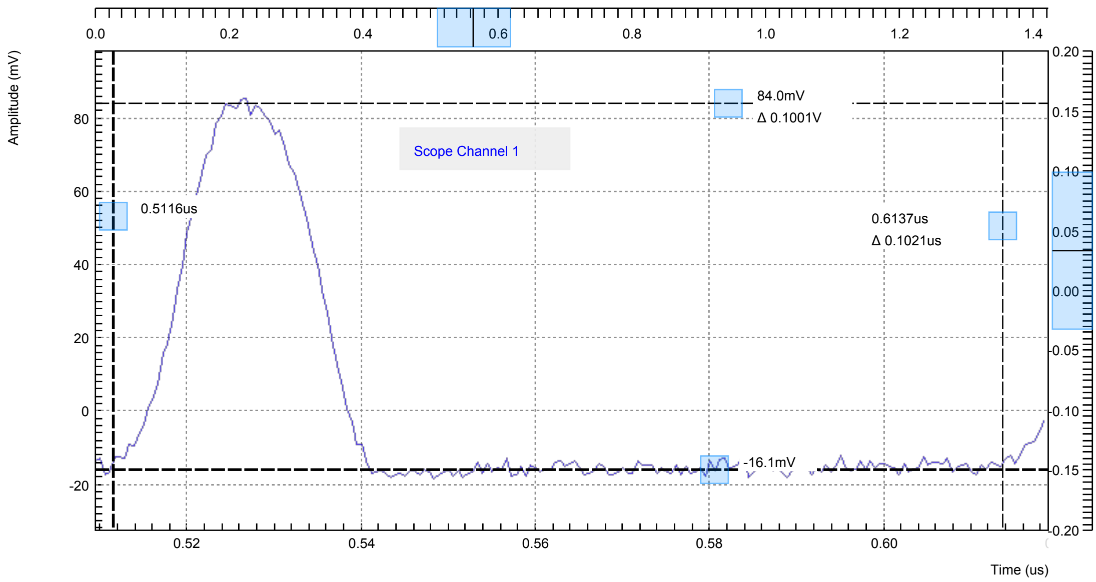


Figure 3.25. Digitized pulse waveform in Scope

Low Duty Cycle Analysis with Period Waveform Analyzer

To analyze the pulse waveform using the PWA, the UHFLI first has to lock to the trigger signal of the pulses. This is done using the Ext Ref mode of the UHFLI. The trigger signal is fed to the Ref / Trigger connector on the front panel which can be an analog signal or a TTL signal. The trigger level can be adjusted in the DIO tab as shown in Section 3.2.5. To lock to the trigger signal, the Lock-in tab should have the following settings; the goal is to lock the internal oscillator 1 to the external trigger from the AWG. The frequency of oscillator 1 in the Lock-in tab should now display 9.7 MHz, with the green light on to indicate a lock condition.

Table 3.20. Settings: lock oscillator 1 to external trigger 1

Tab	Section	#	Label	Setting / Value / State
Lock-in	Demodulators	4	Reference Mode	ExtRef
Lock-in	Demodulators		Input Signal	Trigger 1

Then, to activate the PWA function, place one instance of the Boxcar tool in the LabOne web interface. To display the 9.7 MHz pulse over a single period, the following parameters need to be set.

Table 3.21. Settings: activate PWA

Tab	Section	#	Label	Setting / Value / State
Boxcar	PWA/Signal Input	1	Input Signal	Sig In 1/On
Boxcar	PWA	1	Run/Stop	On

Immediately, one can see in the PWA a very stable and smooth peak in one pulse period. The horizontal axis is shown in phase over 360 degrees to represent one period of the pulse waveform. The position of the peak also indicates the precise phase delay with respect to the trigger signal. In this phase representation, the PWA sub-divide the full 360 degrees into 1024 bins. The phase resolution is therefore about 0.35 deg; for a signal of 9.7 MHz this corresponds to a time resolution of about 100 ps.

.

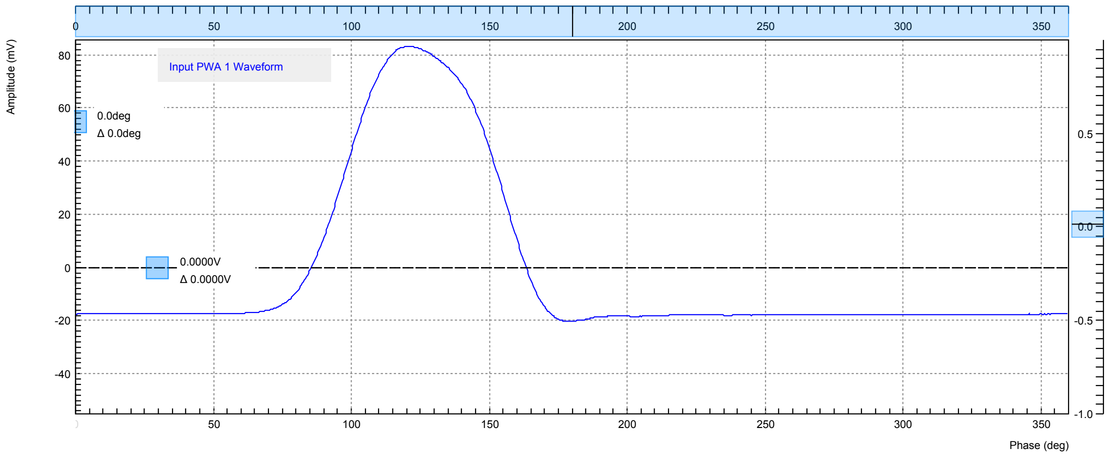


Figure 3.26. Pulse waveform in PWA

If this resolution is not sufficient, one can use the Zoom mode. Then by changing the Width (deg), one can then get more details of the characteristics of the pulse. The redefined phase range will then again be subdivided into 1024 bins. To acquire the same number of samples for a smaller range will increase acquisition time.

Note

The Zoom mode references internally the input signal to a higher harmonic of the reference frequency which allows zooming into the region of interest, and hence increasing the temporal resolution down to millidegrees. This gives a precise analysis for pulsed signals with low duty cycles or any other periodically repeating transient. Of course the real resolution is still limited by the signal input bandwidth, as in the case of the Scope.

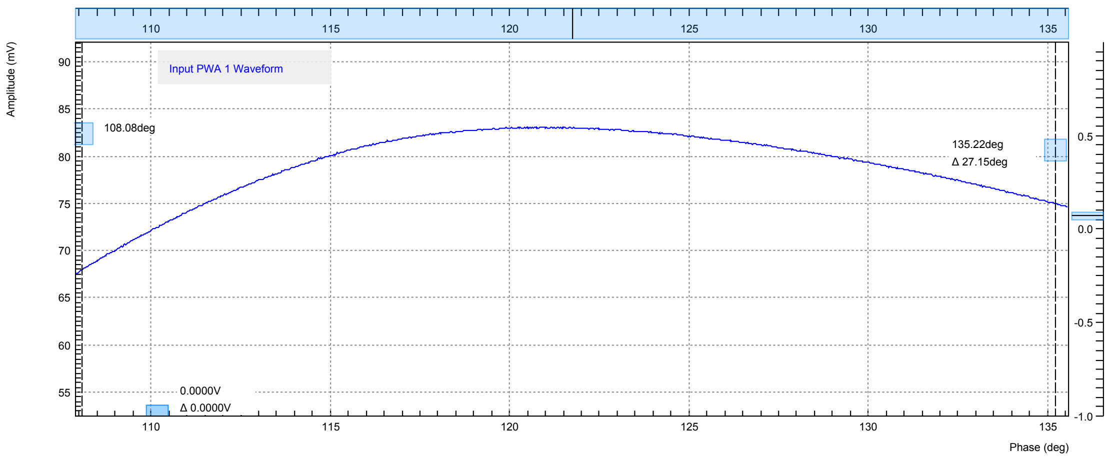


Figure 3.27. Pulse waveform in PWA with a zoom width of 27 degrees

Beside the phase domain display, one can also choose the horizontal display axis in the unit of time or frequency. The harmonics of the pulse waveform can also be analyzed by setting Mode to Harmonics. These options are all part of the multi-channel, multi-domain PWA for peak analysis.

The frequency of 9.7 MHz is not chosen accidentally. In general, one should avoid choosing a modulation frequency that shares the same divisor as the maximum UHF-BOX repetition rate of

450 MHz i.e. the two numbers should not be commensurable. For example, 10 MHz and 450 MHz are commensurable since they can be both divided by 10. This commensurability issue arises from the internal UHF sampling effect which may cause certain bins to get filled constantly but not others. Such an example is shown in the figure below. A red warning indicator will be switched on when a potential commensurability problem is detected.

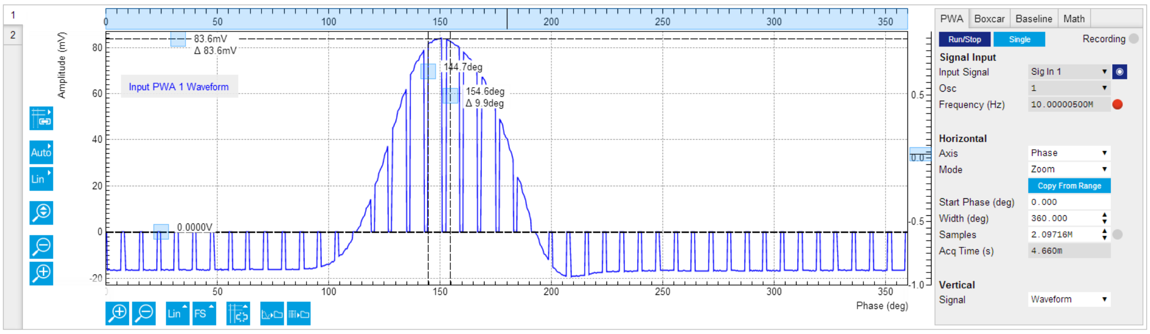


Figure 3.28. Problem of commensurability with the choice of the modulation frequency

Low Duty Cycle Analysis with Scope

The digitized waveform in the Scope can be jittery and noisy. One must remember that the pulse is sampled at 1.8 GSa/s which corresponds to a minimum resolution of 555 ps. This resolution implies that in the zero-crossing triggering, the triggered point on the waveform will not be the same for every pulse. This is indeed one major source of jitter observed.

The Scope comes with averaging and the persistence function which can in theory help to minimize jitter and noise. To use the averaging mode, one simply has to set Avg Filter field under the Scope Control tab to Exp Moving Avg. Then one can choose the number of Averages desired. Below is the averaged pulse waveform at 10 points. Compared to the previous non-averaged waveform, it can be seen that now the spikes are smoothed out.

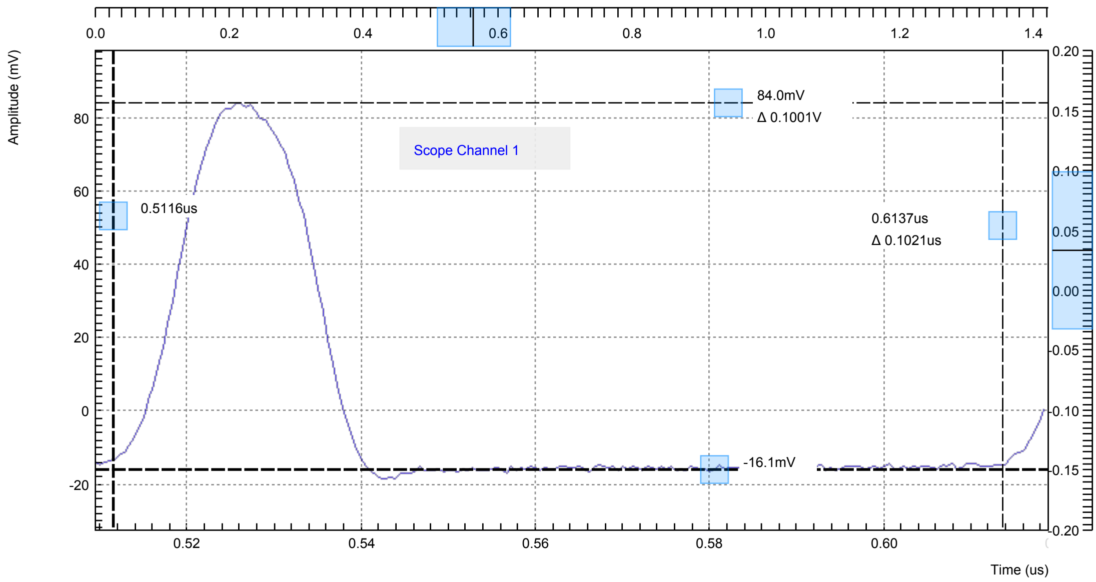


Figure 3.29. Scope waveform with 10 exponential moving averages

In order to observe the extent of jitter and noise, one can use the Persistence mode. Persistence can be enabled in the Advanced tab. Enabling persistence causes each triggered waveform to be superimposed on top of the previous ones. The result of the persistence is shown in the graph below where the superimposed traces are in red. One can measure an amplitude variation of about

7 mV and a time jitter of about 1.6 ns from the thickness of the red trace. Under this condition, the Scope method can be said to be not an ideal tool to analyze a narrow peak, especially when the peak width would be below a nanosecond.

Note

The vertical axis of the Scope needs to be in manual mode in Persistence mode. Persistence cannot be used simultaneously with averaging.

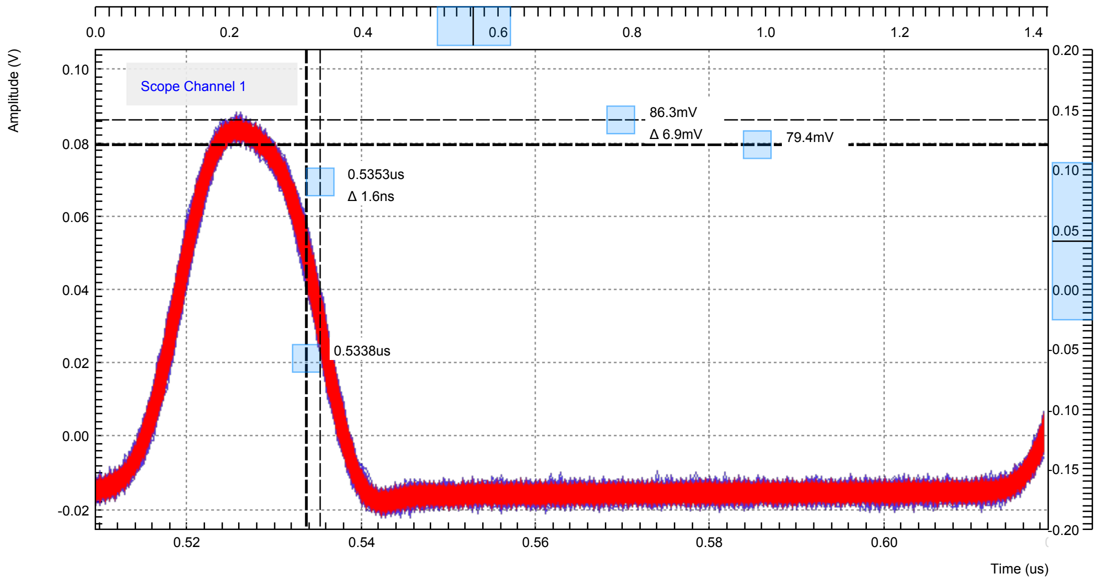


Figure 3.30. Scope waveform with persistence

Comparison shows that the PWA tool is certainly a more precise and elegant way to analyze this type of narrow pulse waveform.

Boxcar Integration

To use the boxcar averager, one can simply click on the Boxcar sub-tab. The boxcar averager integrates a section of the signal and has the output has a unit of volt-second (Vs). The integrated gate can be set either manually in the Start Phase (deg) and Width (deg) fields, or by positioning to vertical cursors and then by pressing Copy From Cursor. The integrated value is updated in the Value (Vs) field. An example boxcar setting is shown below. The integration width is chosen to be 10 degrees around the maximum peak.

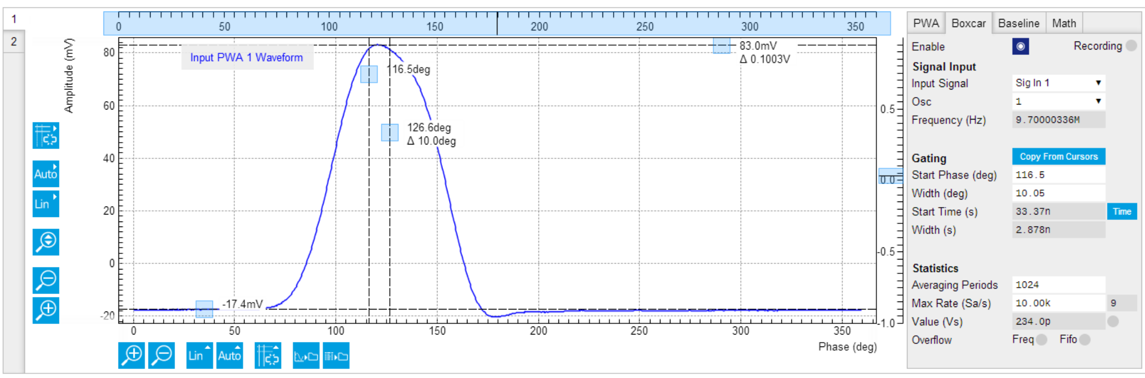


Figure 3.31. Boxcar integration of the pulse waveform

The result of the integration can also be shown graphically using the Plotter tool, as shown below.

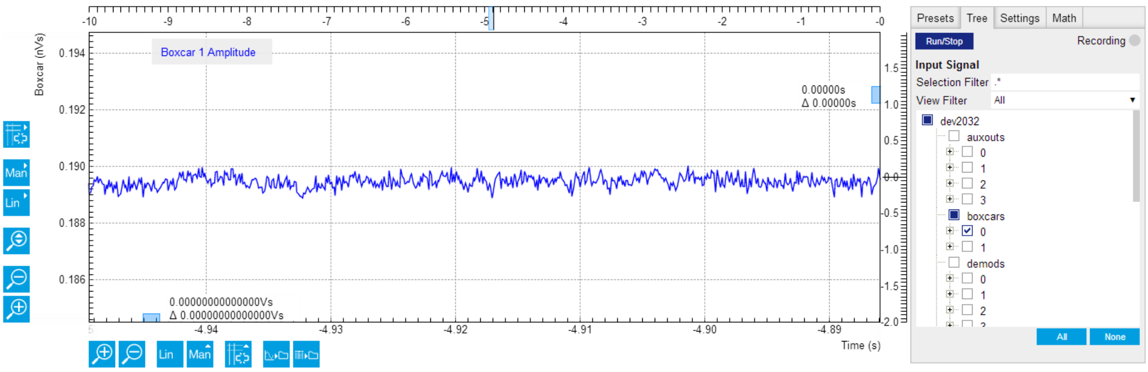


Figure 3.32. Boxcar integration result on Plotter output

Baseline Subtraction

It may happen that sometimes a noise signal is superimposed on the measured boxcar output. This noise can come from the power supply, emf noise coupled through the external wirings or even from the experiment itself. In this case, the baseline subtraction function can be applied to remove the undesired noise found in the Boxcar integration. To show the benefits of the baseline subtraction, the following connections can be made to simulate an undesired period noise injection. In this example, the UHF Signal Output 1 is used to generated a 10 kHz sine wave superimposed on top of the AWG waveform through a T-connector.

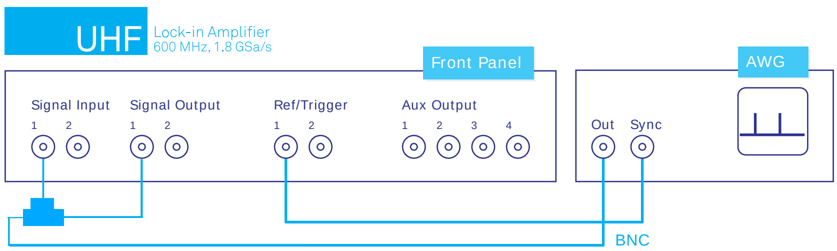


Figure 3.33. UHF connection for baseline subtraction test

Table 3.22. Settings: superpose a sine wave on top of the pulse waveform

Tab	Section	#	Label	Setting / Value / State
Lock-in	Oscillators	2	Frequency	10.0 k
Lock-in	Output Amplitudes	2		1.5
Lock-in	Signal Outputs		Output 1	On

When this is done, the Plotter tool will display an integrated value with the 10 kHz sine component instead of the flat line shown previously.

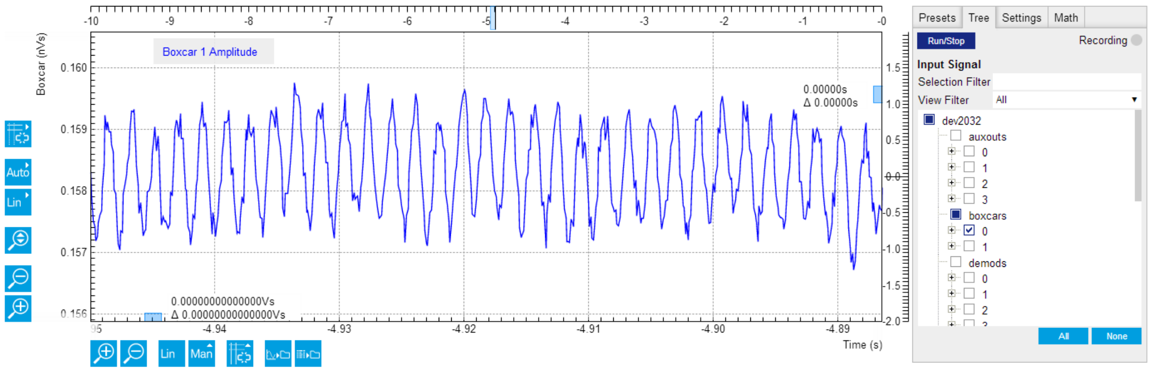


Figure 3.34. Boxcar output without baseline subtraction

In order to eliminate this undesired sine variation, one can simply go to the Baseline sub-tab in the Boxcar tool. The important point is to select a baseline window with the cursor with the same width as the Boxcar integration window (e.g. 10 degrees in this tutorial). The baseline window is chosen to center around the zero crossing value of the PWA waveform, when possible. This is done so the baseline integration only integrates the superimposed sine and not the pulse waveform itself. The subtraction will then be only on the sine component.

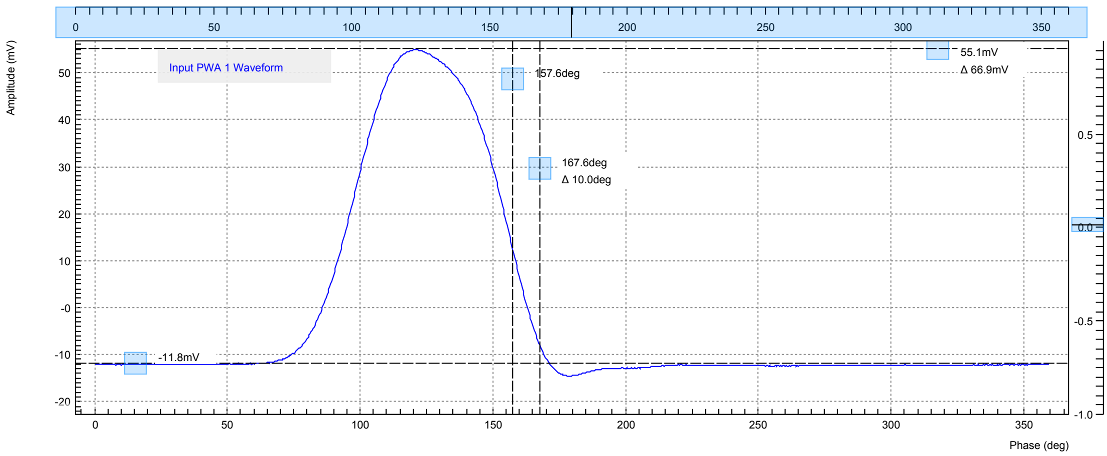


Figure 3.35. Baseline subtraction setup

Once the cursors are defined, one simply clicks on Run/Stop in the Baseline sub tab. One will see right away in the Plotter window that the sine component disappears. The trace that is left is again the original Boxcar averager value.

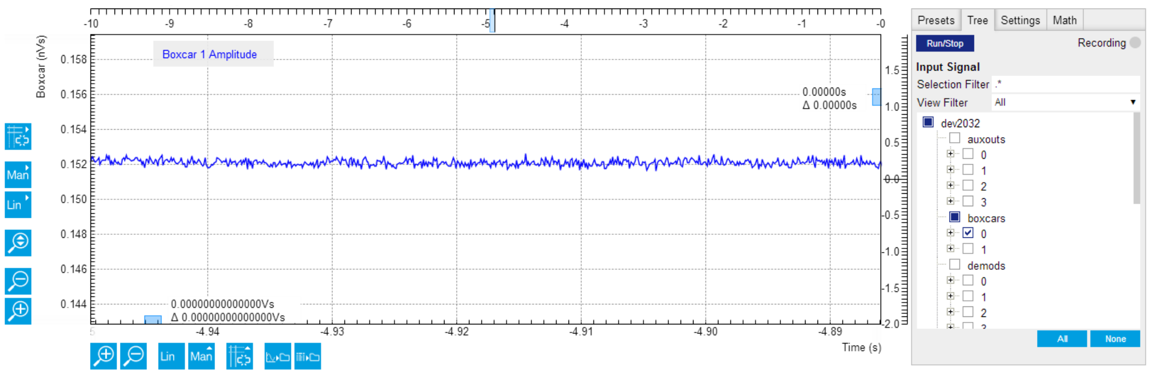


Figure 3.36. Boxcar output with baseline subtraction

3.7. Multi-channel Boxcar Averager

Note

This tutorial is applicable to UHF Instruments with the UHF-BOX Boxcar Averager option installed.

3.7.1. Goals and Requirements

This tutorial explains how to extract the envelope of an amplitude modulated carrier in the Out PWA tool from the boxcar averager. More generally, the multi-channel boxcar feature serves to measure signals that are modulated with two time bases: the fast time base produces the pulses as measured by the boxcar averager, and the slow time base corresponds to a change of the envelope. A typical application would be an amplitude modulated narrow laser pulse waveform.

To perform this tutorial, an external arbitrary waveform generator with an external AM modulation capability is required. In this section you will learn how to measure a narrow pulse waveform that is amplitude modulated. Both the boxcar averager and the output PWA tools will be utilized in this example. First, one needs to generate a test signal.

3.7.2. Preparation

Connect the cables as illustrated below. Make sure the UHFLI is powered on, and then connect the UHFLI through the USB to your PC, or to your local area network (LAN) where the host computer resides. After starting LabOne the default web browser opens with the LabOne graphical user interface.

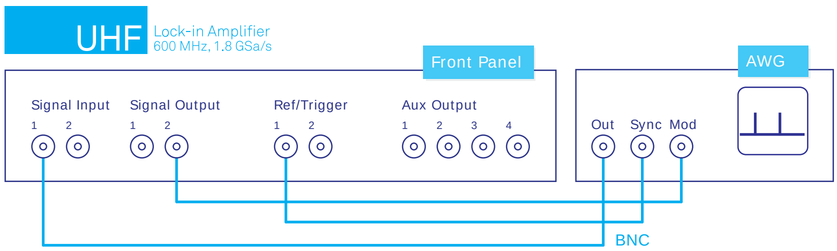


Figure 3.37. UHF connections to an external arbitrary wave generator

The tutorial can be started with the default instrument configuration (e.g. after a power cycle) and the default user interface settings (e.g. as is after pressing F5 in the browser).

3.7.3. Amplitude Modulated Narrow Pulse Measurement

AM Modulated Narrow Pulse Test Signal Generation

Using the external arbitrary waveform generator, a pulse waveform with the following specification should be generated.

Table 3.23. Narrow pulse signal specifications

Pulse Specification	Section
Pulse Type	Square

Pulse Specification	Section
Amplitude	100 mVpp
Frequency	9.7 MHz
Duty Cycle	< 16%

Note

An Agilent 33500B Truefrom waveform generator is used in this example. The minimum duty cycle for a 10 MHz signal is limited to about 16%. An external AM modulation scheme is activated with 100% AM depth.

Furthermore, a sine wave should be generated from the UHF to amplitude modulate the AWG output. The output settings of the UHF are given below.

Table 3.24. Settings: observe the pulse waveform

Tab	Section	#	Label	Setting / Value / State
Lock-in	Oscillators		Frequency (Hz)	10.0 kHz
Lock-in	Signal Outputs	2	Amp (Vpk)	1.5 V
Lock-in	Signal Outputs	2	On	On
Scope	Display		Sampling Rate	28.1 MHz
Scope	Trigger		Signal	Signal Input 1/On
Scope	Trigger		Enable	On
Scope	Trigger		Run/Stop	On

Now, one should be able to see a waveform in Scope that is similar to the one shown below.

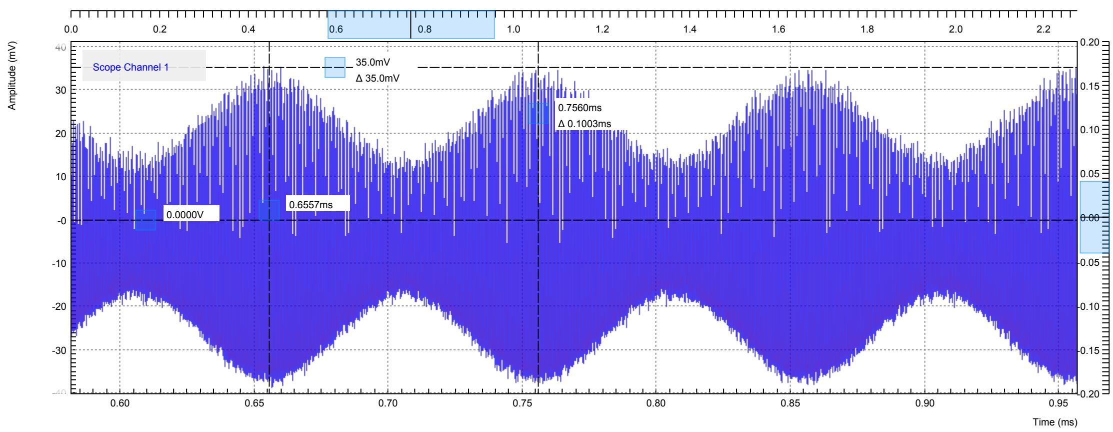


Figure 3.38. AM modulated pulse waveform

Envelope Recovery with the Output PWA

Just like the previous tutorial in the section called “Low Duty Cycle Analysis with Period Waveform Analyzer”, the PWA can be used to observe the pulse train. Although the measured result is similar to the previous tutorial, one can see in the PWA screen shot below that the amplitude is no longer 80 mV peak but rather around 40 mV. One has to remember that we have now an amplitude modulated pulse, and the PWA is showing the average amplitude of these pulses over time. If one decreases the number of averages in PWA then an amplitude-fluctuating behavior can be observed more clearly.

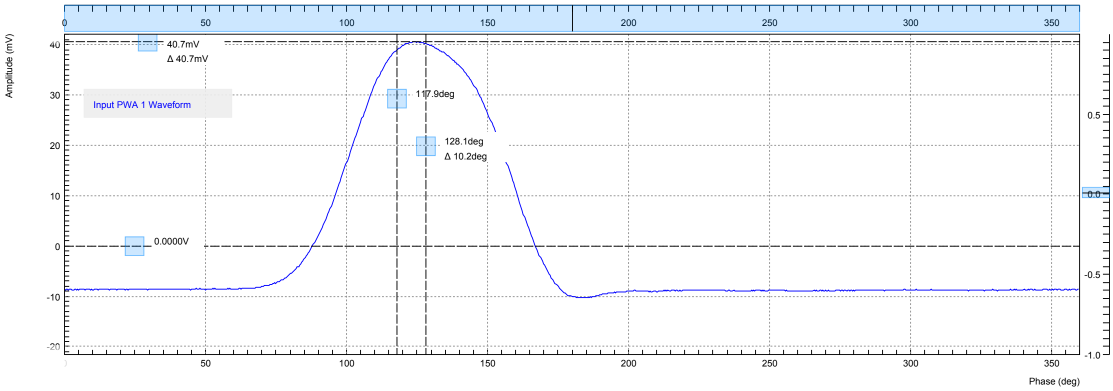


Figure 3.39. Carrier pulse in PWA

As shown previously, the Boxcar tool can be used to obtain the integrated pulse energy over a pre-defined gate width. This integrated value will of course be amplitude modulated as well. Now, the Output PWA can be used to recover this envelope of the integrated value. To do this, one now has to place an instance of the Out PWA tool on the LabOne web interface. The settings of the Output PWA are given below.

Table 3.25. Settings: observe the pulse waveform

Tab	Section	#	Label	Setting / Value / State
Out PWA	Settings/Signal Input	1	Input Signal	Boxcar 1
Out PWA	Settings/Signal Input	1	Osc Select	2
Out PWA	Settings/Signal Input	1		Run/Stop

One should be able to observe a sine wave similar to the one shown below. The Vs magnitude is proportional to the AM modulation depth. One can verify this by changing the AM depth to 50% (see second screen shot). The envelope magnitude indeed decreased by a factor of 2. Out PWA acts like a multi-channel boxcar that can be used to do multiple sideband analysis. The UHF-MF option may be required to observe more than one modulation frequency.

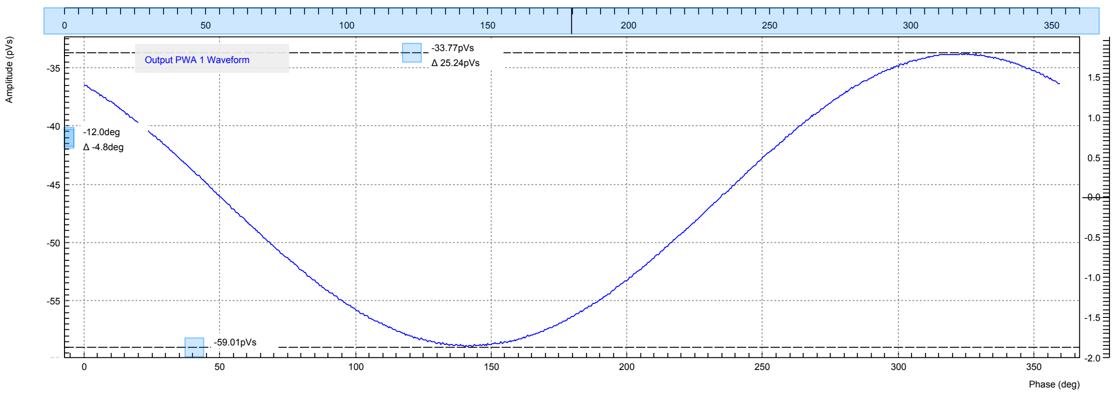


Figure 3.40. AM envelope in Out PWA with 100% and 50% AM depth

3.8. Arbitrary Waveform Generator

Note

This tutorial is applicable to UHFLI Lock-in Amplifier Instruments with the UHF-AWG Arbitrary Waveform Averager option installed and to UHFAWG Arbitrary Waveform Generator Instruments. Where indicated, additional options such as UHF-DIG, UHF-BOX, UHF-CNT, UHF-MF or the UHFLI Lock-in Amplifier are required.

3.8.1. Goals and Requirements

The goal of this tutorial is to demonstrate the basic use of the AWG. We demonstrate waveform generation and playback, triggering and synchronization, carrier modulation, and sequence branching. We conclude with a list of tips for operating the AWG. The tutorial can be done using simple loop back connections.

3.8.2. Preparation

Connect the cables as illustrated below. Make sure the UHF instrument is powered on, and then connect the UHF instrument through the USB to your PC, or to your local area network (LAN) where the host computer resides. After starting LabOne, the default web browser opens with the LabOne graphical user interface.

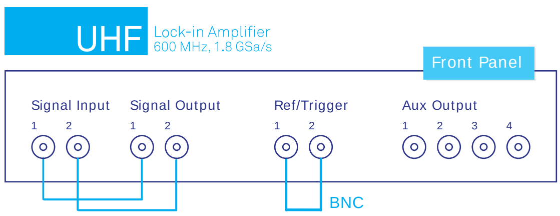


Figure 3.41. UHF connections for the arbitrary waveform generator tutorial

The tutorial can be started with the default instrument configuration (e.g. after a power cycle) and the default user interface settings (e.g. as is after pressing F5 in the browser).

3.8.3. Waveform Generation and Playback

In this tutorial we generate arbitrary signals with the AWG and visualize them with the Scope. In a first step we enable the Signal Outputs, but disable all sinusoidal signals generated by the lock-in unit by default. We also configure the Scope signal input and triggering and arm it by clicking on Run/Stop in the Scope. The following table summarizes the necessary settings.

Table 3.26. Settings: enable the output and configure the Scope

Tab	Sub-tab	Section	#	Label	Setting / Value / State
In/Out		Signal Outputs	1	Enable	ON
In/Out		Signal Outputs	2	Enable	ON
Lock-in		Output Amplitudes	1-8	Amp 1 Enable	OFF

Tab	Sub-tab	Section	#	Label	Setting / Value / State
Lock-in		Output Amplitudes	1-8	Amp 2 Enable	OFF
Scope	Control	Vertical		Channel 1	Signal Input 1
Scope	Trigger	Trigger		Enable	ON
Scope	Trigger	Trigger		Signal	Signal Input 1
Scope	Trigger	Trigger		Level	0.1 V
Scope	Control			Run/Stop	ON

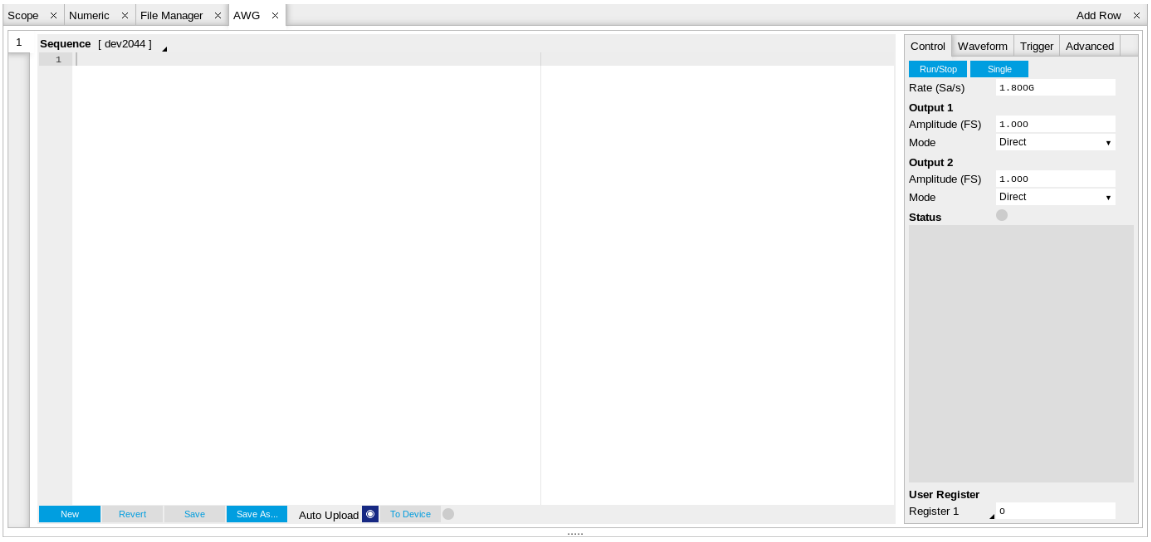


Figure 3.42. LabOne UI: AWG tab

In the AWG tab, we configure both channels to output signals at the full scale (FS) in direct output mode as summarized in the following table.

Table 3.27. Settings: configure the AWG output

Tab	Sub-tab	Section	#	Label	Setting / Value / State
AWG	Control			Rate (Sa/s)	1.8 G
AWG	Control	Output 1		Amplitude (FS)	1.0
AWG	Control	Output 1		Mode	Direct
AWG	Control	Output 2		Amplitude (FS)	1.0
AWG	Control	Output 2		Mode	Direct

Operating the AWG means first of all to specify a sequence program. This can be done interactively by typing the program in the Sequence Editor window. Let's start by typing the following code into the Sequence Editor.

```
wave w_gauss = 1.0*gauss(8000, 4000, 1000);
playWave(1, w_gauss);
```

In the first line of the program, we generate a waveform with a Gaussian shape with a length of 8000 samples and store the waveform under the name `w_gauss`. The peak center position 4000 and the standard deviation 1000 are both defined in units of samples. You can convert them into time by dividing by the chosen Rate (1.8 GSa/s by default). The waveform generated by the `gauss` function has a peak amplitude of 1. This amplitude is dimensionless and the physical signal amplitude is given by this number multiplied with the signal output range (e.g. 1.5 V). We put a scaling factor of 1.0 in place which can be replaced by any other value below 1. The code line is terminated by a semicolon according to C conventions. In the second line, the generated waveform `w_gauss` is played on AWG channel 1.

Note

For the purpose of this tutorial, we will keep the description of the Sequencer commands short. You can find the full specification of the LabOne Sequencer language in Section 4.22.3

If we now click on Save, the program gets compiled. This means the program is translated into instructions for the LabOne Sequencer on the UHF instrument, see Section 4.22.2. If no error occurs (due to wrong program syntax, for example), the Status LED lights up green, and the resulting program as well as the waveform data is written to the instrument memory. If an error or warning occurs, messages in the Status field will help in debugging the program. If we now have a look at the Waveform sub-tab, we see that our Gaussian waveform appeared in the list. The Memory Usage field at the bottom of the Waveform sub-tab shows what fraction of the instrument memory is filled by the waveform data.

By clicking on Single, we have the AWG execute our program once. Since we have armed the Scope previously with a suitable trigger level, it has captured our Gaussian pulse with a FWHM of about 1.33 µs as shown in Figure 3.43 .

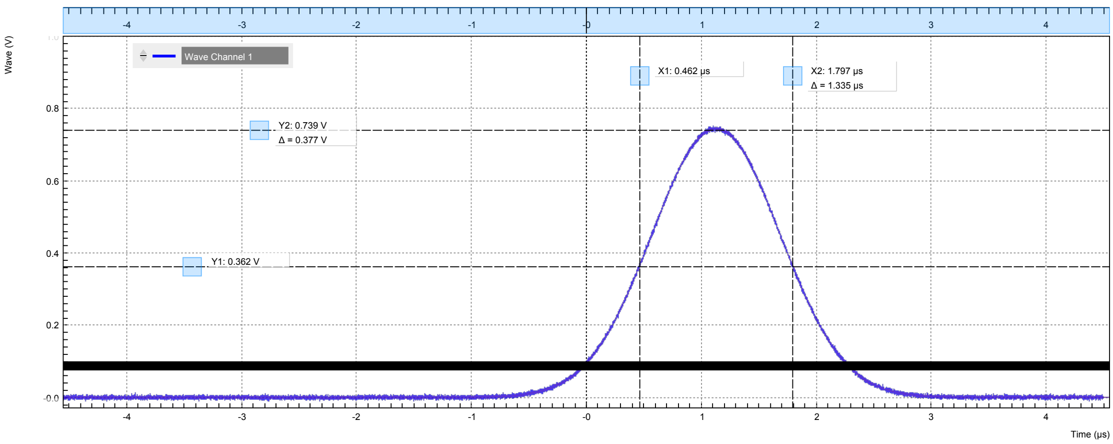


Figure 3.43. Gaussian pulse as generated by the AWG and captured by the LabOne Scope

The LabOne Sequencer language offers a lot of execution control. The basic functionality is to repeat a waveform several times. In the following example, all the code within the curly brackets `{…}` is repeated 5 times. Upon clicking Save and Single, you should observe 5 short Gaussian pulses in a new scope shot, see Figure 3.44 .

```
wave w_gauss = 1.0*gauss(640, 320, 50);
repeat (5) {
    playWave(1, w_gauss);
}
```

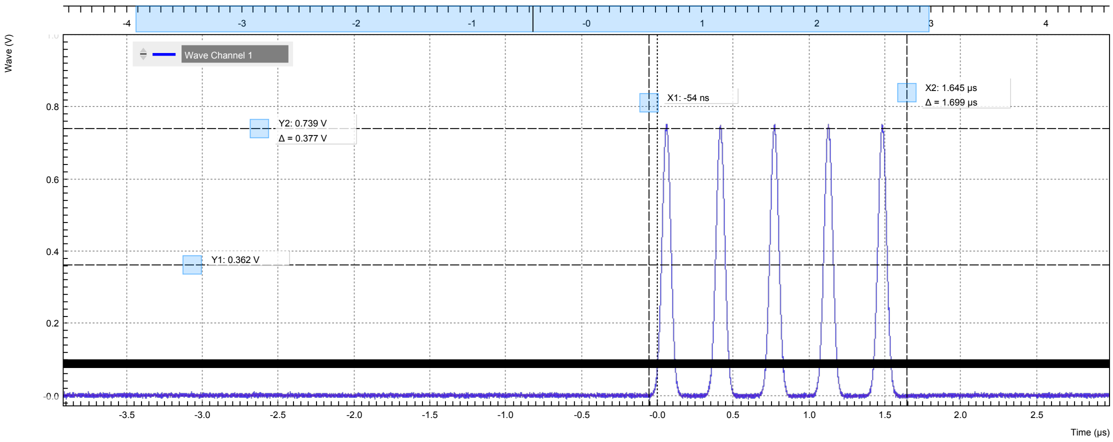


Figure 3.44. Burst of Gaussian pulses generated by the AWG and captured by the LabOne Scope

In order to generate more complex waveforms, the LabOne Sequencer programming language offers a rich toolset for waveform editing. On the basis of a selection of standard waveform generation functions, waveforms can be added, multiplied, scaled, concatenated, and truncated. It's also possible to use compile-time evaluated loops to generate pulse series with systematic parameter variations – see Section 4.22.3 for more precise information. In the following code example, we make use of these tools to generate a pulse with a smooth rising edge, a flat plateau, and a smooth falling edge. We use the `cut` function to cut a waveform at defined sample indices, the `rect` function to generate a waveform with constant level 1.0 and length 320, and the `join` function to concatenate three (or arbitrarily many) waveforms.

```
wave w_gauss = gauss(640, 320, 50);
wave w_rise = cut(w_gauss, 0, 319);
wave w_fall = cut(w_gauss, 320, 639);
wave w_flat = rect(320, 1.0);

wave w_pulse = join(w_rise, w_flat, w_fall);

while (true) {
    playWave(1, w_pulse);
}
```

Note that we replaced the finite repetition by an infinite repetition by using a `while` loop. Loops can be nested in order to generate complex playback routines. The output generated by the program above is shown in Figure 3.45.

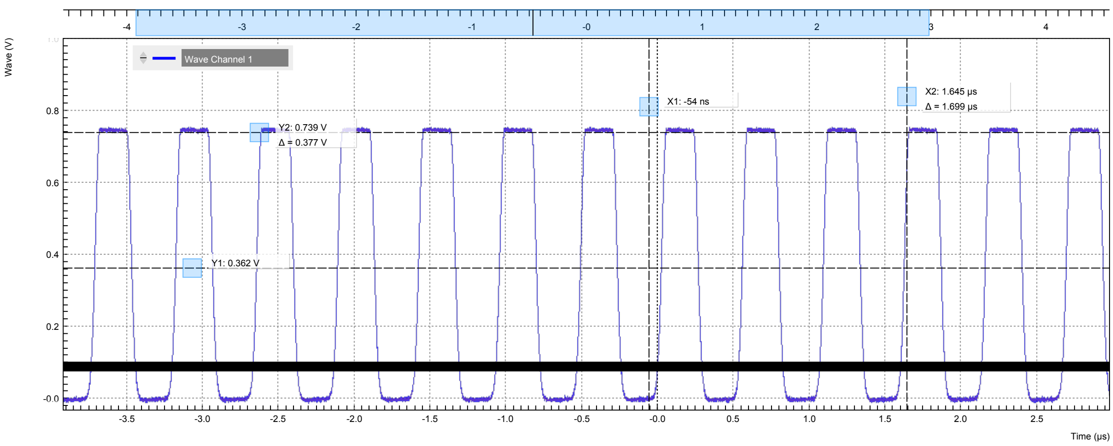


Figure 3.45. Infinite pulse series generated by the AWG and captured by the LabOne Scope

One pitfall when using loops has to do with the nature of the `playWave` and related commands. This command initiates the waveform playback, but during the playback the sequencer will start to execute the next command on his list. This is useful as it allows you to execute commands during playback. In a loop, it means the sequencer can jump back to the beginning of the loop while the waveform is still being played. You can easily change this behavior by adding `waitWave` as the last command in the loop.

As programs get longer, it becomes useful to store and recall them. Clicking on Save as... allows you to store the present program under a newly chosen file name. Clicking on Save then saves your program to the file name displayed at the top of the editor.

The LabOne AWG Sequencer language comes with waveform generation and editing tools that approach the possibilities of standard tools such as Matlab or Python. Should you nonetheless require more customization, you can import any waveform from a comma-separated value (CSV) file. The CSV file should contain floating-point values in the range from −1.0 to +1.0 and contain one (single-channel) or two (dual-channel) columns. The following could be the contents of a file `wave_file.csv` specifying a dual-channel wave with a length of 16 samples:

```
-1.0   0.0
-0.8   0.0
-0.7   0.1
-0.5   0.2
-0.2   0.3
-0.1   0.2
 0.1   0.0
 0.2  -0.1
 0.7  -0.3
 1.0  -0.2
 0.9  -0.3
 0.8  -0.2
 0.4  -0.1
 0.0  -0.1
-0.5  -0.1
-0.8   0.0
```

Store the file in the location of `C:\Users\<user name>\Documents\Zurich Instruments\LabOne\WebServer\awg\waves\wave_file.csv` under Windows or `~/Zurich Instruments/LabOne/WebServer/awg/waves/wave_file.csv` under Linux. In the sequence program you can then play back the wave by referring to the file name without extension:

```
playWave("wave_file");
```

If you prefer, you can also store it in a `wave` data type first and give it a new name:

```
wave w = "wave_file";
playWave(w);
```

The external wave file can have arbitrary content, but consider that the final signal will pass through the 600 MHz low-pass filter of the instrument. This means that signal components exceeding the filter bandwidth are not reproduced exactly as suggested for example by looking at a plot of the waveform data. In particular, this concerns sharp transitions from one sample to the next.

In order to obtain digital marker data (see below) from a file, specify a second wave file with integer instead of floating-point values. The marker bits are encoded in the binary representation of the integer (i.e., integer 1 corresponds to the first marker high, 2 corresponds to the second marker high, and 3 corresponds to both bits high). Later in the program add up the analog and the marker waveforms. For instance, if the floating-point analog data are contained in `wave_file_analog.csv` and the integer marker data in `wave_file_digital.csv`, the following code can be used to combine and play them.

```
wave w_analog  = "wave_file_analog";
wave w_digital = "wave_file_digital";
wave w = w_analog + w_digital;
playWave(w);
```

3.8.4. Triggering and Synchronization

Now we have a look at the triggering functionality of the AWG. In this section we will explain how to deal with the most important use cases:

- Triggering the AWG with an external TTL signal
- Generating a TTL signal with the AWG to trigger an external device
- Control the AWG repetition rate by an internal oscillator

We will simulate these situations with on-board means of the UHF instrument for the sake of simplicity, but the inclusion of external equipment is straightforward in practice.

The AWG's trigger channels can be freely linked to a variety of connectors, such as the bidirectional Ref/Trigger connectors on the front panel, and other functional units inside the instrument, such as the Scope or the Demodulators. This freedom of configuration is enabled by the Cross-Domain Trigger feature and enables triggering and execution control that goes beyond the synchronization between AWG and external devices. In Section 3.8.6 we will discuss how to use this possibility to synchronize the detection tools of the UHF platform with the AWG.

Triggering the AWG

In this section we show how to trigger the AWG with an external TTL signal. We start by generating a TTL signal on the (bidirectional) Ref / Trigger 2 connector on the front panel. This simulates a trigger coming from an external device and is entirely independent of the AWG. The TTL signal has the frequency of the internal oscillator 2 which we set to 300 kHz. Apply the settings listed in the following table.

Table 3.28. Settings: generate a 300 kHz TTL signal on Ref / Trigger 2

Tab	Sub-tab	Section	#	Label	Setting / Value / State
DIO		Ref / Trigger	2	Output Signal	Osc φ Demod 8
DIO		Ref / Trigger	2	Drive	ON
Lock-in	All	Oscillators	2	Frequency	300 kHz
Lock-in	All	Demodulators	8	Osc	2

The AWG has 4 trigger input channels. As discussed, these are not directly associated with physical device inputs but can be freely configured to probe a variety of internal or external signals. Here, we link the AWG Analog Trigger 1 to the physical Ref / Trigger 1 connector.

Table 3.29. Settings: configure the AWG analog trigger input

Tab	Sub-tab	Section	#	Label	Setting / Value / State
AWG	Trigger	Analog Trigger 1		Edge Rise	ON
AWG	Trigger	Analog Trigger 1		Signal	Trig Input 1

Finally, we modify our last AWG program by including a `waitAnaTrigger` command just before the `playWave` command. The result is that upon every repetition inside the infinite `while` loop, the AWG will wait for a rising flank on Ref / Trigger input 1.

```
wave w_gauss = gauss(640, 320, 50);
wave w_rise  = cut(w_gauss, 0, 319);
wave w_fall  = cut(w_gauss, 320, 639);
wave w_flat  = rect(320, 1.0);

wave w_pulse = join(w_rise, w_flat, w_fall);

while (true) {
    waitAnaTrigger(1, 1);
    playWave(1, w_pulse);
}
```

Compile and run the above program. Figure 3.46 shows the pulse series as seen in the Scope: the pulses are now spaced by the oscillator period of about 3.3 μs, unlike previously when the period was determined by the length of the waveform `w_pulse`. Try changing the oscillator frequency in the Lock-in tab, or unplugging the trigger cable, to observe the immediate effect on the signal.

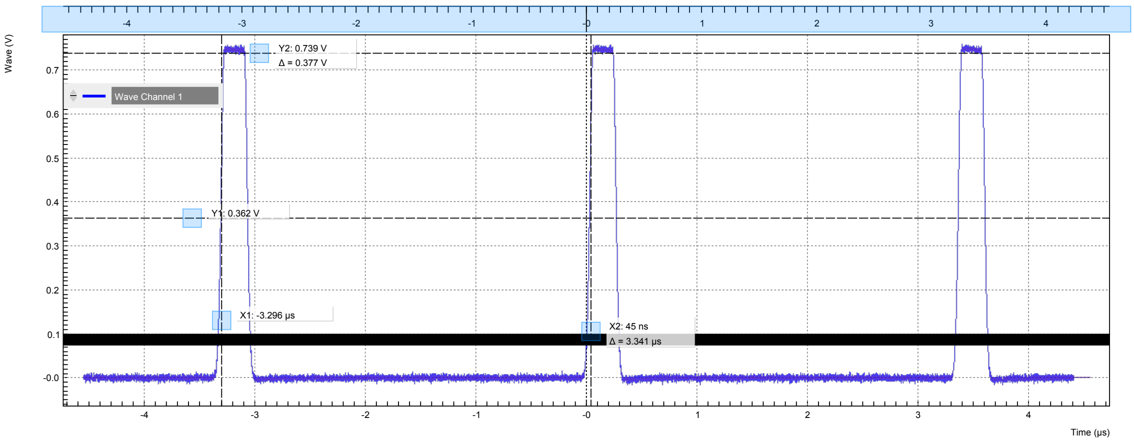


Figure 3.46. Externally triggered pulse series generated by the AWG and captured by the LabOne Scope

Generating Triggers with the AWG

There are two ways of generating trigger output signals with the AWG: as markers, or through sequencer commands.

The method using markers is recommended when precise timing is required, and/or complicated serial bit patterns need to be played on the trigger outputs. Marker bits are part of every waveform which is an array of 16-bit words: 14 bits of each word represent the analog waveform data, and the remaining 2 bits represent two digital marker channels. Upon playback, a digital signal with sample-precise alignment with the analog output is generated.

The method using a sequencer command is simpler, but the timing control is less flexible than when using markers. It is useful for instance to generate a single trigger signal at the start of an AWG program.

Table 3.30. Comparison: AWG markers and triggers

	Marker	Trigger
Implementation	Part of waveform	Sequencer command
Timing control	High	Low
Generation of serial bit patterns	Yes	No

	Marker	Trigger
Cross-device synchronization	Yes	Yes

Let us first demonstrate the use of **markers**. In the following code example we first generate a Gaussian pulse again. The so generated wave does include marker bits – they are simply set to zero by default. We use the `marker` function to assign the desired non-zero marker bits to the wave. The `marker` function takes two arguments, the first is the length of the wave, the second is the marker configuration in binary encoding: the value 0 stands for a both marker bits low, the values 1, 2, and 3 stand for the first, the second, and both marker bits high, respectively. We use this to construct the wave called `w_marker`.

```
const marker_pos = 3000;

wave w_gauss  = gauss(8000, 4000, 1000);
wave w_left   = marker(marker_pos, 0);
wave w_right  = marker(8000-marker_pos, 1);
wave w_marker = join(w_left, w_right);
wave w_gauss_marker = w_gauss + w_marker;

playWave(1, w_gauss_marker);
```

The waveform addition with the '+' operator adds up analog waveform data but also combines marker data. The wave `w_gauss` contains zero marker data, whereas the wave `w_marker` contains zero analog data. Consequentially the wave called `w_gauss_marker` contains the merged analog and marker data. We use the integer constant `marker_pos` to determine the point where the first marker bit flips from 0 to 1 somewhere in the middle of the Gaussian pulse.

Note

The add function and the '+' operator combine marker bits by a logical OR operation. This means combining 0 and 1 yields 1, and combining 1 and 1 yields 1 as well.

The following table summarizes the settings to apply in order to output marker 1 on Ref / Trigger 2, and to configure the scope to trigger on Ref / Trigger 1.

Table 3.31. Settings: configure the AWG marker output and scope trigger

Tab	Sub-tab	Section	#	Label	Setting / Value / State
DIO		Output	2	Signal	AWG Marker 1
DIO		Output	2	Drive	ON
Scope	Trigger	Trigger		Signal	Trig Input 1

Figure 3.47 shows the AWG signal captured by the Scope. The green curve shows the second Scope channel (requires UHF-DIG option) configured to display the Trigger Input 1 signal. Try changing the `marker_pos` constant and re-running the sequence program to observe the effect on the temporal alignment of the Gaussian pulse.

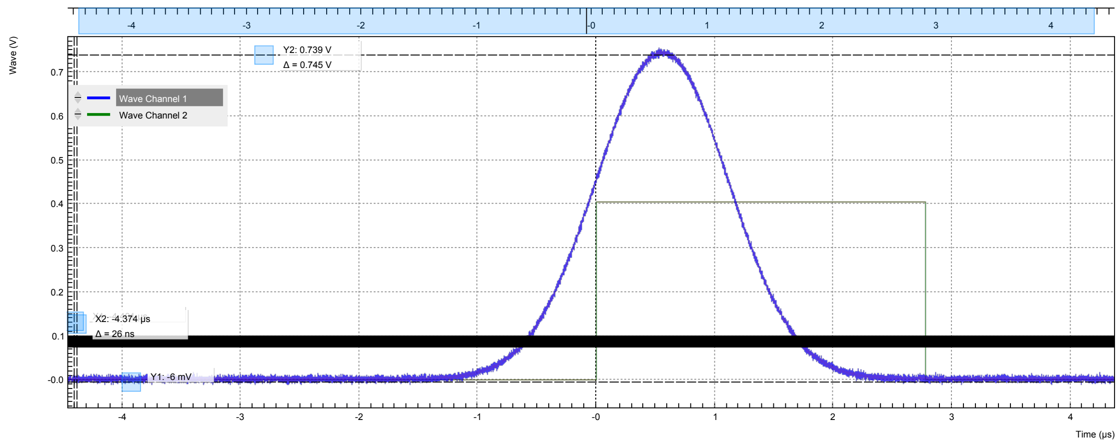


Figure 3.47. Pulse and marker signal generated by the AWG and captured by the LabOne Scope (dual-channel Scope operation requires UHF-DIG option)

Let us now demonstrate the use of **sequencer commands** to generate a trigger signal. Copy and paste the following code example into the Sequence Editor.

```
wave w_gauss  = gauss(8000, 4000, 1000);

setTrigger(1);
playWave(1, w_gauss);
waitWave();
setTrigger(0);
```

The `setTrigger` function takes a single argument encoding the four AWG trigger output states in binary manner – the integer number 1 corresponds to a configuration of 1/0/0/0 for the trigger outputs 1/2/3/4. We included a `waitWave` command after the `playWave` command. It ensures that the subsequent `setTrigger` command is executed only after the Gaussian wave has finished playing, and not during waveform playback.

We reconfigure the Ref / Trigger 2 connector such that it outputs the AWG Trigger 1, instead of the AWG Marker 1. The rest of the settings can stay unchanged.

Table 3.32. Settings: configure the AWG trigger output

Tab	Sub-tab	Section	#	Label	Setting / Value / State
DIO		Output	2	Signal	AWG Trigger 1

Figure 3.48 shows the AWG signal captured by the Scope. This looks very similar to Figure 3.47 in fact. Note that in with this method, we're not so flexible in choosing the trigger time, as the rising trigger edge will always be at the beginning of the waveform. But we don't have to bother about assigning the marker bits to the waveform.

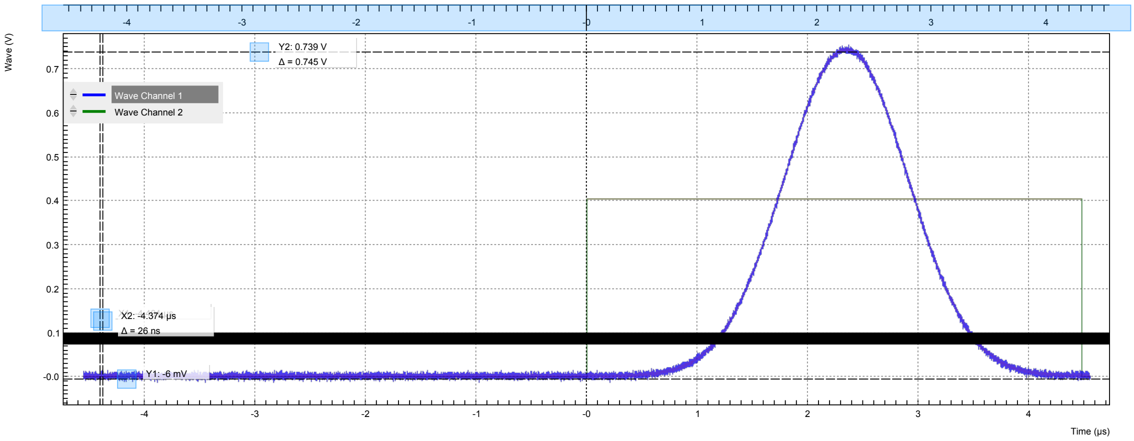


Figure 3.48. Pulse and trigger signal generated by the AWG and captured by the LabOne Scope (dual-channel Scope operation requires UHF-DIG option)

Controlling the AWG Repetition Rate

Finally we show how to synchronize the AWG signal generation with one of the internal oscillators. This enables easy control of the signal repetition rate. It is particularly useful when combining the AWG with synchronous detection methods available on the UHF platform, such as the UHFLI Lock-in Amplifier, or the UHF-BOX Boxcar Averager.

We achieve this by including a `waitOscPhaseOfDemod` command in our Sequencer program. This command works similarly to the `waitAnaTrigger` command. In the following example, the AWG will wait in each repetition until the oscillator phase of demodulator 8 passes through zero.

```
wave w_gauss = gauss(640, 320, 50);

while (true) {
    waitOscPhaseOfDemod(8);
    playWave(1, w_gauss);
}
```

The oscillator frequency of demodulator 8 should still be set to 300 kHz from previous examples. Playing the above AWG program produces a signal similar to that shown in Figure 3.46. However, the AWG is now independent of the external trigger signal which simplifies the setup.

3.8.5. Direct and Modulation Mode

One of the key features of the AWG is the ability to work in amplitude modulation mode, where the output of the AWG is multiplied with the amplitude of one or more of the internal oscillator signals of the device. There are numerous advantages to using modulation mode in comparison to simply generating the sinusoidal signal directly using the AWG, such as the ability to change the frequency at will or even control the frequency using the PID/PLL, extremely high frequency resolution independent of AWG waveform length, phase-coherent generation of signals (because the oscillator keeps running even when the AWG is off), ability to analyze input signal at the exact frequency of the generated signal using demodulators, Boxcar and PWA, and more. The goal of this section is to demonstrate how to use the modulation mode.

We design this example around a common use case, which is the generation of dual-channel quadrature (I/Q) modulation signals to feed into a microwave mixer. Such signals require the independent control of two envelope waveforms multiplied by a carrier that is shifted by 90° between the two channels. The program below generates two independent waveforms and plays them repeatedly on both channels. For dual-channel playback we can use the same `playWave` function that we used up to now, and simply pass to it two waveforms as arguments. We include the previously used trigger commands for the scope, and include a `wait` command whose argument is in units of the sequencer clock period of about 4.44 ns.

```
wave w_gauss = gauss(8000, 4000, 1000);
wave w_drag  = drag(8000, 4000, 1000);

while (true) {
    setTrigger(1);
    playWave(w_gauss, w_drag);
    waitWave();
    setTrigger(0);
    wait(100);
}
```

For Amplitude Modulation mode, the AWG Channel 1 is assigned to the oscillator signal of demodulator 4, and AWG Channel 2 is assigned to the oscillator signal of demodulator 8. If the UHF-MF Multi-frequency option is installed, we have the freedom to wire the same oscillator to both demodulators, which is an advantage if we want to control the relative carrier phase of the two AWG channels like in this case. Without the UHF-MF option, the two demodulators (and so the two AWG channels) are assigned to independent oscillators. In this case, relative phase control is possible but it requires some manual tuning. The following parameter settings apply to the case with installed UHF-MF option.

Table 3.33. Settings: configure the AWG marker output and scope trigger

Tab	Sub-tab	Section	#	Label	Setting / Value / State
DIO		Output	2	Signal	AWG Trigger 1
DIO		Output	2	Drive	ON
Scope	Trigger	Trigger		Signal	Trig Input 1
Lock-in	All	Oscillators	1	Frequency	5 MHz
Lock-in	All	Demodulators	4	Osc	1
Lock-in	All	Demodulators	8	Osc	1
Lock-in	All	Demodulators	8	Phase	0°
Lock-in	All	Demodulators	4	Phase	90°
AWG		Output 1		Mode	Direct
AWG		Output 2		Mode	Direct

Save and play the Sequencer program with the above settings. The upper plot in Figure 3.49 shows the AWG signals captured by the Scope. We see the expected Gaussian pulse on AWG channel 1 (green) and the DRAG pulse, which corresponds to the derivative of a Gaussian function, on AWG channel 2.

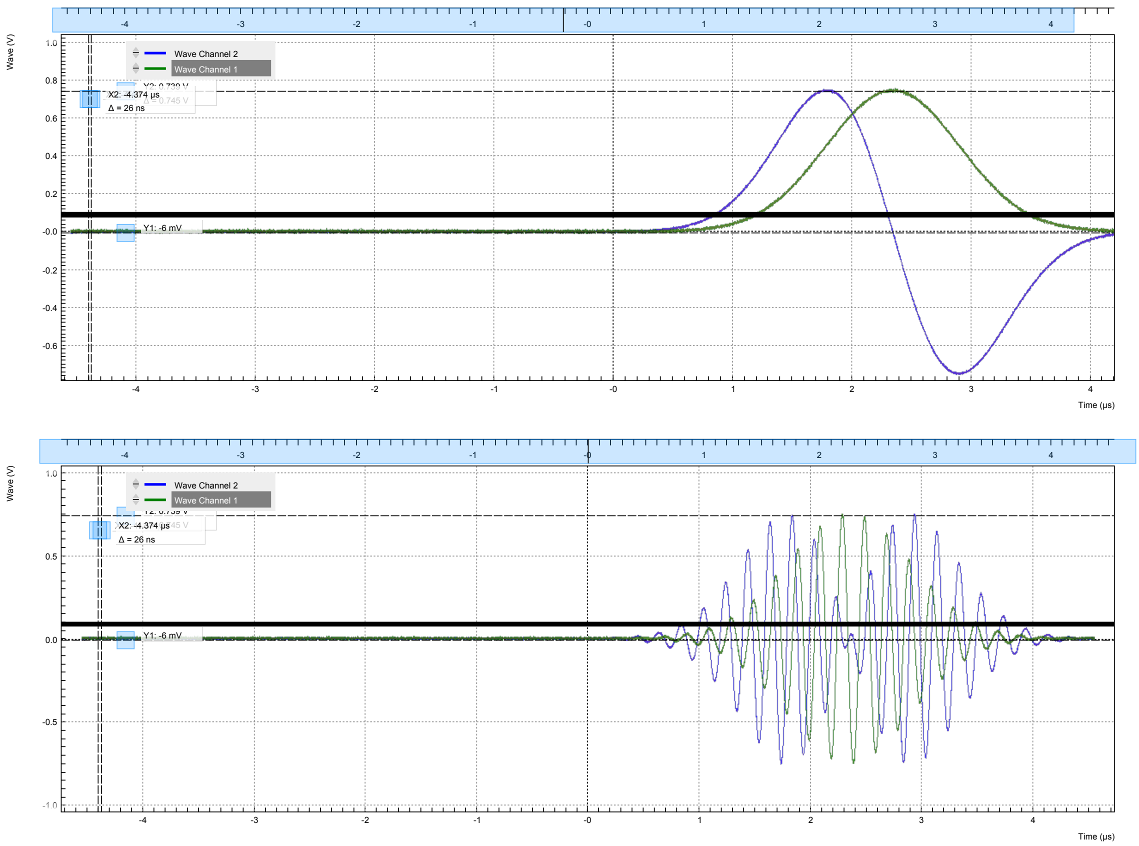


Figure 3.49. Dual-channel signal generated by the AWG and captured by the LabOne Scope (dual-channel Scope operation requires UHF-DIG option). The top figure shows two envelope waveforms played in Direct mode, the bottom figure shows the same envelope waveforms played in Amplitude Modulation mode.

While the AWG is running, you can go ahead now and switch both AWG channels to Modulation mode. The lower plot in Figure 3.49 shows the resulting signals, which are the Gaussian and DRAG pulses multiplied by a 5 MHz carrier with phase shift 0° and 90°, respectively.

Table 3.34. Settings: set both AWG channels to Modulation mode

Tab	Sub-tab	Section	#	Label	Setting / Value / State
AWG		Output 1		Mode	Modulation
AWG		Output 2		Mode	Modulation

Note that in this practical case of I/Q modulation, the two AWG channels typically require further adjustments of the pulse amplitude, DC offset, and inter-channel phase offset in order to compensate for analog mixer imperfections. All these adjustments can now be done on the fly using the AWG Amplitude, the Signal Output Offset, and the Demodulator Phase settings without having to make any changes to the programmed AWG waveforms.

When playing waveforms in modulation mode, it can sometimes be necessary to synchronize the envelope with the phase of the carrier. This will lead to a final pulse shape that is exactly the same in every repetition. This synchronization is easily achieved with the `waitOscPhaseOfDemod` command introduced previously. In the following program, we use this command to align the start of the waveform playback with the oscillator phase of demodulator 4, i.e., the carrier phase.

```
wave w_gauss = gauss(2000, 1000, 200);
```

```
while (true) {
    waitOscPhaseOfDemod(4);
    setTrigger(1);
    playWave(1, w_gauss);
    waitWave();
    setTrigger(0);
    wait(100);
}
```

Look at the generated signal once with and once without the `waitOscPhaseOfDemod` command. As shown in Figure 3.50, you will see that with this synchronization command, every generated pulse looks exactly the same.

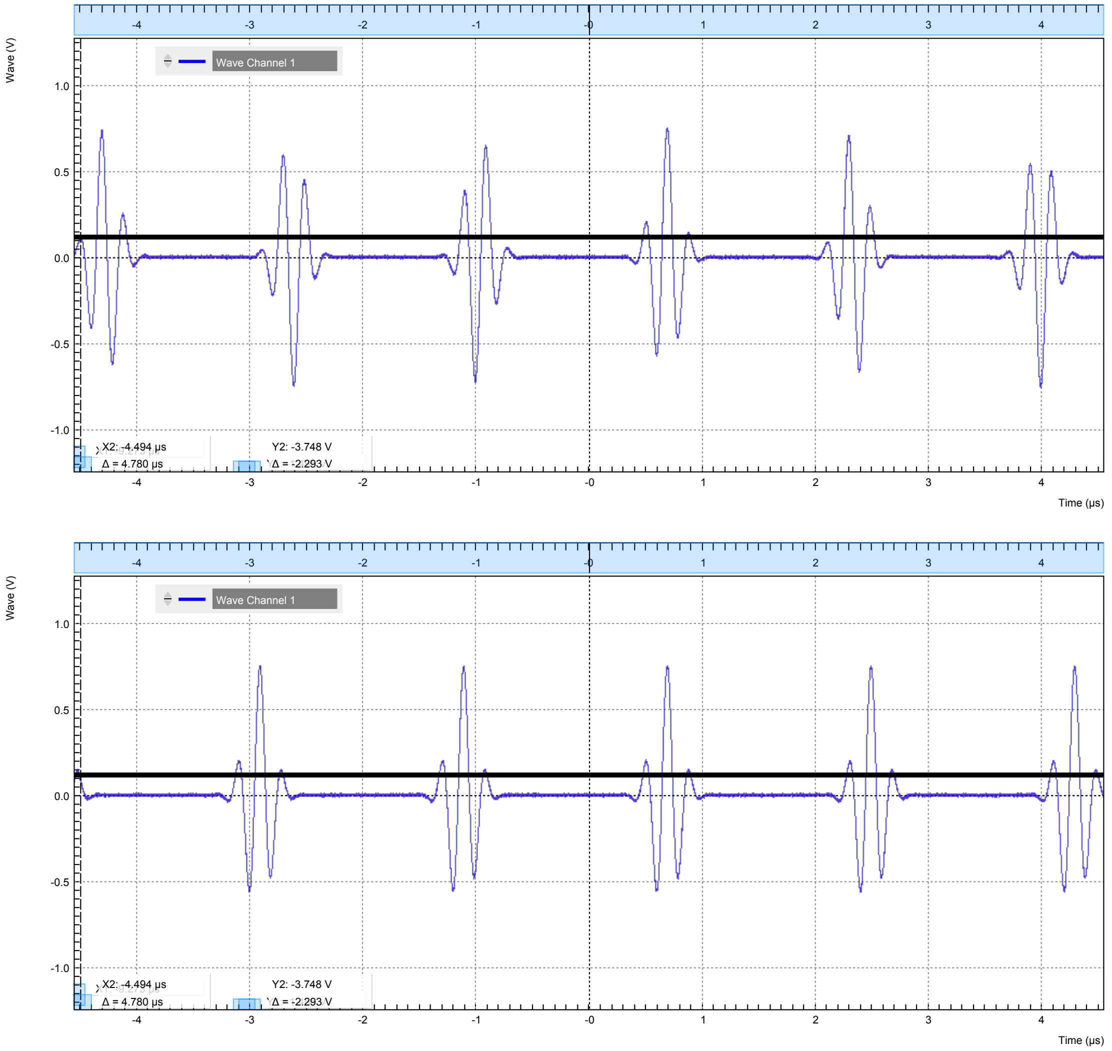


Figure 3.50. Amplitude-modulated signal generated by the AWG and captured by the LabOne Scope. The top figure shows repeated waveform without synchronization between carrier and envelope phase. The bottom figure shows the same signal but with synchronization.

3.8.6. Combining Signal Generation and Detection

The base version of the UHF Arbitrary Waveform Generator contains a single-channel Scope for data acquisition. The UHF hardware platform can furthermore be equipped with a Boxcar Averager, Pulse Counter, or Lock-in Amplifier for more advanced measurement tasks. In this section, we will demonstrate how to use Cross-Domain Triggering and other features in order to combine AWG signal generation and detection in an efficient, precise, and easy way.

Scope/Digitizer: Having the Scope/Digitizer in the same housing as the AWG enables internal routing of trigger and marker signals from the AWG to the Scope. The setup of AWG triggers and markers for internal routing is in no way different than for external use and is explained in the section called “Generating Triggers with the AWG”. Once generated, the generated trigger/marker signals can be selected simply in the Trigger Signal setting of the Scope:

Table 3.35. Settings: configure Scope for internal triggering by AWG

Tab	Sub-tab	Section	#	Label	Setting / Value / State
Scope	Trigger	Trigger		Signal	AWG Trigger 1

Alternatively, AWG Trigger 2–4 or AWG Marker 1–4 can be used.

Boxcar Averager: The UHF-BOX Boxcar averager is an excellent tool for the analysis of signals with low duty cycles. In combination with the AWG, it is well suited whenever the setup response to a short, repeated pulse needs to be measured. Boxcar averager and AWG have to be referenced to the same internal oscillator for proper synchronization. Here, let's consider the case where the AWG generates a signal on Signal Output 1, and the boxcar unit 1 analyzes the return signal on Signal Input 1. Both the AWG and the Boxcar are referenced to oscillator 1. The following table summarizes the necessary settings.

Table 3.36. Settings: configure the Boxcar averager and AWG common reference

Tab	Sub-tab	Section	#	Label	Setting / Value / State
Boxcar	Boxcar	Signal Input	1	Osc	1
Boxcar	Boxcar	Signal Input	1	Input Signal	Sig In 1
AWG	Control	Output 1		Mode	Direct
Lock-in	All	Demodulators	1	Osc	1
Lock-in	All	Oscillators	1	Frequency	200 kHz

Synchronization between the AWG and oscillator 1 is achieved with the `waitOscPhaseOfDemod` used like in the exemplary Sequencer program below. Please refer to the section called “Controlling the AWG Repetition Rate” for more information.

```
wave w_gauss = gauss(640, 320, 50);

while (true) {
    waitOscPhaseOfDemod(1);
    playWave(1, w_gauss);
}
```

Please consider that the duration of the AWG pattern played after the `waitPhaseOfDemod` command should be shorter than one period of the reference oscillator.

The Periodic Waveform Averager (PWA) is a part of the Boxcar averager and usually serves to select a suitable Boxcar integration window. The working principle of the PWA and of the AWG impose some conditions on the repetition frequency. As explained in the section called “Low Duty Cycle Analysis with Period Waveform Analyzer”, the PWA requires a repetition frequency that is incommensurable with the base frequency of 450 MHz in order to faithfully measure a waveform. For the AWG on the contrary, it is preferable to have a repetition rate that is commensurable with the base frequency of 450 MHz, since otherwise the AWG signal jitters by one sequencer period of 4.44 ns.

In choosing between these contradicting requirements, it is usually better to select an repetition frequency that is optimized for a jitter-free AWG signal, i.e., a frequency that is **commensurable** with 450 MHz. Even though this means that the PWA signal will not look smooth, the Boxcar averager is unaffected by this and performs well. The PWA signal is usually still good enough to help adjusting the Boxcar gate start phase and width. Alternatively, you can use the LabOne Sweeper to vary the Boxcar gate parameters in order to maximize the Boxcar averager signal-to-noise ratio.

Pulse Counter: The UHF-CNT Pulse Counter supports two run modes that can make use of trigger signals generated by the AWG, gated free running and gated modes. In gated free running mode, the AWG trigger signal defines a pulse counting period by the rising and falling edge of its trigger signal. In gated mode, the AWG trigger signal resets the periodic Pulse Counter timer. Setting up AWG triggers for this purpose is in no way different than for external use and you can follow the explanations in the section called “Generating Triggers with the AWG”. The following table summarizes the necessary settings in the Counter tab.

Table 3.37. Settings: configure the Counter gating by the AWG

Tab	Sub-tab	Section	#	Label	Setting / Value / State
Counter	1		1	Gate Input	AWG Trigger 1
Counter	1		1	Mode	Gated Free Running or Gated

Alternatively, AWG Trigger 2–4 can be used as Gate Input.

Lock-in Averager: The combination of AWG and lock-in amplifier enables a number of fast sweeper measurement modes described in Section 3.8.8.

3.8.7. Branching and Feed-Forward

Using its branching capabilities, the UHF AWG can select the next waveform based on external conditions such as the state of the 32-bit digital input, or internal conditions such as the value of a demodulated signal quadrature.

- Branching based on external conditions is typically used in automatic testing in order to allow for fast and flexible control of the AWG playback sequence.
- Branching based on internal conditions enables fast feed-forward protocols used for instance in quantum computing.

Inside a Sequencer program, branching is realized with the `if` statement or with the `switch...case` statement (cf. Section 4.22.3 to learn about the timing difference of the two). In the example below, we read the state of the AWG Analog Trigger Input 1 first, and depending on its state (1 or 0) we play a Gaussian waveform with amplitude 0.5 or 1.0.

```
wave w_gauss_low  = 0.5*gauss(8000, 4000, 1000);
wave w_gauss_high = 1.0*gauss(8000, 4000, 1000);

var trigger_state;
while (true) {
    trigger_state = getAnaTrigger(1);
    if (trigger_state == 0) {
        playWave(1, w_gauss_low);
    } else {
        playWave(1, w_gauss_high);
    }
}
```

The AWG output depends on how we configure the AWG Analog Trigger Input 1, and what physical signal we provide on that input. The Trigger source may be chosen with the Analog Trigger 1 Signal setting. For a sequence branching application, the Trigger would normally be used in a level-sensitive (as opposed to edge-sensitive) mode, which means that the Rise and Fall checkboxes would be disabled.

In order to set up branching based on external conditions, the Analog Trigger 1 Signal would be set to a physical trigger input, such the Ref / Trigger 2 input. This setting corresponds to a case in which the playback is controlled by an external TTL signal. Much more complex examples can be constructed by using the 32-bit DIO input. This input can be read using the `getDIO` command that works analogously to the `getAnaTrigger` command used here.

Branching based on internal conditions is available in the combination of UHF-AWG Arbitrary Waveform Generator with signal detection units such as the UHFLI Lock-in Amplifier or the UHF-CNT Pulse Counter. In the following we will look into this unique configuration in more detail.

We shall consider the combination of UHFLI and UHF-AWG and realize the situation where a demodulator output signal figures as the AWG Analog Trigger 1 Signal in order to realize a fast feed-forward protocol. This could correspond to a controlled-reset protocol of a quantum bit (qubit), see Phys. Rev. Lett. 109, 240502 (2012). In this protocol, a qubit state is determined in a fast lock-in measurement, and if the measurement yields that the qubit is in an excited state, a reset pulse is applied immediately afterwards. We use the following Sequencer program in order to demonstrate this method.

```
wave w_gauss_low  = 0.5*gauss(8000, 4000, 1000);
wave w_gauss_high = 1.0*gauss(8000, 4000, 1000);

var trigger_state;
while (true) {
    waitOscPhaseOfDemod(8);
    playWave(1, w_gauss_low);
    waitWave();
    trigger_state = getAnaTrigger(1);
    if (trigger_state == 0) {
        playWave(1, w_gauss_low);
 } else {
        playWave(1, w_gauss_high);
    }

    wait(3000);

    playWave(1, w_gauss_high);
    waitWave();
    trigger_state = getAnaTrigger(1);
    if (trigger_state == 0) {
        playWave(1, w_gauss_low);
    } else {
        playWave(1, w_gauss_high);
    }
}
```

The program consists of two almost identical blocks enclosed in an infinite `while` loop. In each block, we first play a Gaussian pulse - let's call this the measurement pulse. Then we obtain the Analog Trigger 1 state, and then we perform a conditional playback of another Gaussian pulse, let's call this one the reset pulse.

The two blocks only differ by the amplitude of the measurement pulse: it is either 0.5 or 1.0. This difference can be detected by a fast lock-in measurement. We let the AWG run in Modulation mode using a carrier frequency of 5 MHz, and we configure Demodulator 1 to measure at the same frequency. We set the Demodulator filter time constant to 3 μs, a value that is comparable to the width of the measurement pulse. This means that the demodulator filter roughly integrates the signal over the pulse width.

If we configure the AWG Analog Trigger Input 1 with the appropriate Signal (Demodulator 1 R) and Level (140 mV), the AWG will be able to discriminate the high- and low-amplitude measurement pulses. As a consequence, it will play a low-amplitude reset pulse after a low-amplitude measurement pulse, and a high-amplitude reset pulse after a high-amplitude measurement pulse. Note that the `waitWave` command ensures that the subsequent command (the `getAnaTrigger` command which evaluates the measurement value) is executed immediately after (and not during) the playback of the measurement pulse. In our case this is just the right timing to obtain a meaningful demodulator measurement taking into account the demodulator settling time.

The following table summarizes the settings to be made for the feed-forward experiment.

Table 3.38. Settings: configure the AWG and Demodulators for feed-forward

Tab	Sub-tab	Section	#	Label	Setting / Value / State
AWG	Trigger	Analog Trigger	1	Rise	OFF
AWG	Trigger	Analog Trigger	1	Fall	OFF
AWG	Trigger	Analog Trigger	1	Signal	Demodulator 1 R
AWG	Trigger	Analog Trigger	1	Level	140 mV
AWG	Control	Output	1	Mode	Modulation
Lock-in	All	Low-Pass Filter	1	Order	1
Lock-in	All	Low-Pass Filter	1	TC	3 μs
Lock-in	All	Demodulators	1	Osc	1
Lock-in	All	Demodulators	8	Osc	2
Lock-in	All	Oscillators	1	Frequency	5 MHz
Lock-in	All	Oscillators	2	Frequency	1 kHz
Scope	Trigger	Trigger		Signal	Osc φ Demod 8
Scope	Trigger	Trigger		Enable	ON
Scope	Trigger	Trigger		Run / Stop	ON

Figure 3.51 shows the signal generated by the AWG in blue. With the UHF-DIG option, we can simultaneously display the R signal of Demodulator 1 in green. The Y2 cursor position shows the AWG Analog Trigger 1 Level of 140 mV. We can observe that the second and the fourth pulse are indeed played conditionally on the demodulator measurement which is evaluated immediately after the measurement pulse has ended. If you adjust the Trigger Level, you will see the live effect on the second and fourth pulse in the signal.

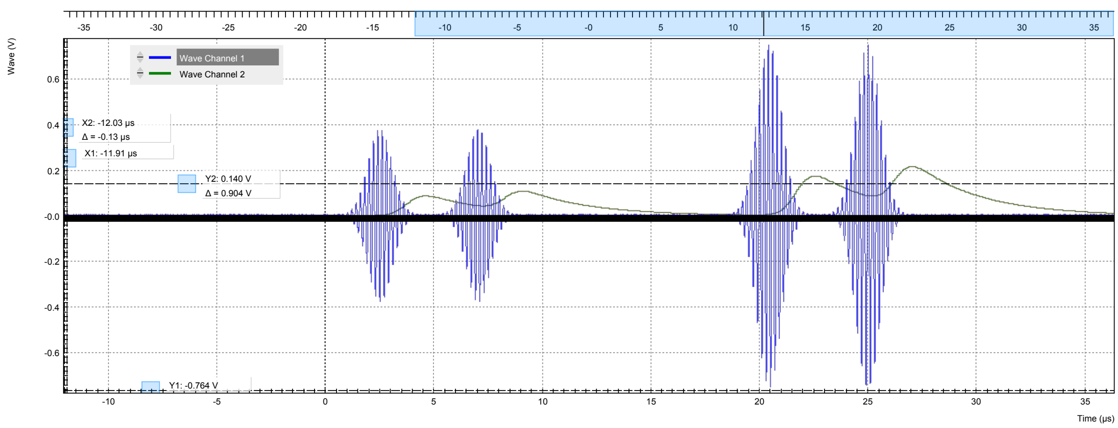


Figure 3.51. Signal generated by the AWG (blue) and demodulated signal (green; displaying this signal requires UHF-DIG option) captured by the LabOne Scope.

3.8.8. Fast AWG Sweeper modes

The LabOne Sweeper offers special operation modes for fast measurement in combination with the UHF-AWG Arbitrary Waveform Generator. These modes take advantage of the powerful execution control of the LabOne AWG Sequencer and of the high measurement speed of the demodulators. Before using the Sweeper in the special modes, it's best to first become familiar with its basic operation. This is described in the functional description of the Sweeper as well as in the PLL tutorial.

Note

For both operation modes described below, the demodulator measurement data rate can be pushed to very high values by gating the demodulator data stream with one of the AWG trigger output channels. Gating is activated using the Trigger setting of the Lock-in tab (collapsed by default). By this method, one can achieve that only the interesting data is transferred to the host PC, but at a much increased peak rate up to 14 MSa/s. This is attractive for applications relying on short and fast measurements interrupted by long dead times.

AWG Index Sweep

The Index Sweep mode allows for recording demodulator samples during a rapid pulse sequence played by the AWG. Typically, a fast series of N pulse patterns is played by the AWG, and in each iteration one parameter is changed, e.g. a pulse length, a pulse amplitude, or a pulse delay. For each iteration, the Sweeper records one demodulator sample. The timing of the demodulator measurement relies on one of the AWG Trigger output channels controlled with the AWG `setTrigger` command. The attribution to a sweep point n=1,...,N relies on the `setID(n)` command in the AWG sequence program. This command tags the demodulator samples with an identifier number n. The Sweeper listens to the AWG Trigger channel selected in the Sweep Parameter setting. When it receives a trigger, it records a demodulator measurement and attributes it to the sweep point corresponding to the current identifier number. To fine-tune the measurement timing relative to the trigger time, it is advised to configure the settling time in the Settings section of the Sweeper (Advanced Mode).

In the example shown below, we play a Gaussian pulse of width ~1 µs in modulation mode and vary the delay of this pulse relative to a trigger in 1000 steps. We demodulate the signal with demodulator 1 with maximum bandwidth and obtain this data with the Sweeper. As a result, we are able to reproduce the envelope of the fast pulse in the Sweeper.

In order for the Index Sweep mode to work as desired, the AWG sequence program needs to be compatible with the settings in the Sweeper and in the Lock-in tab. This means the AWG program should contain a `for` or `while` loop with a loop count identical to the sweep Length parameter (1000 in this example). The `setID` needs to be applied inside the loop with the loop count variable as an argument (in the example the count variable is called `n`). The `setTrigger` command needs to be applied twice inside the loop: once to set the trigger to the high state, and once to set it back to the low state. The demodulator in use needs to be enabled in the Lock-in tab with a sufficiently high bandwidth compared to the AWG pulse pattern. For this example we set the Filter bandwidth to 5.6 MHz (1^{st} order) and the demodulator sample rate to 400 kSa/s. Finally, it's important that the AWG pulse pattern and the demodulator sample rate are compatible: this is easily achieved by using the `waitDemodSample` command with the demodulator number as an argument (here number 1).

The following file shows an example AWG sequence program in which a pulse is generated with a varying delay relative to the AWG Trigger 1. The pulse is to be played in AWG Modulation mode and measured with demodulator 1.

```
var sweepPoints = 1000;
```

```
// define user utility function
void wait_us(const us) {
    wait(us/4.444e-3);
}

wave w_gauss = gauss(8000, 4000, 1000);

// define user function
void user_func(var index) {
    // Set ID for assignment to sweep point
    setID(index);
    // Wait to synchronize AWG and demodulator sampling
    waitDemodSample(1);
    // Set Trigger for Sweeper measurement
    setTrigger(1);
    // Wait a variable time (sweep parameter)
    wait(index);
    // Play waveform
    playWave(1, w_gauss);
    waitWave();
    wait_us(3);
    setTrigger(0);
}

// Loop over sweeper variable (waiting time)
var n;
for (n = 1; n <= sweepPoints; n = n + 1) {
    user_func(n);
    setID(0);
    wait_us(100);
}
setID(0);
```

Once a suitable sequence program is loaded, configure the Sweeper for the measurement. Select AWG 1 → Index Sweep Triggers → AWG Trigger 1 as the Sweep Parameter, corresponding to the trigger channel we used in the Sequence program. Set the sweep Length to 1000. In the Sweeper settings sub-tab, set the Filter to Advanced Mode in order to enable "AWG Control".

Note

The AWG Index Sweep mode is implicitly enabled when the following two settings are made in the Sweeper: 1) AWG Control is enabled 2) either one of AWG Index Trigger 1–4 is selected as the sweep parameter.

If you start now the Sweeper, it will automatically start the AWG and normally capture the data very quickly. The following table summarizes the settings to be made for this example.

Table 3.39. Settings: configure the AWG and Sweeper for AWG Index Sweep mode

Tab	Sub-tab	Section	#	Label	Setting / Value / State
AWG	Control	Output 1/2		Mode	Modulation
Lock-in	All	Demodulators	1	Enable	ON
Lock-in	All	Demodulators	1	Rate	400 kSa/s
Lock-in	All	Demodulators	1	Low-Pass Filter order	1
Lock-in	All	Demodulators	1	Low-Pass Filter bandwidth	5.6 MHz

Tab	Sub-tab	Section	#	Label	Setting / Value / State
Sweeper	Control	Horizontal		Sweep Param	AWG Index Sweep Triggers, Trigger 1
Sweeper	Settings			Filter	Advanced Mode
Sweeper	Settings	Statistics		AWG Control	ON
Sweeper	Control			Single	ON

AWG Parameter Sweep

The AWG Parameter Sweep mode allows for precisely timed demodulator measurements as a function of a large selection of device parameters. This mode combines elements of the basic Sweeper mode and of the AWG Index Sweep mode. Like in the basic Sweeper mode, the Sweeper sets a device parameter (such as an oscillator frequency, an AWG amplitude, or an AWG user register value) and starts measuring the data after that with the possibility to average data for some time. Like in the AWG Index Sweep mode, the precise timing is determined by the AWG Trigger output channel which is controlled with the AWG `setTrigger` command. When the Sweeper receives a trigger, it starts recording demodulator data for the defined averaging period. To fine-tune the measurement timing relative to the trigger time, it is advised to configure the settling time in the Settings section of the Sweeper (Advanced Mode).

In the example shown here, we play a long pulse in the AWG modulation mode and vary the amplitude of the AWG output with the Sweeper. We demodulate the signal with demodulator 1 with maximum bandwidth and obtain this data with the Sweeper.

For such a measurement, the AWG sequence program should contain two `setTrigger` commands that define the measurement time: once to set the trigger channel 1 to the high state, and once to set it back to the low state. The demodulator in use needs to be enabled in the Lock-in tab with a sufficiently high bandwidth compared to the AWG pulse pattern speed. Finally, it's important that the AWG pulse pattern and the demodulator sample rate are compatible: this is easily achieved by using the `waitDemodSample` command with the demodulator number as an argument (here number 1). The following sequence program is used in this example.

```
// Play the example at reduced rate of 28 MSa/s
const RATE = 6; //AWG_RATE_28MHZ;
const FS = 1.8e9/pow(2, RATE);

// Wait a number of microseconds
void wait_us(const us) {
    wait(1e-6*us*225e6);
}

void user_func() {
    // Total waveform length, 1 ms
    const N = 1e-3*FS;
    // Length of rising edge, 100 us
    const M = 100e-6*FS;
    // Create waveform
    wave edges = blackman(2*M, 1.0, 0.2);
    wave w_pulse = join(cut(edges, 0, M-1),
        rect(N-2*M, 1.0),
        cut(edges, M, 2*M-1));
    // Synchronize with demodulator data
    waitDemodSample(1);
    // Start waveform playback
    playWave(1, w_pulse, RATE);
}

// Function for enabling sweeper recording, from/to in microseconds
void sweeper_record(const from_us, const to_us) {
    // Wait a bit, then make sweeper record data
```

```
    wait_us(from_us);
    setTrigger(1);
    wait_us(to_us-from_us);
    setTrigger(0);
}

// Execute the subprograms
user_func();
sweeper_record(100, 900);
```

Note

The AWG Parameter Sweep mode is implicitly enabled when the following two settings are made in the Sweeper: 1) AWG Control is enabled 2) the sweep parameter is anything except AWG Index Trigger 1–4.

If you start now the Sweeper, it will automatically start the AWG and capture the data. The following table summarizes the settings to be made for this example.

Table 3.40. Settings: configure the AWG and Sweeper for AWG Parameter Sweep mode

Tab	Sub-tab	Section	#	Label	Setting / Value / State
AWG	Control	Output 1/2		Mode	Modulation
Lock-in	All	Demodulators	1	Enable	ON
Lock-in	All	Demodulators	1	Rate	400 kSa/s
Lock-in	All	Demodulators	1	Low-Pass Filter order	1
Lock-in	All	Demodulators	1	Low-Pass Filter bandwidth	5.6 MHz
Sweeper	Control	Horizontal		Sweep Param	AWG Output 1 Amplitude
Sweeper	Settings			Filter	Advanced Mode
Sweeper	Settings	Statistics		AWG Control	ON
Sweeper	Control			Single	ON

3.8.9. Four-channel Output

For applications requiring more channels and/or higher voltages, the UHF-AWG can generate signals on the auxiliary outputs of the UHF instrument. To this end the AWG resources for one fast channel (1.8 GSa/s) can be reallocated so as to generate four independent signals at 28 MSa/s and 16 bit resolution in a ±10 V range.

In the sequence program, the functionality is available through the `playAuxWave` command. The command arguments are four waveforms of equal length.

We configure the multi-purpose Auxiliary Outputs for AWG signal generation. This is done by setting the Auxiliary Output Signal to AWG in the Auxiliary tab. Each Auxiliary Output corresponds to one of the four rows in the tab. The Channel setting allows you to route one of the four AWG channels (the four waveforms of the `playAuxWave` command) to the given Auxiliary Output. Typically for the first row the Channel is set to 1, for the second row to 2, and so forth.

We intend to monitor the individual Auxiliary signals with the Scope on Signal Input 1. Before making the corresponding BNC connections, it's good practice to adjust the Auxiliary Output

Lower and Upper Limits in order to prevent damage to the Signal Input. You can use the Scale and the Offset setting in order to modify the signal.

The Auxiliary Outputs have a much lower analog bandwidth than the Signal Outputs. It is therefore necessary to work at a sampling rate below 28 MSa/s. In order to combine slow Auxiliary Output signals with fast signals on the Signal Outputs, it's useful to set the sampling rate for every individual waveform play command. In the following example, we first play four waveforms in parallel on the Auxiliary Outputs at reduced sampling rate, and then one waveform on the Signal Output 2 at full sampling rate.

```
// Sampling rate of the system, adjust accordingly if the rate is reduced
const FS = 1800e6;
// Frequency of the 'sine' in the SINC waveform
const F_SINC = 42e6;

// Generate the four-channel auxiliary output waveform
wave aux_ch1 =  1.0*gauss(8000, 4000, 1000);
wave aux_ch2 =  0.5*gauss(8000, 4000, 1000);
wave aux_ch3 = -0.5*gauss(8000, 4000, 1000);
wave aux_ch4 = -1.0*gauss(8000, 4000, 1000);

// Generate a waveform to be played on Signal Output 2
wave w_sinc = sinc(8000, 4000, FS/F_SINC);

while (true) {
  // play the four Aux Output channels at reduced rate
  playAuxWave(aux_ch1, aux_ch2, aux_ch3, aux_ch4, AWG_RATE_14MHZ);
  // play a wave on Signal Output 2
  playWave(w_sinc, AWG_RATE_1800MHZ);
}
```

The following table summarizes the settings to be made for this example.

Table 3.41. Settings: configure the AWG for generating signals on the Auxiliary Outputs

Tab	Sub-tab	Section	#	Label	Setting / Value / State
AWG	Control	Output 1/2		Mode	Direct
Auxiliary		Aux Output	1-4	Lower/Upper Limit	−1.5 V/+1.5 V
Auxiliary		Aux Output	1-4	Signal	AWG
Auxiliary		Aux Output	1	Channel	1
Auxiliary		Aux Output	2	Channel	2
Auxiliary		Aux Output	3	Channel	3
Auxiliary		Aux Output	4	Channel	4

Note

The four-channel AWG mode features a sample hold functionality: the output voltage of the last sample of a waveform remains fixed after the waveform playback is over. This can be used to control the output voltage between pulses.

3.8.10. Debugging Sequencer Programs

When generating fast signals and observing them with the LabOne Scope, in some configurations you may observe timing jitter or unexpected delays in the generated signal. There are two main reasons for that. The first reason is linked to the AWG's memory architecture, which is based on a main memory and a cache memory. Waveform data stored in the main memory (128 MSa per channel) must be copied to the cache memory (32 kSa per channel) prior to playback. The bandwidth available for this data transfer is less than that required by the AWG for dual-channel operation at 1.8 GSa/s. Therefore, if the AWG is configured to play waveforms longer than what fits in the cache memory in dual-channel mode at 1.8 GSa/s, interruptions in the generated signal may be observed. The second reason is connected to the AWG compiler concept explained in Section 4.22.2. When a program in the Sequence Editor is compiled into machine code that can be executed by the Sequencer hardware, single lines of code may be expanded into several machine instructions. Each instruction requires one clock cycle (4.44 ns) for execution. Therefore, the final timing of the generated waveform may not always be completely apparent from looking solely at the high-level sequencer program. The compiled program, which defines the actual timing, is displayed in the Advanced sub-tab.

Please take the following tips into consideration when operating the UHF-AWG. They should help you prevent and solve timing problems.

- The Scope and the AWG share the same memory, which means that operating them together at high sampling rates affects the performance of both of them. Note that this is only a concern when the AWG is playing back waveforms that are too large to fit in the cache memory. If this is the case it may prove difficult to visualize the generated AWG signal using the LabOne Scope. One option for visualizing such long waveforms is to reduce the sampling rate of both the AWG and the Scope to 225 MHz, which allows both the AWG and the Scope to operate in dual-channel mode simultaneously. The overall shape of the generate AWG signal can then be visualized and evaluated. The sampling rate of the AWG can then be increased once you are satisfied with the shape of the generated signals.
- Minimize waveform memory (1): use the possibility to vary the sample rate during playback. The `playWave` command (and related commands) accept a sampling rate parameter, which means slow and fast signal components can be played at different rates.
- Minimize waveform memory (2): take advantage of the amplitude modulation mode in order to generate signals at the full bandwidth, but with reduced envelope sampling rate.
- Minimize waveform memory (3): in four-channel (Auxiliary Output) mode, the signal amplitude of the last sample after a waveform playback is held. This eliminates the need for long waveforms with constant amplitude, e.g. on a pulse plateau.
- Check the occupied waveform cache memory in the Waveform sub-tab. If you stay below 100%, the performance is best and there is no interference with the LabOne Scope.
- Take advantage of the AWG state signals available on the Trigger outputs. In the DIO tab you can select from a number of options for outputting the AWG state as TTL signals, such as "fetching", or "playing". Monitoring these signals on a scope can help in understanding the AWG timing.
- When possible, use the `repeat` loop instead of the `for` and `while` loops. The `for` and `while` loops evaluate and compare run-time variables, which makes them slower to execute in comparison to the `repeat` loop.
- Fill up sequencer waiting time with useful commands. Placing commands and run-time variable operations just before a `wait` command (and related commands) in the sequence program means they will be executed when the sequencer has time.
- When you need sample-precise timing between analog and digital output signals, use the AWG Markers rather than the AWG Triggers or the DIO outputs.
- When using the four-channel Auxiliary Output mode, be aware that the timing between Signal Output and Aux outputs is not well-defined. Use a scope to adjust inter-channel delays.
- Be aware that the sequencer instruction memory is also segmented into a cache memory and main memory. Very long sequence programs therefore require fetching operations, which costs some time. You can read the memory usage in the Advanced sub-tab.

Chapter 4. Functional Description LabOne User Interface

This chapter gives a detailed description of the functionality available in the LabOne User Interface (UI) for the Zurich Instruments UHFLI. LabOne provides a data server and a web server to control the Instrument with any of the most common web browsers (e.g. Firefox, Chrome, Edge, etc.). This platform-independent architecture supports interaction with the Instrument using various devices (PCs, tablets, smartphones, etc.) even at the same time if needed.

On top of standard functionality like acquiring and saving data points, this UI provides a wide variety of measurement tools for time and frequency domain analysis of measurement data as well as for convenient servo loop implementation.

4.1. User Interface Overview

4.1.1. UI Nomenclature

This section provides an overview of the LabOne User Interface, its main elements and naming conventions. The LabOne User Interface is a browser-based UI provided as the primary interface to the UHFLI. Multiple browser sessions can access the instrument simultaneously and the user can have displays on multiple computer screens. Parallel to the UI the Instrument can be controlled and read out (possibly concurrently) by custom programs written in any of the supported languages (e.g. LabVIEW, MATLAB, Python, C) connecting through the LabOne APIs.

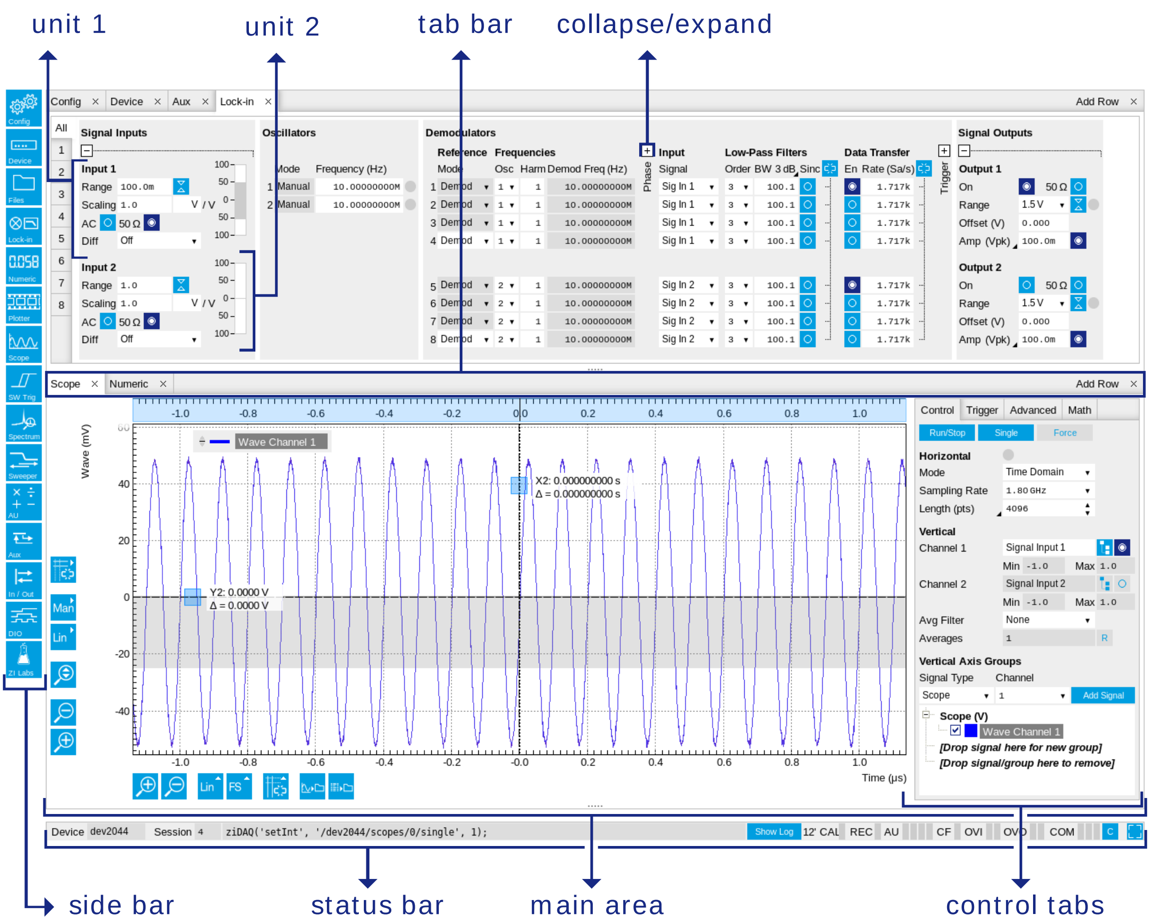


Figure 4.1. LabOne User Interface (default view)

Figure 4.1 shows the LabOne User Interface with the tabs opened by default after a new UI session has been started. The UI is by default divided into two tab rows, each containing a tab structure that gives access to the different LabOne tools. Depending on display size and application, tab rows can be freely added and deleted with the control elements on the right-hand side of each tab bar. Similarly the individual tabs can be deleted or added by selecting app icons from the left side bar. A simple click on an icon adds the corresponding tab to the display, alternatively the icon can be dragged and dropped into one of the tab rows. Moreover, tabs can simply be displaced by drag-and-drop within a row or across rows. Further items are highlighted in Figure 4.2.

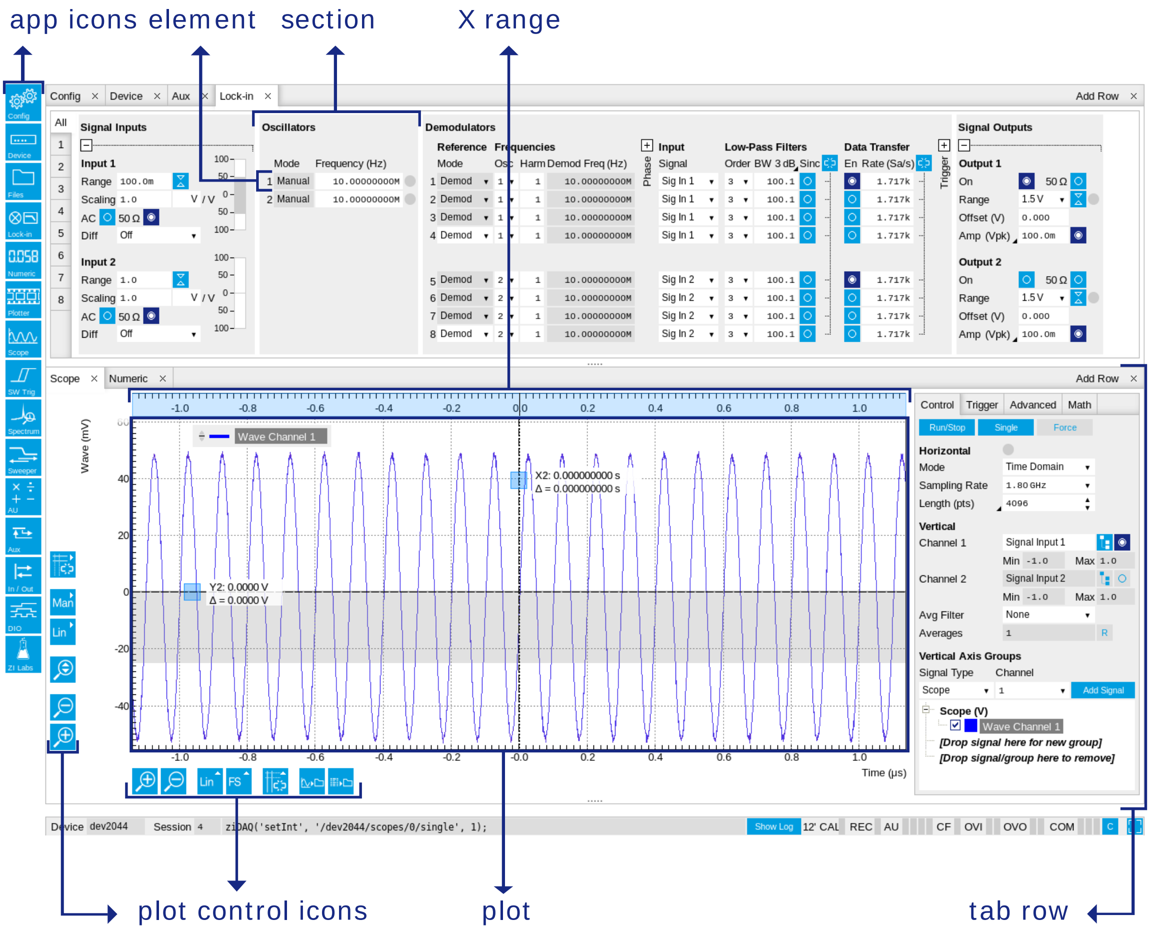


Figure 4.2. LabOne User Interface (more items)

Table 4.1 gives brief descriptions and naming conventions for the most important UI items.

Table 4.1. LabOne User Interface features

Item name	Position	Description	Contains
side bar	left-hand side of the UI	contains app icons for each of the available tabs - a click on an icon adds or activates the corresponding tab in the active tab row	app icons
status bar	bottom of the UI	contains important status indicators, warning lamps, device and session information and access to the command log	status indicators
main area	center of the UI	accommodates all active tabs – new rows can be added and removed by using the control elements in	tab rows, each consisting of tab bar and the active tab area

Item name	Position	Description	Contains
		the top right corner of each tab row	
tab area	inside of each tab	provides the active part of each tab consisting of settings, controls and measurement tools	sections, plots, control tabs, unit selections

4.1.2. Unique Set of Analysis Tools

All Instruments feature a comprehensive tool set for time and frequency domain analysis for both raw signals and demodulated signals. Note that the selection of app icons is limited by the upgrade options installed on a particular instrument.

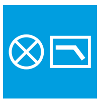
The app icons on the left side of the UI can be roughly divided into two categories: settings and tools. Settings-related tabs are in direct connection of the instrument hardware allowing the user to control all the settings and instrument states. Tools-related tabs place a focus on the display and analysis of gathered measurement data. There is no strict distinction between settings and tools, e.g. will the Sweeper change certain demodulator settings while performing a frequency sweep. Within the tools one can further discriminate between time domain and frequency domain analysis, moreover, a distinction between the analysis of fast input signals - typical sampling rate of 1.8 GSa/s - and the measurement of orders of magnitude slower data - typical sampling rate of <28 MSa/s - derived for instance from demodulator outputs and auxiliary inputs. Table 4.2 provides a brief classification of the tools.

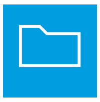
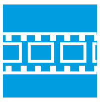
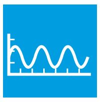
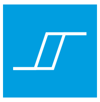
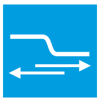
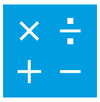
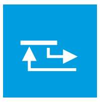
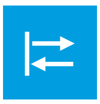
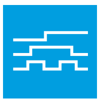
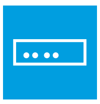
Table 4.2. Tools for time domain and frequency domain analysis

	Time Domain	Frequency Domain
Fast signals (1.8 GSa/s)	Oscilloscope (Scope Tab)	FFT Analyzer (Scope Tab)
	Periodic Waveform Analyzer (Boxcar Tab)	Multi-Harmonic Analyzer (Boxcar Tab)
Slow signals (<28 MSa/s)	Numeric	Spectrum Analyzer (Spectrum Tab)
	Plotter	Sweeper
	Software Trigger	Multi-harmonic Analyzer (Out PWA Tab)
	Periodic Waveform Analyzer (Out PWA Tab)	-

The following table gives the overview of all app icons.

Table 4.3. Overview of app icons and short description

Control/Tool	Option/Range	Description
Lock-in		Quick overview and access to all the settings and properties for signal generation and demodulation.
Lock-in MF		Quick overview and access to all the settings and properties for signal generation and demodulation.

Control/Tool	Option/Range	Description
Files		Access settings and measurement data files on the host computer.
Numeric		Access to all continuously streamed measurement data as numerical values.
Plotter		Displays various continuously streamed measurement data as traces over time (roll-mode).
Scope		Displays shots of data samples in time and frequency domain (FFT) representation.
SW Trig		Provides complex trigger functionality on all continuously streamed data samples and time domain display.
Spectrum		Provides FFT functionality to all continuously streamed measurement data.
Sweeper		Allows to scan one variable (of a wide choice, e.g. frequency) over a defined range and display various response functions including statistical operations.
AU		Real-time arithmetic operations on demodulator outputs.
Aux		Controls all settings regarding the auxiliary inputs and auxiliary outputs.
In/Out		Access to all controls relevant for the main Signal Inputs and Signal Outputs on the instrument's front.
DIO		Gives access to all controls relevant for the digital inputs and outputs including the Ref/ Trigger connectors.
Config		Provides access to software configuration.
Device		Provides instrument specific settings.

Control/Tool	Option/Range	Description
PID		Features all control and analysis capabilities of the PID controllers.
PLL		Features all control and analysis capabilities of the phase-locked loops.
MOD		Control panel to enable (de)modulation at linear combinations of oscillator frequencies.
Boxcar		Boxcar settings and periodic waveform analyzer for fast input signals.
Out PWA		Multi-channel boxcar settings and measurement analysis for boxcar outputs.
AWG		Generate arbitrary signals using sequencing and sample-by-sample definition of waveforms.
Counter		Configure the Pulse Counters for analysis of pulse trains on the digital signal inputs.
ZI Labs		Experimental settings and controls.

Table 4.4 gives a quick overview over the different status bar elements along with a short description.

Table 4.4. Status bar description

Control/Tool	Option/Range	Description
Command log	last command	Shows the last command. A different formatting (Matlab, Python, ..) can be set in the config tab. The log is also saved in [User]\Documents \Zurich Instruments\LabOne \WebServer\Log
Show Log	Show Log	Show the command log history in a separate browser window.
Session	integer value	Indicates the current session identifier.
Device	devXXX	Indicates the device serial number.
Next Calibration	Time or "M"	Remaining minutes until the first calibration is executed or a recalibration is requested.

Control/Tool	Option/Range	Description
		A time interval longer than 99 minutes is not displayed. Manual calibration mode is indicated by an "M".
CAL	grey/yellow/red	State of device self calibration. Yellow: device is warming up and will automatically execute a self calibration after 16 minutes. Grey: device is warmed-up and self calibrated. Red: it is recommended to manually execute a self calibration to assure operation according to specifications.
REC	grey/green	A green indicator shows ongoing data recording (related to global recording settings in the Config tab).
AU	grey/green/red	Arithmetic Unit - Green: indicates which of the arithmetic units is enabled. Red: indicates owerflow.
CF	grey/yellow/red	Clock Failure - Red: present malfunction of the external 10 MHz reference oscillator. Yellow: indicates a malfunction occurred in the past.
OVI	grey/yellow/red	Signal Input Overflow - Red: present overflow condition on the signal input also shown by the red front panel LED. Yellow: indicates an overflow occurred in the past.
OVO	grey/yellow/red	Overflow Signal Output - Red: present overflow condition on the signal output. Yellow: indicates an overflow occurred in the past.
COM	grey/yellow/red	Warning flags related to instrument communication. From left to right: Packet Loss, Sample Loss, Stall. Packet Loss - Red: present loss of data between the device and the host PC. Yellow: indicates a loss occurred in the past. Sample Loss - Red: present loss of sample data between the device and the host PC. Yellow: indicates a loss occurred in the past.

Control/Tool	Option/Range	Description
		Stall - Red: indicates that the sample transfer rates have been reset to default values to prevent severe communication failure. This is typically caused by high sample transfer rates on a slow host computer.
C	C	Reset status flags: Clear the current state of the status flags
RUB	grey/yellow/green	Rubidium Clock - Grey: no rubidium clock is installed. Yellow: Rubidium clock is warming up (takes approximately 300 s). Green: Rubidium clock is warmed up and locked.
BOX	grey/green	Boxcar - Green: indicates which of the boxcar units is enabled.
MOD	grey/green	MOD - Green: indicates which of the modulation kits is enabled.
PID	grey/green	PID - Green: indicates which of the PID units is enabled.
PLL	grey/green	PLL - Green: indicates which of the PLLs is enabled.
Full Screen		Toggles the browser between full screen and normal mode.

4.1.3. Plot Functionality

Several tools - Plotter, Scope, SW Trigger, Spectrum, Sweeper, Boxcar, and outPWA - provide a graphical display of measurement data in the form of plots. These are multi-functional tools with zooming, panning and cursor capability. This section introduces some of the highlights.

Plot area elements

Plots consist of the plot area, the X range and the range controls. The X range (above the plot area) indicates which section of the wave is displayed by means of the blue zoom region indicators. The two ranges show the full scale of the plot which does not change when the plot area displays a zoomed view. The two axes of the plot area instead do change when zoom is applied.

The mouse functionality inside of plot is summarized in Table 4.5

Table 4.5. Mouse functionality inside plots

Name	Action	Description	Performed inside
Panning	left click on any location and move around	moves the waveforms	plot area

Name	Action	Description	Performed inside
Zoom X axis	mouse wheel	zooms in and out the X axis	plot area
Zoom Y axis	shift + mouse wheel	zooms in and out the Y axis	plot area
Window zoom	shift and left mouse area select	selects the area of the waveform to be zoomed in	plot area
Absolute jump of zoom area	left mouse click	moves the blue zoom range indicators	X and Y range, but outside of the blue zoom range indicators
Absolute move of zoom area	left mouse drag-and-drop	moves the blue zoom range indicators	X and Y range, inside of the blue range indicators
Full Scale	double click	set X and Y axis to full scale	plot area

Each plot area contains a legend that lists all the shown signals in the respective color. The legend can be moved to any desired position by means of drag-and-drop.

The X range and Y range plot controls are described in Table 4.6.

Table 4.6. Plot control description

Control/Tool	Option/Range	Description
Axis scaling mode		Selects between automatic, full scale and manual axis scaling.
Axis mapping mode		Select between linear, logarithmic and decibel axis mapping.
Axis zoom in		Zooms the respective axis in by a factor of 2.
Axis zoom out		Zooms the respective axis out by a factor of 2.
Rescale axis to data		Rescale the foreground Y axis in the selected zoom area.
Save figure		Generates an SVG of the plot area or areas for dual plots to the local download folder.
Save data		Generates a TXT consisting of the displayed set of samples. Select full scale to save the complete wave. The save data function only saves one shot at a time (the last displayed wave).
Cursor control		Cursors can be switch On/ Off and set to be moved both

Control/Tool	Option/Range	Description
		independently or one bound to the other one.

Cursors and Math

The plot area provides two X and two Y cursors which appear as dashed lines inside of the plot area. The four cursors are selected and moved by means of the blue handles individually by means of drag-and-drop. For each axis there is a primary cursor indicating its absolute position and a secondary cursor indicating both absolute and relative position to the primary cursor.

Cursors have an absolute position which does not change by pan or zoom events. In case the cursors move out of the zoom area, the corresponding handle is displays on the related side of the plot area. Unless the handle is moved, the cursor keeps the current position. This functionality is very effective to measure large deltas with high precision (as the absolute position of the other cursors does not move).

The cursor data can also be used to define the input data for the mathematical operations performed on plotted data. This functionality is available in the Math sub-tab of each tool. The following Table 4.7 gives an overview of all the elements and their functionality. It is important to know that the Signals and Operations defined will always be performed only on the currently chosen active trace.

Table 4.7. Plot math description

Control/Tool	Option/Range	Description
Source Select		Select from a list of input sources for math operations.
	Cursor Loc	Cursor coordinates as input data.
	Cursor Area	Consider all plot data inside the rectangle defined by the cursor coordinates as input for statistical functions (Min, Max, Avg, Std, Int).
	Tracking	Output plot value at current cursor position. Options are X1 and X2.
	Wave	Consider all plot data currently displayed in the Plot as input for statistical functions (Min, Max, Avg, Std, Int).
	Peak	Find and determine the various peaks in the plotted data and their associated values.
	Histogram	Select statistical data as Math input and display a histogram in the plot area.
Operation Select		Select from a list of mathematical operations to be performed on the selected signals. Choice offered

Control/Tool	Option/Range	Description
		depends on input signals selected.
	X1, X2, X2-X1, Y1, Y2, Y2-Y1	Cursors values and their differences.
	Min, Max, Avg, Std, Int	Statistical Functions applied to a set of samples.
	Pos, Level	Finds the Position (x-values) and the Levels (y-values) of Peaks on a set of samples.
Add	Add	Add the selected math function to the result table below.
Add All	Add All	Add all operations for the selected signal to the result table below.
Select All	Select All	Select all lines from the result table above.
Clear Selected	Clear Selected	Clear selected lines from the result table above.
Unit Prefix	Unit Prefix	Adds a suitable prefix to the SI units to allow for better readability and increase of significant digits displayed.
CSV	CSV	Values of the current result table are saved as a text file into the download folder.
Link	Link	Provides a LabOne Net Link to use the data in tools like Excel, Matlab, etc.
Help	Help	Opens the LabOne User Interface help.

Note

For calculation of the standard deviation the corrected sample standard deviation is used as defined by $\sqrt{\frac{1}{N-1}\sum_{i=1}^{N}(x_i-\overline{x})^2}$ with a total of N samples x_i and an arithmetic average $\overline{x}$.

Tree Sub-Tab

The Numeric tab and Plotter tab are able to display so many different types of signals that a number of different options are provided to access them. One of them is the Tree sub-tab that allows one to access all streamed measurement data in a hierarchical structure by checking the boxes of the signal that should be displayed.

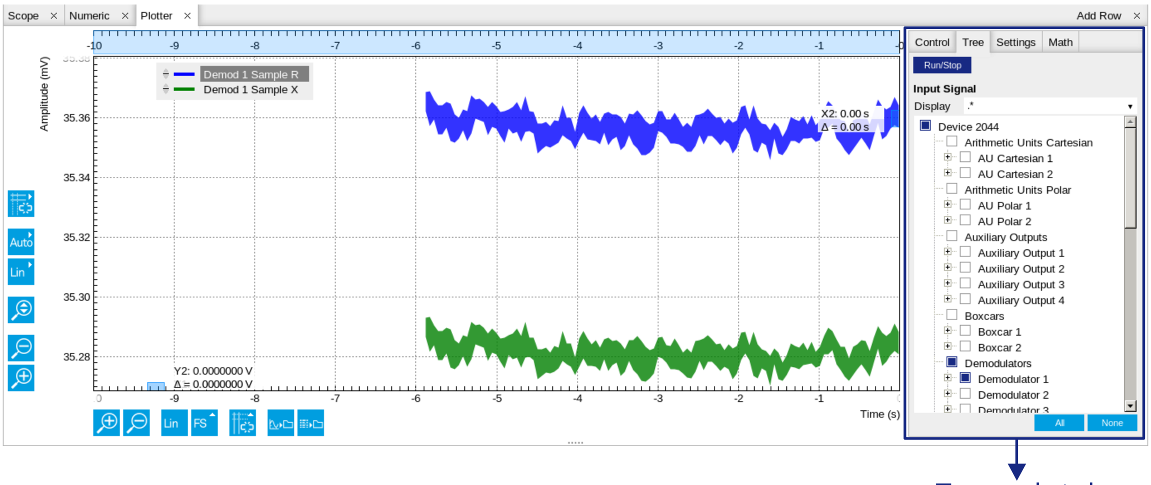


Figure 4.3. Tree sub-tab in Plotter tab

Table 4.8. Tree description

Control/Tool	Option/Range	Description
Display	Preset filter or regular expression	Predefined filters that limit the view to specific signal groups. The display filter does not select any nodes.
All	All	Select all nodes that can be selected in the relevant context.
None	None	Unselect all nodes.

Vertical Axis Groups

Vertical Axis groups are available in the Plotter tab, SW Trigger tab, and Sweeper tab. These tools are able to show signals with different axis properties within the same plot. As a frequency and amplitude axis have fundamentally different limits they have each their individual axis which allows for correct auto scaling. However, signals of the same type e.g. Cartesian demodulator results should preferably share one scaling. This allows for fast signal strength comparison. To achieve this the signals are assigned to specific axis group. Each axis group has its own axis system. This default behavior can be changed by moving one or more signals into a new group.

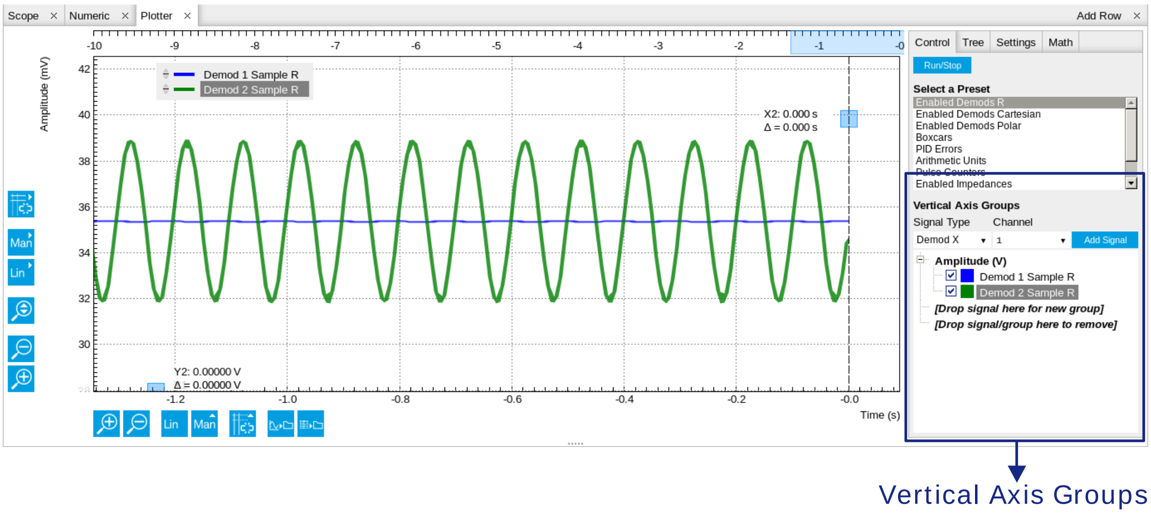


Figure 4.4. Vertical Axis Group in Plotter tool

The tick labels of only one axis group can be shown at once. This is the foreground axis group. To define the foreground group click on one of the group names in the Vertical Axis Groups box. The current foreground group gets a high contrast color.

Select foreground group: Click on a signal name or group name inside the Vertical Axis Groups. If a group is empty the selection is not performed.

Split the default vertical axis group: Use drag-and-drop to move one signal on the field [Drop signal here to add a new group]. This signal will now have its own axis system.

Change vertical axis group of a signal: Use drag-and-drop to move a signal from one group into another group that has the same unit.

Group separation: In case a group hosts multiple signals and the unit of some of these signals changes, the group will be split in several groups according to the different new units.

Remove a signal from the group: In order to remove a signal from a group drag-and-drop the signal to a place outside of the Vertical Axis Groups box.

Remove a vertical axis group: A group is removed as soon as the last signal of a custom group is removed. Default groups will remain active until they are explicitly removed by drag-and-drop. If a new signal is added that match the group properties it will be added again to this default group. This ensures that settings of default groups are not lost, unless explicitly removed.

Rename a vertical axis group: New groups get a default name "Group of ...". This name can be changed by double-clicking on the group name.

Hide/show a signal: Uncheck/check the check box of the signal. This is faster than fetching a signal from a tree again.

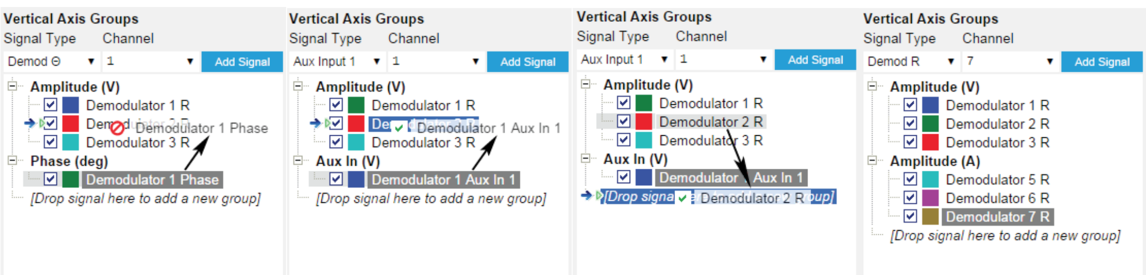


Figure 4.5. Vertical Axis Group typical drag and drop moves

Table 4.9. Vertical Axis Groups description

Control/Tool	Option/Range	Description
Vertical Axis Group		Manages signal groups sharing a common vertical axis. Show or hide signals by changing the check box state. Split a group by dropping signals to the field [Drop signal here to add new group]. Remove signals by dragging them on a free area. Rename group names by editing the group label. Axis tick labels of the selected group are shown in the plot. Cursor elements of the active wave (selected) are added in the cursor math tab.
Signal Type	Demod X, Y, R, Theta	Select signal types for the Vertical Axis Group.
	Frequency	
	Aux Input 1, 2	
	HW Trigger	
	PID Error	
	PID Shift	
	PID Value	
	Boxcar	
	AU Cartesian	
	AU Polar	
Channel	integer value	Selects a channel to be added.
Add Signal	Add Signal	Adds a signal to the plot. The signal will be added to its default group. It may be moved by drag and drop to its own group. All signals within a group share a common y-axis. Select a group to bring its axis to the foreground and display its labels.

4.1.4. Saving and Recording Data

In this section we discuss how to save and record measurement data with the UHFLI Instrument using the LabOne user interface. A quick way of doing this was already introduced in the previous section: in any plot (in the Plotter, Scope, Spectrum, and other tabs), you can save the currently displayed curves as a comma-separated value (CSV) file to the download folder of your web browser. Just click on the corresponding icon at the bottom of the plot. Clicking on will save a vector graphics instead.

The record functionality in comparison allows you to monitor and store measurement data continuously, as well as to track instrument settings over time. The Config tab gives you access to the main settings for this function. The Format selector defines which format is used: CSV or Matlab binary file format. This global setting also applies to the storage format used by the Sweeper and the Software Trigger tab. The CSV delimiter character can be changed in the User Preferences section. The default option is Semicolon.

The node tree display of the Record Data section allows you to browse through the different measurement data and instrument settings, and to select the ones you would like to record. For instance, the demodulator 1 measurement data is accessible under the path DeviceXXXX/ Demodulators/Demod 1/Sample. An example for an instrument setting would be the filter time constant, accessible under the path DeviceXXXX/Demodulators/Demod 1/Filter Time Constant.

The default storage location is the LabOne Data folder which can for instance be accessed via the Windows Start menu. The exact path is displayed in the Folder field whenever a file has been written.

Clicking on the Record checkbox will initiate the recording to the hard drive. In case of demodulator and boxcar data, ensure that the corresponding data stream is enabled, as otherwise no data will be saved.

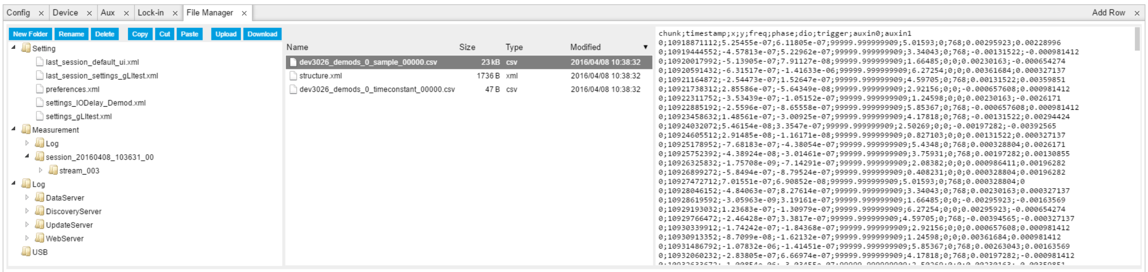


Figure 4.6. Browsing and inspecting files in the LabOne File Manager tab

For each of the selected nodes, at least one file is created. Its location is indicated in the Folder field of the Record Data section. For longer recording periods, LabOne may distribute the data over several files. The size of the files can be controlled using the Window Length parameter in the Settings of the Plotter tab.

The File Manager (Files) tab is a good place to inspect the resulting CSV data files. The file browser on the left of the tab allows you to navigate to the location of the data files and gives you the usual functionalities for managing files in the LabOne Data folder structure. In addition, you can conveniently transfer files between the folder structure and your preferred location using the Upload/Download buttons. The file viewer on the right side of the tab displays the contents of text files up to a certain size limit. Figure 4.6 shows the Files tab after recording Demodulator Sample and Filter Time Constant for a few seconds. The file viewer shows the contents of the demodulator data file.

Note

The structure of files containing instrument settings and of those containing streamed data is the same. Streaming data files contain one line per sampling period, whereas in the case of instrument settings, the file usually only contains a few lines, one for each change in the settings. More information on the file structure can be found in the LabOne Programming Manual.

4.2. Lock-in Tab

This tab is the main lock-in amplifier control panel. Users with instruments with UHF-MF Multi-frequency option installed are kindly referred to Section 4.3.

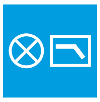
4.2.1. Features

- Functional block diagram with access to main input, output and demodulator controls
- Parameter table with main input, output and demodulator controls
- Control elements for 8 configurable demodulators
- Auto ranging, scaling, arbitrary input units for both input channels
- Control for 2 oscillators
- Settings for main signal inputs and signal outputs
- Flexible choice of reference source, trigger options and data transfer rates

4.2.2. Description

The Lock-in tab is the main control center of the instrument and open after start up by default. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.10. App icon and short description

Control/Tool	Option/Range	Description
Lock-in		Quick overview and access to all the settings and properties for signal generation and demodulation.

The default view of the Lock-in tab is the parameter table view. It is accessible under the side-tab labeled All and provides controls for all demodulators in the instrument. Moreover, for each individual demodulator there is a functional block diagram available. It is accessible under the side-tab labeled with the corresponding demodulator number.

Parameter Table

The parameter table (see Figure 4.7) consists of 4 vertical sections: Signal Inputs, Oscillators, Demodulators and Signal Outputs. The Demodulator section is horizontally divided into two identical groups. The upper group is tied to oscillator 1 and the lower group is tied to oscillator 2. That means demodulators 1 to 4 (5 to 8) can demodulate input signals at the frequency of oscillator 1 (2) and higher multiples. Demodulators 4 and 8 can be used for external referencing. Every demodulator can be connected to any of the possible inputs and outputs. Signal Input 1 and 2 are identical in all aspects, the same holds for the Signal Outputs 1 and 2.

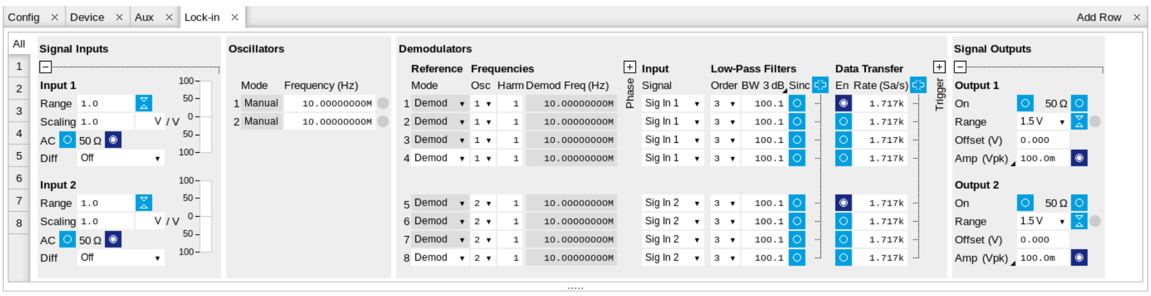


Figure 4.7. LabOne User Interface Lock-in tab - Parameter table (All)

The **Signal Inputs section** allows the user to define all relevant settings specific to the signal entered as for example input coupling, range, etc. Some of the available options like phase adjustment and the trigger functionality are collapsed by default. It takes one mouse click on the "+" icon in order to expand those controls. On the right-hand side of the Lock-in tab the Signal Outputs section allows defining signal amplitudes, offsets and range values.

The Scaling field below the Range field can be used to multiply the Signal Input data for instance to account for the gain of an external amplifier. In case there is a transimpedance gain of 10 V/A applied to the input signal externally, then the Scaling field can be set to 0.1 and the Units field can be set to A in order to show the actual current readings through the entire user interface.

Below the Scaling field there is the AC/DC button and the 50 Ω/1 MΩ. The AC/DC button sets the coupling type: AC coupling has a high-pass cutoff frequency that can be used to block large DC signal components to prevent input signal saturation during amplification. The 50 Ω/1 MΩ button toggles the input impedance between low (50 Ω) and high (approx. 1 MΩ) input impedance. 50 Ω input impedance should be selected for signal frequencies above 10 MHz to avoid artifacts generated by multiple signal reflections within the cable. With 50 Ω input impedance, one will expect a reduction of a factor of 2 in the measured signal if the signal source also has an output impedance of 50 Ω.

The **Oscillator section** indicates the frequencies of both internal oscillators. Where the Mode indicator shows Manual, the user can define the oscillator frequency manually defined by typing a frequency value in the field. In case the oscillator is referenced to an external source, the Mode indicator will show ExtRef and the frequency field is set to read-only. External reference requires a PLL to do the frequency mapping onto an internal oscillator. Successful locking is indicated by a green light right next to the frequency field. When the Modulation unit or the PID controller determine the frequency value of an oscillator, MOD or PID are indicated in the Mode field and the user cannot change the frequency manually.

In the following, we discuss the **Demodulators settings** in more detail. The block diagram displayed in Figure 4.8 indicates the main demodulator components and their interconnection. The understanding of the wiring is essential for successfully operating the instrument.

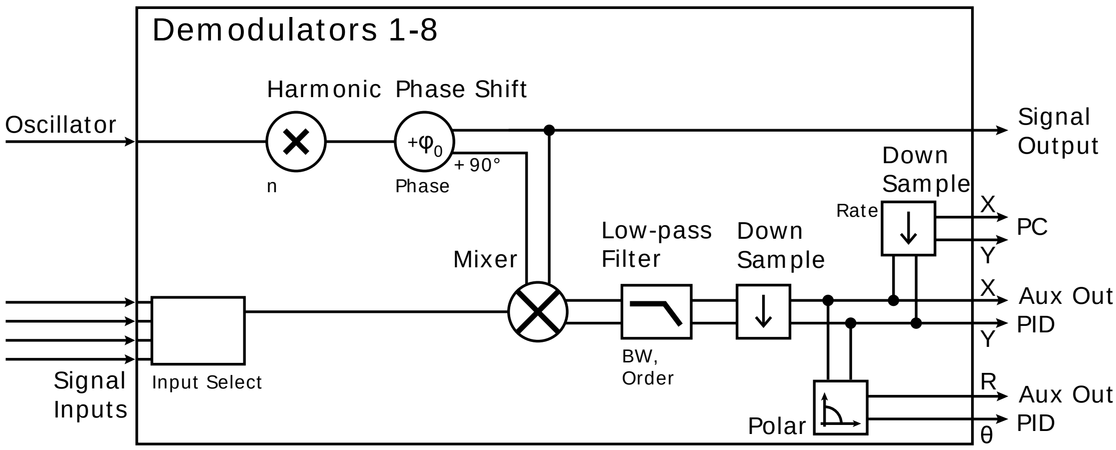


Figure 4.8. Demodulator block diagram without UHF-MF Multi-frequency option.

Every line in the Demodulators section represents one demodulator. The Mode column is read-only for all demodulators except 4 and 8, which can be to set to either internal reference (Demod) or external reference mode (ExtRef). When internal reference mode is selected, it is possible to demodulate the input signal with 4 demodulators simultaneously, using different filter settings or at different harmonic frequencies of the reference frequency. For external reference mode, one demodulator is used for the reference recovery and a few settings are greyed-out, and therefore 3 demodulators remain for simultaneous measurements.

In the Input Signal column one defines the signal that is taken as input for a given demodulator. A wide choice of signals can be selected: Signal Inputs, the Trigger Inputs, the Auxiliary Inputs and Auxiliary Outputs. This allows using the instrument for many different measurement topologies.

For each demodulator an additional phase shift can be introduced to the associated oscillator by entering the phase offset in the Phase column. This phase is added both to the reference channel and to the output of the demodulator. Hence, when the frequency is generated and detected using the same demodulator, signal phase and reference phase change by the same amount and no change will be visible in the demodulation result. Demodulation of frequencies that are integer multiples of any of the oscillator frequencies is achieved by entering the desired factor in the Harm column. The result of the demodulation, i.e. the amplitude and phase can be read e.g. using the Numeric tab which is described in Section 4.4.

In the middle of the Lock-in tab is the Low-Pass Filters section where the filter order can be selected in the drop-down list for each demodulator and the filter bandwidth (BW 3dB) can be chosen by typing a numerical value. Alternatively, the time constant of the filter (TC) or the noise equivalent power filter bandwidth (BW NEP) can be chosen by clicking on the column's header. For example, setting the filter order to 4 corresponds to a roll off of 24 dB/oct or 80 dB/dec i.e. an attenuation of 10^4 for a tenfold frequency increase. If the Low-Pass Filter bandwidth is comparable to or larger than the demodulation frequency, the demodulator output may contain frequency components at the frequency of demodulation and its higher harmonics. In this case, the additional Sinc Filter should be enabled. It attenuates those unwanted harmonic components in the demodulator output. The Sinc Filter is useful when measuring at low frequencies, since it allows one to apply a Low-Pass Filter bandwidth closer to the demodulation frequency, thus speeding up the measurement time.

The data transfer of demodulator outputs is activated by the En button in the Data Transfer section where also the sampling rate (Rate) for each demodulator can be defined.

The Trigger section next to the Data Transfer allows for setting trigger conditions in order to control and initiate data transfer from the Instrument to the host PC by the application of logic signals (e.g. TTL) to either Trigger Input 3 or 4 on the instrument back panel.

In the **Signal Outputs section** the On buttons are used to activate the Signal Outputs. This is also the place where the output amplitudes for each of the Signal Outputs can be set in adjustable units (Vpk, Vrms, or dBm). The Range drop-down list is used to select the proper output range setting. On each Signal Output a digital offset voltage (Offset) can be defined. The maximum output signal permitted is ±1.5 V.

Block Diagram

The block diagram view of the main instrument functions is also sometimes called the "Graphical Lock-in Tab". A set of indexed side-tabs in the Lock-in Tab give access to a block diagram for each demodulator. The block diagrams are fully functional and provide the user with a visual feedback of what is going on inside the instrument. All control elements that are available in the Parameter Table detailed in the previous section are also present in the graphical representation.

The block diagram in Figure 4.9 shows the signal path through the instrument for the case when the internal oscillator is used as reference. The Signal Inputs and Reference/Internal Frequency are shown on the left-hand side. The actual demodulation, i.e. the mixing and low-pass filtering is represented in the center of the tab. On the bottom right the user can set Signal Output parameters. On the top right there are the settings related to the output of the measurement data, either by digital means (PC Data Transfer) or by analog means (Auxiliary Outputs 1 to 4).

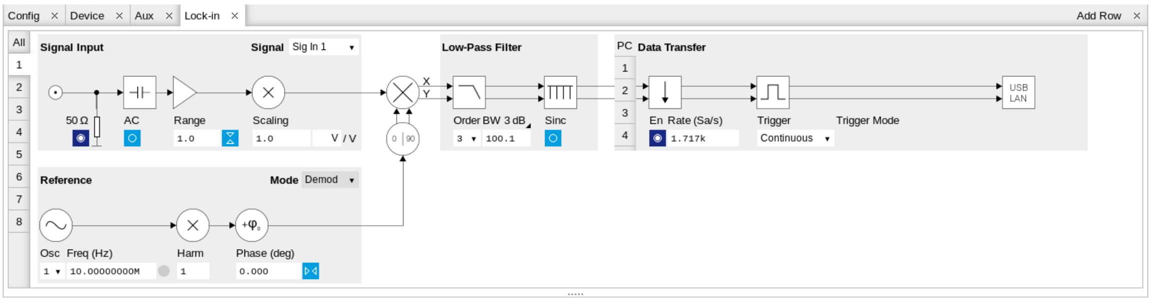


Figure 4.9. LabOne User Interface Lock-in tab - Graphical Lock-in tab in Internal Reference mode

The block diagram in Figure 4.10 shows the signal path through the instrument for the case when an external reference is used. This setting is only available for demodulators 4 and 8. In order to map an external frequency to oscillator 1/2 go to the Reference section of demodulator 4/8 and change the mode to ExtRef. This demodulator will then be used as a phase detector within the phase locked loop. The software will choose the appropriate filter settings according to the frequency and properties of the reference signal. Once demodulator 4/8 is used to map an external frequency on to one of the internal oscillators, it is no longer available for other measurements.

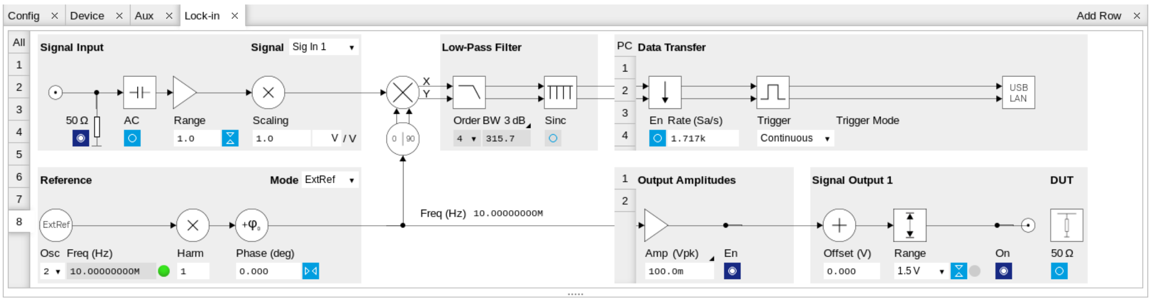


Figure 4.10. LabOne User Interface Lock-in tab - Graphical Lock-in tab in External Reference mode

4.2.3. Functional Elements

Table 4.11. Lock-in tab

Control/Tool	Option/Range	Description
Range	10 mV to 1.5 V	Defines the gain of the analog input amplifier. The range should exceed the incoming signal by roughly a factor two including a potential DC offset. Note 1: the value inserted by the user may be approximated to the nearest value supported by the Instrument. Note 2: a proper choice of range setting is crucial in order to achieve good accuracy and best possible signal to noise ratio as it targets to use the full dynamic range of the input ADC.

Control/Tool	Option/Range	Description
Auto		Automatic adjustment of the Range to about two times the maximum signal input amplitude measured over about 100 ms.
Scaling	numeric value	Applies an arbitrary scale factor to the input signal.
Measurement Unit	unit acronym	Defines the physical unit of the input signal. Use *, / and ^ operators, e.g., m or m/s^2. The value in this field modifies the readout of all measurement tools in the user interface. Typical uses of this field is to make measurements in the unit before the sensor/ transducer, e.g. to take an transimpedance amplifier into account and to directly read results in Ampere instead of Volts.
AC	ON: AC coupling	Defines the input coupling for the Signal Inputs. AC coupling inserts a high-pass filter.
	OFF: DC coupling	
50 Ω	ON: 50 Ω	Switches between 50 Ω (ON) and 1 MΩ (OFF).
	OFF: 1 MΩ	
Diff	Off	Switch input mode between normal (OFF), inverted, and differential. The differential modes are implemented digitally and are not suited for analog common-mode rejection.
	Inverted	
	Input 2 - Input 1	
	Input 1 - Input 2	
Mode		Indicates how the frequency of the corresponding oscillator is controlled (manual, external reference, PLL, PID). Read only flag.
	Manual	The user setting defines the oscillator frequency.
	ExtRef	An external reference is mapped onto the oscillator frequency.
	PLL	The UHF-PID option controls the oscillator frequency.
	PID	The UHF-PID option controls the oscillator frequency.
Frequency (Hz)	0 to 600 MHz	Frequency control for each oscillator.

Control/Tool	Option/Range	Description
Locked	ON / OFF	Oscillator locked to external reference when turned on.
Mode		Indicates the unit that uses the demodulator (Demod stands for regular lock-in amplifier, external reference, PLL)
	Demod	Default operating mode with demodulator used for lock-in demodulation.
	ExtRef	The demodulator is used for external reference mode and tracks the frequency of the selected reference input.
	PLL	The demodulator is used in PLL mode for frequency tracking of the signal. Note this function requires the UHF-PID option to be installed and active on your instrument.
	Mod	The demodulator is used by the UHF-MOD option, e.g. for the direct demodulation of carrier and sideband signals.
Osc	oscillator index	Connects the selected oscillator with the demodulator corresponding to this line. Number of available oscillators depends on the installed options.
Harm	1 to 1023	Multiplies the demodulator's reference frequency with the integer factor defined by this field. Multiplies the demodulator's reference frequency by an integer factor. If the demodulator is used as a phase detector in external reference mode (PLL), the effect is that the internal oscillator locks to the external frequency divided by the integer factor.
Demod Freq (Hz)	0 to 600 MHz	Indicates the frequency used for demodulation and for output generation. The demodulation frequency is calculated with oscillator frequency times the harmonic

Control/Tool	Option/Range	Description
		factor. When the UHF-MOD option is used linear combinations of oscillator frequencies including the harmonic factors define the demodulation frequencies.
Phase (deg)	-180° to 180°	Phase shift applied to the reference input of the demodulator.
Zero		Adjust the demodulator phase automatically in order to read zero degrees. Shifts the phase of the reference at the input of the demodulator in order to achieve zero phase at the demodulator output. This action maximizes the X output, zeros the Y output, zeros the Θ output, and leaves the R output unchanged.
Signal		Selects the signal source to be associated to the demodulator.
	Sig In 1	Signal Input 1 is connected to the corresponding demodulator.
	Sig In 2	Signal Input 2 is connected to the corresponding demodulator.
	Trigger 1	Trigger 1 is connected to the corresponding demodulator.
	Trigger 2	Trigger 2 is connected to the corresponding demodulator.
	Aux Out 1	Auxiliary Output 1 is connected to the corresponding demodulator.
	Aux Out 2	Auxiliary Output 2 is connected to the corresponding demodulator.
	Aux Out 3	Auxiliary Output 3 is connected to the corresponding demodulator.
	Aux Out 4	Auxiliary Output 4 is connected to the corresponding demodulator.
	Aux In 1	Auxiliary Input 1 is connected to the corresponding demodulator.

Control/Tool	Option/Range	Description
	Aux In 2	Auxiliary Input 2 is connected to the corresponding demodulator.
	Oscillator Phase Demod 4	Oscillator Phase of Demod 4 is connected to the corresponding demodulator.
	Oscillator Phase Demod 8	Oscillator Phase of Demod 8 is connected to the corresponding demodulator.
Order		Selects the filter roll off between 6 dB/oct and 48 dB/oct.
	1	1st order filter 6 dB/oct
	2	2nd order filter 12 dB/oct
	3	3rd order filter 18 dB/oct
	4	4th order filter 24 dB/oct
	5	5th order filter 30 dB/oct
	6	6th order filter 36 dB/oct
	7	7th order filter 42 dB/oct
	8	8th order filter 48 dB/oct
TC/BW Select		Defines the display unit of the low-pass filters: time constant (TC), noise equivalent power bandwidth (BW NEP), 3 dB bandwidth (BW 3 dB).
	TC	Defines the low-pass filter characteristic using time constant of the filter.
	BW NEP	Defines the low-pass filter characteristic using the noise equivalent power bandwidth of the filter.
	BW 3 dB	Defines the low-pass filter characteristic using the cut-off frequency of the filter.
TC/BW Value	numeric value	Defines the low-pass filter characteristic in the unit defined above.
Sinc	ON / OFF	Enables the sinc filter. When the filter bandwidth is comparable to or larger than the demodulation frequency, the demodulator output may contain frequency components at the frequency of demodulation and its higher harmonics. The sinc is an additional filter

Control/Tool	Option/Range	Description
		that attenuates these unwanted components in the demodulator output.
Lock		Makes all demodulators filter settings equal (order, time constant, bandwidth). Pressing the lock copies the settings from demodulator one into the settings of all demodulators. When the lock is pressed, any modification to a field is immediately changing all other settings. Releasing the lock does not change any setting, and permits to individually adjust the filter settings for each demodulator.
Enable Streaming		Enables the data acquisition for the corresponding demodulator. Note: increasing number of active demodulators increases load on physical connection to the host computer.
	ON: demodulator active	Enables the streaming of demodulated samples in real time to the host computer. The streaming rate is defined in the field on the right hand side. As a consequence demodulated samples can be visualized on the plotter and a corresponding numeric entry in the numerical tool is activated.
	OFF: demodulator inactive	Disables the streaming of demodulated samples to the host computer.
Rate (Sa/s)	0.42 Sa/s to 14 MSa/s	Defines the demodulator sampling rate, the number of samples that are sent to the host computer per second. A rate of about 7-10 higher as compared to the filter bandwidth usually provides sufficient aliasing suppression. This is also the rate of data received by LabOne Data Server and saved to the computer hard disk. This setting has no impact on the

Control/Tool	Option/Range	Description
		sample rate on the auxiliary outputs connectors. Note: the value inserted by the user may be approximated to the nearest value supported by the instrument.
Demodulator Output Rate Lock		Makes all demodulator output rates equal. Pressing the lock copies the settings from demodulator one into the settings of all demodulators. When the lock is pressed, any modification to a field is immediately changing all other settings. Releasing the lock does not change any setting, and permits to individually adjust the demodulator output rate for each demodulator.
Trigger		Selects the acquisition mode of demodulated samples. Continuous trigger means data are streamed to the host computer at the Rate indicated.
	Continuous	Selects continuous data acquisition mode. The demodulated samples are streamed to the host computer at the Rate indicated on the left hand side. In continuous mode the numerical and plotter tools are continuously receiving and display new values.
	Trigger 3	Selects external triggering by means of the Trigger 3 connector. Demodulated samples are sent to the host computer for each event defined in the Trig Mode field. When edge trigger is selected the rate field is greyed out and has no meaning. Note: some UHF Instruments feature Trigger 1/2 on the back panel instead of Trigger 3/4.
	Trigger 4	Selects external triggering by means of the Trigger 4 connector. Demodulated samples are sent to the host

Control/Tool	Option/Range	Description
		computer for each event defined in the Trig Mode field. When edge trigger is selected the rate field is greyed out and has no meaning. Note: some UHF Instruments feature Trigger 1/2 on the back panel instead of Trigger 3/4.
	Trigger 3|4	Same functionality as above, but triggering is based on a logical OR function of Trigger 3 and Trigger 4. Note: some UHF Instruments feature Trigger 1/2 on the back panel instead of Trigger 3/4.
Trig Mode		Defines the edge or level trigger mode for the selected Trigger input. Note: this field only appears when a non-continuous trigger is selected in the Trigger field.
	Rising	Selects triggered sample acquisition mode on rising edge of the selected Trigger input.
	Falling	Selects triggered sample acquisition mode on falling edge of the selected Trigger input.
	Both	Selects triggered sample acquisition mode on both edges of the selected Trigger input.
	High	Selects continuous sample acquisition mode on high level of the selected Trigger input. In this selection, the sample rate field determines the frequency in which demodulated samples are sent to the host computer.
	Low	Selects continuous sample acquisition mode on low level of the selected Trigger input. In this selection, the sample rate field determines the frequency in which demodulated samples are sent to the host computer.
Amplitude Unit	Vpk, Vrms, dBm	Select the unit of the displayed amplitude value. The dBm

Control/Tool	Option/Range	Description
		value is only valid for a system with 50 Ω termination.
Amp Enable	ON / OFF	Enables individual output signal amplitude. When the UHF-MF option is used, it is possible to generate signals being the linear combination of the available demodulator frequencies.
On	ON / OFF	Main switch for the Signal Output corresponding to the blue LED indicator on the instrument front panel.
50Ω	ON / OFF	Select the load impedance between 50Ω and HiZ. The impedance of the output is always 50Ω. For a load impedance of 50Ω the displayed voltage is half the output voltage to reflect the voltage seen at the load.
Range		Defines the maximum output voltage that is generated by the corresponding Signal Output. This includes the potential multiple Signal Amplitudes and Offsets summed up. Select the smallest range possible to optimize signal quality. This setting ensures that no levels or peaks above the setting are generated, and therefore it limits the values that can be entered as output amplitudes. Therefore selected output amplitudes are clipped to the defined range and the clipping indicator turns on. If 50 Ω target source or differential output is enabled the possible maximal output range will be half.
	150 mV	Selects output range ±150 mV.
	1.5 V	Selects output range ±1.5 V.
Auto Range		Selects the most suited output range automatically.
Output Clipping	grey/red	Indicates that the specified output amplitude(s) exceeds the range setting. Signal

Control/Tool	Option/Range	Description
		clipping occurs and the output signal quality is degraded. Adjustment of the range or the output amplitudes is required.
Offset	-range to range	Defines the DC voltage that is added to the dynamic part of the output signal.
Output	-range to range	Defines the output amplitude for each demodulator frequency as rms or peak-to-peak value. A negative amplitude value is equivalent to a phase change of 180 degree. Demodulator 4 is the signal source for Signal Output 1, demodulator 8 is the source for Signal Output 2.

4.3. Lock-in Tab (UHF-MF option)

This tab is the main lock-in amplifier control panel for UHFLI Instruments with the UHF-MF Multi-frequency option installed. Users with instruments without this option installed are kindly referred to Section 4.2.

4.3.1. Features

- Functional block diagram with access to main input, output and demodulator controls
- Parameter table with main input, output and demodulator controls
- Controls for 8 individually configurable demodulators
- Auto ranging, scaling, arbitrary input units for both input channels
- Control for 8 oscillators
- Settings for main signal inputs and signal outputs
- Choice of reference source, trigger options and data transfer rates

4.3.2. Description

The Lock-in tab is the main control center of the instrument and open after start up by default. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.12. App icon and short description

Control/Tool	Option/Range	Description
Lock-in MF		Quick overview and access to all the settings and properties for signal generation and demodulation.

The default view of the Lock-in tab is the parameter table view. It is accessible under the side-tab labeled All and provides controls for all demodulators in the instrument. Moreover, for each individual demodulator there is a functional block diagram available. It is accessible under the side-tab labeled with the corresponding demodulator number.

Parameter Table

The parameter table (see Figure 4.11) consists of 5 vertical sections: Signal Inputs, Oscillators, Demodulators, Output Amplitudes and Signal Outputs. The Demodulator section contains 8 rows each of them providing access to the settings of one dual phase demodulator. Demodulators 4 and 8 can be used for external referencing. Every demodulator can be connected to any of the possible inputs, outputs and oscillators. Signal Input 1 and 2 are identical in all aspects, the same holds for the Signal Outputs 1 and 2.

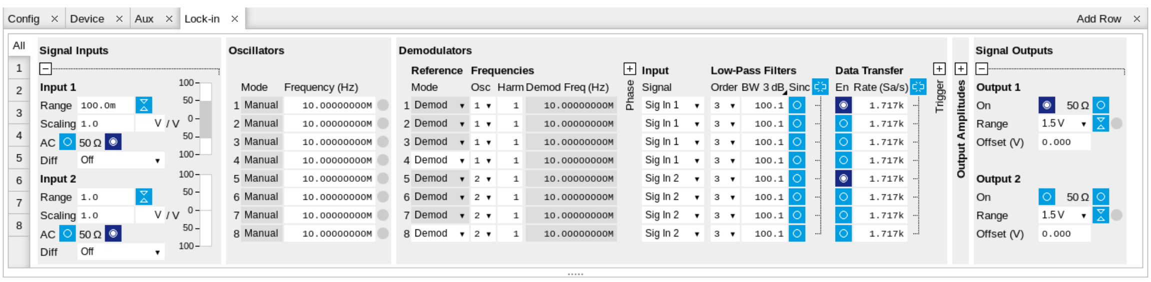


Figure 4.11. LabOne User Interface Lock-in tab with UHF-MF Multi-frequency option.

The **Signal Inputs section** allows the user to define all relevant settings specific to the signal entered as for example input coupling, range, etc. Some of the available options like phase adjustment and the trigger functionality are collapsed by default. It takes one mouse click on the "+" icon in order to expand those controls. On the right-hand side of the Lock-in tab the Signal Outputs section allows to define signal amplitudes, offsets and range values.

The Scaling field below the Range field can be used to multiply the Signal Input data for instance to account for the gain of an external amplifier. In case there is a transimpedance gain of 10 V/A applied to the input signal externally, then the Scaling field can be set to 0.1 and the Units field can be set to A in order to show the actual current readings through the entire user interface.

There are two buttons below the Scaling field that can be toggled: the AC/DC button and the 50 Ω/1 MΩ. The AC/DC button sets the coupling type: AC coupling has a high-pass cutoff frequency that can be used to block large DC signal components to prevent input signal saturation during amplification. The 50 Ω/1 MΩ button toggles the input impedance between low (50 Ω) and high (approx. 1 MΩ) input impedance. 50 Ω input impedance should be selected for signal frequencies above 10 MHz to avoid artifacts generated by multiple signal reflections within the cable. With 50 Ω input impedance, one will expect a reduction of a factor of 2 in the measured signal if the signal source also has an output impedance of 50 Ω.

The **Oscillator section** indicates the frequencies of all 8 internal oscillators. Where the Mode indicator shows Manual the user can define the oscillator frequency manually defined by typing a frequency value in the field. In case the oscillator is referenced to an external source the Mode indicator will show ExtRef and the frequency field is set to read-only. External reference requires a PLL to do the frequency mapping onto an internal oscillator. Successful locking is indicated by a green light right next to the frequency field. When the MOD option or the PID determine the frequency value of an oscillator, MOD and PID are indicated in the Mode field and the user cannot change the frequency manually.

The next section contains the **Demodulators settings**. The block diagram displayed in Figure 4.12 indicates the main demodulator components and their interconnection. The understanding of the wiring is essential for successfully operating the instrument.

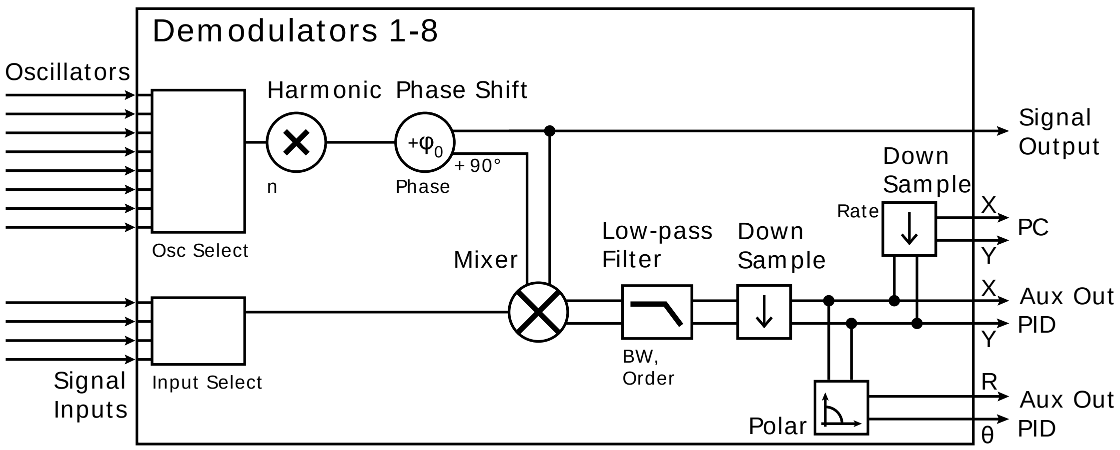


Figure 4.12. Demodulator block diagram with UHF-MF Multi-frequency option.

Every line in the Demodulators section represents one demodulator. The Mode column is read-only for all demodulators except 4 and 8, which can be to set to either internal reference (Demod) or external reference mode (ExtRef). When internal reference mode is selected, it is possible to demodulate the input signal with 8 demodulators simultaneously at 8 independent frequencies and using different filter settings. For external reference mode, one demodulator is used for the reference recovery and a few settings are greyed-out, and therefore 7 demodulators remain for simultaneous measurements.

In the Input Signal column one defines the signal that is taken as input for the demodulator. A wide choice of signals can be selected: Signal Inputs, the Trigger Inputs, the Auxiliary Inputs and Auxiliary Outputs. This allows to use the instrument for many different measurement topologies.

For each demodulator an additional phase shift can be introduced to the associated oscillator by entering the phase offset in the Phase column. This phase is added both, to the reference channel and the output of the demodulator. Hence, when the frequency is generated and detected using the same demodulator, signal phase and reference phase change by the same amount and no change will be visible in the demodulation result. Demodulation of frequencies that are integer multiples of any of the oscillator frequencies is achieved by entering the desired factor in the Harm column. The demodulator readout can be obtained using the Numeric tab which is described in Section 4.4.

In the middle of the Lock-in tab is the Low-Pass Filters section where the filter order can be selected in the drop down list for each demodulator and the filter bandwidth (BW 3dB) can chosen by typing a numerical value. Alternatively the time constant of the filter (TC) or the noise equivalent power filter bandwidth (BW NEP) can be chosen by clicking on the column's header. For example, setting the filter order to 4 corresponds to a roll off of 24 dB/oct or 80 dB/dec i.e. an attenuation of 10^4 for a tenfold frequency increase. If the Low-Pass Filter bandwidth is comparable to or larger than the demodulation frequency, the demodulator output may contain frequency components at the frequency of demodulation and its higher harmonics. In this case, the additional Sinc Filter can be enabled. It attenuates those unwanted harmonic components in the demodulator output. The Sinc Filter is also useful when measuring at low frequencies, since it allows to apply a Low-Pass Filter bandwidth closer to the demodulation frequency, thus speeding up the measurement time.

The data transfer of demodulator outputs is activated by the En button in the Data Transfer section where also the sampling rate (Rate) for each demodulator can be defined.

The Trigger section next to the Data Transfer allows for setting trigger conditions in order to control and initiate data transfer from the Instrument to the host PC by the application of logic signals (e.g. TTL) to either Trigger Input 3 or 4 on the back panel.

The **Output Amplitudes section** is only available for Instruments with the UHF-MF option installed and allows for the flexible adjustment of output amplitudes of different demodulators and their summation on either Signal Output 1 or Signal Output 2. In order to avoid signal clipping the sum of amplitudes of each signal output needs to be smaller than the range defined in the Signal Outputs section on the right. By clicking the headline of each column one can switch between amplitude definitions in terms of root mean square values, peak-to-peak values or even units of dBm, when the 50 Ω option in the Signal Output section is activated.

In the **Signal Outputs section** the On buttons allow to activate each of the Signal Outputs of the front panel. The Range drop down list is used to select the proper output range setting. On each Signal Output a digital offset voltage (Offset) can be defined. The maximum output signal permitted is ±1.5 V.

Block Diagram

The block diagram view of the main instrument functions is also sometimes referred to as the "Graphical Lock-in Tab". Depending how many demodulators are available in the instrument a set of numbered side-tabs occur giving access to a Graphical Lock-in Tab for each demodulator. The block diagrams are fully functional and provide the user with a visual feedback of what is going on inside the instrument. All control elements that are available in the Parameter Table detailed in the previous section are also present in the graphical representation.

The block diagram in Figure 4.13 describes the signal path throughout the instrument for the case when the internal oscillator is used as reference. In this case the tab consists of 6 functional sections. The Signal Inputs and Reference/Internal Frequency are described on the left side, the core of demodulation with the mixer and low-pass filter is located in the center of the tab and the Signal Outputs, the Auxiliary Outputs as well as the data transfer to the PC is sketched on the right.

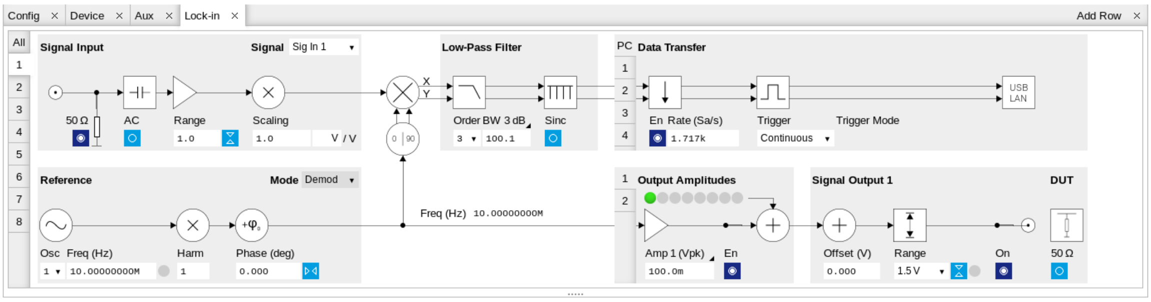


Figure 4.13. LabOne User Interface Lock-in tab - Graphical Lock-in tab in Internal Reference mode

The block diagram in Figure 4.14 describes the signal path throughout the instrument for the case when an external reference is used. This setting is only available for demodulators 4 and 8. In order to map an external frequency to any of the oscillators, go to the Reference section of demodulator 4 and 8 and change the mode to ExtRef. This demodulator will then be used as a phase detector within the phase-locked loop. The software will choose the appropriate filter settings according to the frequency and properties of the reference signal. Once a demodulator is used to map an external frequency on to one of the internal oscillators, it is no longer available for other measurements.

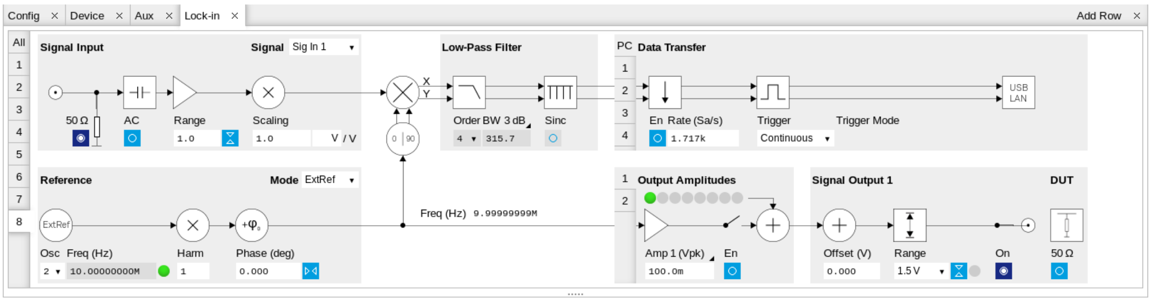


Figure 4.14. LabOne User Interface Lock-in tab - Graphical Lock-in tab in External Reference mode

4.3.3. Functional Elements

Table 4.13. Lock-in MF tab

Control/Tool	Option/Range	Description
Range	10 mV to 1.5 V	Defines the gain of the analog input amplifier. The range should exceed the incoming signal by roughly a factor two including a potential DC offset. Note 1: the value inserted by the user may be approximated to the nearest value supported by the Instrument. Note 2:

Control/Tool	Option/Range	Description
		a proper choice of range setting is crucial in order to achieve good accuracy and best possible signal to noise ratio as it targets to use the full dynamic range of the input ADC.
Auto		Automatic adjustment of the Range to about two times the maximum signal input amplitude measured over about 100 ms.
Scaling	numeric value	Applies an arbitrary scale factor to the input signal.
Measurement Unit	unit acronym	Defines the physical unit of the input signal. Use *, / and ^ operators, e.g., m or m/s^2. The value in this field modifies the readout of all measurement tools in the user interface. Typical uses of this field is to make measurements in the unit before the sensor/ transducer, e.g. to take an transimpedance amplifier into account and to directly read results in Ampere instead of Volts.
AC	ON: AC coupling	Defines the input coupling for the Signal Inputs. AC coupling inserts a high-pass filter.
	OFF: DC coupling	
50 Ω	ON: 50 Ω	Switches between 50 Ω (ON) and 1 MΩ (OFF).
	OFF: 1 MΩ	
Diff	Off	Switch input mode between normal (OFF), inverted, and differential. The differential modes are implemented digitally and are not suited for analog common-mode rejection.
	Inverted	
	Input 2 - Input 1	
	Input 1 - Input 2	
Mode		Indicates how the frequency of the corresponding oscillator is controlled (manual, external reference, PLL, PID). Read only flag.
	Manual	The user setting defines the oscillator frequency.
	ExtRef	An external reference is mapped onto the oscillator frequency.

Control/Tool	Option/Range	Description
	PLL	The UHF-PID option controls the oscillator frequency.
	PID	The UHF-PID option controls the oscillator frequency.
Frequency (Hz)	0 to 600 MHz	Frequency control for each oscillator.
Locked	ON / OFF	Oscillator locked to external reference when turned on.
Mode		Indicates the unit that uses the demodulator (Demod stands for regular lock-in amplifier, external reference, PLL)
	Demod	Default operating mode with demodulator used for lock-in demodulation.
	ExtRef	The demodulator is used for external reference mode and tracks the frequency of the selected reference input.
	PLL	The demodulator is used in PLL mode for frequency tracking of the signal. Note this function requires the UHF-PID option to be installed and active on your instrument.
	Mod	The demodulator is used by the UHF-MOD option, e.g. for the direct demodulation of carrier and sideband signals.
Osc	oscillator index	Connects the selected oscillator with the demodulator corresponding to this line. Number of available oscillators depends on the installed options.
Harm	1 to 1023	Multiplies the demodulator's reference frequency with the integer factor defined by this field. Multiplies the demodulator's reference frequency by an integer factor. If the demodulator is used as a phase detector in external reference mode (PLL), the effect is that the internal oscillator locks to the external frequency divided by the integer factor.

Control/Tool	Option/Range	Description
Demod Freq (Hz)	0 to 600 MHz	Indicates the frequency used for demodulation and for output generation. The demodulation frequency is calculated with oscillator frequency times the harmonic factor. When the UHF-MOD option is used linear combinations of oscillator frequencies including the harmonic factors define the demodulation frequencies.
Phase (deg)	-180° to 180°	Phase shift applied to the reference input of the demodulator.
Zero		Adjust the demodulator phase automatically in order to read zero degrees. Shifts the phase of the reference at the input of the demodulator in order to achieve zero phase at the demodulator output. This action maximizes the X output, zeros the Y output, zeros the Θ output, and leaves the R output unchanged.
Signal		Selects the signal source to be associated to the demodulator.
	Sig In 1	Signal Input 1 is connected to the corresponding demodulator.
	Sig In 2	Signal Input 2 is connected to the corresponding demodulator.
	Trigger 1	Trigger 1 is connected to the corresponding demodulator.
	Trigger 2	Trigger 2 is connected to the corresponding demodulator.
	Aux Out 1	Auxiliary Output 1 is connected to the corresponding demodulator.
	Aux Out 2	Auxiliary Output 2 is connected to the corresponding demodulator.
	Aux Out 3	Auxiliary Output 3 is connected to the corresponding demodulator.

Control/Tool	Option/Range	Description
	Aux Out 4	Auxiliary Output 4 is connected to the corresponding demodulator.
	Aux In 1	Auxiliary Input 1 is connected to the corresponding demodulator.
	Aux In 2	Auxiliary Input 2 is connected to the corresponding demodulator.
	Oscillator Phase Demod 4	Oscillator Phase of Demod 4 is connected to the corresponding demodulator.
	Oscillator Phase Demod 8	Oscillator Phase of Demod 8 is connected to the corresponding demodulator.
Order		Selects the filter roll off between 6 dB/oct and 48 dB/oct.
	1	1st order filter 6 dB/oct
	2	2nd order filter 12 dB/oct
	3	3rd order filter 18 dB/oct
	4	4th order filter 24 dB/oct
	5	5th order filter 30 dB/oct
	6	6th order filter 36 dB/oct
	7	7th order filter 42 dB/oct
	8	8th order filter 48 dB/oct
TC/BW Select		Defines the display unit of the low-pass filters: time constant (TC), noise equivalent power bandwidth (BW NEP), 3 dB bandwidth (BW 3 dB).
	TC	Defines the low-pass filter characteristic using time constant of the filter.
	BW NEP	Defines the low-pass filter characteristic using the noise equivalent power bandwidth of the filter.
	BW 3 dB	Defines the low-pass filter characteristic using the cut-off frequency of the filter.
TC/BW Value	numeric value	Defines the low-pass filter characteristic in the unit defined above.
Sinc	ON / OFF	Enables the sinc filter. When the filter bandwidth is comparable to or larger

Control/Tool	Option/Range	Description
		than the demodulation frequency, the demodulator output may contain frequency components at the frequency of demodulation and its higher harmonics. The sinc is an additional filter that attenuates these unwanted components in the demodulator output.
Lock		Makes all demodulators filter settings equal (order, time constant, bandwidth). Pressing the lock copies the settings from demodulator one into the settings of all demodulators. When the lock is pressed, any modification to a field is immediately changing all other settings. Releasing the lock does not change any setting, and permits to individually adjust the filter settings for each demodulator.
Enable Streaming		Enables the data acquisition for the corresponding demodulator. Note: increasing number of active demodulators increases load on physical connection to the host computer.
	ON: demodulator active	Enables the streaming of demodulated samples in real time to the host computer. The streaming rate is defined in the field on the right hand side. As a consequence demodulated samples can be visualized on the plotter and a corresponding numeric entry in the numerical tool is activated.
	OFF: demodulator inactive	Disables the streaming of demodulated samples to the host computer.
Rate (Sa/s)	0.42 Sa/s to 14 MSa/s	Defines the demodulator sampling rate, the number of samples that are sent to the host computer per second. A rate of about 7-10 higher as compared to the filter bandwidth usually

Control/Tool	Option/Range	Description
		provides sufficient aliasing suppression. This is also the rate of data received by LabOne Data Server and saved to the computer hard disk. This setting has no impact on the sample rate on the auxiliary outputs connectors. Note: the value inserted by the user may be approximated to the nearest value supported by the instrument.
Demodulator Output Rate Lock		Makes all demodulator output rates equal. Pressing the lock copies the settings from demodulator one into the settings of all demodulators. When the lock is pressed, any modification to a field is immediately changing all other settings. Releasing the lock does not change any setting, and permits to individually adjust the demodulator output rate for each demodulator.
Trigger		Selects the acquisition mode of demodulated samples. Continuous trigger means data are streamed to the host computer at the Rate indicated.
	Continuous	Selects continuous data acquisition mode. The demodulated samples are streamed to the host computer at the Rate indicated on the left hand side. In continuous mode the numerical and plotter tools are continuously receiving and display new values.
	Trigger 3	Selects external triggering by means of the Trigger 3 connector. Demodulated samples are sent to the host computer for each event defined in the Trig Mode field. When edge trigger is selected the rate field is greyed out and has no meaning. Note:

Control/Tool	Option/Range	Description
		some UHF Instruments feature Trigger 1/2 on the back panel instead of Trigger 3/4.
	Trigger 4	Selects external triggering by means of the Trigger 4 connector. Demodulated samples are sent to the host computer for each event defined in the Trig Mode field. When edge trigger is selected the rate field is greyed out and has no meaning. Note: some UHF Instruments feature Trigger 1/2 on the back panel instead of Trigger 3/4.
	Trigger 3|4	Same functionality as above, but triggering is based on a logical OR function of Trigger 3 and Trigger 4. Note: some UHF Instruments feature Trigger 1/2 on the back panel instead of Trigger 3/4.
Trig Mode		Defines the edge or level trigger mode for the selected Trigger input. Note: this field only appears when a non-continuous trigger is selected in the Trigger field.
	Rising	Selects triggered sample acquisition mode on rising edge of the selected Trigger input.
	Falling	Selects triggered sample acquisition mode on falling edge of the selected Trigger input.
	Both	Selects triggered sample acquisition mode on both edges of the selected Trigger input.
	High	Selects continuous sample acquisition mode on high level of the selected Trigger input. In this selection, the sample rate field determines the frequency in which demodulated samples are sent to the host computer.
	Low	Selects continuous sample acquisition mode on low level of the selected Trigger input. In this selection, the

Control/Tool	Option/Range	Description
		sample rate field determines the frequency in which demodulated samples are sent to the host computer.
Amplitude Unit	Vpk, Vrms, dBm	Select the unit of the displayed amplitude value. The dBm value is only valid for a system with 50 Ω termination.
Amp Enable	ON / OFF	Enables individual output signal amplitude. When the UHF-MF option is used, it is possible to generate signals being the linear combination of the available demodulator frequencies.
Amp (V)	-range to range	Defines the output amplitude for each demodulator frequency as rms or peak-to-peak value. A negative amplitude value is equivalent to a phase change of 180 degree. Linear combination of multiple amplitude settings on the same output are clipped to the range setting. Note: the value inserted by the user may be approximated to the nearest value supported by the Instrument.
AWG	AWG is ON	Indicates that the output amplitude is generated by the AWG.
On	ON / OFF	Main switch for the Signal Output corresponding to the blue LED indicator on the instrument front panel.
50Ω	ON / OFF	Select the load impedance between 50Ω and HiZ. The impedance of the output is always 50Ω. For a load impedance of 50Ω the displayed voltage is half the output voltage to reflect the voltage seen at the load.
Range		Defines the maximum output voltage that is generated by the corresponding Signal Output. This includes the potential multiple Signal Amplitudes and Offsets

Control/Tool	Option/Range	Description
		summed up. Select the smallest range possible to optimize signal quality. This setting ensures that no levels or peaks above the setting are generated, and therefore it limits the values that can be entered as output amplitudes. Therefore selected output amplitudes are clipped to the defined range and the clipping indicator turns on. If 50 Ω target source or differential output is enabled the possible maximal output range will be half.
	150 mV	Selects output range ±150 mV.
	1.5 V	Selects output range ±1.5 V.
Auto Range		Selects the most suited output range automatically.
Output Clipping	grey/red	Indicates that the specified output amplitude(s) exceeds the range setting. Signal clipping occurs and the output signal quality is degraded. Adjustment of the range or the output amplitudes is required.
Offset	-range to range	Defines the DC voltage that is added to the dynamic part of the output signal.

4.4. Numeric Tab

The Numeric Tab provides a powerful time domain based measurement display as introduced in Section 4.1.2. It is available in all UHFLI Instruments.

4.4.1. Features

- Display of demodulator output data and other streamed data, e.g. auxiliary inputs, PID errors, Boxcar data, demodulator frequencies, AU data, etc.
- Graphical and numerical range indicators
- Polar and Cartesian formats
- Support for Input Scaling and Input Units

4.4.2. Description

The numeric tab serves as the main numeric overview display of multiple measurement data. The display can be configured by both choosing the values displayed and also rearrange the display tiles by drag-and-drop. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.14. App icon and short description

Control/Tool	Option/Range	Description
Numeric		Access to all continuously streamed measurement data as numerical values.

The numeric tab (see Figure 4.15) is divided into a display section on the left and a configuration section on the right. The configuration section is further divided into a number of sub-tabs.

Figure 4.15. LabOne UI: Numeric tab

The numeric tab can be deployed to display the demodulated signal, phase, frequency as well as the signal levels at the auxiliary inputs. By default, the user can display the demodulated data either in polar coordinates (R, Θ) or in Cartesian coordinates (X, Y) which can be toggled using the presets. To display other measurement quantities as available from any of the presets simply click on the tree tab next to the preset tab. The desired display fields can be selected under each demodulator's directory tree structure.

4.4.3. Functional Elements

Table 4.15. Numeric tab: Presets sub-tab

Control/Tool	Option/Range	Description
Select a Preset		Select numerical view based on a preset. Alternatively, the displayed value may also selected based on tree elements.
	Demods Polar	Shows R and Phase of all demodulators.
	Enabled Demods Polar	Shows R and Phase of enabled demodulators.
	Demods Cartesian	Shows X and Y of all demodulators.
	Enabled Demods Cartesian	Shows X and Y of enabled demodulators.
	Demods R	Shows R of all demodulators.
	Boxcars	Shows amplitude of all boxcars.
	PID Errors	Shows error of all PID.
	Arithmetic Units	Shows output of all Cartesian and polar arithmetic units.
	Unpopulated	Shows no signals.
	Manual	If additional signals are added or removed the active preset gets manual.

For the Tree sub-tab please see Table 4.8 in the section called “Tree Sub-Tab”.

Table 4.16. Numeric tab: Settings sub-tab

Control/Tool	Option/Range	Description
Name	text label	Name of the selected plot(s). The default name can be changed to reflect the measured signal.
Mapping		Mapping of the selected plot(s)
	Lin	Enable linear scaling.
	Log	Enable logarithmic scaling.
	dB	Enable logarithmic scaling in dB.
Scaling	Manual/Full Scale	Scaling of the selected plot(s)
Zoom To Limits		Adjust the zoom to the current limits of the displayed histogram data.

Control/Tool	Option/Range	Description
Start Value	numeric value	Start value of the selected plot(s). Only visible for manual scaling.
Stop Value	numeric value	Stop value of the selected plot(s). Only visible for manual scaling.

4.5. Plotter Tab

The Plotter is one of the powerful time domain measurement tools as introduced in Section 4.1.2 and is available in all UHFLI Instruments.

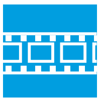
4.5.1. Features

- Plotting of all streamed data, e.g. demodulator data, auxiliary inputs, auxiliary outputs, Boxcar data, PID, etc.
- Plotting of Scope data, e.g. Signal Inputs (requires UHF-DIG option)
- Vertical axis grouping for flexible axis scaling
- Polar and Cartesian data format
- Histogram and Math functionality for data analysis
- 4 cursors for data analysis
- Support for Input Scaling and Input Units

4.5.2. Description

The Plotter serves as graphical display for time domain data in a roll mode, i.e. continuously without triggering. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.17. App icon and short description

Control/Tool	Option/Range	Description
Plotter		Displays various continuously streamed measurement data as traces over time (roll-mode).

The Plotter tab (see Figure 4.16) is divided into a display section on the left and a configuration section on the right.

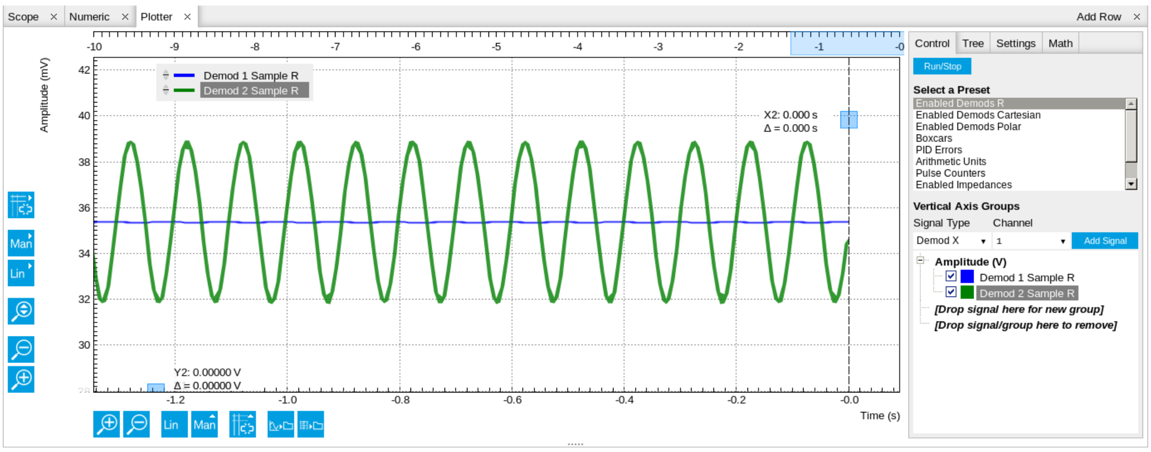


Figure 4.16. LabOne UI: Plotter tab

The Plotter can be used to monitor the evolution of demodulated data and other streamed data continuously over time. Just as in the numeric tab any continuously streamed quantity can be displayed, for instance R, Θ, X, Y, frequency, and others. New signals can be added by either using the presets in the Control sub-tab or by going through the tree and selecting the signals of interest in the tree structure. The vertical and horizontal axis can be displayed in Lin, Log or dB scale. The Plotter display can be zoomed in and out with the magnifier symbols, or through Man (Manual), Auto (Automatic) and FS (Full Scale) button settings (see also Section 4.1.3.

The maximum duration data is kept in the memory can be defined as window length parameter in the Settings sub-tab. The window length also determines the file size for the Record Data functionality.

Note

Setting the window length to large values when operating at high sampling rates can lead to memory problems at the computer hosting the data server.

The sampling rate of the demodulator data is determined by the Rate value in Sa/s set in the Lock-in tab; similarly the rates for PID and Boxcar related data are set in the associated tabs. The Plotter data can be continuously saved to disk by clicking the record button in the Config tab which will be indicated by a green Recording (REC) LED in the status bar.

4.5.3. Functional Elements

Table 4.18. Plotter tab: Control sub-tab

Control/Tool	Option/Range	Description
Run/Stop	Run/Stop	Start and stop continuous data plotting (roll mode)
Select a Preset		Select a pre-defined group signals. A signal group is defined by a common unit and signal type. They should have the same scaling behavior as they share a y-axis. Split a group if the signals have different scaling properties.
	Enabled Demods R	Selects the amplitude of all enabled demodulators.
	Enabled Demods Cartesian	Selects X and Y of all enabled demodulators.
	Enabled Demods Polar	Selects amplitude and phase of all enabled demodulators.
	Boxcars	Selects the amplitude of boxcar 1 and 2.
	PID Errors	Selects the error of all PID.
	Arithmetic Units	Selects the output of all Cartesian and polar arithmetic units.
	Unpopulated	Shows no signals.
	Manual	Selects the signals as defined in the tree sub-tab.

For the Vertical Axis Groups, please see Table 4.9 in the section called “Vertical Axis Groups”.

For the Tree sub-tab please see Table 4.8 in the section called “Tree Sub-Tab”.

Table 4.19. Plotter tab: Settings sub-tab

Control/Tool	Option/Range	Description
Window Length	10 s to 12 h	Plotter memory depth. Values larger than 10 s may cause excessive memory consumption for signals with high sampling rates. Auto scale or pan causes a refresh of the display for which only data within the defined window length are considered.
Histogram	ON / OFF	Shows the histogram in the display.
Rate	27.5 kHz to 28.1 MHz	Streaming Rate of the scope channels. The streaming rate can be adjusted independent from the scope sampling rate. The maximum rate depends on the interface used for transfer. Note: scope streaming requires the DIG option.
Enable	ON / OFF	Enable scope streaming for the specified channel. This allows for continuous recording of scope data on the plotter and streaming to disk. Note: scope streaming requires the DIG option.

For the Math sub-tab please see Table 4.7 in the section called “Cursors and Math”.

4.6. Scope Tab

The Scope is a powerful time domain and frequency domain measurement tool as introduced in Section 4.1.2 and is available for all UHFLI Instruments.

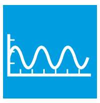
4.6.1. Features

- One input channel with 64 kSa of memory; upgradable to two channels with 128 MSa memory per channel (UHF-DIG option)
- 12 bit nominal resolution
- Fast Fourier Transform (FFT): up to 900 MHz span, spectral density and power conversion, choice of window functions
- Sampling rates from 27 kSa/s to 1.8 GSa/s; up to 36 μs acquisition time at 1.8 GSa/s or 2.3 s at 27 kSa/s
- 8 signal sources including Signal Inputs and Trigger Inputs; up to 8 trigger sources and 2 trigger methods
- Independent hold-off, hysteresis, pre-trigger and trigger level settings
- Support for Input Scaling and Input Units
- Simultaneous display of both input channels with up to 1.8 GSa/s (requires UHF-DIG option)
- Segmented recording (requires UHF-DIG option)
- Continuous recording of both input channels at up to 7 MSa/s over USB and 14 MSa/s over 1GbE

4.6.2. Description

The Scope tab serves as the graphical display for time domain data. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.20. App icon and short description

Control/Tool	Option/Range	Description
Scope		Displays shots of data samples in time and frequency domain (FFT) representation.

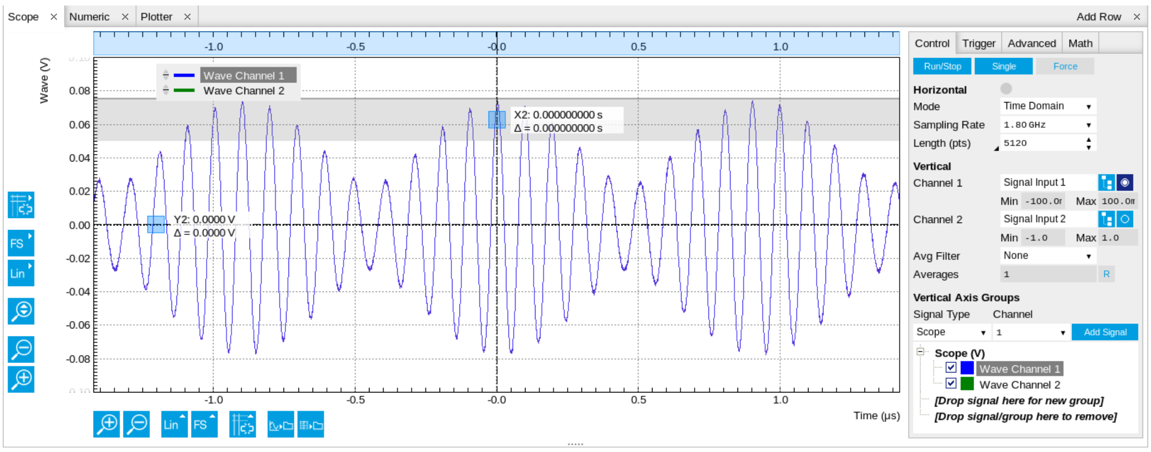


Figure 4.17. LabOne UI: Scope tab - Time domain

The Scope tab consists of a plot section on the left and a configuration section on the right. The configuration section is further divided into 4 sub-tabs. It gives access to a single-channel oscilloscope that can be used to monitor a choice of signals in the time or frequency domain. Hence the X axis of the plot area is time (for time domain display, Figure 4.17) Ar frequency (for frequency domain display, Figure 4.19). It is possible to display the time trace and the associated FFT simultaneously by opening a second instance of the Scope tab. The Y axis displays the selected signal that can be modified and scaled using the arbitrary input unit feature found in the Lock-in tab.

The Scope records data from a single channel at up to 1.8 GSa/s. The channel can be selected among the two Signal Inputs, Auxiliary Inputs, Trigger Inputs and Demodulator Oscillator Phase. The Scope records data sets of up to 64 kSa samples in the standard configuration, which corresponds to an acquisition time of 36 µs at the highest sampling rate. The performance of the Scope is comparable to that of entry level GHz sampling rate oscilloscopes. The Scope may be upgraded with the UHF-DIG Digitizer option, which enables two channels to be recorded in parallel, increases the available memory to 128 MSa/channel, and allows recording of data in a segmented fashion. The UHF-DIG Digitizer option also enables a continuous recording mode with a sampling rate of up to 28 MSa/s.

The product of the inverse sampling rate and the number of acquired points (Length) determines the total recording time for each shot. Hence, longer time intervals can be captured by reducing the sampling rate. The Scope can perform sampling rate reduction either using decimation or BW Limitation as illustrated in Figure 4.18. BW Limitation is activated by default, but it can be deactivated in the Advanced sub-tab. The figure shows an example of an input signal at the top, followed by the Scope output when the highest sample rate of 1.8 GSa/s is used. The next signal shows the Scope output when a rate reduction by a factor of 4 (i.e. 450 MSa/s) is configured and the rate reduction method of decimation is used. For decimation, a rate reduction by a factor of N is performed by only keeping every Nth sample and discarding the rest. The advantage to this method is its simplicity, but the disadvantage is that the signal is undersampled because the input filter bandwidth of the UHFLI instrument is fixed at 600 MHz. As a consequence, the Nyquist sampling criterion is no longer satisfied and aliasing effects may be observed. The default rate reduction mechanism of BW Limitation is illustrated by the lowermost signal in the figure. BW Limitation means that for a rate reduction by a factor of N, each sample produced by the Scope is computed as the average of N samples acquired at the maximum sampling rate. The effective signal bandwidth is thereby reduced and aliasing effects are largely suppressed. As can be seen from the figure, with a rate reduction by a factor of 4, every output sample is simply computed as the average of 4 consecutive samples acquired at 1.8 GSa/s.

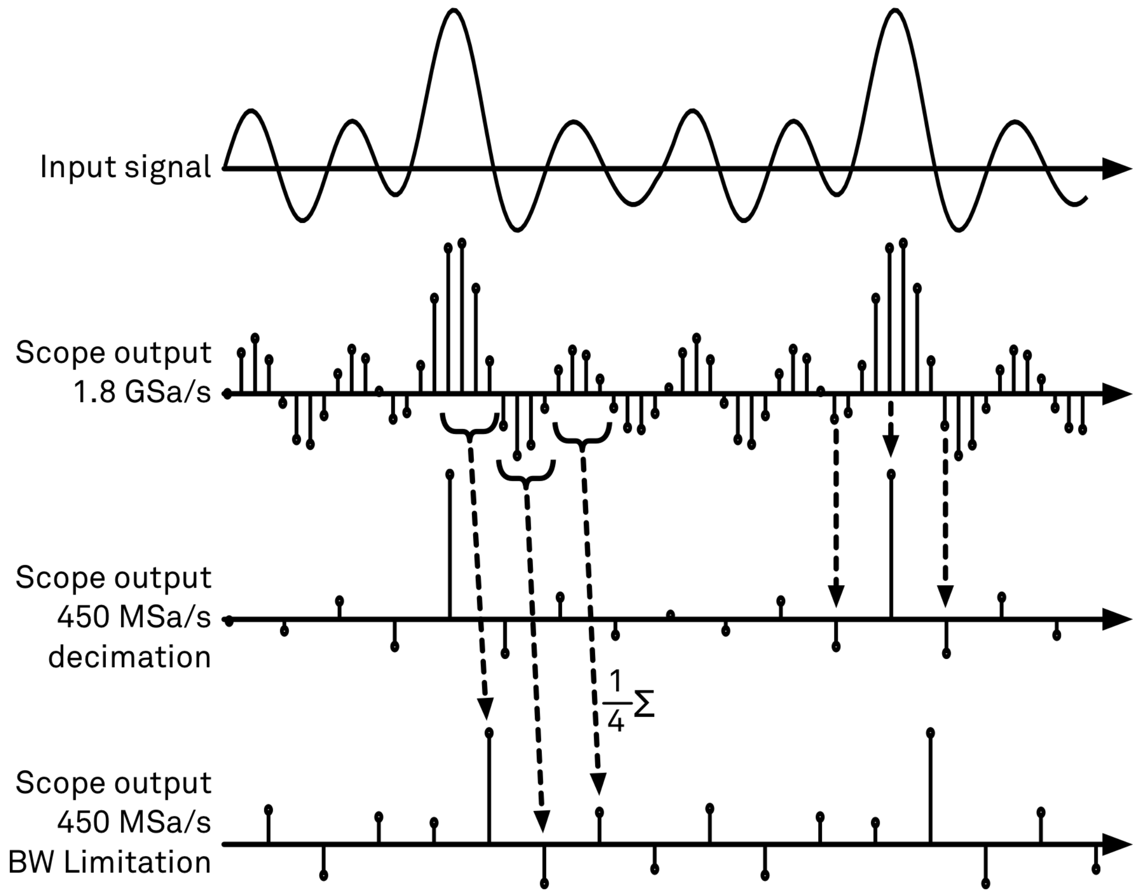


Figure 4.18. Illustration of how the Scope output is generated in BW Limitation and decimation mode when the sampling rate is reduced from the default of 1.8 GSa/s to 450 MSa/s .

The Scope also offers an averaging filter that works on a shot-to-shot basis. The functionality is implemented by means of an exponential moving average filter with configurable filter depth. The averaging filter can help suppress noise components that are uncorrelated with the main signal. It is particularly useful when the spectrum of the signal is considered as it can help to reveal harmonic signals and disturbances that might otherwise be hidden below the noise floor.

The frequency domain representation is activated in the Control sub-tab by selecting Freq Domain FFT as the Horizontal Mode. It allows the user to observe the spectrum of the acquired shots of samples. All controls and settings are shared between the time domain and frequency domain representations.

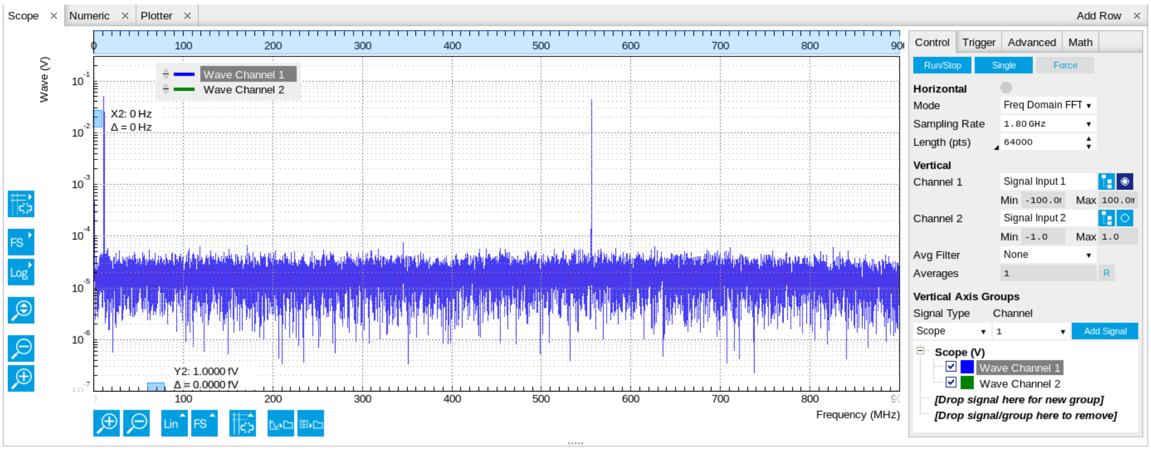


Figure 4.19. LabOne UI: Scope tab - Frequency domain

The Trigger sub-tab offers all the controls necessary for triggering on different signal sources. When the trigger is enabled, then oscilloscope shots are only acquired when the trigger conditions are met. Trigger and Hysteresis levels can be indicated graphically in the plot. A disabled trigger is equivalent to continuous oscilloscope shot acquisition.

Digitizer upgrade option

The UHF-DIG Digitizer option greatly enhances the performance of the Scope with the addition of the following features

- Simultaneous recording of two Scope channels
- Memory depth of 128 MSa for both Scope channels
- Additional input signal sources (Boxcar, Demodulator, Arithmetic Unit and PID data)
- Trigger gating
- Additional trigger input sources that allow for cross-domain triggering
- Additional trigger/marker output sources based on the state of the Scope
- Segmented data recording
- Continuous scope data streaming (Plotter tool)

This additional functionality can be enabled on any UHFLI instrument by uploading an option key. Please contact Zurich Instruments to get more information. The following sections explain the Digitizer features in more detail.

Two channels and extended memory depth

With the UHF-DIG option enabled it is possible to record two channels simultaneously. The two channels are sampled at the same time. This allows for very exact time difference measurements. Each channel can be assigned a different signal source. Enabled triggering will control when the recording of both channels start. The sampling rate and recording length settings are shared between both channels. A single shot length of up to 128 MSa can be recorded. Compared to the standard memory depth of 64 kSa this allows for longer recording times and FFTs with finer frequency resolution.

Additional input sources

Besides the Signal Input, Trigger Input, Auxiliary Input, and Oscillator Phase the UHF-DIG option also allows for recording of Demodulator, PID, Boxcar and Arithmetic Unit signals. This functionality is very powerful in that it allows short bursts to be recorded with very high sampling rates. In order to achieve the best possible utilization of the scope vertical resolution the upper and lower limit of these input signals should be specified. Before sampling, a scaling and an offset are applied to the input signal in order to get 12 bit resolution between the lower and upper limit. The applied scaling and offset values are transferred together with the scope data, which allows for recovery of the original physical signal strength in absolute values. For directly sampled input signals like the Signal Inputs or Trigger Inputs, the limits are read-only values and reflect the selected input range.

Trigger gating

With the UHF-DIG option installed the user can make full use of the Trigger Engine. If trigger gating is enabled, a trigger event will only be accepted if the gating input is active.

Additional trigger input sources

By using a Demodulator , PID, Boxcar, or Arithmetic Unit signal as trigger source, the Scope can be used in a cross-domain triggering mode. This allows, for example, for time domain signals to be recorded in a synchronous fashion triggered by the result from analyzing a signal in the frequency domain by means of a demodulator.

Note

Adjust a negative delay (pre-trigger) to compensate for the delay of the Demodulator, PID, Boxcar or Arithmetic Unit.

Segmented data recording

The scope sends the result of each shot to the PC over an interface with limited data transfer bandwidth. As a consequence, a hold-off time is required between individual scope shots to allow the recorded data to be transferred to the PC. The segmented data recording mode can be enabled if the user requires a small hold-off time between shots. The mode allows a burst of up to 32768 scope shots, called segments, to be recorded into the device memory. The hold-off time in this mode can be less than 100 μs between each shot, because the Scope does not have to wait for the data transfer to complete before the next shot can be started. The segmented data recording is most powerful when used over the API. The data of each shot will contain information on the segment number.

Continuous Scope data streaming

Normal scope operation records scope shots into the device memory. This allows for recording of up to 1.8 GSa/s until the memory is full. After each scope shot there will be a dead time, also known as hold-off time, to re-arm the trigger, address the next memory block and transfer the data to the PC. Due to this dead time scope shots cannot be recorded back to back. In order to record very long scope shots (digitizer mode) the Scope data can be streamed directly to the client bypassing the device memory. This allows for continuous recording of very long Scope traces that exceed the available memory depth of the instrument. The streamed Scope data will be shown in the Plotter tab together with all other streaming data. Due to the limited transfer bandwidth over the TCPIP or USB interface the maximal sampling rate is restricted. The sampling rate for the Scope streaming channels and the enabling of each channel is controlled in the Settings sub-tab of the Plotter. As the sampling rate of the Scope streaming can be adjusted independently from the Scope shot sampling rate it is possible to record continuous data together with triggered high sampling rate Scope shots. The Scope streaming in the Plotter can be very useful for monitoring of the inputs.

Scope state output on Trigger Output

The UHF-DIG option extends the list of available Trigger Outputs by the six elements: Scope Trigger, Scope Armed, Scope Active and their logically inverse signals. The Trigger Output signals are controlled on the DIO tab (Section 4.13). Figure 4.20 shows an illustration of the signal that will be generated on the Trigger Output when one of the six new Scope-related sources is selected. An example input signal is shown at the top of the figure. It is assumed that the Scope is configured to trigger on this input signal on a rising edge crossing the level indicated by the stippled line.

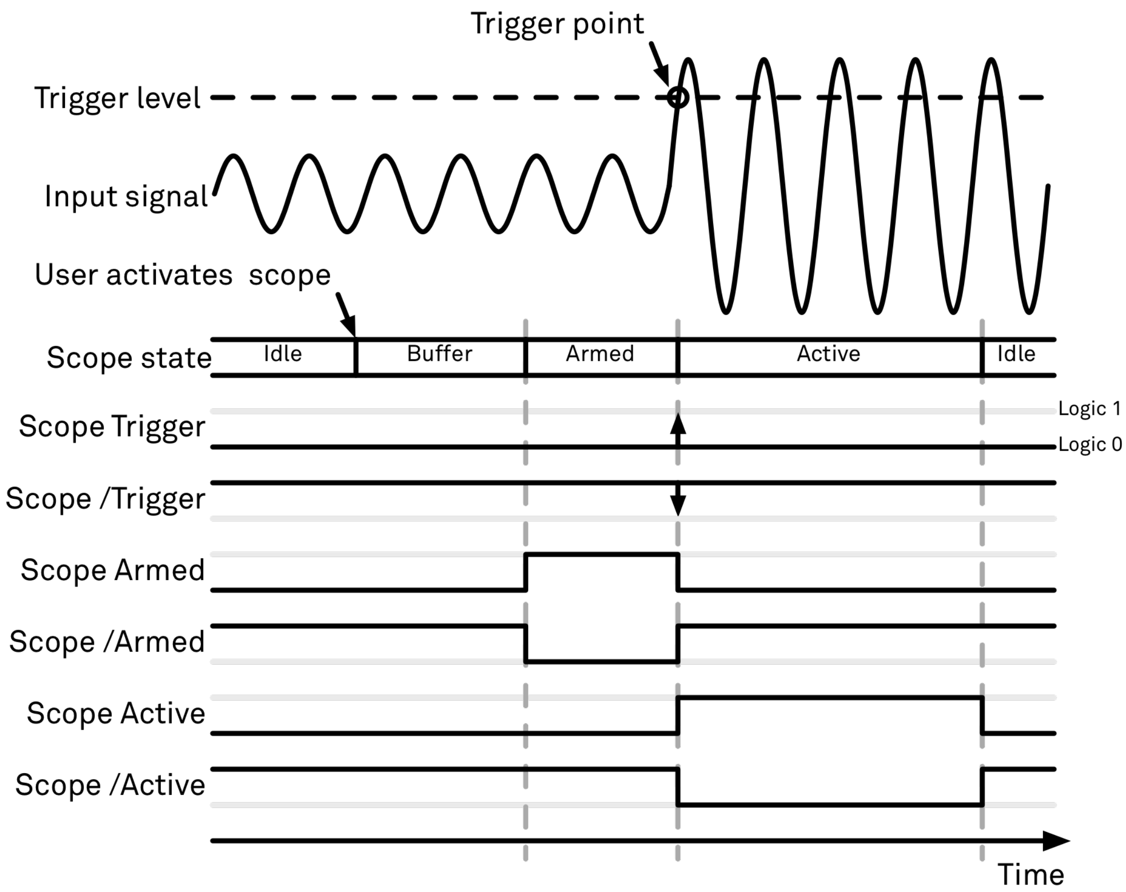


Figure 4.20. Illustration of the signal that will appear on the Trigger Output when one of the six Scope related sources is selected.

The Scope can be thought of as having a state, which changes over time. The state is shown below the input signal in the figure. When the Scope is completely inactive, it is said to be in the Idle state. When the user then activates the Scope, it will transition into a Buffer state. In this state the Scope will start to record the input signal. It will remain in this state until sufficient data has been recorded to fulfill the user requirement for recording data prior to the trigger point as controlled by the trigger Reference and Delay fields in the user interface. Once sufficient data has been recorded, the Scope will transition to the Armed state. In this state the Scope is ready to accept the trigger signal. Note that the Scope will continue to record data for as long as it is in the Armed state, and that if no trigger is defined, the Scope will simply pass straight through the Armed state. Once the input signal passes the Trigger level the Scope will trigger, and at the same time its state will change from Armed to Active. The Scope will remain in the Active state, where it also records data, until sufficient data has been recorded to fulfill the Length requirement configured in the user interface. Once enough data has been acquired, the Scope will transition back into the Idle state where it will wait for the time configured with the Hold-off time before it either starts the next measurement automatically (in case Run is active) or waits for the user to reactivate it.

The trigger source selector allows information about the Scope state to be reproduced on the Trigger Output in a number of ways. The signal that will appear on the output is shown with the six bottommost traces in the figure. Note that these traces are shown as digital signals with symbolic values of logic 0 and 1. These values will of course be actual voltages when measured on the device itself.

First, if Scope Trigger is selected then the trigger output will have a signal that is asserted, which means that it goes high, when the scope triggers, i.e. changes from the Armed to the Active state. The signal will normally have a very short duration and, therefore, it is shown with an arrow in the figure. The duration can be increased by means of the Width input field, which can be found next to the Output Signal selector on the DIO tab. If Scope/Trigger is selected, then the same signal will appear on the output, but it will simply be inverted logically.

Next, if the Scope Armed source is selected, the trigger output will be asserted as long as the Scope is in the Armed state. Again, this means that the Scope has recorded enough data to proceed with the acquisition and is waiting for the trigger condition to become satisfied. In this example, since a rising edge trigger is defined, the trigger condition becomes satisfied when the input signal goes from below the trigger level to above the trigger level.

Similarly, if Scope /Armed is selected, the trigger output will be asserted (i.e. at logic 1) whenever the Scope is in a state different from the Armed state. The same explanation holds for the remaining two configuration options, except here the trigger output is asserted when the Scope is in the Active state or when it is not in the Active state.

4.6.3. Functional Elements

Table 4.21. Scope tab: Control sub-tab

Control/Tool	Option/Range	Description
Run/Stop	Run/Stop	Runs the scope/FFT continuously.
Single	Single	Acquires a single shot of samples.
Force	Force	Force a trigger event.
Mode	Time Domain	Switches between time and frequency domain display.
	Freq Domain (FFT)	
Sampling Rate	27.5 kSa/s to 1.8 GSa/s	Defines the sampling rate of the scope.
Length Mode		Switches between length and duration display.
	Length (pts)	The scope shot length is defined in number of samples. The duration is given by the number of samples divided by the sampling rate. The UHF-DIG option greatly increases the available length.
	Duration (s)	The scope shot length is defined as a duration. The number of samples is given by the duration times the sampling rate.
Length (pts) or Duration (s)	numeric value	Defines the length of the recorded scope shot. Use the Length Mode to switch

Control/Tool	Option/Range	Description
		between length and duration display.
Channel 1/2	Signal Inputs, Trigger Inputs, Auxiliary Inputs, Demodulator Oscillator Phase, Demodulator X/Y/R/Theta, PID, Boxcar, AU	Selects the source for scope channel. Navigate through the tree view that appears and click on the required signal. Note: Channel 2 requires the DIG option.
Min	numeric value	Lower limit of the scope full scale range. For demodulator, PID, Boxcar, and AU signals the limit should be adjusted so that the signal covers the specified range to achieve optimal resolution.
Max	numeric value	Upper limit of the scope full scale range. For demodulator, PID, Boxcar, and AU signals the limit should be adjusted so that the signal covers the specified range to achieve optimal resolution.
Enable	ON / OFF	Activates the display of the corresponding scope channel. Note: Channel 2 requires the DIG option.
Avg Filter		Selects averaging filter type that is applied when the average of several scope shots is computed and displayed.
	None	Averaging is turned off.
	Exponential Moving Avg	Consecutive scope shots are averaged with an exponential weight.
Averages	integer value	The number of shots required to reach 63% settling. Twice the number of shots yields 86% settling.
Reset	R	Resets the averaging filter.

For the Vertical Axis Groups, please see Table 4.9 in the section called “Vertical Axis Groups”.

Table 4.22. Scope tab: Trigger sub-tab

Control/Tool	Option/Range	Description
Trigger	grey/green/yellow	When flashing, indicates that new scope shots are being captured and displayed in the plot area. The Trigger must not necessarily be enabled for this indicator to flash. A

Control/Tool	Option/Range	Description
		disabled trigger is equivalent to continuous acquisition. Scope shots with data loss are indicated by yellow. Such an invalid scope shot is not processed.
Enable		When triggering is enabled scope data are acquired every time the defined trigger condition is met.
	ON	Trigger based scope shot acquisition
	OFF	Continuous scope shot acquisition
Signal	Signal Inputs	Selects the trigger source signal. Navigate through the tree view that appears and click on the required signal.
	Trigger Inputs	
	Auxiliary Inputs	
	Demodulator Oscillator Phase	
	Demodulator X/Y/R/Theta	
	PID	
	Boxcar	
	AU	
Edge Rise	ON / OFF	Performs a trigger event when the source signal crosses the trigger level from low to high. For dual edge triggering, select also the falling edge.
Edge Fall	ON / OFF	Performs a trigger event when the source signal crosses the trigger level from high to low. For dual edge triggering, select also the rising edge.
Level (V)	trigger signal range (negative values permitted)	Defines the trigger level.
Hysteresis Mode		Selects the mode to define the hysteresis strength. The relative mode will work best over the full input range as long as the analog input signal does not suffer from excessive noise.
	Hysteresis (V)	Selects absolute hysteresis.
	Hysteresis (%)	Selects a hysteresis relative to the adjusted full scale signal input range.
Hysteresis (V)	trigger signal range (positive values only)	Defines the voltage the source signal must deviate from the trigger level before the trigger

Control/Tool	Option/Range	Description
		is rearmed again. Set to 0 to turn it off. The sign is defined by the Edge setting.
Hysteresis (%)	numeric percentage value (positive values only)	Hysteresis relative to the adjusted full scale signal input range. A hysteresis value larger than 100% is allowed.
Show Level	ON / OFF	If enabled shows the trigger level as grey line in the plot. The hysteresis is indicated by a grey box. The trigger level can be adjusted by drag and drop of the grey line.
Trigger Gating		Select the signal source used for trigger gating if gating is enabled. This feature requires the UHF-DIG option.
	Trigger In 3 High	Only trigger if the Trigger Input 3 is at high level.
	Trigger In 3 Low	Only trigger if the Trigger Input 3 is at low level.
	Trigger In 4 High	Only trigger if the Trigger Input 4 is at high level.
	Trigger In 4 Low	Only trigger if the Trigger Input 4 is at low level.
Trigger Gating Enable	ON / OFF	If enabled the trigger will be gated by the trigger gating input signal. This feature requires the UHF-DIG option.
Holdoff Mode		Selects the holdoff mode.
	Holdoff (s)	Holdoff is defined as time.
	Holdoff (events)	Holdoff is defined as number of events.
Holdoff (s)	numeric value	Defines the time before the trigger is rearmed after a recording event.
Holdoff (events)	1 to 1048575	Defines the trigger event number that will trigger the next recording after a recording event. A value one will start a recording for each trigger event.
Reference (%)	percent value	Trigger reference position relative to the plot window. Default is 50% which results in a reference point in the middle of the acquired data.
Delay (s)	numeric value	Trigger position relative to reference. A positive delay results in less data being

Control/Tool	Option/Range	Description
		acquired before the trigger point, a negative delay results in more data being acquired before the trigger point.
Enable	ON / OFF	Enable segmented scope recording. This allows for full bandwidth recording of scope shots with a minimum dead time between individual shots. This functionality requires the DIG option.
Segments	1 to 32768	Specifies the number of segments to be recorded in device memory. The maximum scope shot size is given by the available memory divided by the number of segments. This functionality requires the DIG option.
Shown Segment	integer value	Displays the number of recorded segments.
Shown Trigger	integer value	Displays the number of triggered events since last start.

Table 4.23. Scope tab: Advanced sub-tab

Control/Tool	Option/Range	Description
FFT Window	Rectangular	Four different FFT windows to choose from. Each window function results in a different trade-off between amplitude accuracy and spectral leakage. Please check the literature to find the window function that best suits your needs.
	Hann	
	Hamming	
	Blackman Harris	
Resolution (Hz)	mHz to Hz	Spectral resolution defined by the reciprocal acquisition time (sample rate, number of samples recorded).
Spectral Density	ON / OFF	Calculate and show the spectral density. If power is enabled the power spectral density value is calculated. The spectral density is used to analyze noise.
Power	ON / OFF	Calculate and show the power value. To extract power spectral density (PSD) this button should be enabled together with Spectral Density.

Control/Tool	Option/Range	Description
Persistence	ON / OFF	Keeps previous scope shots in the display. The color scheme visualizes the number of occurrences at certain positions in time and amplitude by a multi-color scheme.
BW Limit Ch 1		Selects between sample decimation and sample averaging. Averaging avoids aliasing, but may conceal signal peaks.
	ON	Selects sample averaging for sample rates lower than the maximal available sampling rate.
	OFF	Selects sample decimation for sample rates lower than the maximal available sampling rate.
BW Limit Ch 2		Selects between sample decimation and sample averaging. Averaging avoids aliasing, but may conceal signal peaks. Channel 2 requires the DIG option.
	ON	Selects sample averaging for sample rates lower than the maximal available sampling rate. Channel 2 requires the DIG option.
	OFF	Selects sample decimation for sample rates lower than the maximal available sampling rate. Channel 2 requires the DIG option.

For the Math sub-tab please see Table 4.7 in the section called “Cursors and Math”.

4.7. Software Trigger Tab

The software trigger is one of the powerful time domain measurement tools as introduced in Section 4.1.2 and is available for all UHFLI Instruments.

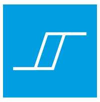
4.7.1. Features

- Time-domain data display for all continuously streamed data
- 6 different trigger types
- Automatic trigger level determination
- Display of multiple traces
- Adjustable record history
- Mathematical toolkit for signal analysis
- Grid mode for imaging support

4.7.2. Description

The Software Trigger tab provides display and recording of shot-wise data sets after configurable trigger events. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.24. App icon and short description

Control/Tool	Option/Range	Description
SW Trig		Provides complex trigger functionality on all continuously streamed data samples and time domain display.

The Software Trigger tab (see Figure 4.21) is divided into a display section on the left and a configuration section on the right. The configuration section is further divided into a number of sub-tabs.

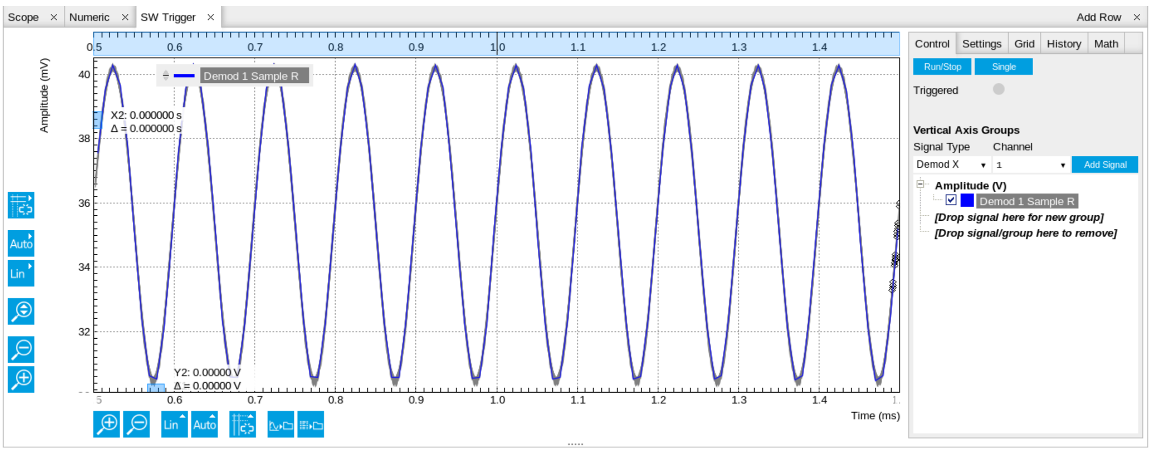


Figure 4.21. LabOne UI: Software trigger tab

The Software Trigger brings the trigger functionality of a scope to the continuously streamed data that can be viewed with the Plotter tool in a roll mode. The user can choose between a variety of different trigger options for the different signal inputs. Each trigger event is indicated by a green LED.

The trigger condition is configured in the **Settings** sub-tab. Among the selection of Trigger Types provided here, Edge and Pulse are most suited for analog trigger sources such as demodulator data, auxiliary voltages, or oscillator frequencies. Instead of manually setting a Trigger Level, you can click on Find to have LabOne find a value by analyzing the data stream. In case of noisy trigger sources, both the Bandwidth and the Hysteresis setting can help preventing false trigger events. The Bandwidth setting provides a configurable low-pass filter applied to the trigger source. When enabling this feature, be sure to choose a high enough bandwidth to let pass the feature that should be triggered upon, i.e., the signal edge or pulse. Note that the Bandwidth setting does not affect the recorded data itself.

For trigger sources with a slowly varying offset, the Tracking Edge and Tracking Pulse Trigger Types provide continuous adjustment of the Level and Hysteresis. In Tracking mode, the Bandwidth setting plays a different role than for the Edge and Pulse trigger types. Here, the Bandwidth should be chosen sufficiently low to filter out all fast features and only let pass the slow offset.

The Trigger Types HW Trigger and Digital are intended for TTL signals on the Trigger Inputs or on the DIO lines. Using the Bits and Bit Mask setting, complex multi-bit trigger conditions on the DIO lines can be defined.

The Horizontal section of the Settings sub-tab contains the settings for shot Duration and Delay (negative delays correspond to pre-trigger time). Also minimum and maximum pulse width for the Pulse and Tracking Pulse trigger types are defined here.

The **Grid** sub-tab provides the functionality for capturing data for imaging applications. With enabled grid mode, the data recorded by the Software Trigger get organized in two-dimensional frames consisting of rows and columns. Every newly captured data shot is assigned to a row, and the total number of rows is defined by the Rows settings. After completion of a full frame, the new data either replace the old or averaging is performed, according to the selected Operation and Repetitions setting. On the horizontal axis, the Duration of a shot is divided into samples according to the Columns setting. If necessary, the streamed data are interpolated, resampled, and aligned with the trigger event. The data of a completed frame after averaging appears as a single entry in the History list. These data are available for saving, but can't be displayed in the plot section. The Software Trigger needs to run continuously for a sufficiently long time to complete the acquisition, and during that time the history entry contains only partial data

By default the plot area keeps the memory and display of the last 100 trigger shots represented in a list in the **History** sub-tab. The button to the left of each list entry controls the visibility of the corresponding trace in the plot; the button to the right controls the color of the trace. Renaming a trace is easily possible by double clicking on a list entry. All measurements in the history can be stored in a file by clicking on Save. Which quantities are saved depends on which demodulator channels have been added to the Vertical Axis Groups section in the Control sub-tab. Naturally, only data from demodulators with enabled Data Transfer in the Lock-in tab can be included in the files. The file format, Matlab or comma-separated value (CSV) depends on the Format setting in the Config tab. The best way to access the saved files is via the File Manager tab.

4.7.3. Functional Elements

Table 4.25. SW Trigger tab: Control sub-tab

Control/Tool	Option/Range	Description
Run/Stop	Run/Stop	Start and stop the software trigger

Control/Tool	Option/Range	Description
Single	Single	Run the SW trigger once (record Count trigger events)
Triggered	grey/green	When green, indicates that new trigger shots are being captured and displayed in the plot area.

For the Vertical Axis Groups, please see Table 4.9 in the section called “Vertical Axis Groups”.

Table 4.26. SW Trigger tab: Settings sub-tab

Control/Tool	Option/Range	Description
Trigger Signal	Demodulator X/Y/R/Theta/ Frequency, Auxiliary Inputs, Trigger Inputs, Trigger Outputs, Demodulator 0 Degree Oscillator Phase Crossing	Source signal for trigger condition.
Trigger Type		Select the type of trigger to use. Selectable options depend on the selected trigger signal.
	Edge	Analog edge triggering based on high and low level. Hysteresis on the levels and low-pass filtering can be used to reduce the risk of wrong trigger for noisy trigger signals.
	Digital	Digital triggering on the 32 bit DIO lines. The bit value defines the trigger conditions. The bit mask controls the bits that are used for trigger evaluation. For triggering just on DIO0 use a bit value 0x0001 and a bit mask 0x0001.
	Pulse	Triggers if a pulse on an analog signal is within the min and max pulse width. Pulses can be defined as either low to high then high to low (positive), the reverse (negative) or both.
	Tracking Edge	Edge triggering with automatic adjustment of trigger levels to compensate for drifts. The tracking speed is controlled by the bandwidth of the low-pass filter. For this filter noise rejection can only be achieved by level hysteresis.

Control/Tool	Option/Range	Description
	HW Trigger	Trigger on one of the four trigger inputs. Ensure that the trigger level and the trigger coupling is correctly adjusted. The trigger input state can be monitored on the plotter.
	Tracking Pulse	Pulse triggering with automatic adjustment of trigger levels to compensate for drifts. The tracking speed is controlled by the bandwidth of the low-pass filter. For this filter noise rejection can only be achieved by level hysteresis.
	Pulse Count	Trigger on trigger events supplied by the pulse counter. This allows for high resolution triggering. The pulse counter must be enabled and configured on the pulse counter tab. This functionality requires the UHF-CNT option.
Force	Force	Forces a single trigger event.
Pulse Type	Positive/Negative/Both	Select between negative, positive or both pulse forms in the signal to trigger on.
Trigger Edge	Positive/Negative/Both	Triggers when the trigger input signal is crossing the trigger level from either high to low, low to high or both. This field is only displayed for trigger type Edge, Tracking Edge and Event Count.
Bits	0 to 2^32-1	Specify the value of the DIO to trigger on. All specified bits have to be set in order to trigger. This field is only displayed for trigger type Digital.
Bit Mask	0 to 2^32-1	Specify a bit mask for the DIO trigger value. The trigger value is bits AND bit mask (bitwise). This field is only displayed for trigger type Digital.
Level	full signal range	Specify the trigger level value.
Find	Find	Automatically find the trigger level based on the current signal.
Hysteresis	full signal range	The hysteresis is important to trigger on the correct edge

Control/Tool	Option/Range	Description
		in the presence of noise. The hysteresis is applied below the trigger level for positive trigger edge selection. It is applied above for negative trigger edge selection, and on both sides for triggering on both edges.
Event Type		Type of event to trigger on for pulse counter trigger signals.
	Any	Trigger on every sample from the pulse counter, regardless of the counter value.
	Increment	Trigger on incrementing counter values.
Count	integer number	Number of trigger events to record (in Single mode)
Trigger progress	0% to 100%	The percentage of triggers already acquired (in Single mode)
Bandwidth (Hz)	0 to 0.5 * Sampling Rate	Bandwidth of the low-pass filter applied to the trigger signal. For edge and pulse trigger use a bandwidth larger than the signal sampling rate divided by 20 to keep the phase delay. For tracking filter use a bandwidth smaller than signal sampling frequency divided by 100 to just track slow signal components like drifts.
Enable	ON / OFF	Enable low-pass filtering of the trigger signal.
Hold Off Time	positive numeric value	Hold off time before the trigger is rearmed. A hold off time smaller than the duration will lead to overlapping trigger frames.
Hold Off Count	integer value	Number of skipped triggers until the next trigger is recorded again.
Delay	-2 s to 2 s	Time delay of trigger frame position (left side) relative to the trigger edge. For delays smaller than 0, trigger edge inside trigger frame (pre trigger). For delays greater than 0, trigger edge before trigger frame (post trigger)
Duration	up to 2 s	Recording length for each triggered data set.

Control/Tool	Option/Range	Description
Pulse Min	0 to 1 s	Minimum pulse width to trigger on.
Pulse Max	0 to 1 s	Maximum pulse width to trigger on.

Table 4.27. SW Trigger tab: Grid sub-tab

Control/Tool	Option/Range	Description
Mode		Enable the grid mode for two-dimensional data recording and select horizontal data resampling method.
	Off	Grid mode disabled.
	Nearest	Grid mode enabled. Resampling is performed using substitution by closest data point.
	Linear	Grid mode enabled. Resampling is performed using linear interpolation.
Operation		Select row update method.
	Replace	New row replaces old row.
	Average	The data for each row is averaged over a number of repetitions.
Columns	numeric value	Number of columns. The data along the horizontal axis are resampled to a number of samples defined by this setting.
Rows	numeric value	Number of rows
Scan Direction		Select the scan direction and mode
	Forward	Scan direction from left to right
	Reverse	Scan direction from right to left
	Bidirectional	Alternate scanning in both directions
Repetitions	numeric value	Number of repetitions used for averaging
AWG Control	ON / OFF	If enabled, the row number is identified based on the digital row ID number set by the AWG. If disabled, every new trigger event is attributed to a new row sequentially.

Table 4.28. SW Trigger tab: History sub-tab

Control/Tool	Option/Range	Description
History	History	Each entry in the list corresponds to a single trigger trace in the history. The number of triggers displayed in the plot is limited to 20. Use the toggle buttons to hide/ display individual traces. Use the color picker to change the color of a trace in the plot. Double click on an entry to edit its name.
Clear All	Clear All	Remove all records from the history list.
All	All	Select all records from the history list.
None	None	Deselect all records from the history list.
Length	integer value	Maximum number of entries stored in the measurement history. The number of entries displayed in the list is limited to the most recent 100.
Save	Save	Save all trigger event based traces in the history to file.Which data is saved depends on the demodulator channels present in the Vertical Axis Groups of the Control sub-tab

For the Math sub-tab please see Table 4.7 in the section called “Cursors and Math”.

4.8. Spectrum Analyzer Tab

The Spectrum Analyzer is one of the powerful frequency domain measurement tools as introduced in Section 4.1.2 and is available in all UHFLI Instruments.

4.8.1. Features

- Fast, high-resolution FFT spectrum analyzer of demodulated data (X+iY, R, Θ, f and dΘ/dt/(2π))
- Variable center frequency, frequency resolution and frequency span
- Auto bandwidth, auto span (sampling rate)
- Choice of 4 different FFT window functions
- Continuous and block-wise acquisition with different types of averaging
- Detailed noise power analysis
- Support for Input Scaling and Input Units
- Mathematical toolbox for signal analysis

4.8.2. Description

The Spectrum Analyzer allows frequency domain analysis of the demodulator data that are streamed to the data server with a user-defined rate. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.29. App icon and short description

Control/Tool	Option/Range	Description
Spectrum		Provides FFT functionality to all continuously streamed measurement data.

The Spectrum tab (see Figure 4.22) is divided into a display section on the left and a configuration section on the right. The configuration section is further divided into a number of sub-tabs.

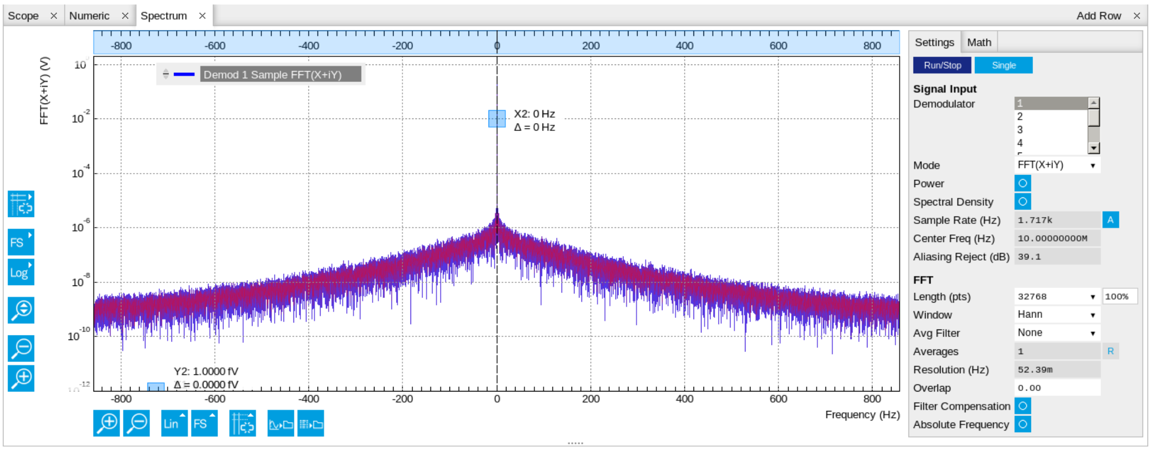


Figure 4.22. LabOne UI: Spectrum analyzer tab

The Spectrum Analyzer allows for spectral analysis of all the demodulator data by performing the fast Fourier transform (FFT) on the complex demodulator data samples X+iY (with i as the imaginary unit). The result of this FFT is a spectrum centered around the demodulation frequency, whereas applying a FFT directly on the raw input data would produce a spectrum centered around zero frequency. The latter procedure corresponds to the Frequency Domain operation in the Scope tab described in Section 4.6. The main difference between the two is that the Spectrum Analyzer tool can acquire data for a much longer periods of time and therefore can achieve very high frequency resolution around the demodulation frequency. By default, the spectrum is displayed centered around zero. Sometimes however it is convenient to shift the frequency axis by the demodulation frequency which allows one to identify the frequencies on the horizontal axis with the physical frequencies at the signal inputs. This can be done by activating Absolute Frequency on the Settings sub-tab.

Another important property of the spectrum is the fact that the data samples have passed a low-pass filter with a well-defined order and bandwidth. This is most clearly noted by the shape of the noise floor. One has to take care that the selected frequency span, which equals the demodulator sampling rate, is in a healthy ratio with respect to the filter bandwidth and order. When in doubt the user can always press the button labeled A next to the sampling rate in order to obtain a default setting that suits the filter settings.

Other than displaying the frequency spectrum of the complex demodulator samples X+iY, the user can also choose to apply an FFT to the polar demodulator values R and Theta. This allows to carefully discriminate between phase noise components and amplitude noise components present in the signal. As these samples are real numbers the spectrum is single-sided with minimum frequency of 0 Hz.

The last option in the drop-down list dΘ/dt allows one to apply the FFT on samples of demodulator frequencies. That is particularly useful when either the PLL or the ExtRef functionalities are used. The FFT of the frequency samples then provide a quantitative view of what frequency noise components are present in the reference signal and also helps to find the optimal PLL bandwidth to track the signal.

4.8.3. Functional Elements

Table 4.30. Spectrum tab: Settings sub-tab

Control/Tool	Option/Range	Description
Run/Stop	Run/Stop	Run the FFT spectrum analysis continuously
Single	Single	Run the FFT spectrum analysis once
Demodulator	demodulator index	Select the input demodulator for FFT spectrum analysis
Mode		Select the source signal for FFT spectrum analysis. Enable spectral density to extract noise values. In order to extract peak values ensure that spectral density is disabled.
	FFT(X+iY)	Complex FFT of the demodulator result (zoom FFT). The center frequency is defined by the oscillator frequency of the demodulator. The span is twice the demodulator sampling rate.

Control/Tool	Option/Range	Description
	FFT(R)	FFT of the demodulator amplitude result sqrt(x² + y²). The FFT is single sided as performed on real data.
	FFT(Θ)	FFT of the demodulator phase result atan2(y, x). The FFT is single sided as performed on real data.
	FFT(f)	FFT of the oscillator frequency of the selected demodulator. This mode is only interesting if the oscillator is controlled by a PID/PLL controller. The FFT is single sided as performed on real data.
	FFT(dΘ/dt)/(2π)	FFT of the demodulator phase derivative. This value is equivalent to the frequency noise observed on the demodulated signal. The FFT is single sided as performed on real data.
Power	ON / OFF	Calculate and show the power value. To extract power spectral density (PSD) this button should be enabled together with spectral density.
Spectral Density	ON / OFF	Calculate and show the spectral density. If power is enabled the power spectral density value is calculated. The spectral density is used to analyze noise.
Sample Rate (Hz)	numeric value	Equivalent to sampling rate of demodulator. The resulting frequency span is equal to the sample rate. Increase the sample rate to reduce aliasing.
Auto	A	Automatic adjustment of the sampling rate. The rate will be selected to achieve good enough anti-aliasing for the selected demodulator bandwidth.
Center Freq (Hz)	numeric value	Demodulation frequency of the selected demodulator used as input for the spectrum. For complex FFT(X+iY) the demodulation frequency defines the center frequency of the displayed FFT.

Control/Tool	Option/Range	Description
Aliasing Reject (dB)	numeric value	Resulting aliasing rejection based on demodulator sampling rate and low-pass filter settings. If the value is too low either increase the sampling rate or lower the filter bandwidth.
Length (pts)	2^8 to 2^23	Number of lines of the FFT spectrum. A higher value increases the frequency resolution of the spectrum.
Sampling Progress	0% to 100%	The percentage of the FFT buffer already acquired.
Window	Rectangular	Four different FFT windows to choose from. Depending on the application it makes a huge difference which of the provided window function is used. Please check the literature to find out the best trade off for your needs.
	Hann	
	Hamming	
	Blackman Harris	
Avg Filter	None	Selects the type of averaging.
	Exp Moving Avg	
Averages	integer value	Defines the number of spectra which are averaged and displayed.
Reset	R	Press once to reset the averaging filter.
Resolution (Hz)	mHz to Hz	Spectral resolution defined by the reciprocal acquisition time (sample rate, number of samples recorded).
Overlap	0 to 1	Overlap of demodulator data used for the FFT transform. Use 0 for no overlap and 0.99 for maximal overlap.
Filter Compensation	ON / OFF	Spectrum is corrected by demodulator filter transfer function. Allows for quantitative comparison of amplitudes of different parts of the spectrum.
Absolute Frequency	ON / OFF	Shifts x-axis labeling to show the demodulation frequency in the center as opposed to 0 Hz, when turned off.

For the Math sub-tab please see Table 4.7 in the section called “Cursors and Math”.

4.9. Sweeper Tab

The Sweeper is a highly versatile measurement tool available in all UHFLI Instruments. The Sweeper allows one to scan one instrument parameter over a defined range and at the same time measure a selection of continuously streamed data. In the special case where the sweep parameter is an oscillator frequency, the Sweeper offers the functionality of a frequency response analyzer (FRA), a well-known class of instruments.

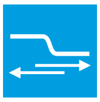
4.9.1. Features

- Full-featured parametric sweep tool for frequency, phase shift, output amplitude, DC output voltages, etc.
- Simultaneous display of data from different sources (Demodulators, PIDs, Boxcar, Arithmetic Unit)
- Different application modes, e.g. Frequency response analyzer (Bode plots), noise amplitude sweeps, etc.
- Different sweep types: single, continuous (run / stop), bidirectional, binary
- Persistent display of previous sweep results
- Normalization of sweeps
- Auto bandwidth, averaging and display normalization
- Support for Input Scaling and Input Units
- Phase unwrap
- Full support of sinc filter

4.9.2. Description

The Sweeper supports a variety of experiments where a parameter is changed stepwise and numerous measurement data can be graphically displayed. Open the tool by clicking the corresponding icon in the UI side bar. The Sweeper tab (see Figure 4.23) is divided into a plot section on the left and a configuration section on the right. The configuration section is further divided into a number of sub-tabs.

Table 4.31. App icon and short description

Control/Tool	Option/Range	Description
Sweeper		Allows to scan one variable (of a wide choice, e.g. frequency) over a defined range and display various response functions including statistical operations.

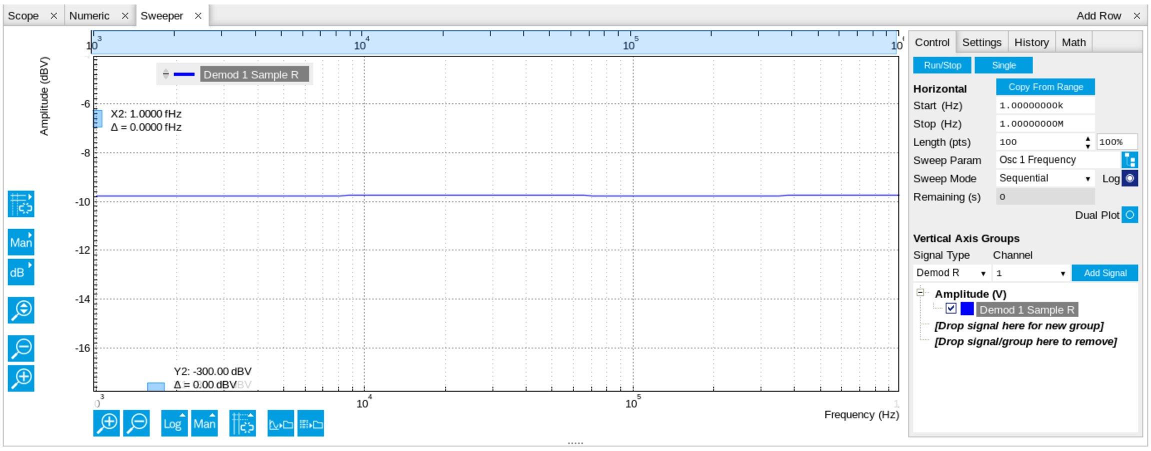


Figure 4.23. LabOne UI: Sweeper tab

The Control sub-tab holds the basic measurement settings such as Sweep Parameter, Start/Stop values, and number of points (Length) in the Horizontal section. Measurement signals can be added in the Vertical Axis Groups section. A typical use of the Sweeper is to perform a frequency sweep and measure the response of the device under test in the form of a Bode plot. Measurement parameters can be added As an example, AFM and MEMS users require to determine the resonance frequency and the phase delay of their oscillators. The Sweeper can also be used to sweep parameters other than frequency, for instance signal amplitudes and DC offset voltages. A sweep of the Auxiliary Output offset can for instance be used to measure current-voltage (I-V) characteristics.

For frequency sweeps the default sweep operation is logarithmic, i.e. with the Log button activated. In this mode, the sweep parameter points are distributed logarithmically - as opposed to equidistant for linear sweeps - between the start and stop values. This feature is particularly useful for sweeps over several decades, which is common for frequency sweeps. The Sweep Mode setting is useful for identifying measurement problems caused by hysteretic sample behavior or too fast sweeping speed. Such problems would cause non-overlapping curves in a bidirectional sweep.

Note

The Sweeper actively modifies the main settings of the demodulators and oscillators. So in particular for situations where multiple experiments are served maybe even from different control computers great care needs to be taken so that the parameters altered by the sweeper module do not have unwanted effects elsewhere.

The Sweeper offers two operation modes differing in the level of detail of the accessible settings: the Application Mode and the Advanced Mode. Both of them are accessible in the Settings sub-tab. The Application Mode provides the choice between six measurement approaches that should help to quickly obtain correct measurement results for a large range of applications. Users who like to be in control of all the settings can access them by switching to the Advanced Mode.

In the Statistics section of the Advanced Mode one can control how data is averaged at each sweep point: either by specifying the Sample count, or by specifying the number of filter time constants (TC). The actual measurement time is determined by the larger of the two settings, taking into account the demodulator sample rate and filter settings. The Algorithm settings determines the statistics calculated from the measured data: the average for general purposes, the deviation for noise measurements, or the mean square for power measurements. The Phase Unwrap features ensures continuity of a phase measurement curve across the +-180 degree boundary. Enabling the Sinc Filter setting means that the demodulator Sinc Filter gets activated for sweep points below 50 Hz in Auto and Fixed mode. This speeds up measurements at small frequencies as explained in the Signal Processing chapter.

In the Settling section one can control the waiting time between a parameter setting and the first measurement. Similarly to the Statistics setting, one has the choice between two different representations of this waiting time. The actual settling time is the maximum of the values set in units of absolute time and a time derived from the demodulator filter and a desired inaccuracy (e.g. 1 m for 0.1%). Let's consider an example. For a 4th order filter and a 3 dB bandwidth of 100 Hz we obtain a step response the attains 90% after about 4.5 ms. This can be easily measured by using the SW Trigger as indicated in Figure 4.24. It is also explained in Section 6.3. In case the full range is set to 1 V this means a measurement has a maximum error caused by imperfect settling of about 0.1 V. However, for most measurements the neighboring values are close compared to the full range and hence the real error caused is usually much smaller.

In the Filter section of the Advanced mode, the Bandwidth Mode setting determines how the filters of the activated demodulators are configured. In Manual mode, the current setting (in the Lock-in tab) remains unchanged by the Sweeper. In Fixed mode, the filter settings can be controlled from within the Sweeper tab. In Auto mode, the Sweeper determines the filter bandwidth for each sweep point based on a desired ω suppression. The ω suppression depends on the measurement frequency and the filter steepness. For frequency sweeps, the bandwidth will be adjusted for every sweep point within the bound set by the Max Bandwidth setting. The Auto mode is particularly useful for frequency sweeps over several decades, because the continuous adjustment of the bandwidth considerably reduces the overall measurement time.

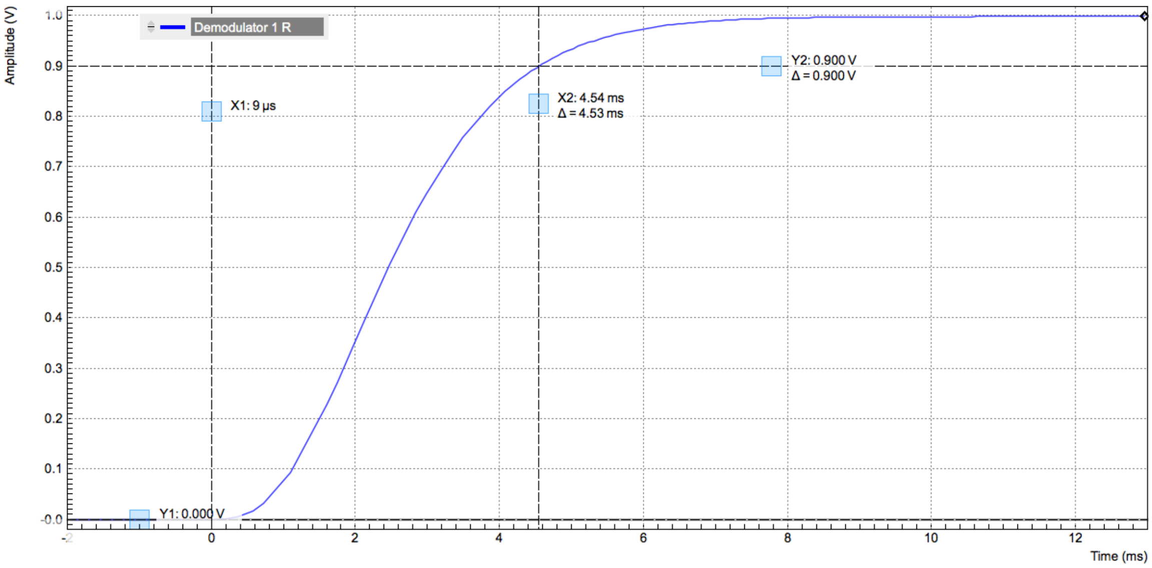


Figure 4.24. Demodulator settling time and inaccuracy

By default the plot area keeps the memory and display of the last 100 sweeps represented in a list in the History sub-tab. The button to the left of each list entry controls the visibility of the corresponding trace in the plot; the button to the right controls the color of the trace. Renaming a trace is easily possible by double clicking on a list entry. All measurements in the history can be stored in a file by clicking on Save. Which quantities are saved depends on which demodulator channels have been added to the Vertical Axis Groups section in the Control sub-tab. Naturally, only data from demodulators with enabled Data Transfer in the Lock-in tab can be included in the files. The file format, Matlab or comma-separated value (CSV) depends on the Format setting in the Config tab. The best way to access the saved files is via the File Manager tab.

With the Reference feature, it is possible to divide all measurements in the history by a reference measurement. This is useful for instance to eliminate spurious effects in a frequency response sweep. To define a certain measurement as the reference, mark it in the list and click on Reference. Then enable the Reference mode with the checkbox below the list to update the plot display. Note that the Reference setting does not affect data saving - saved files always contain raw data.

Note

The Sweeper can get stuck whenever it does not receive any data. A common mistake is to select to display demodulator data without enabling the data transfer of the associated demodulator in the Lock-in tab.

Note

Once a sweep is performed the sweeper stores all data from the enabled demodulators and auxiliary inputs even when they are not displayed immediately in the plot area. These data can be accessed at a later point in time simply by choosing the corresponding signal display settings (Input Channel).

4.9.3. Functional Elements

Table 4.32. Sweeper tab: Control sub-tab

Control/Tool	Option/Range	Description
Run/Stop	Run/Stop	Runs the sweeper continuously.
Single	Single	Runs the sweeper once.
Copy From Range	Copy From Range	Takes over start and stop value from the plot area.
Start (unit)	numeric value	Start value of the sweep parameter. The unit adapts according to the selected sweep parameter.
Stop (unit)	numeric value	Stop value of the sweep parameter. The unit adapts according to the selected sweep parameter.
Length	integer value	Sets the number of measurement points.
Progress	0 to 100%	Reports the sweep progress as ratio of points recorded.
Sweep Param.	Oscillator Frequency	Selects the parameter to be swept. Navigate through the tree view that appears and click on the required parameter. Note: the available selection depends on the configuration of the device.
	Demodulator Phase	
	Signal Output Amplitude	
	Auxiliary Output Offset	
	PID Setpoint	
	Modulation Index	
	Carrier Amplitude	

Control/Tool	Option/Range	Description
	Sideband 1 Amplitude	
	Sideband 2 Amplitude	
	Boxcar Integration Delay	
	Boxcar Integration Time	
	Signal Output Offset	
Sweep Mode		Select the scanning type, default is sequential (incremental scanning from start to stop value)
	Sequential	Sequential sweep from Start to Stop value
	Binary	Non-sequential sweep continues increase of resolution over entire range
	Bidirectional	Sequential sweep from Start to Stop value and back to Start again
	Reverse	Reverse sweep from Stop to Start value
Log	ON / OFF	Selects between linear and logarithmic distribution of the sweep parameter.
Remaining	numeric value	Reporting of the remaining time of the current sweep. A valid number is only displayed once the sweeper has been started. An undefined sweep time is indicated as NaN.
Dual Plot	ON / OFF	Toggle between single plot view and dual plot view

For the Vertical Axis Groups, please see Table 4.9 in the section called “Vertical Axis Groups”.

Table 4.33. Sweeper tab: Settings sub-tab

Control/Tool	Option/Range	Description
Filter		Application Mode: automatic mode. Advanced Mode: manual configuration.
	Application Mode	The sweeper sets the filters and other parameters automatically.
	Advanced Mode	The sweeper uses manually configured parameters.
Application		Select the sweep application mode
	Parameter Sweep	Only one data sample is acquired per sweeper point.

Control/Tool	Option/Range	Description
	Parameter Sweep Averaged	Multiple data samples are acquired per sweeper point of which the average value is displayed.
	Noise Amplitude Sweep	Multiple data samples are acquired per sweeper point of which the standard deviation is displayed (e.g. to determine input noise).
	Freq Response Analyzer	Narrow band frequency response analysis. Averaging is enabled.
	3-Omega Sweep	Optimized parameters for 3-omega application. Averaging is enabled.
	FRA (Sinc Filter)	The sinc filter helps to speed up measurements for frequencies below 50 Hz in FRA mode. For higher frequencies it is automatically disabled. Averaging is off.
	Impedance	This application mode uses narrow bandwidth filter settings to achieve high calibration accuracy.
Precision		Choose between a high speed scan speed or high precision and accuracy.
	Low -> fast sweep	Medium accuracy/precision is optimized for sweep speed.
	High -> slow sweep	High accuracy/precision takes more measurement time.
Bandwidth Mode		Automatically is recommended in particular for logarithmic sweeps and assures the whole spectrum is covered.
	Auto	All bandwidth settings of the chosen demodulators are automatically adjusted. For logarithmic sweeps the measurement bandwidth is adjusted throughout the measurement.
	Fixed	Define a certain bandwidth which is taken for all chosen demodulators for the course of the measurement.

Control/Tool	Option/Range	Description
	Manual	The sweeper module leaves the demodulator bandwidth settings entirely untouched.
Time Constant/Bandwidth Select		Defines the display unit of the low-pass filter to use for the sweep in fixed bandwidth mode: time constant (TC), noise equivalent power bandwidth (NEP), 3 dB bandwidth (3 dB).
	TC	Defines the low-pass filter characteristic using time constant of the filter.
	Bandwidth NEP	Defines the low-pass filter characteristic using the noise equivalent power bandwidth of the filter.
	Bandwidth 3 dB	Defines the low-pass filter characteristic using the cut-off frequency of the filter.
Time Constant/Bandwidth	numeric value	Defines the measurement bandwidth for Fixed bandwidth sweep mode, and corresponds to either noise equivalent power bandwidth (NEP), time constant (TC) or 3 dB bandwidth (3 dB) depending on selection.
Order	numeric value	Selects the filter roll off to use for the sweep in fixed bandwidth mode. Range between 6 dB/oct and 48 dB/oct.
Max Bandwidth (Hz)	numeric value	Maximal bandwidth used in auto bandwidth mode. The effective bandwidth will be calculated based on this max value, the frequency step size, and the omega suppression. The NEP is correctly taken into account for demodulation bandwidths of up to 1.25 MHz.
BW Overlap	ON / OFF	If enabled the bandwidth of a sweep point may overlap with the frequency of neighboring sweep points. The effective bandwidth is only limited by the maximal bandwidth setting and omega suppression. As a result, the bandwidth is independent of

Control/Tool	Option/Range	Description
		the number of sweep points. For frequency response analysis bandwidth overlap should be enabled to achieve maximal sweep speed.
Omega Suppression (dB)	numeric value	Suppression of the omega and 2-omega components. Large suppression will have a significant impact on sweep time especially for low filter orders.
Min Settling Time (s)	numeric value	Minimum wait time in seconds between a sweep parameter change and the recording of the next sweep point. This parameter can be used to define the required settling time of the experimental setup. The effective wait time is the maximum of this value and the demodulator filter settling time determined from the Inaccuracy value specified.
Inaccuracy	numeric value	Demodulator filter settling inaccuracy defining the wait time between a sweep parameter change and recording of the next sweep point. The settling time is calculated as the time required to attain the specified remaining proportion [1e-13, 0.1] of an incoming step function. Typical inaccuracy values: 10 m for highest sweep speed for large signals, 100 u for precise amplitude measurements, 100 n for precise noise measurements. Depending on the order the settling accuracy will define the number of filter time constants the sweeper has to wait. The maximum between this value and the settling time is taken as wait time until the next sweep point is recorded.
Settling Time (TC)	numeric value	Calculated wait time expressed in time constants defined by the specified filter settling inaccuracy.
Algorithm		Selects the measurement method.

Control/Tool	Option/Range	Description
	Averaging	Calculates the average on each data set.
	Standard Deviation	Calculates the standard deviation on each data set.
	Average Power	Calculates the electric power based on a 50 Ω input impedance.
Count (Sa)	integer number	Sets the number of data samples per sweeper parameter point that is considered in the measurement. The maximum between this value and the next setting is taken as effective calculation time.
Count (TC)	0/5/15/50/100 TC	Sets the effective measurement time per sweeper parameter point that is considered in the measurement. The maximum between this value and the previous setting is taken as effective calculation time.
Phase Unwrap	ON / OFF	Allows for unwrapping of slowly changing phase evolutions around the +/-180 degree boundary.
Spectral Density	ON / OFF	Selects whether the result of the measurement is normalized versus the demodulation bandwidth.
Sinc Filter	ON / OFF	Enables sinc filter if sweep frequency is below 50 Hz. Will improve the sweep speed at low frequencies as omega components do not need to be suppressed by the normal low-pass filter.
AWG Control	ON / OFF	If enabled the sweeper starts automatically the AWG when a sweep is started. If sweeps are performed on nodes Index Sweep Triggers the AWG control is enabled automatically. Enable AWG control if some parameters should be recorded based on AWG generated signals.

Table 4.34. Sweeper tab: History sub-tab

Control/Tool	Option/Range	Description
History	History	Each entry in the list corresponds to a single sweep in the history. The number of displayed sweeps is limited to 20. Use the toggle buttons to hide/display individual sweeps. Use the color picker to change the color of a sweep. Double click on an entry to edit its name.
Clear All	Clear All	Remove all records from the history list.
All	All	Select all records from the history list.
None	None	Deselect all records from the history list.
Reference	Reference	Use the selected trace as reference for all active traces.
Length	integer value	Maximum number of entries stored in the measurement history. The number of entries displayed in the list is limited to the most recent 100.
Reference On	ON / OFF	Enable/disable the reference mode.
Reference name	name	Name of the reference trace used.
Save	Save	Save all sweeps in the history to file. Which data is saved depends on the demodulator channels present in the Vertical Axis Groups of the Control sub-tab.

For the Math sub-tab please see Table 4.7 in the section called “Cursors and Math”.

4.10. Arithmetic Unit Tab

The Arithmetic Unit (AU) tab allows the user to define arithmetic operations that are performed on demodulator data in real time. The results of the AUs can be provided to Auxiliary output connectors or to other functional units within the instrument. This functionality and tab is available on all UHF instruments.

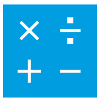
4.10.1. Features

- Four arithmetic units, more than 50 input parameters
- Add and subtract demodulator samples (X, Y, R, Θ) and Boxcar output samples
- Multiply and divide demodulator samples (X, Y, R, Θ) and Boxcar output samples
- Calculate polar coordinates from arbitrary Cartesian demodulator outputs
- Fixed coefficients and auxiliary inputs as scaling factors
- Results available on auxiliary outputs and with that they can also be used as demodulator inputs
- Results available as PID input (requires UHF-PID option)
- Streaming to host computer

4.10.2. Description

The AU tab is the tool used to define and monitor mathematical operations on measurement data in real time. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.35. App icon and short description

Control/Tool	Option/Range	Description
AU		Real-time arithmetic operations on demodulator outputs.

There are four expandable sections (see Figure 4.25), each corresponding to one arithmetic unit. Each unit operates independently and can be considered always ON, hence the defined operation is calculated all the time and the result is available to be used elsewhere in the system. Moreover, when streaming is enabled, the results can be transferred to the host computer, observed in the user interface, and stored to disk. A wide selection of input parameters including demodulator outputs and auxiliary inputs can be taken as operands.

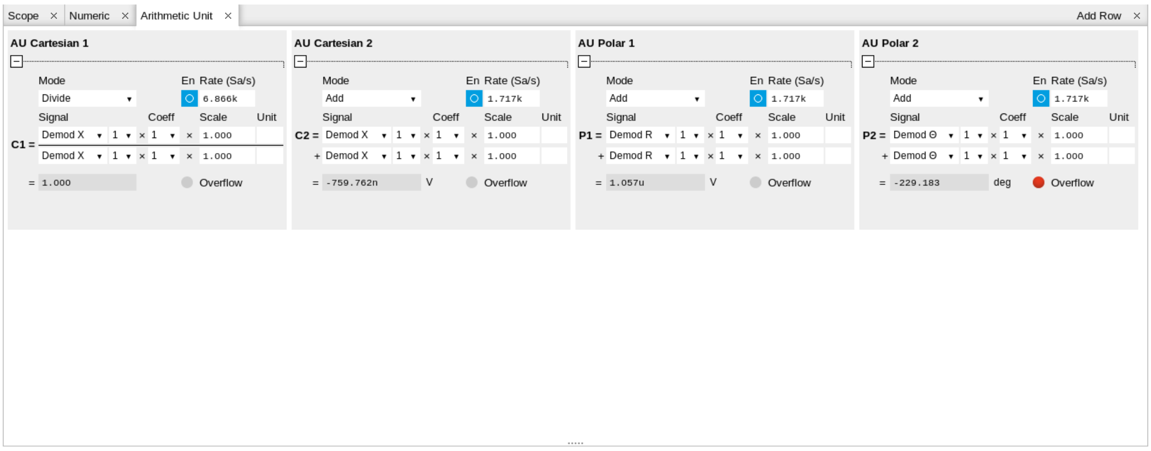


Figure 4.25. LabOne UI: Arithmetic unit tab

In total there are four units, two for Cartesian operations and two for polar operations. Each unit produces a scalar output along with a unit, both indicated in the last line. The Cartesian units can either add, multiply or divide two distinct X and Y values of all demodulators or alternatively the output samples of either Boxcar unit. In addition scaling factors can be applied based on adjustable variables, derived from the auxiliary inputs or even the other Cartesian unit. The polar units can perform similar computations on demodulator magnitude (Demod R) and angle (Demod Θ). In addition, the polar units can also operate on the magnitude and angle of a complex value computed from the two Cartesian units as C1 + iC2 (R(C1+iC2) or Θ(C1+iC2), respectively). Each polar unit must operate entirely on either magnitude or angle values. Similarly to the Cartesian units, the magnitude and angle values can be multiplied with an adjustable variable, a value from one of the auxiliary inputs or even the result of the other Polar arithmetic unit.

4.10.3. Functional Elements

Table 4.36. Arithmetic unit tab

Control/Tool	Option/Range	Description
Mode		Selects the operation mode of the arithmetic unit
	Add	The arithmetic unit is in add mode: two independent demodulator outputs can be added together.
	Divide	The arithmetic unit is in divide mode: two independent demodulator outputs can be divided by each other.
	Multiply	The arithmetic unit is in multiply mode: two independent demodulator outputs can be multiplied with each other.
En		Enables the streaming of results to the host computer.
	ON	The arithmetic unit is operative and results

Control/Tool	Option/Range	Description
		are streamed to the host computer.
	OFF	The arithmetic unit is operative but results are not streamed to the host computer.
Rate	0.2 to 1.75 MSa/s	Defines the number of arithmetic unit result samples that are sent to the host computer per second.
Signal		Select the arithmetic unit input signal
	Demod X	Use demodulator X (for Cartesian AU only).
	Demod Y	Use demodulator Y (for Cartesian AU only).
	Boxcar	Use Boxcar (for Cartesian AU only).
	Demod R	Use demodulator R (for polar AU only).
	Demod Θ	Use demodulator Θ (for polar AU only).
	R(C1 + iC2)	Use the magnitude of C1 + iC2 (for polar AU only).
	Θ(C1 + iC2)	Use the angle of C1 + iC2 (for polar AU only).
Channel	index	Select demodulator and/or Boxcar channel number.
Coeff		Select a coefficient to be applied to the selected Signal. Default: 1.
	1	A coefficient of 1 is used (default).
	Aux In 1	The signal on Aux In 1 is used as coefficient.
	Aux In 2	The signal on Aux In 2 is used as coefficient.
	C1	Output of Cartesian AU 1 (C1) is used as coefficient (for Cartesian AU only).
	C2	Output of Cartesian AU 2 (C2) is used as coefficient (for Cartesian AU only).
	P1	Output of Polar AU 1 (P1) is used as coefficient (for Polar AU only).

Control/Tool	Option/Range	Description
	P2	Output of Polar AU 2 (P2) is used as coefficient (for Polar AU only).
Scale	Real number	Custom scaling factor.
Unit	Text	Unit of "Scale", for example "m/V".
Result value	Real number	Shows the result of the arithmetic unit.
Result unit	Text	Shows the unit of the result of the arithmetic unit. If the unit formula is not valid, it will be indicated as #Invalid! and invalid formula can be corrected by adjusting scaling units.
Overflow	Text	When red, indicates that an overflow has occurred in the arithmetic unit.

4.11. Auxiliary Tab

The Auxiliary tab provides access to the settings of the Auxiliary Inputs and Auxiliary Outputs; it is available for all UHFLI Instruments.

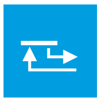
4.11.1. Features

- Monitor signal levels of auxiliary input connectors
- Monitor signal levels of auxiliary output connectors
- Auxiliary output signal sources: Demodulators, PIDs, Boxcars, AUs and manual setting
- Define Offsets and Scaling for auxiliary output values
- Control auxiliary output range limitations

4.11.2. Description

The Auxiliary tab serves mainly as a monitor and control of the auxiliary inputs and outputs. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.37. App icon and short description

Control/Tool	Option/Range	Description
Aux		Controls all settings regarding the auxiliary inputs and auxiliary outputs.

The Auxiliary tab (see Figure 4.26) is divided into three sections. The Aux Input section gives two graphical and two numerical monitors for the signal amplitude applied to the auxiliary inputs on the back panel. In the middle of the tab the Aux Output section allows to associate any of the measured signals to one of the 4 auxiliary outputs on the instrument front panel. With the action buttons next to the Preoffset and Offset values the effective voltage on the auxiliary outputs can be automatically set to zero. The analog output voltages can be limited to a certain range in order to avoid damaging the parts connected to the outputs.

Note

Please note the change of units of the scaling factor depending on what measurement signal is chosen.

Two Aux Output Levels on the right provides 4 graphical and 4 numerical indicators to monitor the voltages currently set on the auxiliary outputs.

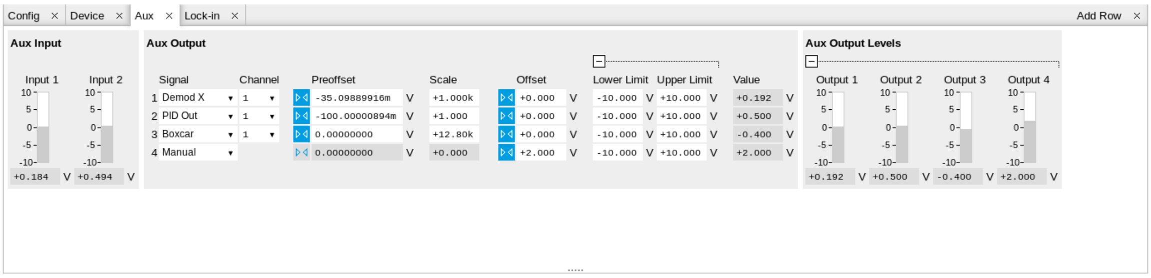


Figure 4.26. LabOne UI: Auxiliary tab

4.11.3. Functional Elements

Table 4.38. Auxiliary tab

Control/Tool	Option/Range	Description
Auxiliary Input Voltage	-10 V to 10 V	Voltage measured at the Auxiliary Input.
Signal		Select the signal source to be represented on the Auxiliary Output.
	X, Y, R, Θ	Select any of the 4 demodulator output quantities of any of the demodulators for auxiliary output.
	PID Out	Use one of the PID controllers output. UHF-PID option needs to be installed.
	PID Shift	Use one of the PID controllers shift results. UHF-PID option needs to be installed.
	Boxcar	Select one of the two Boxcar units for auxiliary output. UHF-BOX option needs to be installed.
	AU Cartesian	Select one of the two Arithmetic Cartesian units for auxiliary output.
	AU Polar	Select one of the two Arithmetic Polar units for auxiliary output.
	AWG	Select one of the AWG channels for auxiliary output when running the AWG in four-channel mode. UHF-AWG option needs to be installed.
	Manual	Manually define an auxiliary output voltage using the offset field.
Channel	index	Select the channel according to the selected signal source.
Preoffset	numerical value in signal units	Add an pre-offset to the signal before scaling is applied. Auxiliary Output Value = (Signal+Preoffset)*Scale + Offset
Auto-zero		Automatically adjusts the Pre-offset to set the Auxiliary Output Value to zero.
Scale	numerical value	Multiplication factor to scale the signal. Auxiliary

Control/Tool	Option/Range	Description
		Output Value = (Signal +Preoffset)*Scale + Offset
Auto-zero	▷◁	Automatically adjusts the Offset to set the Auxiliary Output Value to zero.
Offset	numerical value in Volts	Add the specified offset voltage to the signal after scaling. Auxiliary Output Value = (Signal+Preoffset)*Scale + Offset
Lower Limit	-10 V to 10 V	Lower limit for the signal at the Auxiliary Output. A smaller value will be clipped.
Upper Limit	-10 V to 10 V	Upper limit for the signal at the Auxiliary Output. A larger value will be clipped.
Value	-10 V to 10 V	Voltage present on the Auxiliary Output. Auxiliary Output Value = (Signal +Preoffset)*Scale + Offset

4.12. Inputs/Outputs Tab

The In / Out tab provides access to the settings of the Instrument's main Signal Inputs and Signal Outputs. It is available for all UHFLI Instruments.

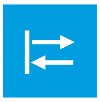
4.12.1. Features

- Signal input configuration
- Signal output configuration

4.12.2. Description

The In / Out tab gives access to the same settings as do the left-most and the right-most sections of the Lock-in tab. It is mainly intended to be used on small screens that can not show the entire the Lock-in tab at once. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.39. App icon and short description

Control/Tool	Option/Range	Description
In/Out		Access to all controls relevant for the main Signal Inputs and Signal Outputs on the instrument's front.

The In / Out tab contains one section for the signal inputs and one for the signal outputs. All of the corresponding connectors are placed on the instrument front panel. The In / Out tab looks differently depending on whether the UHF-MF Multi-frequency option is installed or not. Figure 4.27)

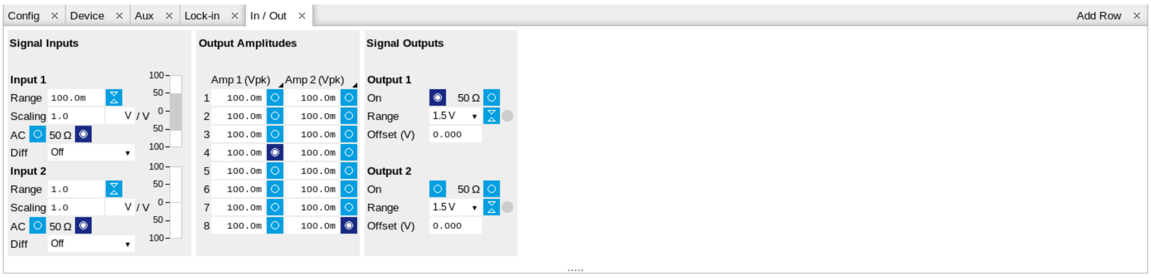


Figure 4.27. LabOne UI: Inputs/Outputs tab (with UHF-MF Multi-frequency option)

Figure 4.28. LabOne UI: Inputs/Outputs tab (without UHF-MF Multi-frequency option)

4.12.3. Functional Elements

All functional elements are equivalent to the ones on the Lock-in tab. See Section 4.2.2 or Section 4.3.2 for a detailed description of the functional elements.

4.13. DIO Tab

The DIO tab provides access to the settings and controls of the digital I/O as well as the Trigger channels and is available for all UHFLI Instruments.

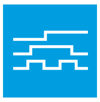
4.13.1. Features

- Monitor and control of digital I/O connectors
- Control settings for external reference and triggering

4.13.2. Description

The DIO tab is the main panel to control the digital inputs and outputs as well as the trigger levels and external reference channels. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.40. App icon and short description

Control/Tool	Option/Range	Description
DIO		Gives access to all controls relevant for the digital inputs and outputs including the Ref/ Trigger connectors.

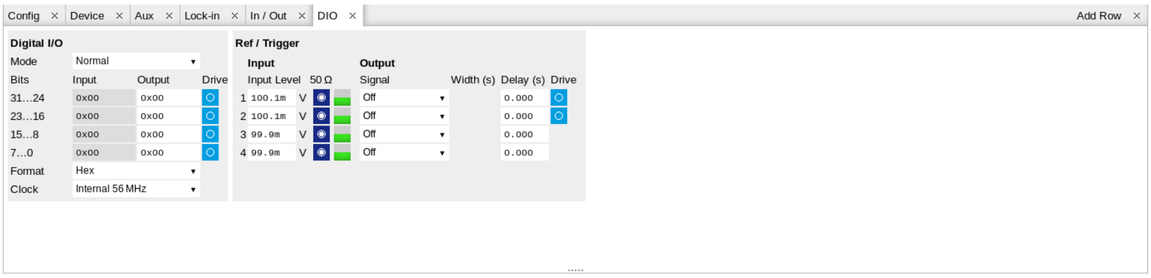


Figure 4.29. LabOne UI: DIO tab

The Digital I/O section provides numerical monitors to observe the states of the digital inputs and outputs. Moreover, with the values set in the Output column and the Drive button activated the states can also be actively set in different numerical formats.

The Ref/Trigger section shows the settings for the 6 reference and trigger inputs and outputs. The two BNC connectors on the front panel are numbered 1 and 2 and can act as inputs as well as outputs. The first two lines in this section are associated to these front panel connectors. On the back panel of the Instrument are 2 more trigger inputs (line 3 and 4, left columns) and 2 more trigger outputs (line 3 and 4, right columns). All four are SMA connectors.

Note

The Input Level determines the trigger threshold for trigger state discrimination. Also a 100 mV hysteresis is applied that cannot be adjusted such that a minimum amplitude of more than 100 mV is needed for the Trigger inputs to work reliably.

4.13.3. Functional Elements

Table 4.41. Digital input and output channels, reference and trigger

Control/Tool	Option/Range	Description
DIO mode		Select DIO mode
	Normal	Manual setting of the DIO output value.
	AWG Sequencer	Enables setting of DIO output values by AWG sequencer commands.
DIO bits	label	Partitioning of the 32 bits of the DIO into 4 buses of 8 bits each. Each bus can be used as an input or output.
DIO input	numeric value in either Hex or Binary format	Current digital values at the DIO input port.
DIO output	numeric value in either hexadecimal or binary format	Digital output values. Enable drive to apply the signals to the output.
DIO drive	ON / OFF	When on, the corresponding 8-bit bus is in output mode. When off, it is in input mode.
Format		Select DIO view format.
	hex	DIO view format is hexadecimal.
	binary	DIO view format is binary.
Clock		Select DIO internal or external clocking.
	Internal 56 MHz	The DIO is internally clocked with a fixed frequency of 56.25 MHz.
	Clk Pin 68	The DIO is externally clocked with a clock signal connected to DIO Pin 68. Available frequency range 1 Hz to 56.25 MHz.
Trigger level	-5 V to 5 V	Trigger voltage level at which the trigger input toggles between low and high. Use 50% amplitude for digital input and consider 100 mV hysteresis.
50 Ω	50 Ω/1 kΩ	Trigger input impedance: When on, the trigger input impedance is 50 Ω, when off 1 kΩ.
Trigger Input status		Indicates the current trigger state.

Control/Tool	Option/Range	Description
	high	A high state has been triggered.
	low	A low state has been triggered.
	toggling	The trigger signal is toggling.
Trigger output signal		Select the signal assigned to the trigger output.
	Off	The output trigger is disabled.
	Osc Phase Demod 4	Trigger event is output for each zero crossing of the oscillator phase used on demodulator 4.
	Osc Phase Demod 8	Trigger event is output for each zero crossing of the oscillator phase used on demodulator 8.
	Scope Trigger	Trigger output is asserted when the scope trigger condition is satisfied.
	Scope /Trigger	Trigger output is deasserted when the scope trigger condition is satisfied.
	Scope Armed	Trigger output is asserted when the scope is waiting for the trigger condition to become satisfied.
	Scope /Armed	Trigger output is deasserted when the scope is waiting for the trigger condition to become satisfied.
	Scope Active	Trigger output is asserted when the scope has triggered and is recording data.
	Scope /Active	Trigger output is deasserted when the scope has triggered and is recording data.
	AWG Trigger 1-4	Trigger output is assigned to one of the AWG Trigger channels controlled by AWG sequencer commands.
	AWG Marker 1-4	Trigger output is assigned to one of the AWG Marker channels attached to AWG waveform data.
	AWG Active	Trigger output is asserted when the AWG is enabled.
	AWG Waiting	Trigger output is asserted when the AWG is waiting for external triggers, for a clock timer, or for other events.

Control/Tool	Option/Range	Description
	AWG Fetching	Trigger output is asserted when the AWG is fetching data from the main waveform and instruction memory.
	AWG Playing	Trigger output is asserted when the AWG is playing waveforms.
Width	0 s to 0.149 s	Defines the minimal pulse width for trigger events signaled on the trigger outputs of the device.
Delay	0 ns to 2.4 ns	Trigger delay, controls the fine delay of the trigger output. The resolution is 78 ps.
Trigger drive	ON / OFF	When on, the bidirectional trigger on the front panel is in output mode. When off, the trigger is in input mode.

4.14. Config Tab

The Config tab provides access to all major LabOne settings and is available for all UHFLI Instruments.

4.14.1. Features

- define instrument connection parameters
- browser session control
- define UI appearance (grids, theme, etc.)
- store and load instrument settings and UI settings
- configure data recording

4.14.2. Description

The Config tab serves mainly as a control panel for all general LabOne related settings and is opened by default on start-up. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.42. App icon and short description

Control/Tool	Option/Range	Description
Config		Provides access to software configuration.

The Config tab (see Figure 4.30) is divided into four sections to control connections, sessions, user interface appearance and data recording.

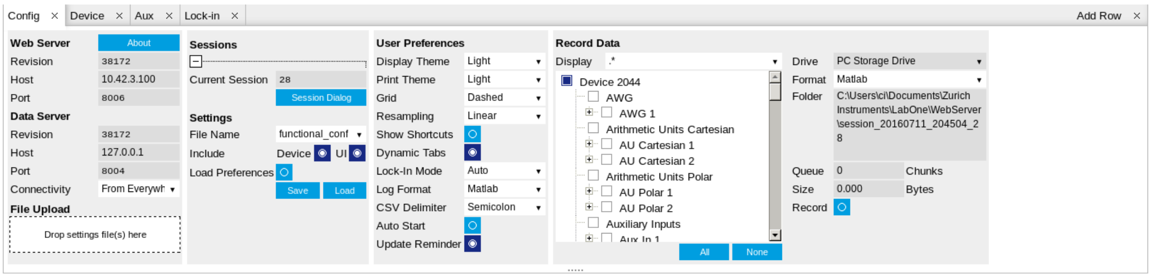


Figure 4.30. LabOne UI: Config tab

The Connection section provides information regarding connection and server versions. Access from remote locations can be restricted with the connectivity setting.

The Session section provides the session number which is also displayed in status bar. Clicking on Session Dialog opens the session dialog window (same as start up screen) that allows one to load different settings files as well as to connect to other instruments.

The Settings section allows one to load and save instrument and UI settings. The saved settings are later available in the session dialogue.

The User Preferences section contains the settings that are continuously stored and automatically reloaded the next time an UHFLI is used from the same computer account. For low ambient light conditions the use of the dark display theme is recommended (see Figure 4.31).

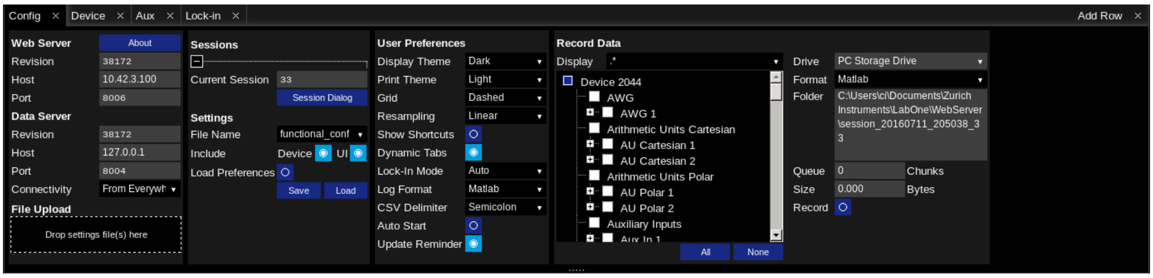


Figure 4.31. LabOne UI: Config tab - dark theme

The Record Data section contains all settings necessary to obtain hard copies of measurement data. The tree structure (see Tree Sub-Tab section) allows one to select among a large number of signals and instrument settings. Use the View Filter in order to reduce the tree structure to the most commonly used nodes such as the demodulator sample nodes.

Whenever the Record button is enabled, all selected nodes get saved continuously as Matlab or comma-separated value (CSV) files. For each selected node at least one file gets generated, but the data may be distributed over several files during long recordings. The quickest way to inspect the files after recording is to use the File Manager tab described in Section 4.16. Apart from the numerical data and settings, the files contain timestamps. These integer numbers encode the measurement time in units of the instrument clock period 1/(1.8 GHz). The timestamps are universal within one instrument and can e.g. be used to align the data from different files.

4.14.3. Functional Elements

Table 4.43. Config tab

Control/Tool	Option/Range	Description
About		Get information about LabOne software.
Web Server Rev	number	Web Server revision number
Host	default is localhost: 127.0.0.1	IP-Address of the LabOne Web Server
Port	4 digit integer	LabOne Web Server TCP/IP port
Data Server Rev	number	Data Server revision number
Host	default is localhost: 127.0.0.1	IP-Address of the LabOne Data Server
Port	default is 8004	TCP/IP port used to connect to the LabOne Data Server.
Connect/Disconnect		Connect/disconnect the LabOne Data Server of the currently selected device. If a LabOne Data Server is connected only devices that are visible to that specific server are shown in the device list.
Connectivity	Localhost Only	Forbid/Allow to connect to this Data Server from other computers.
	From Everywhere	

Control/Tool	Option/Range	Description
File Upload	drop area	Drag and drop files in this box to upload files. Clicking on the box opens a file dialog for file upload. Supported files: Settings (*.xml).
Current Session	integer number	Session identifier. A session is a connection between a client and LabOne Data Server. Also indicated in status bar.
Session Dialog	Session Dialog	Open the session dialog window. This allows for device or session change. The current session can be continued by pressing cancel.
File Name	selection of available file names	Save/load the device and user interface settings to/from the selected file. File location: [user]\AppData\Roaming\Zurich Instruments\LabOne\WebServer\setting
Include Device	ON / OFF	Enable save/load of device settings.
Include UI	ON / OFF	Enable save/load of user interface settings.
Load Preferences	ON / OFF	Enable loading of user preferences from settings file.
Save	Save	Save the user interface and device setting to a file.
Load	Load	Load the user interface and device setting from a file.
Display Theme	Light	Choose theme of the user interface.
	Dark	
Print Theme	Light	Choose theme for printing SVG plots
	Dark	
Grid	Dashed	Select active grid setting for all graphs.
	Solid	
	None	
Resampling Method		Select the resampling interpolation method. Resampling corrects for sample misalignment in subsequent scope shots. This is important when using reduced sample rates with a time resolution below that of the trigger.
	Linear	Linear interpolation

Control/Tool	Option/Range	Description
	pchip	Piecewise Cubic Hermite Interpolating Polynomial
Show Shortcuts	ON / OFF	Displays a list of keyboard and mouse wheel shortcuts for manipulating plots.
Dynamic Tabs	ON / OFF	If enabled, sections inside the application tabs are collapsed automatically depending on the window width.
Lock-In Mode	Auto	Select the display mode for the Graphical Lock-in tab. Auto format will select the format which fits best the current window width.
	Expanded	
	Collapsed	
Log Format	Telnet	Choose the command log format. See status bar and [User]\Documents \Zurich Instruments\LabOne \WebServer\Log
	Matlab	
	Python	
CSV Delimiter	Comma	Select which delimiter to insert for CSV files.
	Semicolon	
	Tab	
Auto Start	ON / OFF	Skip session dialog at start-up if selected device is available. In case of an error or disconnected device the session dialog will be reactivated.
Update Reminder	ON / OFF	Display a reminder on startup if the LabOne software wasn't updated in 180 days.
Drive		Select the drive for data saving.
	PC Storage Drive	Storage of the PC on which the LabOne Web Server is running.
Format	Matlab	Data format of recorded data.
	CSV	
Folder	path indicating file location	Folder containing the saved data.
Queue	integer number	Number of data chunks not yet written to disk.
Size	integer number	Accumulated size of saved data.
Record	ON / OFF	Start and stop saving data to disk as defined in the selection filter. Length of the files is determined by the Window

Control/Tool	Option/Range	Description
		Length setting in the Plotter tab.
Display	filter or regular expression	Display specific tree branches using one of the preset view filters or a custom regular expression.
Tree	ON / OFF	Click on a tree node to activate it.
All		Select all tree elements.
None		Deselect all tree elements.

For more information on the tree functionality in the Record Data section, please see Table 4.8 in the section called “Tree Sub-Tab”.

4.15. Device Tab

The Device tab is the main settings tab for the connected instrument and is available in all UHFLI Instruments.

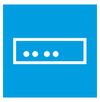
4.15.1. Features

- Option and upgrade management
- External clock referencing (10 MHz)
- Auto calibration settings
- Instrument connectivity parameters
- Device monitor

4.15.2. Description

The Device tab serves mainly as a control panel for all settings specific to the instrument that is controlled by LabOne in this particular session. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.44. App icon and short description

Control/Tool	Option/Range	Description
Device		Provides instrument specific settings.

The Device tab (see Figure 4.32) is divided into five sections: general instrument information, configuration, communication parameters, device presets, and a device monitor.

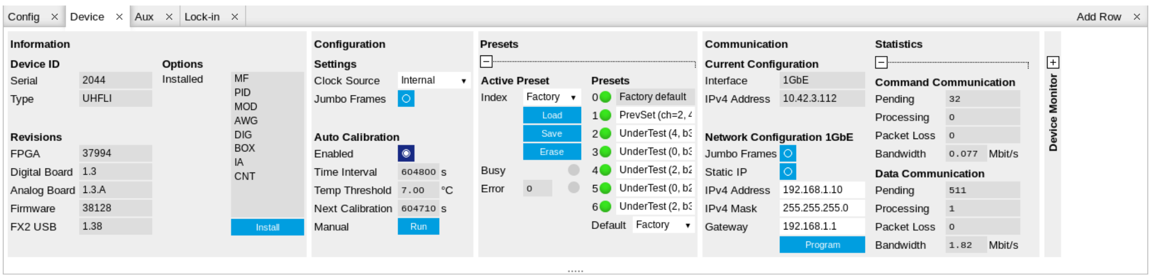


Figure 4.32. LabOne UI: Device tab

The Information section provides details about the instrument hardware and indicates the installed upgrade options. This is also the place where new options can be added by entering the provided option key.

The Configuration section allows one to change the reference from the internal clock to an external 10 MHz reference. The reference is to be connected to the Clock Input on the instrument back panel.

The Presets section allows you to define a custom instrument start-up configuration different from the factory default. This configuration is stored in the instrument itself and are applied independently of the control PC. This saves time in cases where the control PC is not routinely needed, for instance when using only analog interfaces the instrument configuration is fixed.

The Communication section offers access to the instruments TCP/IP settings as well as choosing the connection type.

Note

Activating Jumbo Frames is essential to achieve maximum data rates and also reduces load on the host PC.

The Statistics section gives an overview on communication statistics. In particular the current data rate (Bandwidth) that is consumed.

Note

Packet loss on data streaming over UDP or USB: data packets may be lost if total bandwidth exceeds the available physical interface bandwidth. Data may also be lost if the host computer is not able to handle high-bandwidth data. Network card setting optimization and Jumbo frame enabling may increase the maximal effective bandwidth.

Note

Packet loss on command streaming over TCP or USB: command packets should never be lost as it creates an invalid state.

The Device monitor is collapsed by default and generally only needed for servicing. It displays vitality signals of some of the instrument's hardware components.

Note

The calibration routine takes about 200 ms for that time the transfer of measurement data is stopped. That will lead to the following visible effects on the UI:

- missing data on the plotter
- the UI will shortly freeze
- the data loss flag will not report data loss (as the server intentionally trashed data)
- Sweeper, SW Trigger and Scope will behave as usual and wait until they get data again
- The Spectrum tool will restart as it can only analyze continuously sampled data

Please see also additional remarks regarding calibration in Section 5.6.

4.15.3. Functional Elements

Table 4.45. Device tab

Control/Tool	Option/Range	Description
Serial	1-4 digit number	Device serial number
Type	string	Device type
FPGA	integer number	HDL firmware revision
Digital Board	version number	Hardware revision of the FPGA base board

Control/Tool	Option/Range	Description
Analog Board	version indicator	Hardware revision of the analog board
Firmware	integer number	Revision of the device internal controller software
FX2 USB	version number	USB firmware revision
Installed Options	short names for each option	Options that are installed on this device
Install	Install	Click to install options on this device. Requires a unique feature code and a power cycle after entry.
Clock Source		10 MHz reference clock source.
	Internal	Internal 10 MHz clock is used as the frequency and time base reference.
	Clk 10 MHz	An external 10-MHz clock is used as the frequency and time base reference. Provide a clean and stable 10 MHz reference to the appropriate back panel connector.
Jumbo Frames	ON / OFF	Enables jumbo frames (4k) on the TCP/IP interface. This will reduce the load on the PC and is required to achieve maximal throughput. Make sure that jumbo frames (4k) are enabled on the network card as well. If one of the devices on the network is not able to work with jumbo frames, the connection will fail.
Enabled	ON / OFF	Enables an automatic instrument self calibration about 16 min after start up. In order to guarantee the full specification, it is recommended to perform a self calibration after warm-up of the device.
Time interval	time in seconds	Time interval for which the self calibration is valid. After this time it is recommended to rerun the auto calibration. A LED indicator in the status bar indicates when another self calibration is recommended.

Control/Tool	Option/Range	Description
Calibration temperature threshold	temperature in °C	When the temperature changes by the specified amount, it is recommended to rerun the self calibration. A LED indicator in the status bar indicates when another self calibration is recommended.
Next calibration	time in seconds	Remaining seconds until the first calibration is executed or a recalibration is requested.
Manual self calibration	Run	Initiate self calibration to improve input digitizer linearity.
Index		Select between factory preset or presets stored in internal flash memory.
	Factory	Select factory preset.
	Flash 1-6	Select one of the presets stored in internal flash memory 1-6.
Load	Load	Load the selected preset.
Save	Save	Save the actual setting as preset.
Erase	Erase	Erase the selected preset.
Busy	grey/green	Indicates that the device is busy with either loading, saving or erasing a preset.
Error		Returns a 0 if the last preset operation was successfully completed or 1 if the last preset operation was illegal.
	0	Last preset operation was successfully completed.
	1	Last preset operation was illegal.
Error LED	grey/red	Turns red if the last operation was illegal.
Valid LED	grey/green	Turns green if a valid preset is stored at the respective location.
Presets		Shows a list of available presets including factory preset.
	0	Factory default preset. The name of the factory default preset is given and can not be edited.

Control/Tool	Option/Range	Description
	1	Flash preset 1. The name of this preset can be edited.
	2	Flash preset 2. The name of this preset can be edited.
	3	Flash preset 3. The name of this preset can be edited.
	4	Flash preset 4. The name of this preset can be edited.
	5	Flash preset 5. The name of this preset can be edited.
	6	Flash preset 6. The name of this preset can be edited.
Default		Indicates the preset which is used as default preset at start-up of the device.
	Factory	Select factory preset as default preset.
	Flash 1-6	Select one of the presets stored in internal flash memory 1-6 as default preset.
Interface	1. USB, 2. 1GbE	Active interface between device and data server. In case multiple options are available, the priority as indicated on the left applies.
IPv4 Address	default 192.168.001.010	Current IP address of the device. This IP address is assigned dynamically by a DHCP server, defined statically, or is a fall-back IP address if the DHCP server could not be found (for point to point connections).
Jumbo Frames	ON / OFF	Enable jumbo frames for this device and interface as default.
Static IP	ON / OFF	Enable this flag if the device is used in a network with fixed IP assignment without a DHCP server.
IPv4 Address	default 192.168.001.010	Static IP address to be written to the device.
IPv4 Mask	default 255.255.255.000	Static IP mask to be written to the device.
Gateway	default 192.168.001.001	Static IP gateway
Program	Program	Click to program the specified IPv4 address, IPv4 Mask and Gateway to the device.

Control/Tool	Option/Range	Description
Pending	integer value	Number of buffers ready for receiving command packets from the device.
Processing	integer value	Number of buffers being processed for command packets. Small values indicate proper performance. For a TCP/IP interface, command packets are sent using the TCP protocol.
Packet Loss	integer value	Number of command packets lost since device start. Command packets contain device settings that are sent to and received from the device.
Bandwidth	numeric value	Command streaming bandwidth usage on the physical network connection between device and data server.
Pending	integer value	Number of buffers ready for receiving data packets from the device.
Processing	integer value	Number of buffers being processed for data packets. Small values indicate proper performance. For a TCP/IP interface, data packets are sent using the UDP protocol.
Packet Loss	integer value	Number of data packets lost since device start. Data packets contain measurement data.
Bandwidth	numeric value	Data streaming bandwidth usage on the physical network connection between device and data server.
FW Load	numeric value	Indicates the CPU load on the processor where the firmware is running.

4.16. File Manager Tab

The File Manager tab provides a quick access to measurement files, log files and setting files in the local file system.

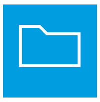
4.16.1. Features

- Quick access to measurement files, log files and settings files
- File preview for settings files and log files

4.16.2. Description

The File Manager tab provides standard tools to see and organize the files relevant for the use of the instrument. Files can be conveniently copied, renamed and deleted. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.46. App icon and short description

Control/Tool	Option/Range	Description
Files		Access settings and measurement data files on the host computer.

The Files tab (see Figure 4.33) provides three windows for exploring. The left window allows one to browse through the directory structure, the center window shows the files of the folder selected in the left window, and the right window displays the content of the file selected in the center window, e.g. a settings file or log file.

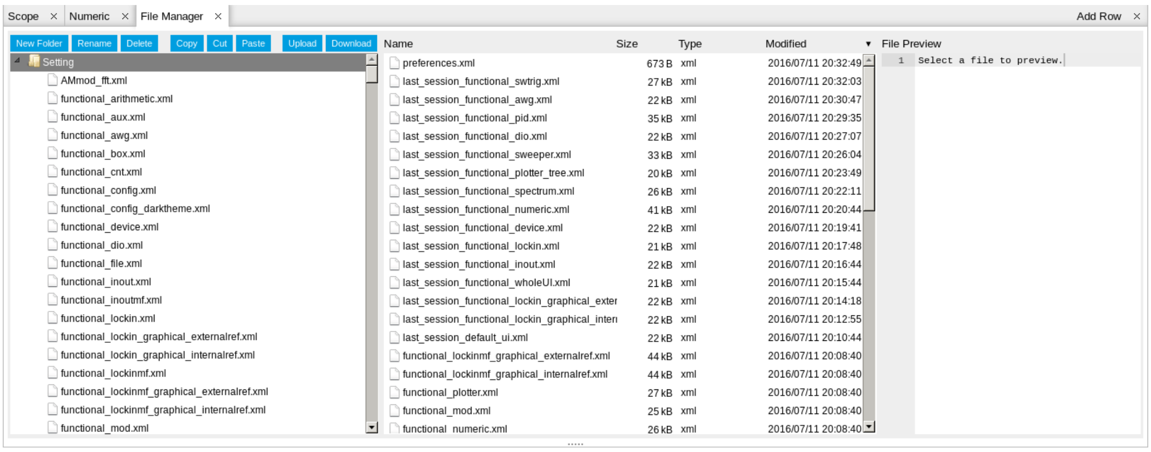


Figure 4.33. LabOne UI: File Manager tab

4.16.3. Functional Elements

Table 4.47. File tab

Control/Tool	Option/Range	Description
New Folder	New Folder	Create new folder at current location.

Control/Tool	Option/Range	Description
Rename	Rename	Rename selected file or folder.
Delete	Delete	Delete selected file(s) and/or folder(s).
Copy	Copy	Copy selected file(s) and/or folder(s) to Clipboard.
Cut	Cut	Cut selected file(s) and/or folder(s) to Clipboard.
Paste	Paste	Paste file(s) and/or folder(s) from Clipboard to the selected directory.
Upload	Upload	Upload file(s) and/or folder(s) to the selected directory.
Download	Download	Download selected file(s) and/or folder(s).

4.17. PLL Tab

The PLL tab allows convenient setup of a phase-locked loop using one of the demodulators as phase detector and one of the PID controllers to provide feedback to an internal oscillator. This tab is only available when the UHF-PID/PLL controller option is installed on the UHF Instrument (see Information section in the Device tab).

Note

Demodulator and PID parameters that are used within an active PLL are set to read-only values on the Lock-in tab and PID tab.

Note

Demodulator that are used within an active PLL are set to read-only values on the Lock-in tab.

4.17.1. Features

- Two fully programmable 600 MHz phased-locked loops
- Programmable PLL center frequency and phase setpoint
- Programmable PLL phase detector filter settings and PID controller parameters
- PLL Advisor for model-based parameter suggestion and transfer function analysis
- Phase unwrap for extended lock range and increased stability
- Auto-zero functions for center frequency and setpoint
- Generation of sub-multiple frequencies by use of harmonic multiplication factor

4.17.2. Description

The PLL tab offers a convenient way to use the combination of PID controllers and demodulators to set up a phase-locked loop. In this way the frequency of an external signal can be mapped onto one of the instrument's internal oscillators. An advisor functionality based on mathematical models helps the user finding and optimizing the PID parameters and quickly optimizing the servo bandwidth for the application. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.48. App icon and short description

Control/Tool	Option/Range	Description
PLL		Features all control and analysis capabilities of the phase-locked loops.

The PLL tab (see Figure 4.34) is divided into two side-tabs corresponding to the two PLL units. It contains a settings sections on the left and a modeling section with graphical display on the right.

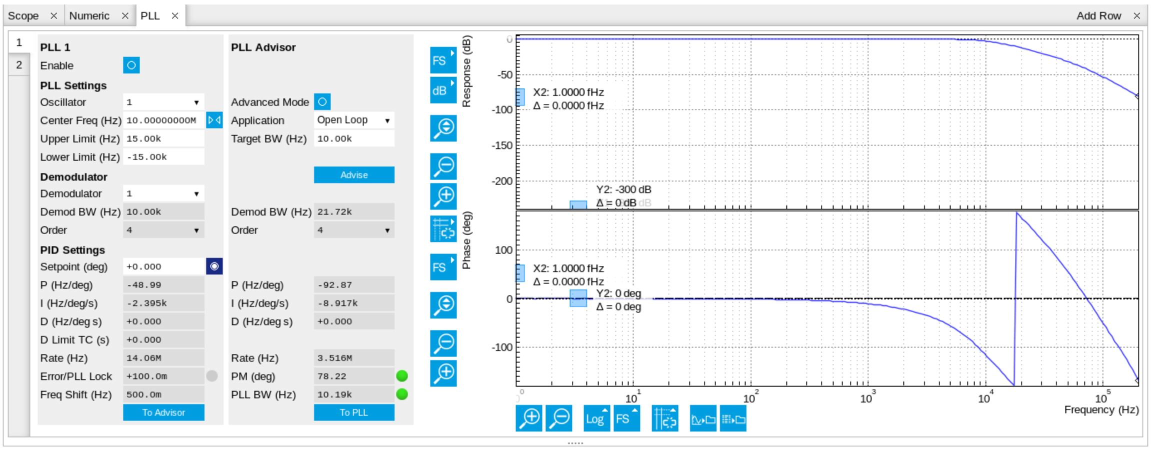


Figure 4.34. LabOne UI: PLL tab

Figure 4.35 shows a block diagram of the PLL with its components, their interconnections and the variables to be specified by the user. The demodulator and the PID controller are slightly simplified for this sketch. Their full detailed block diagrams are given in Figure 4.8, Figure 4.12, and Figure 4.37 respectively.

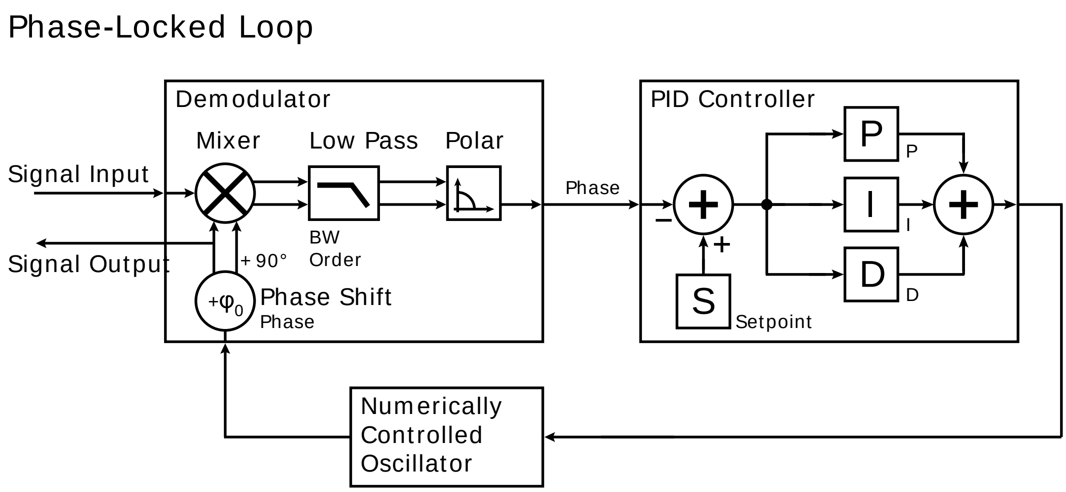


Figure 4.35. Phase-Locked Loop block diagram (components simplified)

In a typical work flow to set up a PLL one would first define the center frequency, and the phase setpoint in the left section. If the frequency is not know beforehand, it can often be measured using the Sweeper or Spectrum tool. Then one would set a target bandwidth in the PLL Advisor section and subsequently click on the Advise button. The feedback parameters calculated by the Advisor will be shown in the fields just below. A graphical representation of the calculated transfer function in shown in the plot on the right-hand side. Once satisfied with the result, one can transfer the values to the instrument by clicking the To PLL button, and then enable the PLL. If the Error/PLL Lock field now displays very small values, the phase lock has been successful.

One can now iterate the process and e.g. play with the target bandwidth in the PLL Advisor to calculate a new set of feedback parameters. Displaying the demodulator phase in the Plotter along with a Histogram and Math function (e.g. standard deviation) can help to characterize residual phase deviations and further improve lock performance by manual tweaking.

Note

The frequency limits in the PLL Settings section should exceed the target bandwidth by at least a factor of 5 to 10.

Note

In the PLL tab you select which of the 8 demodulators you use as a phase detector. Open the Lock-in tab to check if the right Signal Input is associated with the demodulator in use.

4.17.3. Functional Elements

Table 4.49. PLL tab

Control/Tool	Option/Range	Description
Enable	ON / OFF	Enable the PLL (i.e. the associated PID controller)
Oscillator	oscillator index	Oscillator controlled by the PLL
Center Freq (Hz)	0 to 600 MHz	Center frequency of the PLL oscillator. The PLL frequency shift is relative to this center frequency.
Auto Adjust		Adjust the center so that the frequency shift is zero.
Upper Limit (Hz)	numeric value	Upper frequency limit of the PLL oscillator. The PLL frequency is clamped between Center + Lower Limit and Center + Upper Limit.
Lower Limit (Hz)	numeric value	Lower frequency limit of the PLL oscillator. The PLL frequency is clamped between Center + Lower Limit and Center + Upper Limit.
Demodulator	demodulator index	Select the demodulator that is used as the phase detector of the PLL.
Demod BW (Hz)	numeric value	Filter bandwidth of the demodulator used as the phase detector.
Order	1 to 8	Filter order of the input demodulator
Setpoint (deg)	numeric value	Phase set point in degrees (i.e. PID setpoint). Controls the phase difference between the input signal and the generated signal.

Control/Tool	Option/Range	Description
Phase Unwrap	ON / OFF	Enables the phase error unwrapping up to +/-32pi.
P (Hz/deg)	numeric value	PID proportional gain P
I (Hz/deg/s)	numeric value	PID integral gain I
D (Hz/deg*s)	numeric value	PID derivative gain D
D Limit TC	numeric value	Time constant of the low-pass filter for the D gain limitation. When 0, the low-pass filter is disabled.
Rate (Hz)	numeric value	Current sampling rate of the PLL control loop. Note: The numerical precision of the controller is influenced by the loop filter sampling rate. If the target bandwidth is below 1 kHz is starts to make sense to adjust this rate to a value of about 100 to 500 times the target bandwidth. If the rate is set too high for low-bandwidth applications, integration inaccuracies can lead to nonlinear behavior.
Error (deg)	numeric value	Current phase error of the PLL (Set Point - PID Input).
PLL lock LED	grey/green	Indicates when the PLL is locked. The PLL error is sampled at 5 Sa/s and its absolute value is calculated. If the result is smaller than 5 degrees the loop is considered locked.
Freq Shift (Hz)	numeric value	Current frequency shift of the PLL (Oscillator Freq - Center Freq).
To Advisor	To Advisor	Copy the current PLL settings to the PLL Advisor.
Advanced Mode	ON / OFF	Enables manual tuning of the PID parameters. The stability is reported and the frequency response is shown on the plots.
Application	Open Loop	Select PLL Advisor mode. Currently only one mode is supported.
Target BW (Hz)	0.1 Hz to 84 kHz	Requested PLL bandwidth. Higher loop filter bandwidth can be attained by manual tuning only.

Control/Tool	Option/Range	Description
Advise	Advise	Calculate PLL settings based on application mode and given settings.
Demod BW (Hz)	numeric value	Demodulator bandwidth used for the PLL loop filter
Order	1 to 8	Demodulator order used for the PLL loop filter
P (Hz/deg)	numeric value	PLL Advisor proportional gain P
I (Hz/deg/s)	numeric value	PLL Advisor integral gain I
D (Hz/deg*s)	numeric value	PLL Advisor derivative gain D
Rate (Hz)	109.9 kHz to 14 MHz	PLL Advisor sampling rate of the PLL control loop
PM (deg)	numeric value	Simulated phase margin of the PLL with the current settings. The phase margin should be greater than 45 deg and preferably greater than 65 deg for stable conditions.
Advisor stability LED	green/red	When green, the PLL Advisor found a stable solution with the given settings. When red, revise your settings and rerun the PLL Advisor.
PLL BW (Hz)	numeric value	Simulated bandwidth of the PLL with the current settings. The bandwidth is roughly equal to the locking range of the PLL.
Model BW LED	green/red	Red indicates the simulated PLL BW is smaller than the Target BW.
To PLL	To PLL	Copy the PLL Advisor settings to the PLL.

4.18. PID Tab

The PID tab is only available if the UHF-PID Quad PID/PLL Controller option is installed on the UHFLI Instrument (the installed options are displayed in the Device tab).

Note

Some settings in the PID tab are interdependent with settings that are controlled from other tabs. If the PID output controls a certain variable, e.g. Signal Output Offset, this variable will be shown as read-only where it appears in other tabs (i.e. in the Lock-in tab for this case).

Note

Each of the PIDs can also be used from other Instrument entities. In particular when the user selects ExtRef for either Demodulator 4 or 8 (see Lock-in tab, Demodulator section, Mode column) one of the PID controllers will be reserved for that purpose. Similarly using the PLLs will cause one PID controller to be blocked for each enabled PLL and can then only be controller from the PLL tab, however, all the values are still updated in the PID tab as read-only values.

4.18.1. Features

- Four independent proportional, integral, derivative (PID) controllers
- PID Advisor with multiple DUT models, transfer function, and step function modeling
- Auto Tune: Automatic minimization of the amplitude of the PID error signal
- High speed operation with up to 300 kHz loop filter bandwidth
- Input parameters: demodulator data, auxiliary inputs, auxiliary outputs and arithmetic unit
- Output parameters: output amplitudes, oscillator frequencies, demodulator phase, auxiliary outputs and signal output offsets
- Phase unwrap for demodulator Θ data (± 64 π), e.g. for optical phase-locked loops

4.18.2. Description

The PID tab is the main control center of general feedback loop related settings. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.50. App icon and short description

Control/Tool	Option/Range	Description
PID		Features all control and analysis capabilities of the PID controllers.

The PID tab (see Figure 4.36) is divided into four identical sub-tabs, each of them providing access to the settings functionality for one of the four PID controllers and the associated PID Advisor.

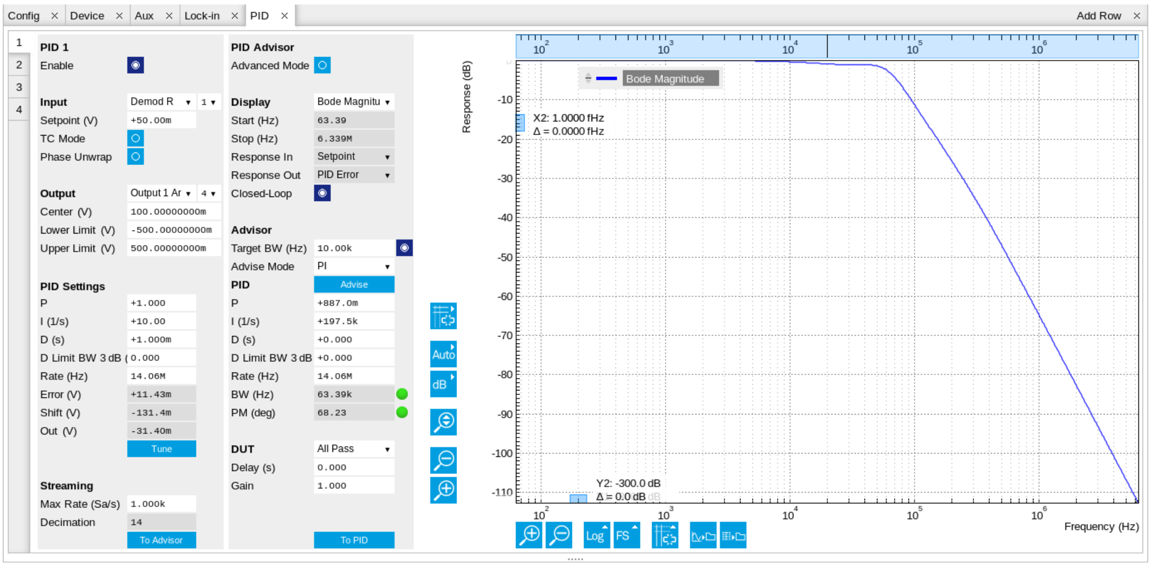


Figure 4.36. LabOne UI: PID tab

With their variety of different input and output connections, the LabOne PID controllers are extremely versatile and can be used over a wide range of different applications. With low internal delays the speed is even high enough to cater to demanding laser locking applications. Figure 4.37 shows a block diagram of all PID controller components, their interconnections and the variables to be specified by the user.

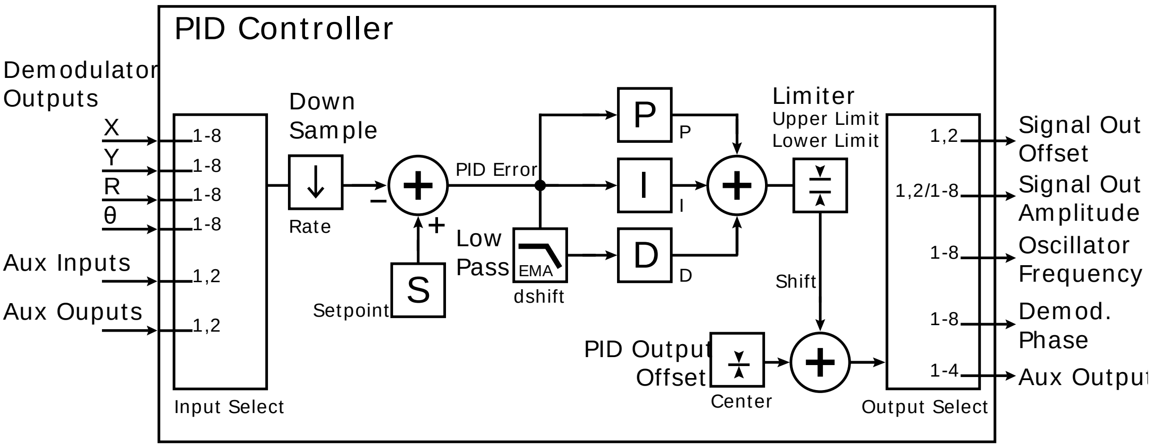


Figure 4.37. PID controller block diagram

Setting up a control loop

Depending on the application there are a number of ways to set up a control loop. Let's consider a few different approaches and see how the Advisor can help to reduce the effort and improve on the result and understanding of the setup.

Manual approach

In cases where the transfer function of the device under test (DUT) is entirely unknown and only little noise couples into the system from the environment, a manual approach is often the quickest way to get going. For manual configuration of a new control loop it is generally recommended to start with a small value for P and set the other parameters (I, D, D Limit) to zero. By enabling the controller one will then immediately see if the sign of P is correct and if the feedback is acting on the correct output parameter for instance by checking the numbers (Error, Shift, Out) displayed in the PID tab. A slow increase of I will then help to zero the PID error signal completely. Care has to be taken when enabling the D part as this often introduces an instable servo behavior which sometimes can be mitigated by activating the associated low-pass filter. At this stage a Plotter tab opened in parallel and displaying the PID error over time can be a great help. The math tools offered by the Plotter allow us to display the standard deviation and the average value of the error. These values should be minimized by tweaking the PID parameters and the associated histogram should have a symmetric (ideally Gaussian) envelope. After a few iterations one can then check the performance by introducing a step response by changing the PID setpoint. The SW Trigger is the ideal tool to record the step function of the control loop by setting the trigger condition half way of the step and the Delay and Duration according to the expected bandwidth. One should also make sure that the data rate set for the transfer of the PID data is high enough to fully resolve the behavior in the time domain.

Auto tune

The Auto Tune feature can now help to fully optimize on the residual noise of the error signal. The implemented simplex algorithm will vary the parameters, as selected in the Advise Mode field in the PID Advisor section, in order to minimize the root mean square of the PID error signal. That is often accompanied by a lowering of the effective servo loop bandwidth and works great as long as there are no occasional large disruptions entering the loop. A typical example where the use of the Auto Tune feature makes no sense are situations where the loop serves to follow a step change of a certain parameter, e.g. the setpoint, that needs to be accommodated within a required time interval. The transfer function of the chosen PID settings can always be checked by copying the values to the Advisor pressing the "To Advisor" button and selecting the Advanced Mode. With the Response In set to Setpoint, the Response Out set to PID Output and with Closed-Loop not activated one can visualize the Bode Magnitude of the PID controller's transfer function. This graph is what is usually given in textbooks and is independent of the model function chosen in the DUT section. However, in order to simulate step responses or to calculate a bandwidth a suitable model for the entire loop is required. If one is only interested in the PID bandwidth one can chose the All Pass DUT model function with Gain 1 and a Delay set to 0. The PID bandwidth will then be indicated below the PID parameters in the Advisor section.

Using DUT model functions with the PID Advisor

For many experimental situations the external device or DUT that needs to be controlled can be well approximated by a simple model. LabOne offers a number of different choices of DUT model functions. Apart from model-specific parameters, all of them have a setting for the delay that occurs outside the instrument. Depending on the targeted servo bandwidth, the external delay can often be the limiting factor and should be sensibly chosen.

Note

The delay specified for each model resembles the earliest possible response for a step change of the instrument output to be seen on the instrument input. It describes the causality of the system and does not affect the shape of the DUT transfer function. Standard coaxial cables cause a signal delay of about 5 ns/m.

The most simple approach to modeling is to assume a DUT with a unity transfer function by using All Pass. The low-pass filters allow for limiting the bandwidth, to set an overall gain and a damping for the second order filter. Resonator Frequency is a model that applies well in situations with a passive external component, e.g. a AFM cantilever or a quartz resonator, whose frequency should be tracked by a PLL over time. In cases where the amplitude of the resonator signal needs to be stabilized with a second control loop (automatic gain control), the Resonator Amplitude model is the right choice. Setting the resonance frequency and the Q factor, both can be obtained before by a frequency scan over the resonance using the sweeper module, allows the Advisor to estimate the gain and low-pass behavior of the resonator. Internal PLL is used whenever an external oscillating signal is provided that shall be followed by one of the internal oscillators. The VCO setting describes a situation where the input variable of the DUT is a voltage and the output is a frequency. The gain is then the conversion factor of how much voltage change on the input causes how much frequency shift on the VCO output. In case the frequency of the VCO can be tracked by using the external reference mode, one can easily obtain this gain with the sweeper by scanning the Auxiliary Output voltage and displaying the resulting oscillator frequency. The gain is given by the slope of the resulting line at the frequency of interest.

With a suitable model chosen and the proper parameter set to best describe the actual measurement situation, one can now continue by defining a target bandwidth for the entire control loop and the Advise Mode, i.e. the parameters that shall be used for the control operation. Whenever the input signal is derived from one of the demodulators it is convenient to activate the box next to target bandwidth. With that in place the Advise algorithm will automatically adjust the demodulator bandwidth to a value about 5 time higher than the target bandwidth in order to avoid to be limited by demodulation speed. With all the model information and the Target Bandwidth the Advise algorithm will now calculate a target step response function that it will try to achieve by adjusting the parameters in the next step. Before doing so in case of a newly set up DUT model the algorithm will first try to estimate the PID parameters by using the Ziegler-Nichols method. When there has been a previous run the user can also change the parameters in the model manually which will the be used as new start parameters of the next Advise run. Starting from the initial parameters, the Advisor will then perform a numerical optimization in order to achieve a least-squares fit of the calculated step response to a target step response determined from the Target Bandwidth. The result is numerically characterized by an achieved bandwidth (BW) and a phase margin (PM). Moreover, the large plot area on the right can be used to characterize the result by displaying transfer functions, magnitude and phase, and step responses between different signal nodes inside the loop. Once the modeling is finished one can copy the resulting parameters to the physical PID by clicking on "To PID".

Table 4.51. DUT transfer functions

Name	Function	Parameters
All pass	$H(s) = g$	1. Gain g
Low-pass 1st	$H(s) = g\frac{1}{t_c s + 1} = g\frac{\omega_n}{s + \omega_n}$	1. Gain g 2. Filter bandwidth (BW) $f_{\text{-3dB}} = \omega_n / 2\pi$
Low-pass 2nd	$H(s) = g\frac{\omega_n^2}{s^2 + 2\omega_n \zeta s + \omega_n^2}$	1. Gain g 2. Resonance frequency $f_{\text{res}} = \omega_n / 2\pi$ 3. Damping ratio ζ with $f_{\text{-3dB}} = 2\zeta f_{\text{res}}$
Resonator frequency	$H(s) = \frac{1}{t_c s + 1}$ with $t_c = \frac{1}{2\pi f_{\text{-3dB}}} = \frac{2Q}{2\pi f_{\text{res}}}$	1. Resonance frequency f_{res} 2. Quality factor Q
Resonator amplitude	$H(s) = g\frac{\omega_{\text{res}}/(2Q)}{s + \omega_{\text{res}}/(2Q)}$ with $\omega_{\text{res}} = 2\pi f_{\text{res}}$	1. Gain g

Name	Function	Parameters
		2. Resonance frequency f_{res} 3. Quality factor Q
Internal PLL	$H(s) = -\frac{360}{s}$	1. none
VCO	$H(s) = g\frac{360}{s(t_c s + 1)}$ with $t_c = \frac{1}{2\pi f_{\mathrm{-3dB}}}$	1. Gain g (Hz/V) 2. Bandwidth (BW) $f_{\mathrm{-3dB}}$

Note

It is generally recommended to use the Advise feature in a stepwise approach where one increases the free parameter from P to PI, to PID, and then to PIDF. This can save time because it prevents optimizing into local minima. Also it can be quite illustrative which of the feedback parameters leads to which effect in the feedback behavior.

Note

The low-pass filter in the differential part is implemented as an exponential moving average filter described by $y_t = (1 - \alpha) y_{t-1} + \alpha x_t$ with $\alpha = 2^{-\mathrm{dshift}}$. The default value for dshift is 0, i.e. no averaging or unity filter transfer function. On the UI the filter properties can be changed in units of bandwidth or a time constant.

In case the feedback output is a voltage applied to sensitive external equipment it is highly recommended to make use of the center value and the upper and lower limit values. This will guarantee that the output stays in the defined range even when the lock fails and the integrator goes into saturation.

4.18.3. Functional Elements

Table 4.52. PID tab

Control/Tool	Option/Range	Description
Enable	ON / OFF	Enable the PID controller
Input	Demodulator X	Select input source of PID controller
	Demodulator Y	
	Demodulator R	
	Demodulator Theta	
	Aux Input	
	Aux Output	
	Arithmetic Unit Cartesian	
	Arithmetic Unit Polar	
Input Channel	index	Select input channel of PID controller.

Control/Tool	Option/Range	Description
Setpoint	numeric value	PID controller setpoint
TC Mode	ON / OFF	Enables time constant representation of PID parameters.
Phase Unwrap	ON / OFF	Enables the phase error unwrapping up to +/-32pi.
Output		Select output of the PID controller
	Sig Out 1/2 Amplitude	Feedback to the main signal output amplitudes
	Oscillator Frequency	Feedback to any of the internal oscillator frequencies
	Demodulator Phase	Feedback to any of the 8 demodulator phase set points
	Aux Output Offset	Feedback to any of the 4 Auxiliary Output's Offset
	Signal Output Offset	Feedback to the main Signal Output offset adjustment
Output Channel	index	Select output channel of PID controller.
Center, Upper, Lower Limit	numeric value	After adding the Center value to the PID output, the signal is clamped to Center + Lower Limit and Center + Upper Limit.
Range	numeric value	Set the range of the PID controller output relative to the center
P (Hz/deg)	numeric value	PID proportional gain P
I (Hz/deg/s)	numeric value	PID integral gain I
D (Hz/deg*s)	numeric value	PID derivative gain D
D Limit TC/BW 3 dB	102 ns to 2.33 ms/68.3 Hz to 1.56 MHz	The cutoff of the low-pass filter for the D limitation, shown as either the filter time constant or the 3 dB cutoff frequency, depending on the selected TC mode. When set to 0, the low-pass filter is disabled.
Rate	109.9 kHz to 14 MHz	PID sampling rate and update rate of PID outputs. Needs to be set substantially higher than the targeted loop filter bandwidth. Note: The numerical precision of the controller is influenced by the loop filter sampling rate. If the target bandwidth is

Control/Tool	Option/Range	Description
		below 1 kHz is starts to make sense to adjust this rate to a value of about 100 to 500 times the target bandwidth. If the rate is set to high for low bandwidth applications, integration inaccuracies can lead to non linear behavior.
Error	numeric value	Error = Set point - PID Input
Shift	numeric value	Difference between the current output value Out and the Center. Shift = P*Error + I*Int(Error, dt) + D*dError/dt
Out	numeric value	Current output value
Tune	Tune	Optimize the PID parameters so that the noise of the closed-loop system gets minimized. The tuning method needs a proper starting point for optimization (away from the limits). The tuning process can be interrupted and restarted. The tuning will try to match the PID bandwidth with the loop bandwidth of the DUT, signal input (demodulator), and signal output.
Max Rate (Sa/s)	1 to 14 MSa/s	Target Rate for PID output data sent to PC. This value defines the applied decimation for sending data to the PC. It does not affect the Aux Output.
Decimation	Integer value, ideally 0	Decimation factor applied to ensure a sampling rate smaller than the Max Rate set.
To Advisor	To Advisor	Copy the current PID settings to the PID Advisor.
Advanced	ON / OFF	Enables manual selection of display and advice properties. If disabled the display and advise settings are automatically with optimized default values.
Display		Select the display mode used for rendering the system frequency or time response.
	Bode Magnitude	Display the Bode magnitude plot.
	Bode Phase	Display the Bode phase plot.
	Step Resp	Display the step response plot.

Control/Tool	Option/Range	Description
Start (Hz)	numeric value	Start frequency for Bode plot display. For disabled advanced mode the start value is automatically derived from the system properties and the input field is read-only.
Stop (Hz)	numeric value	Stop frequency for Bode plot display. For disabled advanced mode the stop value is automatically derived from the system properties and the input field is read-only.
Start (s)	numeric value	Start time for step response display. For disabled advanced mode the start value is zero and the field is read-only.
Stop (s)	numeric value	Stop time for step response display. For disabled advanced mode the stop value is automatically derived from the system properties and the input field is read-only.
Response In		Start point for the plant response simulation for open or closed loops. In closed loop configuration all elements from output to input will be included as feedback elements.
	Demod Input	Start point is at the demodulator input.
	Setpoint	Start point is at the setpoint in front of the PID.
	PID Output	Start point is at PID output.
	Instrument Output	Start point is at the instrument output.
	DUT Output	Start point is at the DUT output and instrument input.
Response Out		End point for the plant response simulation for open or closed loops. In closed loop configuration all elements from output to input will be included as feedback elements.
	PID Output	End point is at PID output.
	Instrument Output	End point is at the instrument output.
	DUT Output	End point is at the DUT output and instrument input.

Control/Tool	Option/Range	Description
	Demod Input	End point is at the demodulator input.
	PID Error	End point is at the PID error calculation of the PID.
Closed-Loop	ON / OFF	Switch the display of the system response between closed or open loop.
Target BW (Hz)	numeric value	Target bandwidth for the closed loop feedback system which is used for the advising of the PID parameters. This bandwidth defines the trade-off between PID speed and noise.
Auto Bandwidth	ON / OFF	Adjusts the demodulator bandwidth to fit best to the specified target bandwidth of the full system. If disabled, a demodulator bandwidth too close to the target bandwidth may cause overshoot and instability. In special cases the demodulator bandwidth can also be selected smaller than the target bandwidth.
Advise Mode		Select the PID coefficients that are optimized. The other PID coefficients remain unchanged but are used during optimization. This allows to force coefficients to a value while optimizing the rest. The advise time will increase significantly with the number of parameters optimized.
	P	Only optimize the proportional gain.
	I	Only optimize the integral gain.
	PI	Only optimize the proportional and the integral gain.
	PID	Optimize the proportional, integral, and derivative gains.
	PIDF	Optimize the proportional, integral, and derivative gains. Also the derivative gain bandwidth will be optimized.

Control/Tool	Option/Range	Description
Advise	Advise	Calculate the PID coefficients based on the used DUT model and the given target bandwidth. If optimized values can be found the coefficients are updated and the response curve is updated on the plot. Only PID coefficients specified with the advise mode are optimized. The Advise mode can be used incremental, means current coefficients are used as starting point for the optimization unless other model parameters are changed in-between.
P (Hz/deg)	numeric value	Proportional gain P coefficient used for calculation of the response of the PID model. The parameter can be optimized with PID advise or changed manually. The parameter only gets active on the PID after pressing the button To PLL.
I (Hz/deg/s)	numeric value	Integral gain I coefficient used for calculation of the response of the PID model. The parameter can be optimized with PID advise or changed manually. The parameter only gets active on the PID after pressing the button To PLL.
D (Hz/deg*s)	numeric value	Derivative gain D coefficient used for calculation of the response of the PID model. The parameter can be optimized with PID advise or changed manually. The parameter only gets active on the PID after pressing the button To PLL.
D Limit TC/BW 3 dB	numeric value	The cutoff of the low-pass filter for the D limitation, shown as either the filter time constant or the 3 dB cutoff frequency, depending on the selected TC mode. When set to 0, the low-pass filter is disabled.
Rate	109.9 kHz to 14 MHz	PID sampling rate used for simulation.

Control/Tool	Option/Range	Description
		The advisor will update the rate to match with the specified target bandwidth. A sampling rate close to the target bandwidth and excessive higher bandwidth will results in a simulation mismatch.
BW (Hz)	numeric value	Simulated bandwidth of the full close loop model with the current PID settings. This value should be larger than the target bandwidth.
Target BW LED	green/red	Green indicates that the target bandwidth can be achieved. For very high PID bandwidth the target bandwidth might be only achieved using marginal stable PID settings. In this case, try to lower the bandwidth or optimize the loop delays of the PID system.
PM (deg)	numeric value	Simulated phase margin of the PID with the current settings. The phase margin should be greater than 45 deg for internal PLL and 60 deg for all other DUT for stable conditions. An Infinite value is shown if no unity gain crossing is available to determine a phase margin.
Stable LED	green/red	Green indicates that the phase margin is fulfilled and the PID system should be stable.
To PID	To PID	Copy the PID Advisor settings to the PID.
DUT Model		Type of model used for the external device to be controlled by the PID. A detailed description of the transfer function for each model is found in the previous section.
	All Pass	The external device is modeled by an all pass filter. Parameters to be configured are delay and gain.
	LP 1st	The external device is modeled by a first-order low-pass filter. Parameters to be configured

Control/Tool	Option/Range	Description
		are delay, gain and filter bandwidth.
	LP 2nd	The external device is modeled by a second-order low-pass filter. Parameters to be configured are delay, gain, resonance frequency and damping ratio.
	Resonator Frequency	The external device is modeled by a resonator. Parameters to be configured are delay, center frequency and quality factor.
	Internal PLL	The DUT is the internal oscillator locked to an external signal through a phase-locked loop. The parameter to be configured is the delay.
	VCO	The external device is modeled by a voltage controlled oscillator. Parameters to be configured are delay, gain and bandwidth.
	Resonator Amplitude	The external device is modeled by a resonator. Parameters to be configured are delay, gain, center frequency and quality factor.
Delay	numeric value	Parameter that determines the earliest response for a step change. This parameter does not affect the shape of the DUT transfer function.
Gain	numeric value	Parameter that determines the gain of the DUT transfer function.
BW (Hz)	numeric value	Parameter that determines the bandwidth of the first-order low-pass filter respectively the bandwidth of the VCO.
Damping Ratio	numeric value	Parameter that determines the damping ratio of the second-order low-pass filter.
Center Frequency	numeric value	Parameter that determines the resonance frequency of the of the modeled resonator.
Q	numeric value	Parameter that determines the quality factor of the modeled resonator.

4.19. MOD Tab

The MOD tab provides access to the settings of the amplitude and frequency modulation units. This tab is only available when the UHF-MOD AM/FM Modulation option is installed on the Instrument (see Information section in the Device tab).

Note

The UHF-MOD AM/FM Modulation option requires the UHF-MF Multi-frequency option.

4.19.1. Features

- Phase coherently add and subtract oscillator frequencies and their multiples
- Control for AM and FM demodulation
- Control for AM and narrow-band FM generation
- Direct analysis of higher order carrier frequencies and sidebands

4.19.2. Description

The MOD tab offers control in order to phase coherently add and subtract the frequencies of multiple numerical oscillators. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.53. App icon and short description

Control/Tool	Option/Range	Description
MOD		Control panel to enable (de)modulation at linear combinations of oscillator frequencies.

The MOD tab (see Figure 4.38) is divided into two horizontal sections, one for each modulation unit.

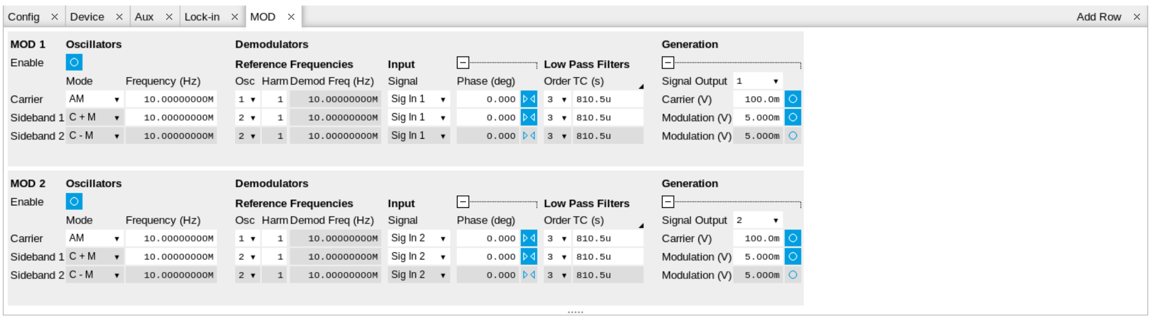


Figure 4.38. LabOne UI: MOD tab

The modulation units are designed for experiments involving multiple frequencies. For many of such experiments the associated spectrum reveals a dominant center frequency, often called the carrier, and one or multiple sidebands symmetrically placed around the carrier. Typical examples are amplitude modulated (AM) signals with one carrier and two sidebands separated from the carrier by the AM modulation frequency. Another example is frequency modulation (FM) where multiple sidebands to the left and right of the carrier can appear. The relative amplitude of the sideband for both AM and FM depends on the modulation depth, which is often expressed by the modulation index.

The classical approach of analyzing such signals (in particular when only analog instruments are available) is to use a configuration called tandem demodulation. This is essentially the serial cascading of lock-in amplifiers. The first device is referenced to the carrier frequency and outputs the in-phase component. This is then fed into the subsequent lock-in amplifiers in order to extract the different sideband components. There are several downsides to this scheme:

- The quadrature component of the first lock-in tuned to the carrier has to be continuously zeroed out by adjusting the reference phase. Otherwise a serious part of the signal power is lost for the analysis which usually leads to a drop in SNR.
- The scheme scales badly in terms of the hardware resources needed, in particular if multiple sideband frequencies need to be extracted.
- Every time a signal enters or exits an instrument the SNR gets smaller (e.g. due to the instrument inputs noise). Multiple such steps can deteriorate signal quality significantly.

All these shortcomings are nicely overcome by providing the ability to generate linear combinations of oscillator frequencies and use these combinations as demodulation references.

The MOD tab contains two sections MOD 1 and MOD 2. Both are identical in all aspects except that MOD 1 is linked to demodulators 1,2 and 3, whereas MOD 2 is linked to demodulators 5, 6, and 7. Each of the MOD units can make use of up to 3 oscillators, which can be even referenced to an external source by using ExtRef or a PLL. Figure 4.39 gives an overview of the different components involved and their interconnections.

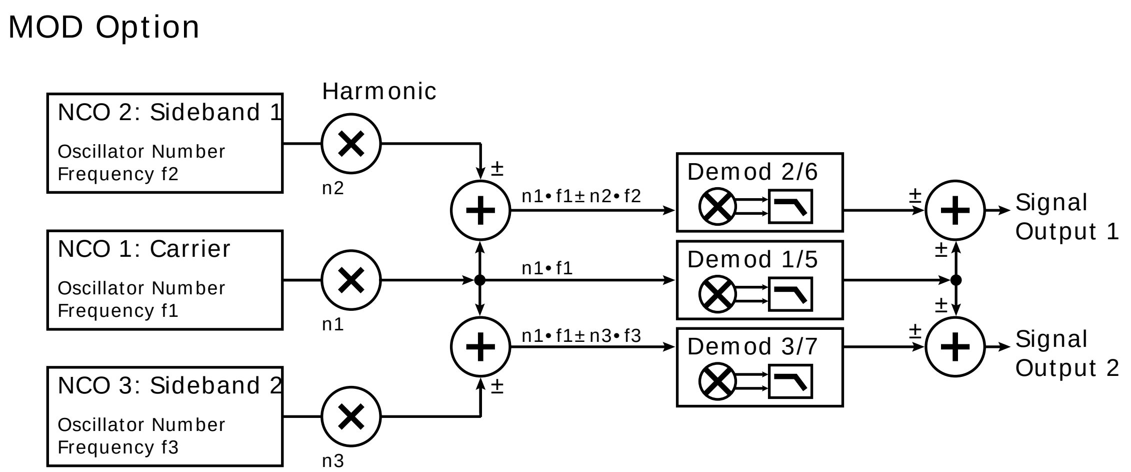


Figure 4.39. Modulation Option block diagram

For convenience the UI provides access to presets for AM and FM in the Mode column. In the Manual Mode all settings can be chosen freely. When there are more than three frequencies present on a single signal one can even associate both sections MOD 1 and 2 to the same Signal Input.

Note

Whenever MOD 1 or 2 is enabled, all the settings in the Lock-in tab that are controlled by the MOD Option will be set to read-only.

On top of signal analysis the MOD option can also be utilized for signal generation. The Generation section provides all the necessary controls to adjust the carrier and sideband amplitudes.

Note

FM signals are generated by coherent superposition of the carrier signal with two sideband frequencies on either side that have the same amplitudes but opposite phases. The phase shift is achieved by using negative amplitudes as displayed in the Lock-in tab. This FM generation method approximates true FM as long as the modulation index is well below 1, i.e. higher-order sidebands can be neglected. For a modulation index of 1 true FM provides more than 13% of signal power in the second and higher order sidebands.

More details regarding AM and FM signal analysis and generation can be found on the Zurich Instruments web page, e.g. http://www.zhinst.com/blogs/sadik/2014/02/sideband-analysis/.

4.19.3. Functional Elements

Table 4.54. MOD tab

Control/Tool	Option/Range	Description
Enable	ON / OFF	Enable the modulation
Mode	AM/FM/manual	Select the modulation mode.
Mode		Enabling of the first sideband and selection of the position of the sideband relative to the carrier frequency for manual mode.
	Off	First sideband is disabled. The sideband demodulator behaves like a normal demodulator.
	C + M	First sideband to the right of the carrier
	C - M	First sideband to the left of the carrier
Mode		Enabling of the second sideband and selection of the position of the sideband relative to the carrier frequency for manual mode.
	Off	Second sideband is disabled. The sideband demodulator behaves like a normal demodulator.
	C + M	Second sideband to the right of the carrier
	C - M	Second sideband to the left of the carrier
Frequency (Hz)	0 to 600 MHz	Sets the frequency of the carrier.

Control/Tool	Option/Range	Description
Frequency (Hz)	0 to 600 MHz	Frequency offset to the carrier from the first sideband.
Frequency (Hz)	0 to 600 MHz	Frequency offset to the carrier from the second sideband.
Carrier	oscillator index	Select the oscillator for the carrier signal.
Sideband 1	oscillator index	Select the oscillator for the first sideband.
Sideband 2	oscillator index	Select the oscillator for the second sideband.
Harm	1 to 1023	Set harmonic of the carrier frequency. 1=Fundamental
Harm	1 to 1023	Set harmonic of the first sideband frequency. 1 = fundamental
Harm	1 to 1023	Set harmonic of the second sideband frequency. 1 = fundamental
Demod Freq (Hz)	0 to 600 MHz	Carrier frequency used for the demodulation and signal generation on the carrier demodulator.
Demod Freq (Hz)	0 to 600 MHz	Absolute frequency used for demodulation and signal generation on the first sideband demodulator.
Demod Freq (Hz)	0 to 600 MHz	Absolute frequency used for demodulation and signal generation on the second sideband demodulator.
Channel	Signal Inputs, Trigger Inputs, Auxiliary Inputs, Auxiliary Outputs, Phase Demod 4, Phase Demod 8	Select Signal Input for the carrier demodulation
Channel	Signal Inputs, Trigger Inputs, Auxiliary Inputs, Auxiliary Outputs, Phase Demod 4, Phase Demod 8	Select Signal Input for the sideband demodulation
Phase	-180° to 180°	Phase shift applied to the reference input of the carrier demodulator and also to the carrier signal on the Signal Outputs
Phase	-180° to 180°	Phase shift applied to the reference input of the sideband demodulator and also to the sideband signal on the Signal Outputs

Control/Tool	Option/Range	Description
Zero		Adjust the carrier demodulator phase automatically in order to read zero degrees. Shifts the phase of the reference at the input of the carrier demodulator in order to achieve zero phase at the demodulator output. This action maximizes the X output, zeros the Y output, zeros the Θ output, and leaves the R output unchanged.
Zero		Adjust the sideband demodulator phase automatically in order to read zero degrees. Shifts the phase of the reference at the input of the sideband demodulator in order to achieve zero phase at the demodulator output. This action maximizes the X output, zeros the Y output, zeros the Θ output, and leaves the R output unchanged.
Order	1 to 8	Filter order used for carrier demodulation
Order	1 to 8	Filter order used for sideband demodulation
TC/BW Value	numeric value	Defines the low-pass filter characteristic in the unit defined above for the carrier demodulation
TC/BW Value	numeric value	Defines the low-pass filter characteristic in the unit defined above for the sideband demodulation
Signal Output	1, 2 or both	Select Signal Output 1, 2 or none
Carrier (V)	-range to range	Set the carrier amplitude
Modulation (V)	-range to range	Set the amplitude of the first sideband component.
Modulation (V)	-range to range	Set the amplitude of the second sideband component.
Index	-range to range	In FM mode, set modulation index value. The modulation index equals peak deviation divided by modulation frequency.

Control/Tool	Option/Range	Description
Peak Dev (Hz)	-range to range	In FM mode, set peak deviation value.
Enable FM Peak Mode	ON / OFF	In FM mode, choose to work with either modulation index or peak deviation. The modulation index equals peak deviation divided by modulation frequency.
Enable	ON / OFF	Enable the signal generation for the first sideband
Enable	ON / OFF	Enable the signal generation for the second sideband
Enable	ON / OFF	Enable the carrier signal

4.20. Boxcar Tab

The Boxcar tab relates to the UHF-BOX Boxcar Averager option and is only available if this option is installed on the UHF Instrument (see Information section in the Device tab).

4.20.1. Features

- 2 equivalent boxcar units with up to 450 MHz repetition rate
- Baseline suppression for each Boxcar unit
- up to 450 MHz repetition rate
- Dead time free operation for frequencies below 450 MHz
- Period waveform analyzer (PWA) allows display of waveform and convenient graphical setting of Boxcar averaging windows
- PWA frequency domain view allows for simultaneous analysis of up to 512 harmonics of the reference frequency

4.20.2. Description

The Boxcar tab provides access to the gated averaging functionality of the UHF Instrument. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.55. App icon and short description

Control/Tool	Option/Range	Description
Boxcar		Boxcar settings and periodic waveform analyzer for fast input signals.

Each Boxcar unit is shown in a separate sub-tab (see Figure 4.40) that consists of a plot area and three control tabs on the right-hand side.

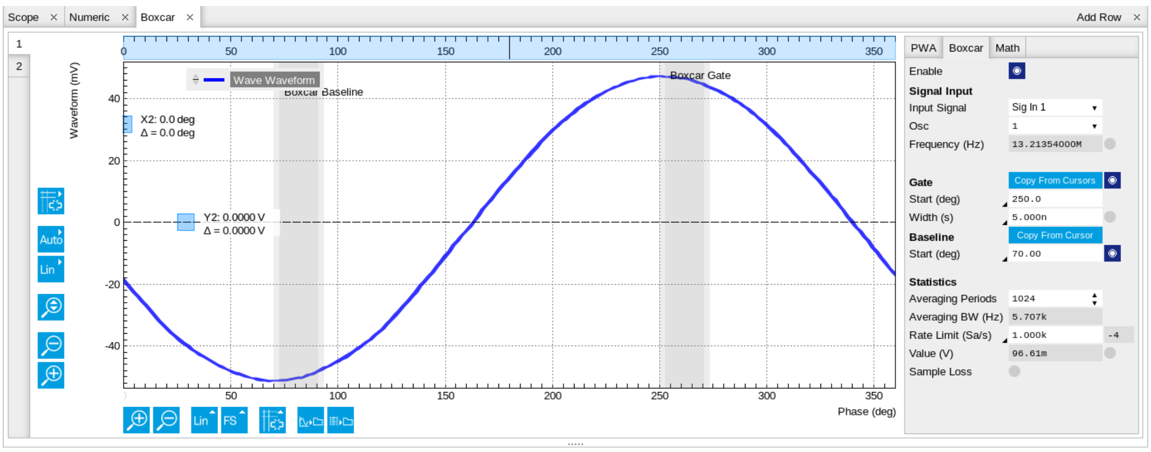


Figure 4.40. LabOne UI: Boxcar tab

Similar to the lock-in amplifier functionality the Boxcar offers a large reduction of the incoming signal bandwidth sampled with 1.8 GSa/s to a regime where much lower sampling rates suffice that can easily be transferred to a PC over USB or Ethernet cable for further analysis and post processing. For both methods ideally no piece of signal information is lost during the data reduction but huge parts if the initial signal are discarded that contain no or a negligible piece of relevant information. The operation of the lock-in amplifier can most easily be understood considering the inputs signal in the frequency domain where the lock-in acts as a sophisticated bandpass filter with adjustable center frequency and bandwidth (if we generously ignore phase sensitivity here for the sake of simplicity). In contrast, the Boxcar does a very similar thing in the time domain where it allows to cut out only the signal components that contain information. A very common use case are experiments with pulsed lasers. In particular when duty cycles are low, the fraction of the time domain signal where there is actual information can be quite small and so the idea is to record only the parts when for instance the laser is on.

In classical analog instruments this is typically realized by a switch, that can by triggered externally, and a subsequent integrator. Most often the trigger functionality also allows to configure a time delay and a certain window for as long as the switch shall open up for each trigger and the signal will be integrated. The signal output from the integrator is then passed through an adjustable low-pass filter for further noise reduction.

One of the biggest limitations of analog boxcar instruments is their trigger re-arm time (caused by the finite time required to erase the integrator) which is usually several 10 ms long. During that time no signals can be acquired. For periodic signals this means a limitation to frequencies of a few 10 kHz when signal loss cannot be afforded, measurement time needs to be minimized while high SNR is crucial.

Note

The Zurich Instruments Boxcar uses a synchronous detection approach instead of the traditional triggering method described above. A reference frequency has to be provided - either from external or an internal oscillator can be used - instead of a trigger signal and the Boxcar window is defined in terms of the phases of that reference frequency.

Note

Using a synchronous detection scheme in combination with a fixed input sampling rate of 1.8 GSa/s excludes all commensurate signal frequencies from proper analysis. The UI provides warnings whenever the reference frequency is anywhere close to any of these. Potential issues can be easily quantified by displaying the bin counts in the PWA sub-tab.

Figure 4.41 shows a detailed block diagram how signal processing is performed.

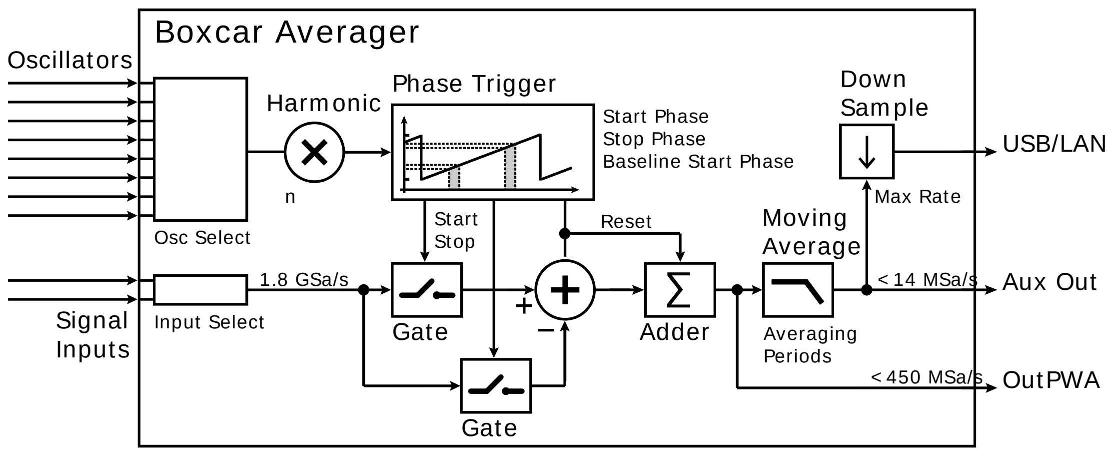


Figure 4.41. Boxcar averager block diagram

The input signal is sampled at a rate of 1.8 GSa/s. Depending on the phase of the reference oscillator and the set Start Phase and Window Width each of these samples is added up and output from the Adder after each period. From there one branch is directly connected to the outPWA (see Section 4.21) for a further step of synchronous detection. The other signal path way is subject to a Moving Average filter that allows to average over an adjustable number of reference oscillator periods.

Note

The moving average filter provides up to 512 intermittent results. That means if Averaging Periods is set to 1024 the Output is updated with a new value every second oscillator period whereas for smaller numbers of averaging Periods this update is performed on every cycle.

Another big advantage of the Zurich Instruments Boxcar is the graphical display of the input signal termed Periodic Waveform Analyzer. Each Boxcar unit is equipped with a PWA unit that can be either bound to the Boxcar settings or used on any other signal input and oscillator independently. Figure 4.42 shows a block diagram of the PWA.

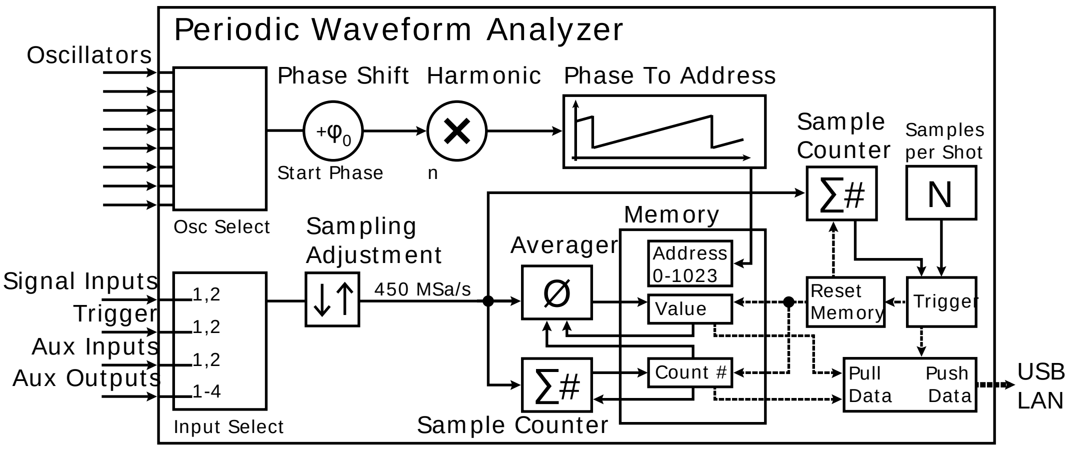


Figure 4.42. Periodic Waveform Analyzer block diagram

The user can select from a variety of different input signals, all of which will be re-sampled - either up or down, where no averaging is provided - at the input to a sampling rate of 450 MSa/s. Depending on the phase of the reference oscillator each data sample is associated to one of 1024 memory units which records the average values and the number of samples. These 1024 can be spread over the entire 360 degree of the reference oscillator period or a smaller span by using the Zoom mode. After an adjustable number of total input samples the entire memory is transferred to the PC and the memory is reset.

Each shot of data contains 1024 average values and sample counts each associated to a certain phase window. In case the reference frequency is sufficiently stable over the course of one shot it makes perfect sense to switch from the phase domain view to the time domain, which for some experiments might be the more natural way of consideration.

4.20.3. Functional Elements

Table 4.56. Boxcar tab: PWA sub-tab

Control/Tool	Option/Range	Description
Run/Stop	Run/Stop	Continuously run and stop PWA acquisition.
Single	Single	Single acquisition of a PWA data set.
Input Signal	Signal Inputs, Trigger Inputs, Auxiliary Inputs, Auxiliary Outputs, Phase Demod 4, Phase Demod 8	Select PWA input signal.
Input Interlock	ON / OFF	Interlock PWA and Boxcar Input settings
Osc	oscillator index	Select reference oscillator for PWA signal acquisition.
PWA Frequency	numeric value	Actual frequency at which the PWA operates based on set oscillator frequency and harmonic scaling factor.
Commensurability	grey/red	Traffic light showing whether the number of samples acquired is evenly distributed over all bins.
Mode	Phase	Measurement data can be interpreted in four different modes and displayed over either phase (native), time, frequency (FFT) or harmonics of the base frequency (FFT).
	Time	
	Freq Domain (FFT)	
	Harmonics (FFT)	
Copy from range	Copy From Range	Change PWA start and span according to plot range.
Reset	R	Reset the start and width value to show the full 360 deg.
Start	numeric value	Defines the start of PWA range in time or phase.
Width	numeric value	Defines width of PWA range in time or phase.
Samples	1 to 2^47	Defines the number of samples acquired of each PWA data set (450 MSa/s).
Acq Time (s)	numeric value	Estimated time needed for recording of the specified number of samples.
Overflow	grey/red	Indicates whether the number of samples collected per bin or the amplitude exceeds the numerical limit. Reduce

Control/Tool	Option/Range	Description
		number of samples and/or change frequency.
Infinite Acq Time	string	The signal source of this unit (Boxcar) is not producing any data. Once it is configured and enabled, this field will indicate the duration of a single measurement.
Progress (%)	0 to 100%	Show state of the PWA acquisition in percent.
Resolution	numeric value	FFT resolution (bin width) in Hz.
Max Harmonics	numeric value	Maximum number of displayed harmonics.
Signal	Waveform	Select signal to be displayed.
	Count	

Table 4.57. Boxcar tab: Boxcar sub-tab

Control/Tool	Option/Range	Description
Enable	ON / OFF	Enable the BOXCAR unit
Input Signal	1/2	Select Signal Input used for the boxcar analysis.
Osc	oscillator index	Selection of the oscillator used for the boxcar analysis
Frequency (Hz)	frequency value	Oscillator frequency used for the boxcar analysis.
Too high frequency	grey/red	Frequency for the boxcar is above or equal 450 MHz. Sticky flag cleared by restarting the boxcar. The boxcar output may not be reliable any more.
Copy from cursors	Copy From Cursors	Take cursor values to define Window Start and Window span values.
Show Gate Opening	ON / OFF	Show gate opening on the PWA plot.
Start Mode		Selects the mode to specify the start of the boxcar averaging gate opening. The phase (deg) is the native mode for the device.
	Start (deg)	Native definition of the boxcar averaging gate start as phase.
	Start (s)	Definition of the boxcar averaging gate start as time. Due to the conversion to

Control/Tool	Option/Range	Description
		phase on the device a small uncertainty window exists.
Start (deg)	0 to 360	Boxcar averaging gate opening start in degrees. It can be converted to time assuming 360 equals to a full period of the driving oscillator.
Start Time (s)	0 to period	Boxcar averaging gate opening start in seconds based on one oscillator frequency period equals 360 degrees. Boxcar must be disabled to edit input field.
Width Mode		Selects the mode to specify the width of the boxcar averaging gate opening. The time (s) is the native mode for the device.
	Width (deg)	Definition of the averaging gate width as phase.
	Width (s)	Native definition of the averaging gate width as time.
	Width (pts)	Definition of the averaging gate width in samples.
Width (deg)	0 to 360	Boxcar averaging gate opening width in degrees based on one oscillator frequency period equals 360 degrees. Boxcar must be disabled to edit input field.
Width (s)	555 ps to period	Boxcar averaging gate opening width in seconds. It can be converted to phase assuming 360 equals to a full period of the driving oscillator.
Width (pts)	Integer value	Boxcar averaging gate opening width in samples at 1.8 GHz rate.
Too large gate width	grey/red	Boxcar averaging gate opening width is more than one cycle of the signal and should be reduced.
Copy from cursor	Copy from cursor	Take cursor value to define Baseline Start value.
Start Mode		Selects the mode to specify the start of the boxcar baseline suppression gate opening. The phase (deg) is the native mode for the device.

Control/Tool	Option/Range	Description
	Start (deg)	Native definition of the boxcar baseline suppression gate start as phase.
	Start (s)	Definition of the boxcar baseline suppression gate start as time.
	Offset (deg)	Definition of the boxcar baseline suppression gate start relative to the gate opening start as phase.
	Offset (s)	Definition of the boxcar baseline suppression gate start relative to the gate opening start as time.
Start (deg)	0 to 360	Boxcar baseline suppression gate opening start in degrees based on one oscillator frequency period equals 360 degrees.
Start (s)	0 to period	Boxcar baseline suppression gate opening start in seconds based on one oscillator frequency period equals 360 degrees.
Start (deg)	0 to 360	Boxcar baseline suppression gate opening start in degrees relative to Gate Start.
Start (s)	0 to period	Boxcar baseline suppression gate opening start in seconds relative to Gate Start.
Enable	ON / OFF	Enable Baseline Suppression
Averaging Periods	1 to 2^20	Number of periods to average. The output will be refreshed up to 512 times during the specified number of periods. This setting has no effect on Output PWAs.
Averaging BW	10 μHz to 7 MHz	The 3 dB signal bandwidth of the Boxcar Averager is determined by the oscillation frequency and the Number of Averaging Periods set. Note: internally the boxcar signal is sampled at a rate of 14 MSa/s and the signal bandwidth of the auxiliary output is 7 MHz.
Rate Limit (Sa/s)	1 to 14.06 MSa/s	Rate Limit for Boxcar output data sent to PC. This value does not affect the Aux Output for which the effective rate is given by min(14 MSa/s ,

Control/Tool	Option/Range	Description
		Frequency / max(1, Averaging Periods/512)).
Rate (Sa/s)	1 to 14.06 MSa/s	Display of the currently effective rate used for data transfer to the PC given by min(14 MSa/s , Frequency / max(1, Averaging Periods/512)). This value is read-only.
Rate Limit (Sa/s) or Rate (Sa/s)		Switches between display of Rate Limit or Rate
	Rate Limit (Sa/s)	Display of the Rate Limit which defines the maximal transfer rate.
	Rate (Sa/s)	Display of the currently active transfer rate.
Oversampling	Integer value, ideally 0	Indicates, in powers of 2, the number of averager outputs sent to the PC while Averaging Periods Boxcar integrations are obtained. Positive integer values indicate oversampling. Negative integer values indicate undersampling. Examples for oversampling values: 0 : 2^0 = 1 averager output is sent to the PC during Averaging Periods Boxcar integrations. 2 : 2^2 = 4 averager outputs are sent to the PC during Averaging Periods Boxcar integrations. -1 : 2^-1 = 0.5, only every other Averaging Periods Boxcar integrations an averager output is sent to the PC.
Value	numeric value	The current boxcar output.
Value Overflow flag	grey/red	Overflow detected. Sticky flag cleared by restarting the boxcar. The boxcar output may not be reliable any more.
Sample Loss	grey/red	Data lost during streaming to PC. Sticky flag cleared by restarting the boxcar.

For the Math sub-tab please see Table 4.7 in the section called “Cursors and Math”.

4.21. Out PWA Tab

The Out PWA tab relates to the UHF-BOX Boxcar Averager option and is only available if this option is installed on the UHF Instrument (see Information section in the Device tab).

4.21.1. Features

- Period waveform analyzer for boxcar output samples (multi-channel boxcar, deconvolution boxcar)
- Support signals derived from asynchronous optical sampling

4.21.2. Description

The Out PWA tab provides access to the period waveform analyzer that acts on boxcar output samples. This feature is also called multi-channel boxcar or deconvolution boxcar. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.58. App icon and short description

Control/Tool	Option/Range	Description
Out PWA		Multi-channel boxcar settings and measurement analysis for boxcar outputs.

The Out PWA tab (see Figure 4.43) consists of a display section on the left and a configuration section on the right. The configuration section is further divided into a number of sub-tabs.

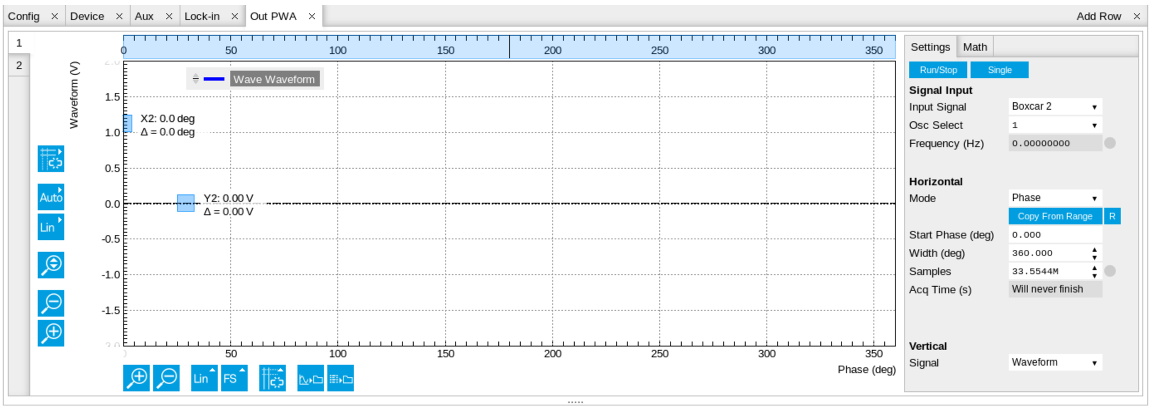


Figure 4.43. LabOne UI: Out PWA tab

Note

The Out PWA works analogously to the PWA supplied in the Boxcar tabs (see Figure 4.42) except that its inputs are limited to the output of the two Boxcar units. It is important to understand that the Boxcar results are directly connected to the input of the Out PWA, in particular that there is no averaging or downsampling applied in between.

4.21.3. Functional Elements

Table 4.59. Out PWA tab: Settings sub-tab

Control/Tool	Option/Range	Description
Run/Stop	Run/Stop	Continuously run and stop PWA acquisition.
Single	Single	Single acquisition of a PWA data set.
Input Signal	Boxcar 1	Select PWA input signal.
	Boxcar 2	
Osc Select	oscillator index	Select reference oscillator for PWA signal acquisition.
Frequency	numeric value	Actual frequency at which the PWA operates based on set oscillator frequency and harmonic scaling factor.
Commensurability	grey/red	Traffic light showing whether the number of samples acquired is evenly distributed over all bins.
Mode	Phase	Measurement data can be interpreted in four different modes and displayed over either phase (native), time, frequency (FFT) or harmonics of the base frequency (FFT).
	Time	
	Freq Domain (FFT)	
	Harmonics (FFT)	
Copy from range	Copy From Range	Change PWA start and span according to plot range.
Reset	R	Reset the start and width value to show the full 360 deg.
Start	numeric value	Defines the start of PWA range in time or phase.
Width	numeric value	Defines width of PWA range in time or phase.
Samples	1 to 2^47	Defines the number of samples acquired of each PWA data set (450 MSa/s).
Acq Time (s)	numeric value	Estimated time needed for recording of the specified number of samples.
Overflow	grey/red	Indicates whether the number of samples collected per bin or the amplitude exceeds the numerical limit. Reduce number of samples and/or change frequency.
Infinite Acq Time	string	The signal source of this unit (Boxcar) is not producing any data. Once it is configured

Control/Tool	Option/Range	Description
		and enabled, this field will indicate the duration of a single measurement.
Progress (%)	0 to 100%	Show state of the PWA acquisition in percent.
Resolution	numeric value	FFT resolution (bin width) in Hz.
Max Harmonics	numeric value	Maximum number of displayed harmonics.
Signal	Waveform	Select signal to be displayed.
	Count	

For the Math sub-tab please see Table 4.7 in the section called “Cursors and Math”.

4.22. AWG Tab

The AWG tab is available on UHF Arbitrary Waveform Generator instruments and on UHFLI Lock-in Amplifier instruments with installed UHF-AWG Arbitrary Waveform Generator option (see Information section in the Device tab).

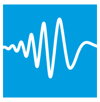
4.22.1. Features

- Dual-channel arbitrary waveform generator
- 128 MSa waveform memory per channel
- Sequence branching
- Direct mode and amplitude modulation
- Cross-trigger engine
- Sequence Editor with code highlighting and auto completion
- High-level programming language with waveform generation and editing toolset

4.22.2. Description

The AWG tab gives access to the arbitrary waveform generator functionality. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.60. App icon and short description

Control/Tool	Option/Range	Description
AWG		Generate arbitrary signals using sequencing and sample-by-sample definition of waveforms.

The AWG tab (see Figure 4.44) consists of a settings section on the right side and the Sequence Editor on the left side. The settings section is further divided into Control, Waveform, Trigger, and Advanced sub-tabs. The **Sequence Editor** is used for displaying, editing and compiling a LabOne sequence program. The sequence program defines which waveforms are played and in which order. The Sequence Editor is the main tool for operating the AWG.

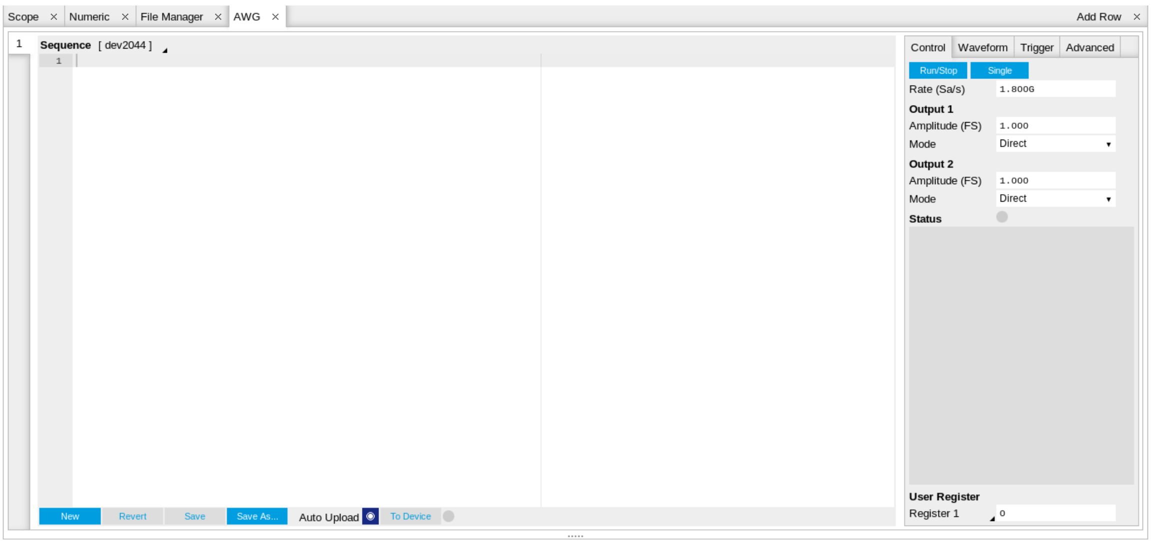


Figure 4.44. LabOne UI: AWG tab

A number of sequence programming examples can be found in Section 3.8. The LabOne sequence programming language is specified in detail in Section 4.22.3. The language comes with a number of predefined waveforms, such as Gaussian, Blackman, sine, or square functions. By combining those predefined waveforms using the waveform editing tools (add, multiply, cut, concatenate, etc), signals with a high level of complexity can be generated directly from the Sequence Editor window. Sample-by-sample definition of the output signal is possible by using comma-separated value (CSV) files specified by the user, see Section 3.8.3 for an example.

The UHF AWG features a compiler, which translates the high-level sequence program into machine instructions and waveform data to be stored in the instrument memory as shown in Figure 4.45. The sequence program is written using high-level control structures and syntax that are inspired by human language, whereas machine instructions reflect exactly what happens on the hardware level. Writing the sequence program using a high-level language represents a more natural and efficient way of working in comparison to writing lists of machine instructions, which is the traditional way of programming AWGs. Concretely, the improvements rely on features such as:

- combination of waveform generation, editing, and playback sequence in a single script
- easily readable syntax and naming for run-time variables and constants
- optimized waveform memory management, reduced transfers upon waveform changes
- definition of user functions and procedures for advanced structuring
- syntax validation

By design, there is no one-to-one link between the list of statements in the high-level language and the list of instructions executed by the Sequencer. There are cases in which a more detailed understanding of the Sequencer instruction list, and in particular its execution timing, is needed. Typically this is the case when observing delays or other signal timing properties that are unexpected from looking at the high-level script. Often such problems can be solved with a few adjustments to the program. Please see Section 3.8.10 for practical advice.

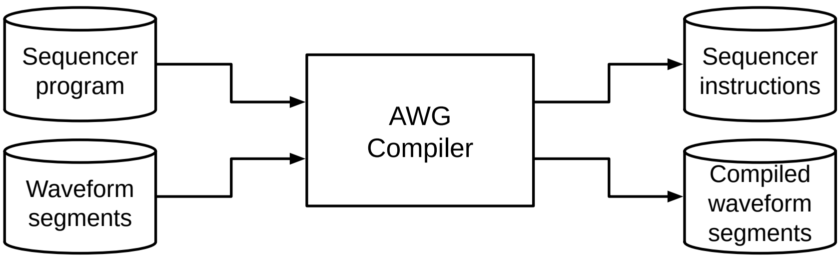


Figure 4.45. AWG sequence program compilation process

The **Sequence Editor** provides the editing, compilation, and transfer functionality for sequence programs. A program typed into the Editor is compiled upon clicking Save. If the compilation is successful and Automatic Upload is enabled, the program including all necessary waveform data is transferred to the device. If the compilation fails, the Status field will display debug messages. Clicking on Save as... allows you to choose a new name for the program. The name of the program that is currently edited is displayed at the top of the editor. External program files as well as waveform data files should be copied to the location shown in Table 4.61 so they become accessible from the user interface. The program name is displayed in a drop-down box. The box allows quick access to all programs in the standard sequence program location. It is possible to quickly switch between programs using the box. Changes made in one program will be preserved when switching to a different program. The file name of a program will be postfixed by an asterisk in case there are unsaved changes in the source file. Note that switching programs in the editor is not sufficient to also update the program in the actual instrument. In order to send a newly selected program to the instrument, the To Device button must be clicked.

Table 4.61. Sequence program and waveform file location

File type	Location
Waveform files (Windows)	C:\Users\<user name>\Documents\Zurich Instruments \LabOne\WebServer\awg\waves
Sequence programs (Windows)	C:\Users\<user name>\Documents\Zurich Instruments \LabOne\WebServer\awg\src
Waveform files (Linux)	~/Zurich Instruments/LabOne/WebServer/awg/waves
Sequence files (Linux)	~/Zurich Instruments/LabOne/WebServer/awg/src

In the **Control sub-tab** the user configures signal parameters and controls the execution of the AWG. The AWG can be started in a continuous mode by clicking on Run/Stop, where the Sequencer will be restarted automatically when its program completes. If Single is clicked, the sequence program will only be allowed to execute once. The continuous mode is a simple way to create an infinite loop, but for better performance it is recommended to specify infinite loops directly in the sequence program.

The Rate field is used to control the default playback sampling rate of the AWG. The default playback rate may be overruled in the sequence program using an optional argument in the waveform playback commands. This is useful when the signal contains both fast and slow components. The two Output sections are used to configure the AWG output mode and signal amplitude. The AWG output channels are not the same as the physical Signal Outputs of the instrument. The AWG output channels are routed to the Signal Outputs of the device. The Amplitude value is a gain parameter, 1.0 by default, that is applied to waveforms on the way from the AWG output channel to the Signal Output. The Amplitude value gives a means to rescale the signal independently of the programmed waveforms. The Output Mode control is used to select between the direct output and amplitude modulation modes. In amplitude modulation mode, the signal of an AWG channel is multiplied with an oscillator signal prior to being sent to the Signal Output. This is useful for the frequent case where the desired signal can be described as a sinusoidal carrier with a shaped envelope. Please read more about use cases, advantages, and practical examples in Section 3.8.5.

The generation of the modulated signal depends on the settings made in the Lock-in tab. Figure 4.46 shows how the signals are routed internally on their way from the oscillators and the AWG to the Signal Outputs. There are two switches in the diagram. The upper switch is related to the AWG Mode selection. In Modulation mode, the signal coming from the AWG unit 1 (2) is multiplied with the oscillator signal of demodulator 4 (8). The phase and harmonic of the oscillator signal can be adjusted in the Lock-in tab. In Direct mode, the AWG signal is multiplied with a constant 1.0, in other words, it remains unchanged. The lower switch is related to the running state of the AWG, i.e, the Run/Stop and Single buttons. When the AWG is idle, the Output Amplitude setting from the Lock-in tab takes the place of the AWG signal. This is the standard configuration for lock-in measurements. It is furthermore useful for defining the voltage appearing on the Signal Output when the AWG is off. The UHF-MF Multi-frequency option provides additional oscillators as well as an Oscillator Select switch matrix at the input of the demodulators, enabling the use of up to 8 independent frequencies for modulation. The Register values may be used as integer variables inside a sequence program, for instance to vary a delay between pulses manually or with the Sweeper .

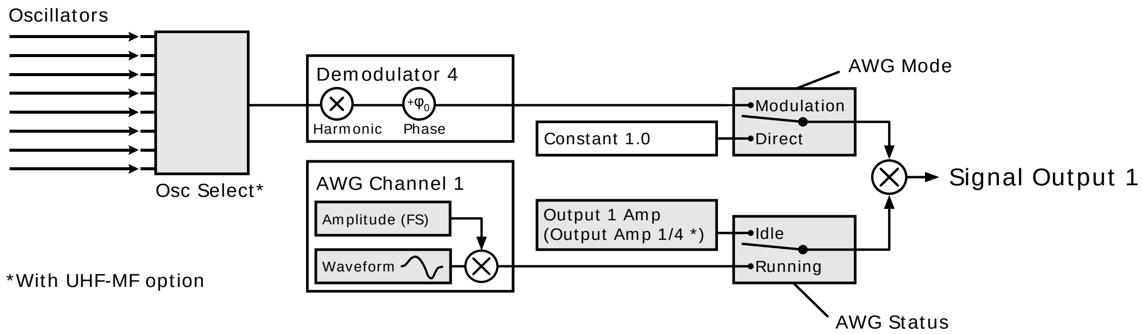


Figure 4.46. Amplitude modulation block diagram for AWG channel 1

The **Waveform sub-tab** displays information about the waveforms that are used by the current sequence program, such as their length and channel number.

On the **Trigger sub-tab** you can configure the Trigger inputs of the AWG and control the Cross-Domain Trigger functionality of the instrument. The AWG has four Trigger input channels which can be configured to probe a variety of signals coming both from internal (e.g. demodulator output data) or external (e.g. Ref/Trigger input) sources. This means that the AWG Trigger input channels are not the same as physical device inputs. Two of the Trigger input channels are called analog (meaning they can accept signals of continuous, analog-like character), and two are called digital (meaning they can accept binary signals). Trigger Level and Hysteresis may be configured for the Analog Triggers, and the user can select between rising and falling edge trigger functionality. The primary use of the triggers is to control the timing of the AWG signal relative to an external device. Another use of triggers is to implement sequence branching. See Section 3.8.4 and Section 3.8.7 for practical examples on how to use the AWG trigger in- and outputs.

The **Advanced sub-tab** displays the compiled list of sequencer instructions and the current state of the sequencer on the instrument. This can help an advanced user in debugging a sequence program and understanding its execution.

4.22.3. LabOne Sequence Programming

A Simple Example

The syntax of the LabOne AWG Sequencer programming language is based on C, but with a few simplifications. Each statement is concluded with a semicolon, several statements can be grouped with curly brackets, and comment lines are identified with a double slash. The following example shows some of the fundamental functionalities: waveform generation, repeated playback, triggering, and single/dual-channel waveform playback. See Section 3.8 for a step-by-step introduction with more examples.

```
// Define an integer constant
const N = 4096;
// Create a Gaussian pulses with length N points,
// amplitude +1.0 (-1.0), center at N/2, and a width of N/8
wave gauss_pos = 1.0 * gauss(N, N/2, N/8);
wave gauss_neg = -1.0 * gauss(N, N/2, N/8);
// execute playback sequence 100 times
repeat (100) {
  // Wait for demod 8 oscillator phase for synchronization
  waitOscPhaseOfDemod(8);
  // Play pulse on AWG channel 1
  playWave(gauss_pos);
  // Wait until waveform playback has ended
  waitWave();
  // Play pulses parallel on both AWG channels
  playWave(gauss_pos, gauss_neg);
```

```
}
```

Keywords and Comments

The following table lists the keywords used in the LabOne AWG Sequencer language.

Table 4.62. Programming keywords

Keyword	Description
`const`	Constant declaration
`var`	Integer variable declaration
`cvar`	Compile-time variable declaration
`string`	Constant string declaration
`true`	Boolean true constant
`false`	Boolean false constant
`for`	For-loop declaration
`while`	While-loop declaration
`repeat`	Repeat-loop declaration
`if`	If-statement
`else`	Else-part of an if-statement
`switch`	Switch-statement
`case`	Case-statement within a switch
`default`	Default-statement within a switch
`return`	Return from function or procedure, optionally with a return value

The following code example shows how to use comments.

```
const a = 10; // This is a line comment.

/* This is a block comment. Everything between the start-of-block-comment
and end-of-block-comment markers is ignored.

For example, the following statement will be ignored by the compiler.
const b = 100;
*/
```

Variables and Constants

Variables may be used for making simple computations during run time, i.e., on the UHF instrument. The Sequencer supports integer variables, addition, and subtraction. Not supported are floating-point variables, multiplication, and division. Typical uses of variables are to step waiting times, to output DIO values, or to tag digital measurement data with a numerical identifier. Variables are declared using the `var` keyword.

```
var b = 100; // Create and initialize a variable

// Repeat the following block of statements 100 times
repeat (100) {
  b = b + 1; // Increment b
  wait(b);   // Wait 'b' cycles
}
```

Compile-time variables may be used in computations and loop iterations during compile time. They may be of integer or floating-point type. They are used in a similar way as constants, except that they can change their value during compile time operations. Compile-time variables are declared using the `cvar` keyword.

Constants may be used to make the program more readable. They may be of integer or floating-point type. It must be possible for the compiler to compute the value of a constant at compile time, i.e., on the host computer. Constants are declared using the `const` keyword.

The following table shows the predefined constants. The symbol << denotes the bit shift operator.

Table 4.63. Predefined Constants

Name	Value	Description
`AWG_RATE_1800MHZ`	0	Constant to set sample rate to 1.8 GHz.
`AWG_RATE_900MHZ`	1	Constant to set sample rate to 900 MHz.
`AWG_RATE_450MHZ`	2	Constant to set sample rate to 450 MHz.
`AWG_RATE_225MHZ`	3	Constant to set sample rate to 225 MHz.
`AWG_RATE_112MHZ`	4	Constant to set sample rate to 112 MHz.
`AWG_RATE_56MHZ`	5	Constant to set sample rate to 56 MHz.
`AWG_RATE_28MHZ`	6	Constant to set sample rate to 28 MHz.
`AWG_RATE_14MHZ`	7	Constant to set sample rate to 14 MHz.
`AWG_RATE_7MHZ`	8	Constant to set sample rate to 7 MHz.
`AWG_RATE_3P5MHZ`	9	Constant to set sample rate to 3.5 MHz.
`AWG_RATE_1P8MHZ`	10	Constant to set sample rate to 1.8 MHz.
`AWG_RATE_880KHZ`	11	Constant to set sample rate to 880 kHz.
`AWG_RATE_440KHZ`	12	Constant to set sample rate to 440 kHz.
`AWG_RATE_220KHZ`	13	Constant to set sample rate to 220 kHz.
`AWG_TRIGGER1`	1	Constant to select trigger line 1.
`AWG_TRIGGER2`	2	Constant to select trigger line 2.
`AWG_TRIGGER3`	4	Constant to select trigger line 3.
`AWG_TRIGGER4`	8	Constant to select trigger line 4.
`AWG_TRIGGER5`	16	Constant to select trigger line 5.
`AWG_TRIGGER6`	32	Constant to select trigger line 6.
`AWG_TRIGGER7`	64	Constant to select trigger line 7.
`AWG_TRIGGER8`	128	Constant to select trigger line 8.
`AWG_ANA_TRIGGER1`	1 << 0	Constant for the analog trigger line 1.
`AWG_ANA_TRIGGER2`	1 << 1	Constant for the analog trigger line 2.
`AWG_DIG_TRIGGER1`	1 << 2	Constant for the digital trigger line 1.
`AWG_DIG_TRIGGER2`	1 << 3	Constant for the digital trigger line 2.
`AWG_DEMOD_TRIGGER1`	1 << 4	Constant for oscillator phase of demodulator 1.
`AWG_DEMOD_TRIGGER2`	1 << 5	Constant for oscillator phase of demodulator 2.
`AWG_DEMOD_TRIGGER3`	1 << 6	Constant for oscillator phase of demodulator 3.
`AWG_DEMOD_TRIGGER4`	1 << 7	Constant for oscillator phase of demodulator 4.

Name	Value	Description
AWG_DEMOD_TRIGGER5	1 << 8	Constant for oscillator phase of demodulator 5.
AWG_DEMOD_TRIGGER6	1 << 9	Constant for oscillator phase of demodulator 6.
AWG_DEMOD_TRIGGER7	1 << 10	Constant for oscillator phase of demodulator 7.
AWG_DEMOD_TRIGGER8	1 << 11	Constant for oscillator phase of demodulator 8.
AWG_DEMODRATEMAX_TRIGGER1	1 << 13	Constant for max oscillator rate of demodulator 1.
AWG_DEMODRATE_TRIGGER1	1 << 13	Constant for oscillator rate of demodulator 1.
AWG_DEMODRATE_TRIGGER2	1 << 14	Constant for oscillator rate of demodulator 2.
AWG_DEMODRATE_TRIGGER3	1 << 15	Constant for oscillator rate of demodulator 3.
AWG_DEMODRATE_TRIGGER4	1 << 16	Constant for oscillator rate of demodulator 4.
AWG_DEMODRATE_TRIGGER5	1 << 17	Constant for oscillator rate of demodulator 5.
AWG_DEMODRATE_TRIGGER6	1 << 18	Constant for oscillator rate of demodulator 6.
AWG_DEMODRATE_TRIGGER7	1 << 19	Constant for oscillator rate of demodulator 7.
AWG_DEMODRATE_TRIGGER8	1 << 20	Constant for oscillator rate of demodulator 8.
AWG_WAIT_TRIGGER	1 << 29	Constant to refer to the internal wait trigger.
AWG_TS_TRIGGER	1 << 30	Constant to refer to the internal ts trigger.
AWG_TIME_TRIGGER	1 << 31	Constant to refer to the internal wait trigger.
AWG_CHAN1	1	Constant to select channel 1.
AWG_CHAN2	2	Constant to select channel 2.
AWG_MARKER1	1	Constant to select marker 1.
AWG_MARKER2	2	Constant to select marker 2.
AWG_SUPPRESS_CHAN1_SIGOUT1	1	Constant to supress channel 1 on the output 1.
AWG_SUPPRESS_CHAN1_SIGOUT2	2	Constant to supress channel 1 on the output 2.
AWG_SUPPRESS_CHAN2_SIGOUT1	4	Constant to supress channel 2 on the output 1.
AWG_SUPPRESS_CHAN2_SIGOUT2	8	Constant to supress channel 2 on the output 2.
AWG_ENABLE_CHAN1_SIGOUT1	1	Constant to enable channel 1 on the output 1.

Name	Value	Description
AWG_ENABLE_CHAN1_SIGOUT2	2	Constant to enable channel 1 on the output 2.
AWG_ENABLE_CHAN2_SIGOUT1	4	Constant to enable channel 2 on the output 1.
AWG_ENABLE_CHAN2_SIGOUT2	8	Constant to enable channel 2 on the output 2.
AWG_OSC_PHASE_START	1	Constant to trigger the oscillator phase on the positive edge.
AWG_OSC_PHASE_MIDDLE	0	Constant to trigger the oscillator phase on the negative edge.
AWG_USERREG_SWEEP_COUNT0	35	Constant for the sweep count register 0.
AWG_USERREG_SWEEP_COUNT1	36	Constant for the sweep count register 1.

Numbers can be expressed using either of the following formatting.

```
const a = 10;             // Integer notation
const b = -10;            // Negative number
const h = 0xdeadbeef;     // Hexadecimal integer
const bin = 0b010101010; // Binary integer
const f = 0.1e-3;         // Floating point number.
const not_float = 10e3;  // Not a floating point number
```

Booleans are specified with the keywords `true` and `false`. Furthermore, all numbers that evaluate to a nonzero value are considered true. All numbers that evaluate to zero are considered false.

Strings are delimited using "" and are interpreted as constants. Strings may be concatenated using the + operator.

```
string AWG_PATH = "awgs/0/";
string AWG_GAIN_PATH = AWG_PATH + "gains/0";
```

Waveform Generation and Editing

The following table contains the definition of functions for waveform generation.

Table 4.64. Waveform Generation

Function	Description

`wave sine(const samples, const amplitude=1.0, const phaseOffset, const nrOfPeriods)`
 	
`ARGUMENTS`
`samples : Number of samples in the waveform`
`amplitude : Amplitude of the signal (optional)`
`phaseOffset : Phase offset of the signal in radians`
`nrOfPeriods : Number of Periods within the defined number of samples`
 	
`RETURN`
`resulting waveform`
	Sine function with arbitrary amplitude (a), phase offset (p), number of periods (f) and number of samples (N).
	$h(x) = a \cdot \sin(2\pi f \frac{x}{N} + p)$

`wave cosine(const samples, const amplitude=1.0, const phaseOffset, const nrOfPeriods)`	Cosine function with arbitrary amplitude (a), phase offset (p), number of periods (f) and number of samples (N).

Function	Description
ARGUMENTS samples : Number of samples in the waveform amplitude : Amplitude of the signal (optional) phaseOffset : Phase offset of the signal in radians nrOfPeriods : Number of Periods within the defined number of samples RETURN resulting waveform	$h(x) = a \cdot \cos(2\pi f \frac{x}{N} + p)$
wave sinc(const samples, const amplitude=1.0, const position, const beta) ARGUMENTS samples : Number of samples in the waveform amplitude : Amplitude of the signal (optional) position : Peak position of the function beta : Width of the function RETURN resulting waveform	Normalized sinc function with control of peak position (p), amplitude (a), width (beta) and number of samples (N). $h(x) = \begin{cases} a & \text{if } x = p \\ a \cdot \frac{\sin(2\pi \cdot beta \cdot \frac{x-p}{N})}{2\pi \cdot beta \cdot \frac{x-p}{N}} & \text{else} \end{cases}$
wave ramp(const samples, const startLevel, const endLevel) ARGUMENTS samples : Number of samples in the waveform startLevel : Peak position of the function endLevel : Width of the function RETURN resulting waveform	Linear ramp from the start (s) to the end level (e) over the number of samples (N). $h(x) = s + \frac{x(e-s)}{N-1}$
wave sawtooth(const samples, const amplitude=1.0, const phaseOffset, const nrOfPeriods) ARGUMENTS samples : Number of samples in the waveform amplitude : Amplitude of the signal phaseOffset : Phase offset of the signal in radians nrOfPeriods : Number of Periods within the defined number of samples RETURN resulting waveform	Sawtooth function with arbitrary amplitude, phase and number of periods.
wave triangle(const samples, const amplitude=1.0, const phaseOffset, const nrOfPeriods) ARGUMENTS samples : Number of samples in the waveform amplitude : Amplitude of the signal phaseOffset : Phase offset of the signal in radians nrOfPeriods : Number of Periods within the defined number of samples RETURN resulting waveform	Triangle function with arbitrary amplitude, phase and number of periods.

<table>
<tr><th>Function</th><th>Description</th></tr>
<tr><td><pre>wave gauss(const samples, const
 amplitude=1.0, const position, const width)

ARGUMENTS
samples : Number of samples in the waveform
amplitude : Amplitude of the signal
 (optional)
position : Peak position of the pulse
width : Width of the pulse

RETURN
resulting waveform</pre></td><td>Gaussian pulse with arbitrary amplitude (a), position (p), width (w) and number of samples (N).
$h(x) = a \cdot e^{-\frac{(x-p)^2}{2 \cdot w^2}}$</td></tr>
<tr><td><pre>wave drag(const samples, const
 amplitude=1.0, const position, const width)

ARGUMENTS
samples : Number of samples in the waveform
amplitude : Amplitude of the signal
 (optional)
position : Center point position of the
 pulse
width : Width of the pulse

RETURN
resulting waveform</pre></td><td>Derivative of Gaussian pulse with arbitrary amplitude (a), position (p), width (w) and number of samples (N).
$h(x) = -a \cdot \frac{\sqrt{e}(p-x)}{w} \cdot e^{-\frac{(x-p)^2}{2 \cdot w^2}}$</td></tr>
<tr><td><pre>wave blackman(const samples, const
 amplitude=1.0, const alpha)

ARGUMENTS
samples : Number of samples in the waveform
amplitude : Amplitude of the signal
 (optional)
alpha : Width of the function

RETURN
resulting waveform</pre></td><td>Blackman window function with arbitrary amplitude (a), alpha parameter and number of samples (N).
$h(x) = -\, a \cdot (\alpha_0 - \alpha_1 \cos(\frac{2\pi x}{N-1}) + \alpha_2 \cos(\frac{4\pi x}{N-1}))$
$\alpha_0 = \frac{1-\alpha}{2}; \quad \alpha_1 = \frac{1}{2}; \quad \alpha_2 = \frac{\alpha}{2};$</td></tr>
<tr><td><pre>wave hamming(const samples, const
 amplitude=1.0)

ARGUMENTS
samples : Number of samples in the waveform
amplitude : Amplitude of the signal
 (optional)

RETURN
resulting waveform</pre></td><td>Hamming window function with arbitrary amplitude (a) and number of samples (N).
$h(x) = a \cdot (\alpha - \beta \cos(\frac{2\pi x}{N-1}))$
$\text{with } \alpha = 0.54 \text{ and } \beta = 0.46$</td></tr>
<tr><td><pre>wave hann(const samples, const
 amplitude=1.0)

ARGUMENTS
samples : Number of samples in the waveform
amplitude : Amplitude of the signal

RETURN
resulting waveform</pre></td><td>Hann window function with arbitrary amplitude (a) and number of samples (N).
$h(x) = a \cdot 0.5 \cdot (1 - \cos(\frac{2\pi x}{N-1}))$</td></tr>
<tr><td><pre>wave rect(const samples, const amplitude)

ARGUMENTS
samples : Number of samples in the waveform
amplitude : Amplitude of the signal

RETURN</pre></td><td>Const amplitude (a) over the defined number of samples.
$h(x) = \mathrm{a}$</td></tr>
</table>

Function	Description
resulting waveform	
wave rand(const samples, const amplitude=1.0, const mean, const stdDev) ARGUMENTS samples : Number of samples in the waveform amplitude : Amplitude of the signal mean : Average signal level stdDev : Standard deviation of the noise signal RETURN resulting waveform	White noise with arbitrary amplitude, power and standard deviation.
wave chirp(const samples, const amplitude=1.0, const startFreq, const stopFreq, const phase) ARGUMENTS samples : Number of samples in the waveform amplitude : Amplitude of the signal (optional) startFreq : Start frequency of the signal stopFreq : Stop Frequency of the signal phase : Initial phase of the signal (optional) RETURN resulting waveform	Frequency chirp function with arbitrary amplitude, start and stop frequency, initial phase and number of samples. Start and stop frequency are expressed in units of the sampling frequency. The amplitude can only be defined if the initial phase is defined as well.
wave marker(const samples, const markerValue)	
wave rrc(const samples, const amplitude=1.0, const position, const beta, const width) ARGUMENTS samples : Number of samples in the waveform amplitude : Amplitude of the signal position : Center point position of the pulse beta : Roll-off factor width : Width of the pulse RETURN Resulting waveform	Root raised cosine function with arbitrary amplitude (a), position (p), roll-off factor (beta) and width (w) and number of samples (N). $h(y) = a\frac{\sin(y\pi(1-\beta)) + 4y\beta\cos(y\pi(1+\beta))}{y\pi(1-(4y\beta)^2)}$ $\text{with } y(x) = 2w\frac{x-p}{N}$
wave vect(const amplitude) ARGUMENTS amplitude : Amplitude of each point in the waveform in the range of -1 to 1 RETURN resulting waveform	Generate a waveform with the specified point-by-point values.

The following table contains the definition of functions for waveform editing.

Table 4.65. Waveform Editing

Function	Description
wave join(wave wave1, wave wave2, const interpolLength=0) ARGUMENTS	Connect two or more waveforms with optional linear interpolation between the waveforms.

Function	Description
wave1 : Input waveforms wave2 : Input waveforms interpolLength : Number of samples to interpolate between waveforms (optional, default 0) RETURN joined waveforms	
`wave join(wave wave1, wave wave2,...)` ARGUMENTS wave1 : Input waveforms wave2 : Input waveforms RETURN joined waveforms	Connect two or more waveforms.
`wave interleave(wave wave1, wave wave2,...)` ARGUMENTS wave1 : Input waveforms wave2 : Input waveforms RETURN interleaved waveforms	Interleave two or more waveforms sample by sample.
`wave add(wave wave1, wave wave2,...)` ARGUMENTS wave1 : Input waveforms wave2 : Input waveforms RETURN sum waveforms	Add two or more waveforms sample by sample.
`wave multiply(wave wave1, wave wave2,...)` ARGUMENTS wave1 : Input waveforms wave2 : Input waveforms RETURN product waveforms	Multiply two or more waveforms sample by sample.
`wave scale(wave waveform, const factor)` ARGUMENTS waveform : Input waveform factor : Scaling factor RETURN scaled waveform	Scale the input waveform with the factor and return the scaled waveform. The input waveform remains unchanged.
`wave flip(wave waveform)` ARGUMENTS waveform : Input waveform RETURN flipped waveform	Flip the input waveform back to front and return the flipped waveform. The input waveform remains unchanged.
`wave cut(wave waveform, const from, const to)` ARGUMENTS waveform : Input waveform from : First sample of the cut waveform to : Last sample of the cut waveform	Cut a segment out of the input waveform and returns it. The input waveform remains unchanged. The segment is flipped in case that "from" is larger than "to".

Function	Description
RETURN cut waveform	

Predefined Functions

The following table contains the definition of functions for waveform playback and other purposes.

Table 4.66. Predefined Functions

Function	Description
void setDIO(var value) ARGUMENTS value : The value to write to the DIO (const or var)	Writes the value as a 32-bit value to the DIO bus. The value can be either a const or a var value. Configure the settings in the DIO tab when using this command.
var getDIO() RETURN var containing the read value	Reads a 32-bit value from the DIO bus.
void setTrigger(var value) ARGUMENTS value : to be written to the trigger output lines	Sets the trigger output signals with the given value. Each trigger output line is represented by one bit of the integer value.
void setID(var id) ARGUMENTS id : The new ID to be attached to streaming data of the device	Sets the ID value that is attached to data streamed from the device to the host PC. The ID value is useful for synchronizing the data acquisition process in combination with the sweeper or the software trigger.
void playWave(const output, wave waveform, const rate=AWG_RATE_DEFAULT) ARGUMENTS output : defines on which output the following waveform is played waveform : waveform to be played rate : sample rate with which the AWG plays the waveforms (default set in the user interface).	Starts to play the given waveforms on the defined output channels. The playback begins as soon as the previous waveform is finished.
void playWave(const output, wave waveform,...) ARGUMENTS output : defines on which output the following waveform is played waveform : waveform to be played	Starts to play the given waveforms on the defined output channels. It can contain multiple waveforms with an output definition. The playback begins as soon as the previous waveform is finished.
void playWave(wave waveform, const rate=AWG_RATE_DEFAULT) ARGUMENTS waveform : waveform to be played	Starts to play the given waveforms, output channels are assigned automatically depending on the number of input waveforms. The playback begins as soon as the previous waveform is finished.

Function	Description
`rate : sample rate with which the AWG plays the waveforms (default set in the user interface).`	

`void playWave(wave waveform,...)`
 	
`ARGUMENTS`
`waveform : waveform to be played`
	Starts to play the given waveforms, output channels are assigned automatically depending on the number of input waveforms. The playback begins as soon as the previous waveform is finished.

`void playWaveNow(const output, wave waveform, const rate=AWG_RATE_DEFAULT)`
 	
`ARGUMENTS`
`output : defines on which output the following waveform is played`
`waveform : waveform to be played`
`rate : sample rate with which the AWG plays the waveforms (default set in the user interface).`
	Starts to play the given waveforms on the defined output channels. It starts immediately even if the AWG is still busy.

`void playWaveNow(const output, wave waveform,...)`
 	
`ARGUMENTS`
`output : defines on which output the following waveform is played`
`waveform : waveform to be played`
	Starts to play the given waveforms on the defined output channels. It can contain multiple waveforms with an output definition. It starts immediately even if the AWG is still busy.

`void playWaveNow(wave waveform, const rate=AWG_RATE_DEFAULT)`
 	
`ARGUMENTS`
`waveform : waveform to be played`
`rate : sample rate with which the AWG plays the waveforms (default set in the user interface).`
	Starts to play the given waveforms, channels are assigned automatically depending on the number of input waveforms. It starts immediately even if the AWG is still busy.

`void playWaveNow(wave waveform,...)`
 	
`ARGUMENTS`
`waveform : waveform to be played`
	Starts to play the given waveforms, channels are assigned automatically depending on the number of input waveforms. It starts immediately even if the AWG is still busy.

`void playWaveIndexed(const output, wave waveform, var offset, const length, const rate=AWG_RATE_DEFAULT)`
 	
`ARGUMENTS`
`output : defines on which output the following waveform is played`
`waveform : waveform to be played`
`offset : offset in samples from the start of the waveform`
`length : number of samples to be played from this waveform`
`rate : sample rate with which the AWG plays the waveforms (default set in the user interface).`
	Starts to play the specified part of the given waveforms on the defined output channels. It can contain multiple waveforms with an output definition. The playback begins as soon as the previous waveform is finished.

Function	Description
`void playWaveIndexed(wave waveform, var offset, const length, const rate=AWG_RATE_DEFAULT)` ARGUMENTS waveform : waveform to be played offset : offset in samples from the start of the waveform length : number of samples to be played from this waveform rate : sample rate with which the AWG plays the waveforms (default set in the user interface).	Starts to play the specified part of the given waveforms, channels are assigned automatically depending on the number of input waveforms. The playback begins as soon as the previous waveform is finished.
`void playWaveIndexedNow(const output, wave waveform, var offset, const length, const rate=AWG_RATE_DEFAULT)` ARGUMENTS output : defines on which output the following waveform is played waveform : waveform to be played offset : offset in samples from the start of the waveform length : number of samples to be played from this waveform rate : sample rate with which the AWG plays the waveforms (default set in the user interface).	Starts to play the specified part of the given waveforms on the defined output channels. It can contain multiple waveforms with an output definition. It starts immediately even if the AWG is still busy.
`void playWaveIndexedNow(wave waveform, var offset, const length, const rate=AWG_RATE_DEFAULT)` ARGUMENTS waveform : waveform to be played offset : offset in samples from the start of the waveform length : number of samples to be played from this waveform rate : sample rate with which the AWG plays the waveforms (default set in the user interface).	Starts to play the specified part of the given waveforms, channels are assigned automatically depending on the number of input waveforms. It starts immediately even if the AWG is still busy.
`void playAuxWave(const output, wave waveform, const rate=AWG_RATE_DEFAULT)` ARGUMENTS output : defines on which output the following waveform is played waveform : waveform to be played rate : sample rate with which the AWG plays the waveforms (default set in the user interface).	Starts to play the given waveforms on the defined output channels with enabled 4-channel-mode. The playback begins as soon as the previous waveform is finished.
`void playAuxWave(const output, wave waveform,...)` ARGUMENTS output : defines on which output the following waveform is played waveform : waveform to be played	Starts to play the given waveforms on the defined output channels with enabled 4-channel-mode. It can contain multiple waveforms with an output definition. The playback begins as soon as the previous waveform is finished.

Function	Description

`void playAuxWave(wave waveform, const rate=AWG_RATE_DEFAULT)`
 	
`ARGUMENTS`
`waveform : waveform to be played`
`rate : sample rate with which the AWG plays the waveforms (default set in the user interface).`
	Starts to play the given waveforms, channels are assigned automatically depending on the number of input waveforms, with enabled 4-channel-mode. The playback begins as soon as the previous waveform is finished.

`void playAuxWave(wave waveform,...)`
 	
`ARGUMENTS`
`waveform : waveform to be played`
	Starts to play the given waveforms, channels are assigned automatically depending on the number of input waveforms, with enabled 4-channel-mode. If the AWG is already busy playing another waveform does it block and start to play as soon as the preview waveform is finished.

`void wait(var cycles)`
 	
`ARGUMENTS`
`cycles : number of cycles to wait`
	Waits for the given number of cycles (min 4 cycles).

`void waitWave()`	Waits until the AWG is done playing the current waveform.

`void waitTrigger(const mask, const value)`
 	
`ARGUMENTS`
`mask : mask to be applied to the input signal`
`value : value to be compared with the trigger input`
	Waits until the masked trigger input is equal to the given reference value.

`void waitAnaTrigger(const index, const value)`
 	
`ARGUMENTS`
`index : index of the analog trigger input to be waited on, can be either 1 or 2`
`value : value to be compared with the analog trigger input, can be either 0 or 1`
	Waits until the indexed analog trigger input is equal to the given value.

`void waitDigTrigger(const index, const value)`
 	
`ARGUMENTS`
`index : index of the digital trigger input to be waited on, can be either 1 or 2`
`value : value to be compared with the digital trigger input, can be either 0 or 1`
	Waits until the indexed digital trigger input is same as the given value. This function reads the current value on one of the internally generated trigger input signals produced by the Cross-Trigger Engine. The physical signal connected to the AWG trigger inputs must be configured in the Cross-Trigger Engine itself.

`void waitDemodOscPhase(const demod, const position=0)`
 	
`ARGUMENTS`
`demod : index of the demodulator to be waited on, can be between 1 and 8`
`position : either start of the phase (0) or middle of the phase (1), default 0`
	Waits until the oscillator phase of the indexed demodulator reaches the defined value.

Function	Description

`void waitDemodSample(const demod)`
 	
`ARGUMENTS`
`demod : index of the demodulator to be waited on, can be between 1 and 8`
	Waits until the indexed demodulator delivers a new sample.

`var getTrigger(const mask)`
 	
`ARGUMENTS`
`mask : mask to apply to the trigger input signals`
 	
`RETURN`
`trigger input value, either 0 or 1`
	Gets the trigger input signals and applies the given mask on it. The trigger input lines are represented as individual bits in the return value.

`var getAnaTrigger(const index)`
 	
`ARGUMENTS`
`index : index of the analog trigger input to be read, can be either 1 or 2`
 	
`RETURN`
`trigger value, either 0 or 1`
	Gets the indexed analog trigger input value.

`var getDigTrigger(const index)`
 	
`ARGUMENTS`
`index : index of the digital trigger input to be read, can be either 1 or 2`
 	
`RETURN`
`trigger value, either 0 or 1`
	Gets the indexed digital trigger input value.

`void setInt(string path, var value)`
 	
`ARGUMENTS`
`path : The node path to be written to`
`value : The integer value to be written`
	Writes a value to one of the nodes in the device. If the path does not start with a device identifier, then the current device is assumed.

`void setDouble(string path, const value)`
 	
`ARGUMENTS`
`path : The node path to be written to`
`value : The integer or floating point value to be written`
	Writes a value to one of the nodes in the device. If the path does not start with a device identifier, then the current device is assumed.

`void setUserReg(const register, var value)`
 	
`ARGUMENTS`
`register : The register to be written to`
`value : The integer value to be written`
	Writes a value to one of the user registers. The user registers may be used for communicating information to the LabOne user interface or a running API program.

`var getUserReg(const register)`
 	
`ARGUMENTS`
`register : The register to be read`
 	
`RETURN`
`current register value`
	Reads the value from one of the user registers. The user registers may be used for communicating information to the LabOne user interface or a running API program.

`var getSweeperLength(const index)`
 	
`ARGUMENTS`
	Reads the length as configured by the LabOne Sweeper. The length is only valid when the AWG is started by the Sweeper.

Function	Description

`index : The index of the Sweeper parameter to get the length of. Currently only the value of 1 is accepted.`
 	
`RETURN`
`length configured by the Sweeper`	

`void setRate(const rate)`
 	
`ARGUMENTS`
`rate : New default rate for the current scope`
	Overwrites the global default rate for the following playWave commands.

`void now()`	Resets the local timer.

`void at(var time)`
 	
`ARGUMENTS`
`time : value to wait for`
	Waits until the local timer reaches the given value.

`void error(string msg)`
 	
`ARGUMENTS`
`msg : Message to be displayed`
	Throws the given error message when reached.

Expressions

Expressions may be used for making computations based on mathematical functions and operators. There are two kinds of expressions: those evaluated at compile time (the moment of clicking "Save" or "Save as..." in the user interface), and those evaluated at run time (after clicking "Run/Stop" or "Start"). Compile-time evaluated expressions only involve constants (`const`) or compile-time variables (`cvar`) and can be computed at compile time by the host computer. Such expressions can make use of standard mathematical functions as well as floating point arithmetic. Run-time evaluated expressions involve variables (`var`) and are evaluated by the Sequencer on the UHF instrument. Due to the limited computational capabilities of the Sequencer, these expressions may only operate on integer numbers and there are less operators available than at compile time.

The following table contains the list of mathematical functions supported at compile time.

Table 4.67. Mathematical Functions

Function	Description
`const abs(const c)`	absolute value
`const acos(const c)`	inverse cosine
`const acosh(const c)`	hyperbolic inverse cosine
`const asin(const c)`	inverse sine
`const asinh(const c)`	hyperbolic inverse sine
`const atan(const c)`	inverse tangent
`const atanh(const c)`	hyperbolic inverse tangent
`const cos(const c)`	cosine
`const cosh(const c)`	hyperbolic cosine
`const exp(const c)`	exponential function
`const ln(const c)`	logarithm to base e (2.71828...)

Function	Description
`const log(const c)`	logarithm to the base 10
`const log2(const c)`	logarithm to the base 2
`const log10(const c)`	logarithm to the base 10
`const sign(const c)`	sign function -1 if x<0; 1 if x>0
`const sin(const c)`	sine
`const sinh(const c)`	hyperbolic sine
`const sqrt(const c)`	square root
`const tan(const c)`	tangent
`const tanh(const c)`	hyperbolic tangent
`const ceil(const c)`	smallest integer value not less than the argument
`const round(const c)`	round to nearest integer
`const floor(const c)`	largest integer value not greater than the argument
`const avg(const c1, const c2,...)`	mean value of all arguments
`const max(const c1, const c2,...)`	maximum of all arguments
`const min(const c1, const c2,...)`	minimum of all arguments
`const pow(const base, const exp)`	first argument raised to the power of second argument
`const sum(const c1, const c2,...)`	sum of all arguments

The following table contains the list of predefined mathematical constants. These can be used for convenience in compile-time evaluated expressions.

Table 4.68. Mathematical Constants

Name	Value	Description
`M_E`	2.71828182845904523536028747135266250	e
`M_LOG2E`	1.44269504088896340735992468100189214	log2(e)
`M_LOG10E`	0.434294481903251827651128918916605082	log10(e)
`M_LN2`	0.693147180559945309417232121458176568	loge(e)
`M_LN10`	2.30258509299404568401799145468436421	loge(10)
`M_PI`	3.14159265358979323846264338327950288	pi
`M_PI_2`	1.57079632679489661923132169163975144	pi/2
`M_PI_4`	0.785398163397448309615660845819875721	pi/4
`M_1_PI`	0.318309886183790671537767526745028724	1/pi
`M_2_PI`	0.636619772367581343075535053490057448	2/pi
`M_2_SQRTPI`	1.12837916709551257389615890312154517	2/sqrt(pi)
`M_SQRT2`	1.41421356237309504880168872420969808	sqrt(2)
`M_SQRT1_2`	0.707106781186547524400844362104849039	1/sqrt(2)

Table 4.69. Operators supported at compile time

<table>
<tr><th>Operator</th><th>Description</th><th>Priority</th></tr>
<tr><td>=</td><td>assignment</td><td>-1</td></tr>
<tr><td>||</td><td>logical OR</td><td>1</td></tr>
<tr><td>&&</td><td>logical AND</td><td>2</td></tr>
<tr><td>|</td><td>bit-wise logical OR</td><td>3</td></tr>
<tr><td>&</td><td>bit-wise logical AND</td><td>4</td></tr>
<tr><td>!=</td><td>not equal</td><td>5</td></tr>
<tr><td>==</td><td>equal</td><td>5</td></tr>
<tr><td><=</td><td>less or equal</td><td>6</td></tr>
<tr><td>>=</td><td>greater or equal</td><td>6</td></tr>
<tr><td>></td><td>greater than</td><td>6</td></tr>
<tr><td><</td><td>less than</td><td>6</td></tr>
<tr><td><<</td><td>left bit shift</td><td>7</td></tr>
<tr><td>>></td><td>right bit shift</td><td>7</td></tr>
<tr><td>+</td><td>addition</td><td>8</td></tr>
<tr><td>-</td><td>subtraction</td><td>8</td></tr>
<tr><td>*</td><td>multiplication</td><td>9</td></tr>
<tr><td>/</td><td>division</td><td>9</td></tr>
<tr><td>~</td><td>bit-wise logical negation</td><td>10</td></tr>
</table>

Table 4.70. Operators supported at run time

<table>
<tr><th>Operator</th><th>Description</th><th>Priority</th></tr>
<tr><td>=</td><td>assignment</td><td>-1</td></tr>
<tr><td>||</td><td>logical OR</td><td>1</td></tr>
<tr><td>&&</td><td>logical AND</td><td>2</td></tr>
<tr><td>|</td><td>bit-wise logical OR</td><td>3</td></tr>
<tr><td>&</td><td>bit-wise logical AND</td><td>4</td></tr>
<tr><td>==</td><td>equal</td><td>5</td></tr>
<tr><td>!=</td><td>not equal</td><td>5</td></tr>
<tr><td><=</td><td>less or equal</td><td>6</td></tr>
<tr><td>>=</td><td>greater or equal</td><td>6</td></tr>
<tr><td>></td><td>greater than</td><td>6</td></tr>
<tr><td><</td><td>less than</td><td>6</td></tr>
<tr><td><<</td><td>left bit shift</td><td>7</td></tr>
<tr><td>>></td><td>right bit shift</td><td>7</td></tr>
<tr><td>+</td><td>addition</td><td>8</td></tr>
<tr><td>-</td><td>subtraction</td><td>8</td></tr>
<tr><td>~</td><td>bit-wise logical negation</td><td>9</td></tr>
</table>

Control Structures

Functions may be declared using the `var` keyword. **Procedures** may be declared using the `void` keyword. Functions must return a value, which should be specified using the `return` keyword. Procedures can not return values. Functions and procedures may be declared with an arbitrary number of arguments. The `return` keyword may also be used without arguments to return from and arbitrary point within the function or procedure. Functions and procedures may contain variable and constant declarations. These declarations are local to the scope of the function or procedure.

```
var function_name(argument1, argument2, ...) {
  // Statements to be executed as part of the function.
  return constant-or-variable;
}

void procedure_name(argument1, argument2, ...) {
  // Statements to be executed as part of the procedure.
  // Optional return statement
  return;
}
```

An **if-then-else** structure is used to create a conditional branching point in a sequencer program.

```
// If-then-else statement syntax
if (expression) {
  // Statements to execute if 'expression' evaluates to 'true'.
} else {
  // Statements to execute if 'expression' evaluates to 'false'.
}

// If-then-else statement short syntax
(expression)?(statement if true):(statement if false)

// If-then-else statement example
const REQUEST_BIT     = 0x0001;
const ACKNOWLEDGE_BIT = 0x0002;
const IDLE_BIT        = 0x8000;
var dio = getDIO();
if (dio & REQUEST_BIT) {
  dio = dio | ACKNOWLEDGE_BIT;
  setDIO(dio);
} else {
  dio = dio | IDLE_BIT;
  setDIO(dio);
}
```

A **switch-case** structure serves to define a conditional branching point similarly to the `if-then-else` statement, but is used to split the sequencer thread into more than two branches. Unlike the `if-then-else` structure, the switch statement is synchronous, which means that the execution time is the same for all branches and determined by the execution time of the longest branch. If no default case is provided and no case matches the condition, all cases will be skipped. The case arguments need to be of type `const`.

```
// Switch-case statement syntax
switch (expression) {
  case const-expression:
    expression;
  ...
  default:
 expression;
}

// Switch-case statement example
switch(getDIO()){
```

```
  case 0:
    playWave(gauss(1024,1.0,512,64));
  case 1:
    playWave(gauss(1024,1.0,512,128));
  case 2:
    playWave(drag(1024,1.0,512,64));
  default:
    playWave(drag(1024,1.0,512,128));
}
```

The **for loop** is used to iterate through a code block several times. The `initialization` statement is executed before the loop starts. The `end-expression` is evaluated at the start of each iteration and determines when the loop should stop. The loop is executed as long as this expression is true. The `iteration-expression` is executed at the end of each loop iteration.

Depending on how the `for` loop is set up, it can be either evaluated at compile time or at run time. Run-time evaluation is typically used to play series of waveforms. Compile-time evaluation is typically used for advanced waveform generation, e.g. to generate a series of waveforms with varying amplitude which later can be iterated through with the `playWaveIndexed` command. For a run-time evaluated `for` loop, use the `var` data type as a loop index. To ensure that a loop is evaluated at compile time, use the `cvar` data type as a loop index. Furthermore, the compile-time `for` loop should only contain waveform generation/editing operations and it can't contain any variables of type `var`. The following code example shows both versions of the loop.

```
// For loop syntax
for (initialization; end-expression; iteration-expression) {
  // Statements to execute while end-expression evaluates to true
}

// For loop example (compile-time execution)
cvar i;
wave w_pulses;
for (i = 0; i < 10; i = i + 1) {
  w_pulses = join(w_pulses, i*0.1*gauss(1000, 500, 100);
}

// For loop example (run-time execution)
var j;
for (j = 0; j < 10; j = j + 2) {
  setDIO(j);
}
```

The **while loop** is a simplified version of the `for` loop. The `end-expression` is evaluated at the start of each loop iteration. The contents of the loop are executed as long as this expression is true. Like the `for` loop, this loop comes in a compile-time version (if the end-expression involves only `cvar` and `const`) and in a run-time version (if the end-expression involves also `var` data types).

```
// While loop syntax
while (end-expression) {
 // Statements to execute while end-expression evaluates to true
}

// While loop example
const STOP_BIT = 0x8000;
var run = 1;
var i = 0;
var dio = 0;
while (run) {
  dio = getDIO();
  run = dio & STOP_BIT;

  dio = dio | (i & 0xff);
  setDIO(dio);
  i = i + 1;
```

```
}
```

The **repeat loop** is a simplified version of the `for` loop. It repeats the contents of the loop a fixed number of times. The `repeat` loop statement takes a constant expression as argument, i.e. it is not possible to use the loop with a variable.

```
// Repeat loop syntax
repeat (constant-expression) {
  // Statements to execute
}

// Repeat loop example
repeat (100) {
  setDIO(0x1);
  wait(10);
  setDIO(0x0);
  wait(10);
}
```

4.22.4. Functional Elements

Table 4.71. AWG tab: Control sub-tab

Control/Tool	Option/Range	Description
Run / Stop	Run/Stop	Runs the AWG continuously.
Single	Single	Runs the AWG once.
Rate (Sa/s)	220 kSa/s to 1.8 GSa/s	AWG sampling rate. This value is used by default and can be overridden in the Sequence program.
Amplitude (FS)	0.0 to 1.0	Amplitude in units of full scale of the given AWG channel. The full scale corresponds to the Range voltage setting of the Signal Outputs.
Mode		Select between direct mode and amplitude modulation mode.
	Direct	Output of AWG goes directly to Signal Output.
	Modulation	Output of AWG channel 1 (2) is multiplied with oscillator signal of demodulator 4 (8).
Status		Display compiler errors and warnings.
Compile Status	grey/green/yellow/red	Sequence program compilation status. Grey: No compilation started yet. Green: Compilation successful. Yellow: Compiler warnings (see status field). Red: Compilation failed (see status field).
Register selector		Select the number of the user register value to be edited.

Control/Tool	Option/Range	Description
Register	0 to 2^32	Integer user register value. The sequencer has reading and writing access to the user register values during run time.
Input File		External source code file to be compiled.
New	New	Create a new sequence program.
Revert	Revert	Undo the changes made to the current program and go back to the contents of the original file.
Save (Ctrl+S)	Save	Compile and save the current program displayed in the Sequence Editor. Overwrites the original file.
Save as... (Ctrl+Shift+S)	Save as...	Compile and save the current program displayed in the Sequence Editor under a new name.
Automatic upload	ON / OFF	If enabled, the sequence program is automatically uploaded to the device after clicking Save and if the compilation was successful.
To Device	To Device	Sequence program will be compiled and, if the compilation was successful, uploaded to the device.
Sync Status	grey/green/yellow	Sequence program synchronization status. Grey: No program loaded on device. Green: Program in sync with device. Yellow: Sequence program in editor differs from the one running on the device.

Table 4.72. AWG tab: Waveform sub-tab

Control/Tool	Option/Range	Description
Waveforms		Lists all waveforms used by the current sequence program.
Mem Usage (%)	0 to 100	Amount of the used waveform data relative to the device cache memory. The cache memory provides space for 32 kSa of waveform data. Mem Usage > 100% means that waveforms must be loaded from the main memory (128

Control/Tool	Option/Range	Description
		MSa per channel) during playback, which can lead to delays.

Table 4.73. AWG tab: Trigger sub-tab

Control/Tool	Option/Range	Description
Level (V)	numeric value	Defines the analog trigger level.
Hysteresis Mode		Selects the mode to define the hysteresis size. The relative mode will work best over the full input range as long as the analog input signal does not suffer from excessive noise.
	Hysteresis (V)	Selects absolute hysteresis.
	Hysteresis (%)	Selects a hysteresis relative to the adjusted full scale signal input range.
Hysteresis (V)	trigger signal range (positive values only)	Defines the voltage the source signal must deviate from the trigger level before the trigger is rearmed again. Set to 0 to turn it off. The sign is defined by the Edge setting.
Hysteresis (%)	numeric percentage value (positive values only)	Hysteresis relative to the adjusted full scale signal input range. A hysteresis value larger than 100% is allowed.
Rise	ON / OFF	Trigger on the rising edge.
Fall	ON / OFF	Trigger on the falling edge.
Signal	Signal Inputs	Selects the analog trigger source signal. Navigate through the tree view that appears and click on the required signal.
	Trigger Inputs	
	Auxiliary Inputs	
	Demodulator Oscillator Phase	
	Demodulator X/Y/R/Theta	
	PID	
	Boxcar	
	AU	
Force		Enforce a trigger event.
Gating	Trigger In 3	Select the signal source used for trigger gating if gating is enabled.
	Trigger In 4	
Gating enable	ON / OFF	If enabled the trigger will be gated by the trigger gating input signal.

Control/Tool	Option/Range	Description
Signal	Trigger In 1	Selects the digital trigger source signal.
	Trigger In 2	
	Trigger In 3	
	Trigger In 4	
	Trigger Out 1	
	Trigger Out 2	
	Trigger Out 3	
	Trigger Out 4	

Table 4.74. AWG tab: Advanced sub-tab

Control/Tool	Option/Range	Description
Counter		Current position in the list of sequence instructions during execution.
Status	Running, Idle, Waiting	Displays the status of the sequencer on the instrument.
Sequence Editor		Display and edit the sequence program.
Assembly	Text display	Displays the current sequence program in compiled form. Every line corresponds to one hardware instruction and requires one clock cycle (4.44 ns) for execution.
Mem Usage (%)	0 to 100	Size of the current sequence program relative to the device cache memory. The cache memory provides space for 1024 instructions. Mem Usage > 100% means that instructions must be loaded from the main memory during playback, which can lead to delays.

4.23. Pulse Counter Tab

The Pulse Counter tab relates to the UHF-CNT Pulse counter option and is only available if this option is installed on the UHF Instrument (see Information section in the Device tab).

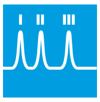
4.23.1. Features

- 4 counter modules
- 225 MHz maximum count rate
- 4 modes: free running, gated, gated free running, and pulse tagging
- 4 analog signal inputs with adjustable discriminator level
- 32 digital signal inputs
- Background subtraction
- Analog output of counter data
- Count integration

4.23.2. Description

The Pulse Counter tab provides access to the pulse counter settings. Whenever the tab is closed or an additional one of the same type is needed, clicking the following icon will open a new instance of the tab.

Table 4.75. App icon and short description

Control/Tool	Option/Range	Description
Counter		Configure the Pulse Counters for analysis of pulse trains on the digital signal inputs.

The Pulse Counter tab shown in Figure 4.47 consists of four side-tabs, one for each Counter module. The Enable button and the Mode selector are the main controls that determine if and how a Counter unit generates an output. The output is displayed in the Value field and is available in the Plotter, Numeric, and Software Trigger tab. They can also be output as analog values on the Auxiliary Outputs.

The counter Input signal is selectable among the four analog Trigger inputs as well as any of the 32 DIO channels on the VHDCI connector on the instrument rear panel. The trigger level of the analog trigger inputs is configurable in the DIO tab. The following operation modes are available.

- Free running: the counter is active during repeated periods defined by the a configurable timer. The timer period is controlled by the Period field. At the beginning of the period the counter is reset, and at the end, the accumulated number of counts is output.
- Gated: the counter is controlled with the Gate Input signal. The counter is enabled at the rising edge of the Gate Input signal and disabled at the falling edge. Pulses are counted as long as the counter is enabled. The accumulated number of counts is output on the falling edge of the Gate Input signal.
- Gated free running: the counter runs on a repetitive time base defined by the Period field. The Gate Input signal controls when the counter is allowed to run. The counter as well as the timer is reset when the Gate Input signal is low. The counter will only deliver new values if the Gate Input signal is high for a time longer than the configured Period.

- Time tagging: every single event is counted and transmitted to the server along with a time tag.

Background subtraction or summation of data from two counter modules is controlled by the Operation field. For add and subtract operations, counter units 1 is grouped with unit 2, and unit 3 is grouped with unit 4. The Pulse Counter supports integration of counter data over time.

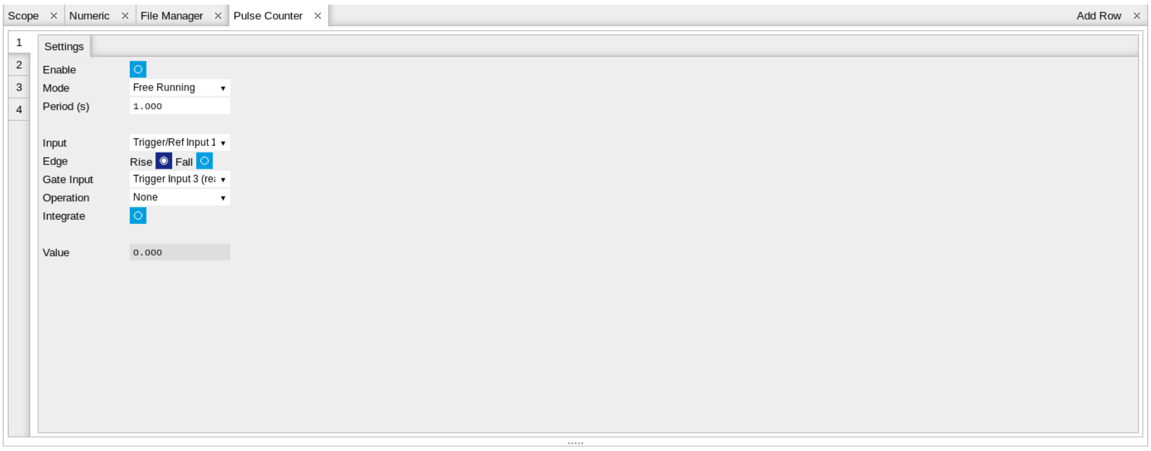


Figure 4.47. LabOne UI: Counter tab

4.23.3. Functional Elements

Table 4.76. Pulse Counter tab

Control/Tool	Option/Range	Description
Enable	ON / OFF	Enable the pulse counter unit.
Mode		Select the run mode of the counter unit.
	Free Running	The counter runs on a repetitive time base defined by the Period field. At the beginning of each period the counter is reset, and at the end, the accumulated number of counts is output.
	Gated Free Running	The counter runs on a repetitive time base defined by the Period field. The Gate Input signal controls when the unit counter is allowed to run. The counter as well as the timer is reset when the Gate Input signal is low. The counter will only deliver new values if the Gate Input signal is high for a time longer than the configured Period.
	Gated	The counter is controlled with the Gate Input signal. The counter is enabled at the rising

Control/Tool	Option/Range	Description
		edge of the Gate Input signal and disabled at the falling edge. Pulses are counted as long as the counter is enabled. The accumulated number of counts is output on the falling edge of the Gate Input signal.
	Time Tagging	Every pulse is detected individually and tagged with the time of the event. The Period defines the minimum hold-off time between the tagging of two subsequent pulses. If more than one pulse occurs within the window defined by the Period, then the pulses are accumulated and output at the end of the window. The Period effectively determines the maximum rate at which pulse information can be transmitted to the host PC.
Period	44.4 ns to 19 s	Set the period used for the Free Running and Gated Free Running modes. Also sets the hold-off time for the Time Tagging mode.
Input	Ref/Trigger Input 1/2, Trigger Input 3/4, DIO Bit 0-31	Select the counter signal source.
Gate Input	Ref/Trigger Input 1/2, Trigger Input 3/4, AWG internal Trigger 1-4	Select the signal source used for enabling the counter in the Gated Free Running and Gated modes.
Operation	None	Select the arithmetic operation (addition, subtraction) applied to the counter unit outputs. "Other counter" refers to the grouping of the counter units: 1 with 2, and 3 with 4.
	Add Other Counter	
	Subtract Other Counter	
Integrate	ON / OFF	Sum up counter values over time.
Value		Displays the counter output value.

4.24. ZI Labs Tab

The ZI Labs tab contains experimental LabOne functionalities added by the ZI development team. The settings found here are often relevant to special applications, but have not yet found their definitive place in one of the other LabOne tabs. Naturally this tab is subject to frequent changes, and the documentation of the individual features would go beyond the scope of this user manual. Clicking the following icon will open a new instance of the tab.

Table 4.77. App Icon and short description

Control/Tool	Option/Range	Description
ZI Labs		Experimental settings and controls.

Chapter 5. Specifications

Important

Unless otherwise stated, all specifications apply after 30 minutes of instrument warm-up.

Important

An internal calibration is performed 10 minutes after powering the instrument. This internal calibration is essential to achieve the specifications of the system. Further it is required to perform the internal calibration after 7 days of instrument use. This automatic calibration is turned on by default and can be configured in the Device tab.

Important

Important changes in the specification parameters are explicitly mentioned in the revision history of this document.

5.1. General Specifications

Table 5.1. General and storage

Parameter	min	typ	max
storage temperature	-25 °C	-	65 °C
storage relative humidity (non-condensing)	-	-	95%
operating temperature	5 °C	-	40 °C
operating relative humidity (non-condensing)	-	-	90%
specification temperature	18 °C	-	28 °C
power consumption	-	-	150 W
operating environment	IEC61010, indoor location, installation category II, pollution degree 2		
operating altitude	up to 2000 meters		
power inlet fuses	250 V, 2 A, fast acting, 5 x 20 mm		
power supply AC line	100-240 V (±10%), 50/60 Hz		
dimensions with handles and feet	45.0 x 34.5 x 10.0 cm, 17.7 x 13.6 x 3.9 inch, 19 inch rack compatible		
weight	6.4 kg		
recommended calibration interval	2 years		

Table 5.2. Maximum ratings

Parameter	min	typ	max
damage threshold Signal Input 1 and 2	-5 V	-	5 V
damage threshold Signal Output 1 and 2	-2.5 V	-	2.5 V
damage threshold Ref / Trigger 1 and 2	-6 V	-	6 V
damage threshold Trigger Out 1 and 2	-1 V	-	6 V
damage threshold Trigger In 1 and 2	-6 V	-	6 V
damage threshold Aux Output 1, 2, 3, 4	-12 V	-	12 V
damage threshold Aux In 1 and 2	-12 V	-	12 V
damage threshold DIO (digital I/O)	-1 V	-	6 V
damage threshold Clk In and Clk Out	-5 V	-	5 V

Table 5.3. Host computer requirements

Parameter	Description
supported Windows operating systems	32-bit and 64-bit versions of XP, Vista, Windows 7, Windows 8
supported Linux distribution	32-bit and 64-bit of Linux, Ubuntu 12.04 LTS (i386, AMD64), 64-bit systems require the IA32 extension

Parameter	Description
minimum host computer requirements (for low bandwidth data transfer)	Windows XP 32-bit Dual Core CPU with SSE2 support 4 GB DRAM 1 Gbit/s Ethernet controller
recommended host computer requirements	Windows 7 64-bit or Linux 64-bit Quad Core CPU (i7) or better 8 GB DRAM or better 1 Gbit/s Ethernet controller with receive side scaling and Jumbo Frame support (4k); high data transfer rates can be obtained by using for instance Intel Ethernet Server Adapter I210-T1 SSD HD drive (for high-bandwidth data saving)
supported processors (requiring SSE2)	AMD K8 (Athlon 64, Sempron 64, Turion 64, etc.), AMD Phenom, Intel Pentium 4, Xeon Celeron, Celeron D, Pentium M, Celeron M, Core, Core 2, Core i5, Core i7, Atom

Table 5.4. Demodulator output sample rate to host computer.

Host computer connection	Active demodulators	Maximum sample rate per demodulator	Comments
1 GbE	1	1.6 MSa/s	To achieve highest rates, it is advised to remove all other data transfer that loads the LAN/USB interface. It is recommended to check the sample loss flag (in the status tab) from time to time when using high readout rate settings.
	2 - 4	800 kSa/s	
	5 - 8	400 kSa/s	
USB 2.0	1 - 2	400 kSa/s	
	3 - 6	200 kSa/s	
	7 - 8	100 kSa/s	

Note

The sample readout rate is the rate at which demodulated samples are transferred from the Instrument to the host computer. This rate has to be set to at least 2 times the signal bandwidth of the related demodulator in order to satisfy the Nyquist sampling theorem. As the total rate is limited by the USB/LAN interface, the maximum rate becomes smaller when the number of active demodulators is increased. This is summarized in the table above. An up-to-date and performing host computer is required to achieve these rates.

5.2. Analog Interface Specifications

Table 5.5. UHF signal inputs

Parameter	Conditions	min	typ	max
connectors	-	BNC, front panel single-ended		
input impedance	low value	-	50 Ω	-
	high value	-	1 MΩ // 16 pF	-
input frequency range	50 Ω termination	DC	-	600 MHz
input frequency range	1 MΩ termination	DC	-	100 MHz
input A/D conversion	-	12 bit, 1.8 GSa/s		
input noise amplitude	> 100 kHz, 10 mV range, 50 Ω termination	-	4 nV/√Hz	-
input bias current	50 Ω termination	-	10 µA	-
	1 MΩ termination	-	-	1 nA
input full range sensitivity (10 V lock-in amplifier output)	-	1 nV	-	1.5 V
input AC ranges	-	10 mV	-	1.5 V
input range (AC + common mode)	DC coupling	-1.5 V	-	1.5 V
	AC coupling	-3.5 V	-	3.5 V
AC coupling cutoff frequency	50Ω termination	-	320 kHz	-
	1 MΩ termination	-	80 Hz	-
input amplitude accuracy	< 100 MHz	-	3 %	-
	> 100 MHz	-	10 %	-
input amplitude stability	-	-	0.1 %/°C	-
input offset amplitude	with respect to range	-	-	5%
input harmonic distortion (HD2/HD3)	1 Vpp, 50 Ω termination, 10 minutes after manual input calibration < 1 MHz	-	-75 dB	-
	< 10 MHz	-	-70 dB	-
	< 100 MHz	-	-60 dB	-
	> 100 MHz	-	-50 dB	-
dynamic reserve		-	90 dB	100 dB

Table 5.6. UHF signal outputs

Parameter	Conditions	min	typ	max
connectors	-	BNC, front panel single-ended		
output impedance	-	-	50 Ω	-
output frequency range	-	DC	-	600 MHz

Parameter	Conditions	min	typ	max
output frequency resolution	-	-	6 µHz	-
output phase range	-	-180 °	-	180 °
output phase resolution	-	-	1.0 µ°	-
output D/A conversion	-	14 bit, 1.8 GSa/s		
output amplitude ranges	-	±150 mV, ±1.5 V		
output power	-	-	-	7.5 dBm
output amplitude accuracy	< 100 MHz	-	2%	-
	> 100 MHz	-	5%	-
output harmonic distortion (HD2/HD3)	1 Vpp, 50 Ω termination, < 1 MHz	-	-70 dB	-
	< 10 MHz	-	-70 dB	-
	< 100 MHz	-	-55 dB	-
	> 100 MHz	-	-42 dB	-
output noise amplitude	> 100 kHz	-	25 nV/√Hz	-
output phase noise	10 MHz, BW = 0.67 Hz, offset 100 Hz	-	-120 dBc/Hz	-
	10 MHz, BW = 0.67 Hz, offset 1 kHz	-	-130 dBc/Hz	-
output random jitter (RMS)	100 MHz, 6 dBm sine	-	4.5 ps	-
output offset amplitude	-	-5 mV	-	5 mV
output drive current	-	-	-	100 mA

Table 5.7. Reference signals and reference modes.

Parameter	Conditions	min	typ	max
connectors	-	BNC, front panel bidirectional SMA, back panel input SMA, back panel output		
input impedance (front and back panel)	low value	-	50 Ω	-
	high value	-	1 kΩ	-
input level at Ref / Trigger (front panel) and Trigger In (back panel)	low input impedance	-2.5 V	-	2.5 V
	high input impedance	-5 V	-	5 V
output impedance (front and back panel)	-	-	50 Ω	-
output level (front and back panel)	-	-	-	3.3 V TTL
input trigger hysteresis	-	-	100 mV	-
internal reference mode, output of reference on UHF outputs	frequency range	1 mHz	-	600 MHz
	reference orthogonality	-	0 °	-

Parameter	Conditions	min	typ	max
	reference acquisition time, lock time	instantaneous		
internal reference mode, output of reference on Ref / Trigger	frequency range	1 mHz	-	200 MHz
	reference orthogonality	-	0 °	-
	reference acquisition time, lock time	instantaneous		
external reference mode and auto reference mode, reference input at Signal Input 1 and 2	frequency range	10 Hz	-	600 MHz
	amplitude, note: for low-swing input signals the gain should be set to full-swing range to achieve best performance	100 mV	-	-
	amplitude (using UHF-PID option), note: for low-swing input signals the gain should be set to full-swing range to achieve best performance	10 mV	-	-
	reference acquisition time, lock time	-	-	100 reference cycles or 1.2 ms whatever is larger
external reference mode, reference input at Ref / Trigger	signal type	arbitrary, active at rising edge		
	frequency range	10 Hz	-	600 MHz
	amplitude	250 mV	-	-
	reference acquisition time, lock time	-	-	100 reference cycles or 1.2 ms, whatever is larger

Note

The UHF Instrument uses the same connectors for reference and trigger signals. This applies to input signals as well as output signals. Overall, the instrument features 2 output, 2 input, and 2 bidirectional connectors for reference and triggering purposes.

Table 5.8. Demodulators

Parameter	Details	min	typ	max
demodulator number	-		8	
demodulator harmonic setting range	-	1	-	1023
demodulator filter time constant	-	30 ns	-	76 s
demodulator measurement bandwidth	-	80 μHz	-	5 MHz
demodulator filter slope / roll-off	-		6, 12, 18, 24, 30, 36, 42, 48 dB/oct, consisting of up to 8 cascaded critical damping filters	
demodulator output resolution	-		X, Y, R, Θ with 64-bit resolution	
demodulator output sample rate (readout rate), for detailed specifications refer to Table 5.4	on auxiliary outputs	-	-	28 MSa/s
	USB 2.0 high speed	-	-	400 kSa/s
	1GbE, 1 Gbit/s LAN	-	-	1.6 MSa/s
demodulator harmonic rejection	-	110 dBc	-	-
group delay (lag time from Signal Input to Aux Output)	30 ns time constant and 1st order filter	-	-	3 μs

Table 5.9. Auxiliary Inputs and Outputs

Parameter	Details	min	typ	max
auxiliary output	connectors		BNC, 4 outputs on front-panel	
	sampling		28 MSa/s, 16-bit	
	bandwidth	-	-	7 MHz
	impedance	-	50 Ω	-
	amplitude	-10 V	-	10 V
	resolution	0.3 mV	-	-
	drive current	-	-	100 mA
auxiliary input	connectors		SMA, 2 inputs on back-panel	
	sampling		400 kSa/s, 16-bit	
	bandwidth	-	-	100 kHz
	impedance	-	1 MΩ	-
	amplitude	-10 V	-	10 V
	resolution	0.3 mV	-	-

Table 5.10. Oscillator and clocks

Parameter	Details	min	typ	max
internal clock (ovenized crystal)	initial accuracy	-	±0.5 ppm	±1 ppm
	long term accuracy / aging	-	-	±0.4 ppm/year
	short term stability (1 s)	0.00005 ppm	-	-

Parameter	Details	min	typ	max
	short term stability (100 s)	0.0005 ppm	-	-
	temperature coefficient (23° ± 5°)	-	-	±0.03 ppm/°
	phase noise (at 100 Hz)	-	-130 dBc/Hz	-
	phase noise (at 1 kHz)	-	-140 dBc/Hz	-
	warm-up time	-	-	60 s
UHF-RUB Rubidium clock (option)	initial accuracy at 25°	-	-	±0.0005 ppm
	long term accuracy / aging	-	-	±5e-6 ppm/day ±0.0005 ppm/year
	short term stability, AVAR (1 s)	0.00008 ppm	-	-
	short term stability, AVAR (100 s)	0.000008 ppm	-	-
	temperature coefficient (25° ± 25°)	-	-	±0.0005 ppm/°
	phase noise (at 100 Hz)	-	-	-
	phase noise (at 1 kHz)	-	-140 dBc/Hz	-
	warm-up time	-	-	300 s @ 25°C
clock input	connector	SMA, on back-panel		
	impedance	-	50 Ω	-
	amplitude	200 mV	320 mV	1 V
	frequency	9.98 MHz	10 MHz	10.02 MHz
clock output	connector	SMA, on back-panel		
	impedance	-	50 Ω	-
	amplitude, 50 Ω	250 mV	500 mV	1 V
	frequency	-	10 MHz	-

5.3. Digital Interface Specifications

Table 5.11. Digital interfaces

Parameter	Description
host computer connection	USB 2.0 high-speed, 480 Mbit/s
	1GbE, LAN / Ethernet, 1 Gbit/s
DIO port	4 x 8 bit, general purpose digital input/output port, 5 V TTL specification
ZCtrl peripheral port	2 connectors for ZI proprietary bus to control external peripherals

5.3.1. DIO Port

The DIO port is a VHDCI 68 pin connector as introduced by the SPI-3 document of the SCSI-3 specification. It is a female connector that requires a 32 mm wide male connector. The DIO port features 32 bits that can be configured byte-wise as inputs or outputs.

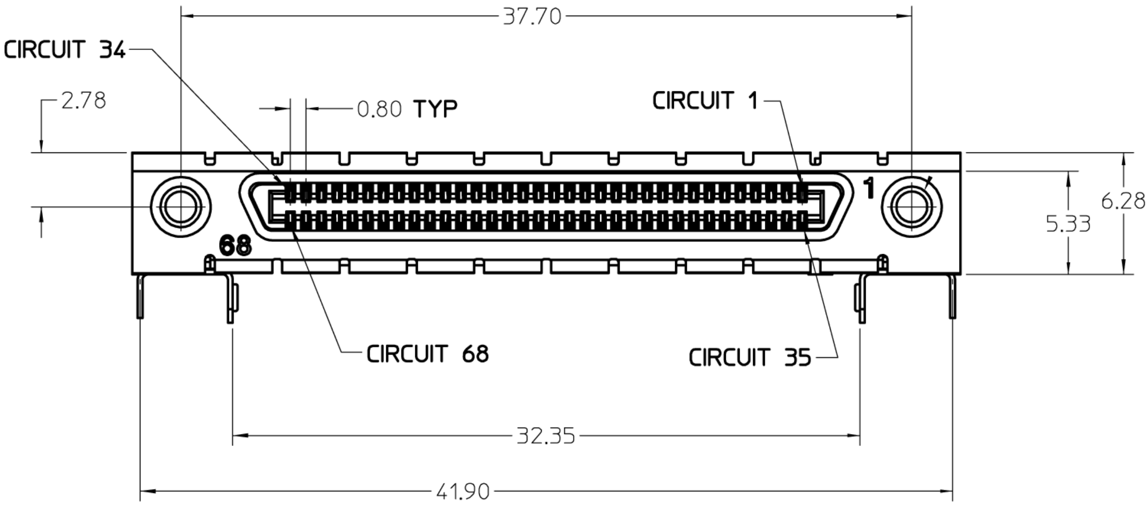


Figure 5.1. DIO HD 68 pin connector

Table 5.12. DIO pin assignment

Pin	Name	Description	Range specification
68	CLKI	clock input, used to latch signals at the digital input ports - can also be used to retrieve digital signals from the output port using an external sampling clock	5 V CMOS/TTL
67	DOL	DIO output latch, 56.25 MHz clock signal, the digital outputs are synchronized to the falling edge of this signal	5 V CMOS
66-59	DI[31:24]	digital input or output (set by user)	output CMOS 5 V, input is CMOS/TTL

Pin	Name	Description	Range specification
58-51	DIO[23:16]	digital input or output (set by user)	output CMOS 5 V, input is CMOS/TTL
50-43	DIO[15:8]	digital input or output (set by user)	output CMOS 5 V, input is CMOS/TTL
42-35	DIO[7:0]	digital input or output (set by user)	output CMOS 5 V, input is CMOS/TTL
34-1	GND	digital ground	-

The figure below shows the architecture of the DIO input/output. The DIO port features 32 bits that can be configured byte-wise as inputs or outputs by means of a drive signal. The digital output data is latched synchronously with the falling edge of the internal clock, which is running at 56.25 MHz. The internal sampling clock is available at the DOL pin of the DIO connector. Digital input data can either be sampled by the internal clock or by an external clock provided through the CLKI pin. A decimated version of the input clock is used to sample the input data. The Decimation unit counts the clocks to decimation and then latches the input data. The default decimation is 5625000, corresponding to a digital input sampling rate of 1 sample per second.

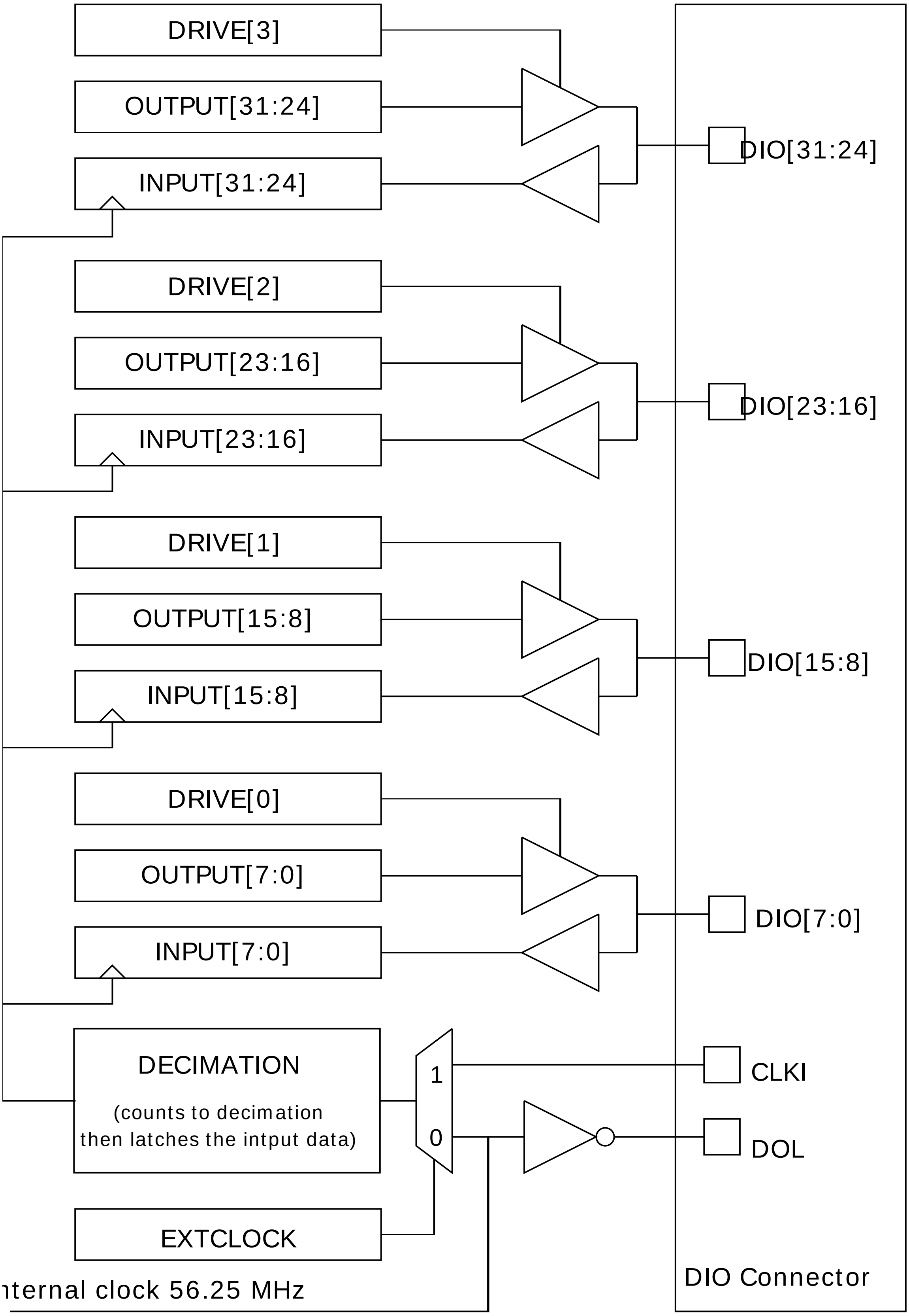


Figure 5.2. DIO input/output architecture

5.3.2. ZCtrl Peripheral Port

The ZCtrl port serves to power and communicate to external equipments, such as pre-amplifiers: the port provides a floating power supply with ±14.5 V and 100 mA per port. After Instrument power-on, the port is not active and must be switched on in order to be used. Two activation methods are supported:

- Manual switch in the user interface
- Manual switch by shorting the ZCtrl_Detect and Device_Ground - these pins should be floating against ZCtrl_GND and ZCtrl_PWR

Th ZCtrl port can be connected with an RJ45 connector, therefore non-crossed Ethernet cables can be used for convenient interfacing.

Warning

Connection to a Ethernet might damage the UHF Instrument.

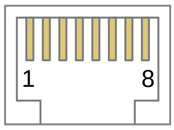


Figure 5.3. The pinout of the ZCtrl port

Table 5.13. DIO port pin assignment

Pin	Name	Description	Range specification
1	ZCtrl_Power+	power pin, for external use	14.5 V, 100 mA
2	ZCtrl_Detect	connection detection	-
3	Device_Ground	ground of UHF Instrument, connected to earth pin	-
4	ZCtrl_Power-	power pin, for external use	-14.5 V, 100 mA
5	ZCtrl_D	proprietary function	-
6	ZCtrl_C	proprietary function	-
7	ZCtrl_GND	floating input	-
8	ZCtrl_GND	reference ground pin for ZCtrl_Power+ and ZCtrl_Power-	-

5.4. Performance Diagrams

Many of the parameters mentioned in Section 5.2 are valid without specific conditions. Other parameters instead are typical specifications, because they depend on several parameters, such as the input range setting, the input termination and/or the frequency. This section completes the previous chapters with detailed performance diagrams in order to support the validation of applications.

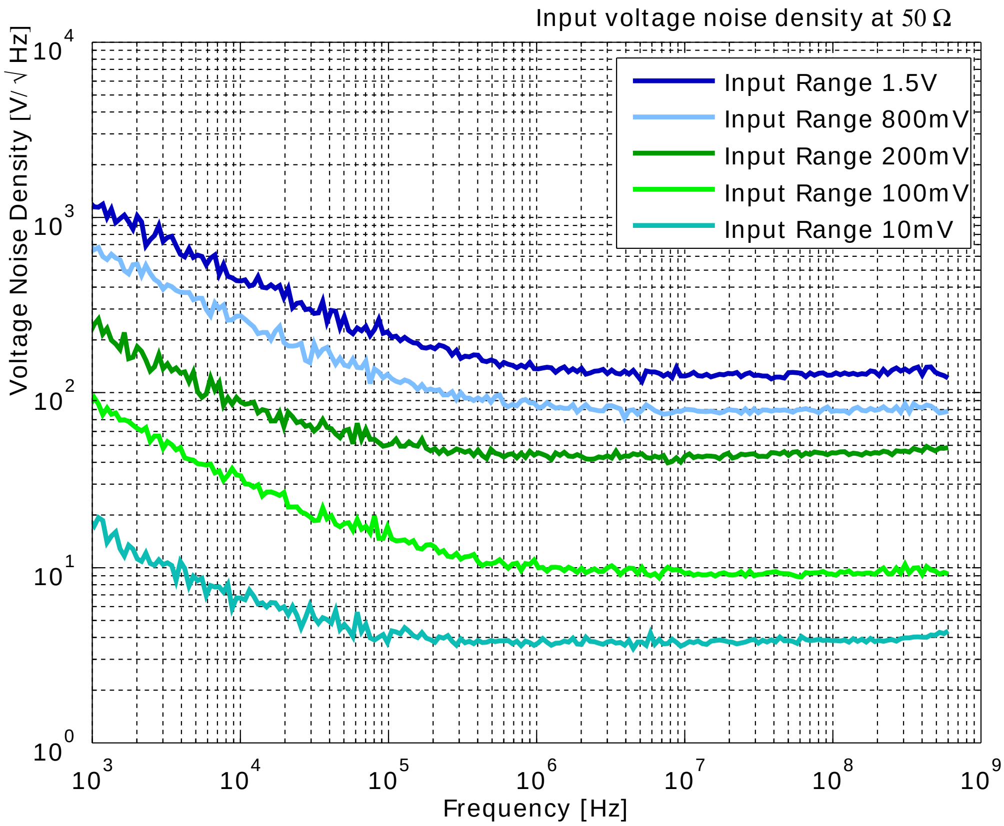


Figure 5.4. UHFLI input noise with 50Ω input impedance

Input noise amplitude depends on several parameters, and in particular on the frequency and the input range setting. The input noise is lower for smaller input ranges, and it is recommended to use small ranges especially for noise measurements. Only the noise with DC input coupling is shown here as the input noise with AC coupling is the same, as long as the frequency is above the AC cutoff frequency (see Table 5.5).

The input noise does depend on the input impedance setting, which can be 50 Ω or 1 MΩ. The performance diagrams for 50 Ω and 1 MΩ input impedance are shown in Figure 5.4 and in Figure 5.5, respectively. For both, the corner frequency of the 1/f noise is in the range of 100 kHz.

For 50 Ω input impedance, the white noise floor is around 4 nV/√Hz for the smallest input range. For 1 MΩ input impedance, the white noise floor is below 8 nV/√Hz for the smallest input range.

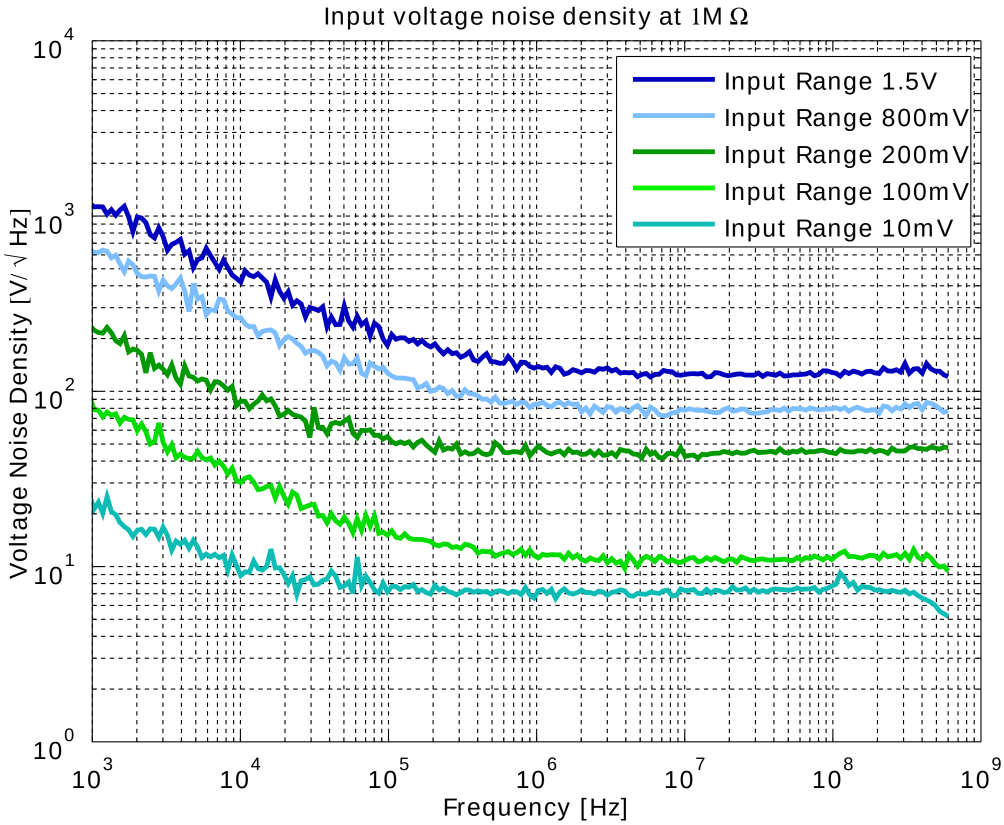


Figure 5.5. UHFLI input noise with 1 MΩ input impedance

5.5. Clock 10 MHz

A 10 MHz clock input and output is provided for synchronization with other instruments. The figure explains the internal routing of the different clock signals. An internal clock generation unit receives a 10 MHz clock reference and generates all necessary device internal sampling clocks. The clock reference either comes from the internal quartz/Rubidium oscillator or from an external clock source connected to the Clock 10 MHz In connector. The user can define if the clock is taken from the internal or external source. The Clock 10 MHz Out connector always provides the 10 MHz clock of the internal quartz/Rubidium oscillator.

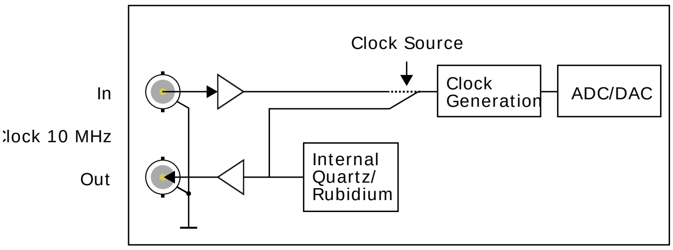


Figure 5.6. Clock routing

5.6. Device Self Calibration Procedure

The device requires a self calibration after a short warm-up period to ensure operation according to specifications. During this self calibration process, components of the sensitive analog front-end are calibrated to account for temperature variations and drift. It is worth noting, that self calibration has nothing to do with the device calibration, which is done at the manufacturer site. The self calibration lasts about one second and only applies a fine-tuning.

The first self calibration after warm-up is executed automatically. Any further self calibration needs to be manually executed by the user. The self calibration process can be executed by means of clicking the "Run" button of the Auto Calibration in the Device tab of the user interface.

The user can disable the calibration procedure completely if necessary. This can be done by changing the Enabled button of the Auto Calibration in the device tab. If this flag is disabled, no calibration is executed after warm-up time.

The default self calibration procedure can be divided into three different states, which are also indicated by the CAL flag in the footer of the user interface. The CAL flag can be either yellow, gray/off, or red.

- Yellow: The yellow CAL flag indicates, that the calibration has not been executed yet. After a warm-up and temperature settling period of approximately 16 minutes, a self calibration is executed and the CAL flag turns gray. If the self calibration is disabled, the CAL flag turns red after the warm-up period to indicate that no calibration was run.
- Gray/off: The gray CAL flag indicates, that the device is self calibrated. The CAL flag turns red when the temperature change is larger than a given threshold or the time since the last calibration is longer than a given time interval. The values of these thresholds are indicated in the device tab.
- Red: The red CAL flag indicates, that it is recommended to run a self calibration. The self calibration is never executed automatically in this state. The CAL flag is red, either, when the device experienced a temperature change, which is larger than a given threshold, or, when the time since the last calibration is longer than a given time interval. By executing a self calibration, the CAL flag will turn gray.

Chapter 6. Signal Processing Basics

This chapter provides insights about several lock-in amplifier principles not necessarily linked to a specific instrument from Zurich Instruments. Since the appearance of the first valve-based lock-in amplifiers in the 1930s the physics have not changed, but the implementation and the performance have evolved greatly. Many good lock-in amplifier primers have appeared in the past decades, and some of them appear outdated now because they were written with analog instruments in mind. This section does not aim to replace any existing primer, but to complete them with a preferred emphasis on digital lock-in amplifiers.

The first subsection describes the principles of lock-in amplification, followed by the description of the function of discrete-time filters. After, we discuss the definition of the full range sensitivity, a specification parameter particularly important for analog lock-in amplifiers but with somewhat reduced importance for digital instruments. In the following, we describe the function and use of sinc filtering in particular for low-frequency lock-in measurements. The last section is dedicated to the zoom FFT feature. Innovative in the context of lock-in amplifiers, zoom FFT offers a fast and high-resolution spectral analysis around the lock-in operation frequency.

6.1. Principles of Lock-in Detection

Lock-in demodulation is a technique to measure the amplitude A_s and the phase Θ of a periodic signal with the frequency $\omega_s = 2\pi f_s$ by comparing the signal to a reference signal. This technique is also called phase-sensitive detection. By averaging over time the signal-to-noise ratio (SNR) of a signal can be increased by orders of magnitude, allowing very small signals to be detected with a high accuracy making the lock-in amplifier a tool often used for signal recovery. For both signal recovery and phase-sensitive detection, the signal of interest is isolated with narrow band-pass filtering therefore reducing the impact of noise in the measured signal.

Figure 6.1 shows a basic measurement setup: a reference V_r signal is fed to the device under test. This reference signal is modified by the generally non-linear device with attenuation, amplification, phase shifting, and distortion, resulting in a signal $V_s = A_s \cos(\omega_s t + \Theta_s)$ plus harmonic components.

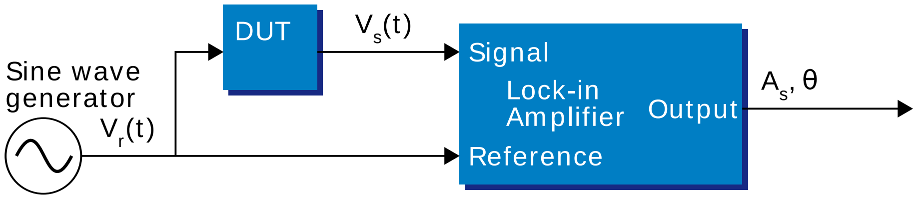


Figure 6.1. Basic measurement setup incorporating a lock-in amplifier

For practical reasons, most lock-in amplifiers implement the band-pass filter with a mixer and a low-pass filter (depicted in Figure 6.2): the mixer shifts the signal of interest into the baseband, ideally to DC, and the low-pass filter cuts all unwanted higher frequencies.

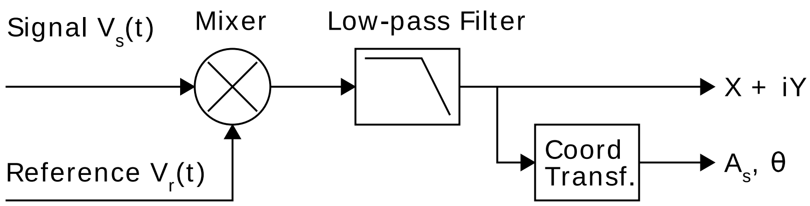


Figure 6.2. Mixing and low-pass filtering performed by the lock-in amplifier

The input signal $V_s(t)$ is multiplied by the reference signal $V_r(t) = \sqrt{2}\, e^{-i\omega_r t}$, where $\omega_r = 2\pi f_r$ is the demodulation frequency and i is the imaginary unit. This is the complex representation of a sine and cosine signal (phase shift 90°) forming the components of a quadrature demodulator, capable of measuring both the amplitude and the phase of the signal of interest. In principle it is possible to multiply the signal of interest with any frequency, resulting in a heterodyne operation. However the objective of the lock-in amplifier is to shift the signal as close as possible to DC, therefore the frequency of the reference and the signal is chosen similar. In literature this is called homodyne detection, synchrodyne detection, or zero-IF direct conversion.

The result of the multiplication is the signal

$$V_s(t)\cdot V_r(t) = V_s(t)\cdot\sqrt{2}\,e^{-i\omega_r t} = \frac{A_s}{\sqrt{2}}\,e^{+i[(\omega_s-\omega_r)t+\Theta]} + \frac{A_s}{\sqrt{2}}\,e^{-i[(\omega_s+\omega_r)t+\Theta]}$$

Equation 6.1. Multiplication of signal of interest with reference signal

It consists of a slow component with frequency $\omega_s-\omega_r$ and a fast component with frequency $\omega_s+\omega_r$.

The demodulated signal is then low-pass filtered with an infinite impulse response (IIR) RC filter, indicated by the symbol $\langle \cdot \rangle$. The frequency response of the filter F(ω) will let pass the low frequencies $F(\omega_s-\omega_r)$ while considerably attenuating the higher frequencies $F(\omega_s+\omega_r)$. Another way to consider the low-pass filter is an averager.

$$X + iY = \left\langle V_s(t) \cdot \sqrt{2}\, e^{-i\omega_r t} \right\rangle \approx F(\omega_s - \omega_r) \frac{A_s}{\sqrt{2}} e^{i[(\omega_s-\omega_r)t+\Theta]}$$

Equation 6.2. Averaging the result of the signal multiplication

The result after the low-pass filter is the demodulated signal X+iY, where X is the real and Y is the imaginary part of a signal depicted on the complex plane. These components are also called in-phase and quadrature components. The transformation of X and Y into the amplitude R and phase Θ information of $V_s(t)$ can be performed with trigonometric operations.

It is interesting to note that the value of the measured signal corresponds to the RMS value of the signal, which is equivalent to $R = A_s/\sqrt{2}$.

Most lock-in amplifiers output the values (X,Y) and (R, Θ) encoded in a range of −10 V to +10 V of the auxiliary output signals.

6.1.1. Lock-in Amplifier Applications

Lock-in amplifiers are employed in a large variety of applications. In some cases the objective is measuring a signal with good signal-to-noise ratio, and then that signal could be measured even with large filter settings. In this context the word phase sensitive detection is appropriate. In other applications, the signal is very weak and overwhelmed by noise, which forces to measure with very narrow filters. In this context the lock-in amplifier is employed for signal recovery. Also, in another context, a signal modulated on a very high frequency (GHz or THz) that cannot be measured with standard approaches, is mixed to a lower frequency that fits into the measurement band of the lock-in amplifier.

One example for measuring a small, stationary or slowly varying signal which is completely buried in the 1/f noise, the power line noise, and slow drifts. For this purpose a weak signal is modulated to a higher frequency, away from these sources of noise. Such signal can be efficiently mixed back and measured in the baseband using a lock-in amplifier. In Figure 6.3 this process is depicted. Many optical applications perform the up-mixing with a chopper, an electro-optical modulator, or an acousto-optical modulator. The advantage of this procedure is that the desired signal is measured in a spectral region with comparatively little noise. This is more efficient than just low-pass filtering the DC signal.

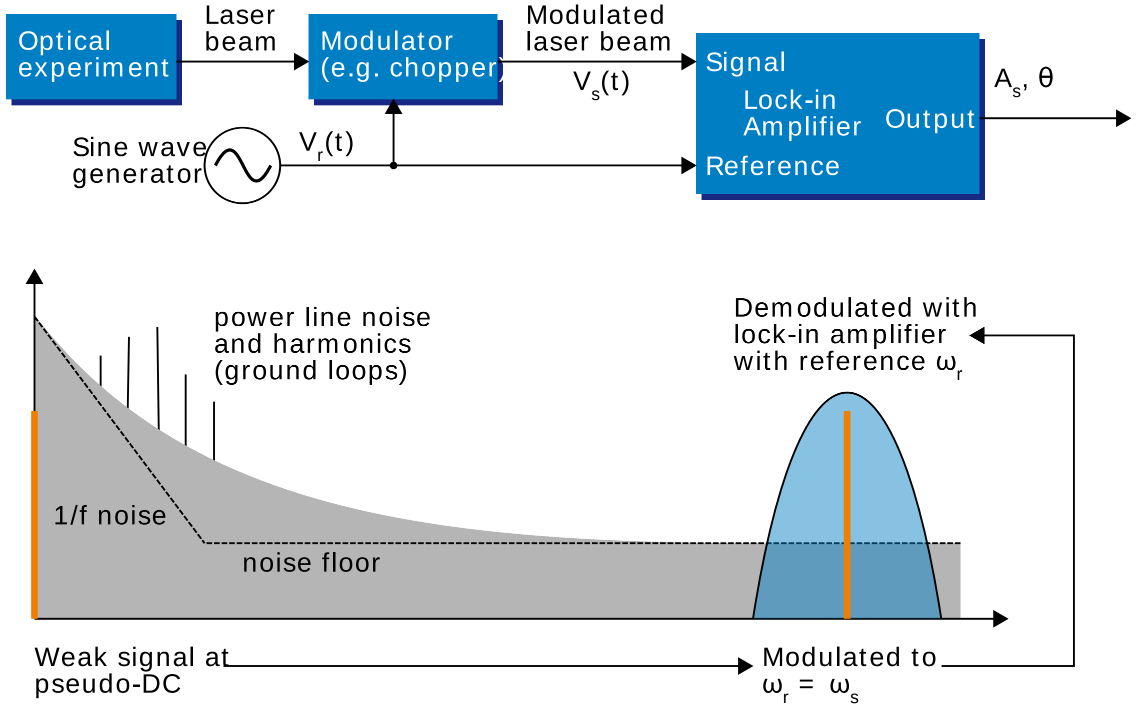


Figure 6.3. Lock-in measurement of a noisy DC signal

6.2. Signal Bandwidth

The signal bandwidth (BW) theoretically corresponds to the highest frequency components of interest in a signal. In practical signals, the bandwidth is usually quantified by the cut-off frequency. It is the frequency at which the transfer function of a system shows 3 dB attenuation relative to DC (BW = $f_{\text{cut-off}}$ = $f_{\text{-3dB}}$); that is, the signal power at $f_{\text{-3dB}}$ is half the power at DC. The bandwidth, equivalent to cut-off frequency, is used in the context of dynamic behavior of a signals or separation of different signals. This is for instance the case for fast-changing amplitudes or phase values like in a PLL or in a imaging application, or when signals closely spaced in frequency need to be separated.

The noise equivalent power bandwidth (NEPBW) is also a useful figure, and it is distinct from the signal bandwidth. This unit is typically used for noise measurements: in this case one is interested in the total amount of power that passes through a low-pass filter, equivalent to the area under the solid curve in Figure 6.4. For practical reasons, one defines an ideal brick-wall filter that lets pass the same amount of power under the assumption that the noise has a flat (white) spectral density. This brick-wall filter has transmission 1 from DC to f_{NEPBW}. The orange and blue areas in Figure 6.4 then are exactly equal in a linear scale.

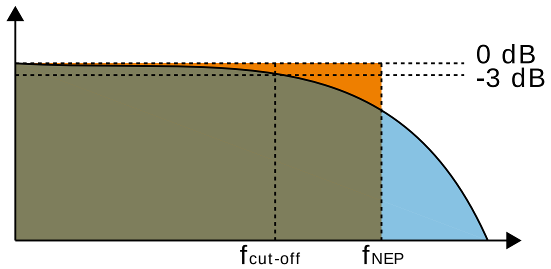


Figure 6.4. Signal bandwidth and noise equivalent power bandwidth

It is possible to establish a simple relation between the $f_{\text{cut-off}}$ and the f_{NEPBW} that only depends on the slope (or roll-off) of the filter. As the filter slope actually depends on the time constant (TC) defined for the filter, it is possible to establish the relation also to the time constant. It is intuitive to understand that for higher filter orders, the $f_{\text{cut-off}}$ is closer to the f_{NEPBW} than for smaller orders.

The time constant is a parameter used to interpret the filter response in the time domain, and relates to the time it takes to reach a defined percentage of the final value. The time constant of a low-pass filter relates to the bandwidth according to the formula

$$\mathrm{TC} = \frac{\mathrm{FO}}{2\pi f_{\text{cut-off}}} \tag{6.3}$$

where FO is said factor that depends on the filter slope. This factor, along with other useful conversion factors between different filter parameters, can be read from the following table.

Table 6.1. Summary of conversion factors for bandwidth definitions

filter order	filter roll-off	FO	$f_{\text{cut-off}}$	f_{NEPBW}	f_{NEPBW} / $f_{\text{cut-off}}$
1st	6 dB/oct	1.000	0.159 / TC	0.250 / TC	1.57
2nd	12 dB/oct	0.644	0.102 / TC	0.125 / TC	1.22
3rd	18 dB/oct	0.510	0.081 / TC	0.094 / TC	1.15
4th	24 dB/oct	0.435	0.068 / TC	0.078 / TC	1.12
5th	30 dB/oct	0.386	0.062 / TC	0.068 / TC	1.11

filter order	filter roll-off	FO	$f_{cut\text{-}off}$	f_{NEPBW}	f_{NEPBW} / $f_{cut\text{-}off}$
6th	36 dB/oct	0.350	0.056 / TC	0.062 / TC	1.10
7th	42 dB/oct	0.323	0.051 / TC	0.056 / TC	1.10
8th	48 dB/oct	0.301	0.048 / TC	0.052 / TC	1.09

6.3. Discrete-Time Filters

6.3.1. Discrete-Time RC Filter

There are many options how to implement digital low-pass filters. One common filter type is the exponential running average filter. Its characteristics are very close to those of an analog resistor-capacitor RC filter, which is why this filter is sometimes called a discrete-time RC filter. The exponential running average filter has the time constant TC = τ_N as its only adjustable parameter. It operates on an input signal $X_{in}[n,T_S]$ defined at discrete times nT_S, $(n+1)T_S$, $(n+2)T_S$, etc., spaced at the sampling time T_S. Its output $X_{out}[n,T_S]$ can be calculated using the following recursive formula,

$$X_{out}[n,T_S] = e^{-T_S/\tau_N} X_{out}[n-1,T_S] + \left(1-e^{-T_S/\tau_N}\right) X_{in}[n,T_S]$$

Equation 6.4. Time domain response of the discrete-time RC filter

The response of that filter in the frequency domain is well approximated by the formula

$$H_1(\omega) = \frac{1}{1+i\cdot\omega\cdot\tau_N}$$

Equation 6.5. Frequency domain response of the first-order discrete-time RC filter

The exponential filter is a first-order filter. Higher-order filters can easily be implemented by cascading several filters. For instance the 4th order filter is implemented by chaining 4 filters with the same time constant TC = τ_N one after the other so that the output of one filter stage is the input of the next one. The transfer function of such a cascaded filter is simply the product of the transfer functions of the individual filter stages. For an n-th order filter, we therefore have

$$H_n(\omega) = \frac{1}{(1+i\cdot\omega\cdot\tau_N)^n}$$

Equation 6.6. Frequency domain response of the n-th order discrete-time RC filter

The attenuation and phase shift of the filters can be obtained from this formula. Namely, the filter attenuation is given by the absolute value squared $|H_n(\omega)|^2$. The filter transmission phase is given by the complex argument $\arg[H_n(\omega)]$.

6.3.2. Filter Settling Time

The low-pass filters after the demodulator cause a delay to measured signals depending on the filter order and time constant TC = τ_N. After a change in the signal, it will therefore take some time before the lock-in output reaches the correct measurement value. This is depicted in Figure 6.5 where the response of cascaded filters to a step input signal this is shown.

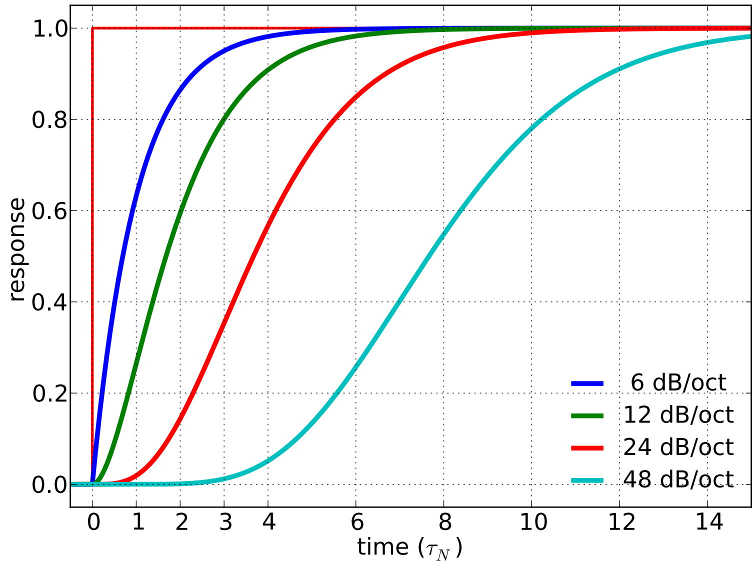


Figure 6.5. Time domain step response of the RC low-pass filters

More quantitative information on the settling time can be obtained from Table 6.2. In this table, you find settling times in units of the filter time constant τ_N for all filter orders available with the UHFLI Lock-in Amplifier. The values tell the time you need to wait for the filtered demodulator signal to reach 5%, 95% and 99% of the final value. This can help in making a quantitatively correct choice of filter parameters for example in a measurement involving a parameter sweep.

Table 6.2. Summary of Filter Rise Times

filter order	Setting time to		
	5%	95%	99%
1^{st}	0.025 · TC	3.0 · TC	4.6 · TC
2^{nd}	0.36 · TC	4.7 · TC	6.6 · TC
3^{rd}	0.82 · TC	6.3 · TC	8.4 · TC
4^{th}	1.4 · TC	7.8 · TC	10 · TC
5^{th}	2.0 · TC	9.2 · TC	12 · TC
6^{th}	2.6 · TC	11 · TC	12 · TC
7^{th}	3.3 · TC	12 · TC	15 · TC
8^{th}	4.0 · TC	13 · TC	16 · TC

6.4. Full Range Sensitivity

The sensitivity of the lock-in amplifier is the RMS value of an input sine that is demodulated and results in a full scale analog output. Traditionally the X, Y, or R components are mapped onto the 10 V full scale analog output. In such a case, the overall gain from input to output of the lock-in amplifier is composed of the input and output amplifier stages. Many lock-in amplifiers specify a sensitivity between 1 nV and 1 V. In other words the instrument permits an input signal between 1 nV and 1 V to be amplified to the 10 V full range output.

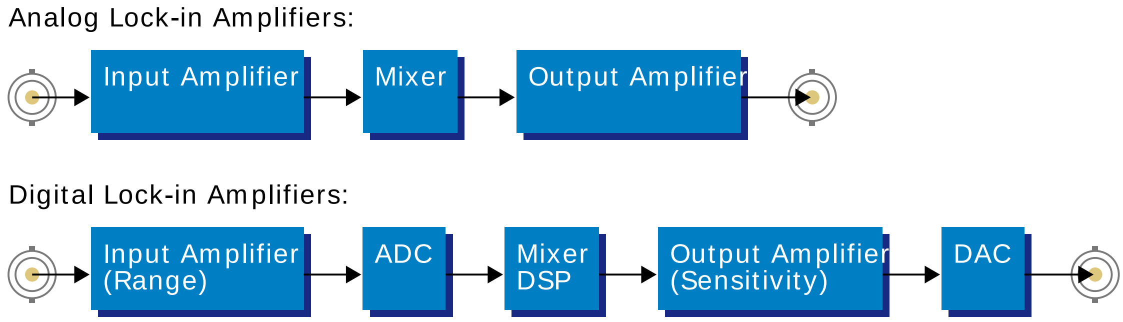


Figure 6.6. Sensitivity from signal input to signal output

In analog lock-in amplifiers the sensitivity is simple to understand. It is the sum of the analog amplification stages between in the input and the output of the instrument: in particular the input amplifier and the output amplifier.

In digital lock-in amplifiers the sensitivity less straightforward to understand. Analog-to-digital converters (ADC) operate with a fixed input range (e.g. 1 V) and thus require a variable-gain amplifier to amplify the input signal to the range given by the ADC. This variable-gain amplifier must be in the analog domain and its capability determines the minimum input range of the instrument. A practical analog input amplifier provides a factor 1000 amplification, thus 1 V divided by 1000 is the minimum input range of the instrument.

The input range is the maximum signal amplitude that is permitted for a given range setting. The signal is internally amplified with the suited factor, e.g. (1 mV)·1000 to result in a full swing signal at the ADC. For signals larger than the range, the ADC saturates and the signal is distorted – the measurement result becomes useless. Thus the signal should never exceed the range setting.

But the input range is not the same as the sensitivity. In digital lock-in amplifiers the sensitivity is only determined by the output amplifier, which is an entirely digital signal processing unit which performs a numerical multiplication of the demodulator output with the scaling factor. The digital output of this unit is then fed to the output digital-to-analog converter (DAC) with a fixed range of 10 V. It is this scaling factor that can be retrofitted to specify a sensitivity as known from the analog lock-in amplifiers. A large scaling factor, and thus a high sensitivity, comes at a relatively small expense for digital amplification.

One interesting aspect of digital lock-in amplifiers is the connection between input resolution and sensitivity. As the ADC operates with a finite resolution, for instance 14 bits, the minimum signal that can be detected and digitized is for instance 1 mV divided by the resolution of the ADC. With 14 bits the minimum level that can be digitized would be 122 nV. How is it possible to reach 1 nV sensitivity without using a 21 bit analog-to-digital converter? In a world without noise it is not possible. Inversely, thanks to noise and current digital technology it is possible to achieve a sensitivity even below 1 nV.

Most sources of broadband noise, including the input amplifier, can be considered as Gaussian noise sources. Gaussian noise is equally distributed in a signal, and thus generates equally distributed disturbances. The noise itself can be filtered by the lock-in amplifier down to a level where it does not impact the measurement. Still, in the interplay with the signal, the noise does have an effect on the measurement. The input of the ADC is the sum of the noise and the signal amplitude. Every now and then, the signal amplitude on top of the large noise will be able to toggle the least significant bits even for very small signals, as low as 1 nV and below. The resulting digital signal has a component at the signal frequency and can be detected by the lock-in amplifier.

There is a similar example from biology. Rod cells in the human eye permit humans to see in very low light conditions. The sensitivity of rod cells in the human eye is as low as a single photon. This sensitivity is achieved in low light conditions by a sort of pre-charging of the cell to be sensitive to the single photon that triggers the cell to fire an impulse. In a condition with more surround light, rod cells are less sensitive and need more photons to fire.

To summarize, in digital lock-in amplifiers the full range sensitivity is only determined by the scaling factor capability of the digital output amplifier. As the scaling can be arbitrary big, 1 nV minimum full range sensitivity is achievable without a problem. Further, digital lock-in amplifiers exploit the input noise to heavily increase the sensitivity without impacting the accuracy of the measurement.

6.5. Sinc Filtering

As explained in Section 6.1, the demodulated signal in an ideal lock-in amplifier has a signal component at DC and a spurious component at twice the demodulation frequency. The components at twice the demodulation frequency (called the 2ω component) is effectively removed by regular low-pass filtering. By selecting filters with small bandwidth and faster roll-offs, the 2ω component can easily be attenuated by 100 dB or more. The problem arises at low demodulation frequencies, because this forces the user to select long integration times (e.g. >60 ms for a demodulation frequency of 20 Hz) in order to achieve the same level of 2ω attenuation.

In practice, the lock-in amplifier will modulate DC offsets and non-linearities at the signal input with the demodulation frequency, resulting in a signal at the demodulation frequency (called ω component). This component is also effectively removed by the regular low-pass filters at frequencies higher than 1 kHz.

At low demodulation frequencies, and especially for applications with demodulation frequencies close to the filter bandwidth, the ω and 2ω components can affect the measurement result. Sinc filtering allows for strong attenuation of the ω and 2ω components. Technically the sinc filter is a comb filter with notches at integer multiples of the demodulation frequency (ω, 2ω, 3ω, etc.). It removes the ω component with a suppression factor of around 80 dB. The amount of 2ω component that gets removed depends on the input signal. It can vary from entirely (e.g. 80 dB) to slightly (e.g. 5 dB). This variation is not due to the sinc filter performance but depends on the bandwidth of the input signal.

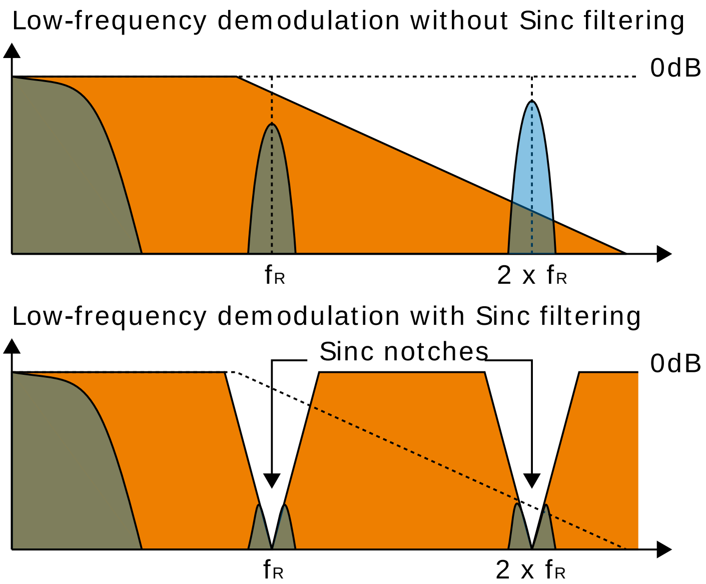


Figure 6.7. Effect of sinc filtering

Table 6.3. Artifacts in the demodulation signal

Input signal	Demodulation result before low-pass filter	Result
Signal at ω	DC component	Amplitude and phase information (wanted signal)
	2ω component	Unwanted component (can additionally be attenuated by sinc filter)

Input signal	Demodulation result before low-pass filter	Result
DC offset	ω component	Unwanted component (can additionally be attenuated by sinc filter)

We can observe the effect of the sinc filter by using the Spectrum Analyzer Tool of the UHFLI Lock-in Amplifier. As an example, consider a 30 Hz signal with an amplitude of 0.1 V that demodulated using a filter bandwidth of 100 Hz and a filter order 8. In addition 0.1 V offset is added to the signal so that we get a significant ω component.

Figure 6.8 shows a spectrum with the sinc filter disabled, whereas for Figure 6.9 the sinc filter is enabled. The comparison of the two clearly shows how the sinc options dampens both the ω and 2ω components by about 100 dB.

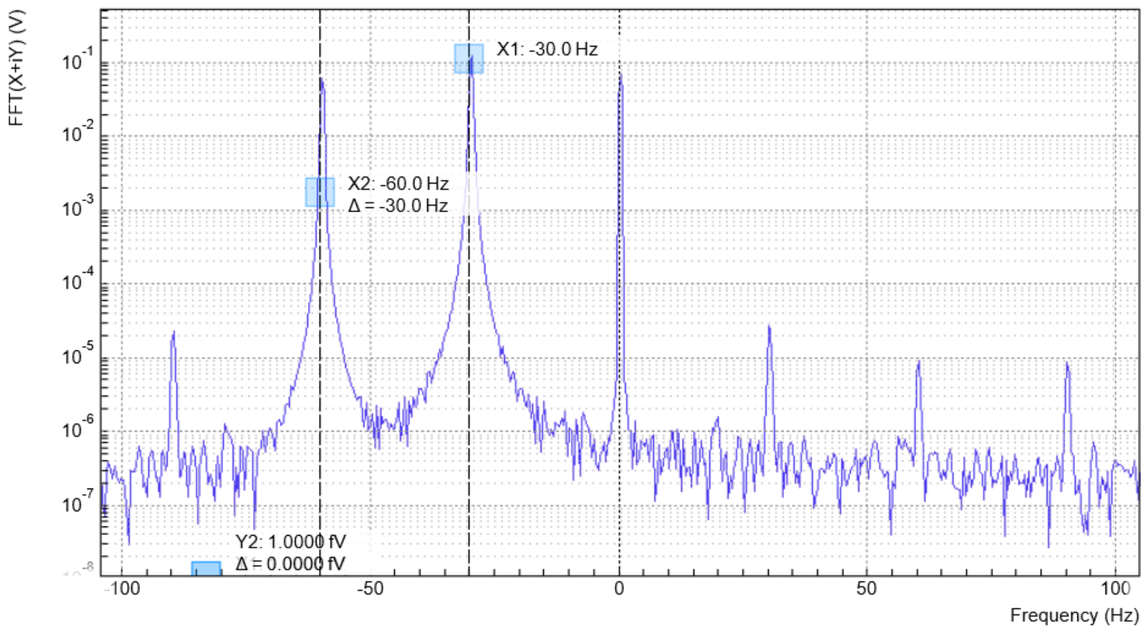


Figure 6.8. Spectrum of a demodulated 30 Hz signal without sinc filter

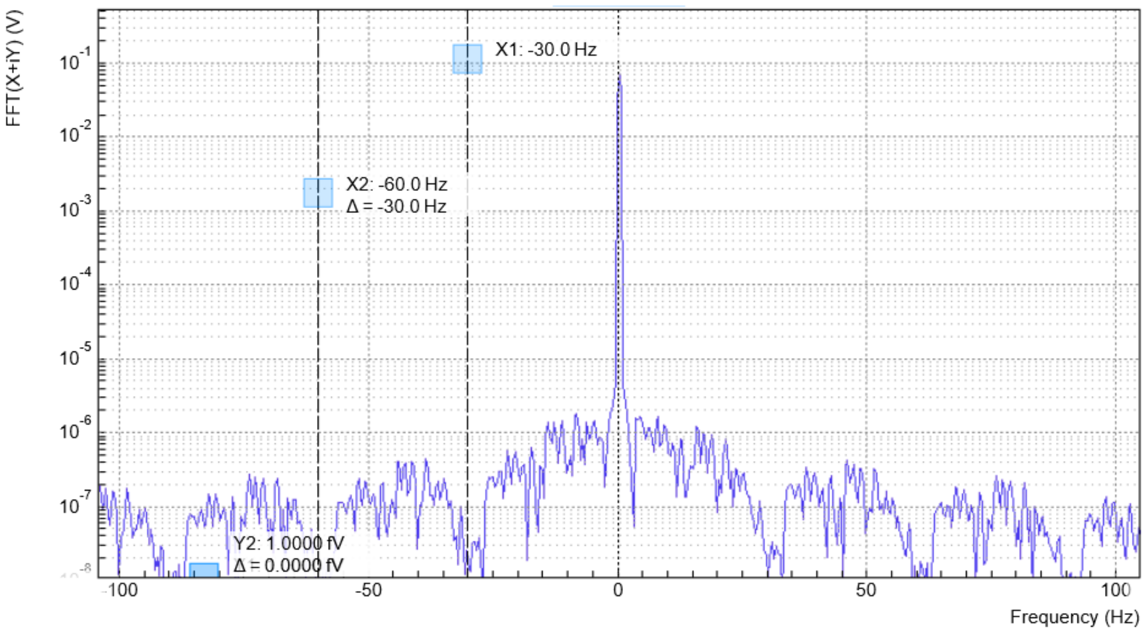


Figure 6.9. Spectrum of a demodulated 30 Hz signal with sinc filter

Note

In order to put the notches of the digital filter to ω and 2ω, the sampling rate of the filter would have to be precisely adjusted to the signal frequency. As this is technically not feasible, the generated signal frequency is adjusted instead by a very small amount.

6.6. Zoom FFT

The concept of zoom FFT allows the user to analyze the spectrum of the input signal around a particular frequency by zooming in on a narrow frequency portion of the spectrum. This is done by performing a Fourier transform of the demodulated in-phase and quadrature (X and Y) components or more precisely, on the complex quantity X+iY, where i is the imaginary unit. In the LabOne user interface, this functionality is available in the Spectrum tab.

In normal FFT, the sampling rate determines the frequency span and the total acquisition time determines the frequency resolution. Having a large span and a fine resolution at the same time then requires long acquisition times at high sample rates. This means that a lot of data needs to be acquired, stored, and processed, only to retain a small portion of the spectrum and discard most of it in the end. In zoom FFT, the lock-in demodulation is used to down-shift the signal frequency, thereby allowing one to use both a much lower sampling rate and sample number to achieve the same frequency resolution. Typically, to achieve a 1 Hz frequency resolution at 1 MHz, FFT would require to collect and process approximately 10^6 points, while zoom FFT only processes 10^3 points. (Of course the high rate sampling is done by the lock-in during the demodulation stage, so the zoom FFT still needs to implicitly rely on a fast ADC.)

In order to illustrate why this is so and what benefits this measurement tool brings to the user, it is useful to remind that at the end of the demodulation of the input signal $V_s(t) = A_s\cos(\omega_s t + \Theta)$, the output signal is $X + iY = F(\omega_s - \omega_r)(A_s/\sqrt{2})e^{i[(\omega_s-\omega_r)t+\Theta]}$ where F(ω) is the frequency response of the filters.

Since the demodulated signal has only one component at frequency $\omega_s-\omega_r$, its power spectrum (Fourier transform modulus squared) has a peak of height $(|A_s|^2/2)\cdot|F(\omega_r - \omega_s)|^2$ at $\omega_s-\omega_r$: this tells us the spectral power distribution of the input signal at frequencies close to ω_r within the demodulation bandwidth set by the filters F(ω).

Note that:

- the ability of distinguish between positive and negative frequencies works only if the Fourier transform is done on X+iY. Had we taken X for instance, the positive and negative frequencies of its power spectrum would be equal. The symmetry relation G(−ω)=G*(ω) holds for the Fourier transform G(ω) of a real function g(t) and two identical peaks would appear at $\pm|\omega_s-\omega_r|$.
- one can extract the amplitude of the input signal by diving the power spectrum by $|F(\omega)|^2$, the operation being limited by the numerical precision. This is implemented in LabOne and is activated by the Filter Compensation button: with the Filter Compensation enabled, the background noise appears white; without it, the effect of the filter roll-off becomes apparent.

The case of an input signal containing a single frequency component can be generalized to the case of multiple frequencies. In that case the power spectrum would display all the frequency components weighted by the filter transfer function, or normalized if the Filter Compensation is enabled.

When dealing with discrete-time signal processing, one has to be careful about aliasing which originates when the signal frequencies higher than the sampling rate Ω are not sufficiently suppressed. Remember that Ω is the user settable readout rate, not the 1.8 GSa/s sampling rate of the UHFLI input. Since the discrete-time Fourier transform extends between −Ω/2 and +Ω/2, the user has to make sure that at ±Ω/2 the filters provide the desired attenuation: this can be done either by increasing the sampling rate or resolving to measure a smaller frequency spectrum (i.e. with a smaller filter bandwidth).

Similarly to the continuous case, in which the acquisition time determines the maximum frequency resolution (2π/T if T is the acquisition time), the resolution of the zoom FFT can be increased by increasing the number of recorded data points. If N data points are collected at a sampling rate Ω, the discrete Fourier transform has a frequency resolution of Ω/N.

Glossary

This glossary provides easy to understand descriptions for many terms related to measurement instrumentation including the abbreviations used inside this user manual.

A

A/D	Analog to Digital See Also ADC.
AC	Alternate Current
ADC	Analog to Digital Converter
AM	Amplitude Modulation
Amplitude Modulated AFM (AM-AFM)	AFM mode where the amplitude change between drive and measured signal encodes the topography or the measured AFM variable. See Also Atomic Force Microscope.
API	Application Programming Interface
ASCII	American Standard Code for Information Interchange
Atomic Force Microscope (AFM)	Microscope that scans surfaces by means an oscillating mechanical structure (e.g. cantilever, tuning fork) whose oscillating tip gets so close to the surface to enter in interaction because of electrostatic, chemical, magnetic or other forces. With an AFM it is possible to produce images with atomic resolution. See Also Amplitude Modulated AFM, Frequency Modulated AFM, Phase modulation AFM.
AVAR	Allen Variance

B

Bandwidth (BW)	The signal bandwidth represents the highest frequency components of interest in a signal. For filters the signal bandwidth is the cut-off point, where the transfer function of a system shows 3 dB attenuation versus DC. In this context the bandwidth is a synonym of cut-off frequency $f_{cut-off}$ or 3dB frequency f_{-3dB}. The concept of bandwidth is used when the dynamic behavior of a signal is important or separation of different signals is required. In the context of a open-loop or closed-loop system, the bandwidth can be used to indicate the fastest speed of the system, or the highest signal update change rate that is possible with the system. Sometimes the term bandwidth is erroneously used as synonym of frequency range. See Also Noise Equivalent Power Bandwidth.
BNC	Bayonet Neill-Concelman Connector

C

CF	Clock Fail (internal processor clock missing)

Common Mode Rejection Ratio (CMRR)	Specification of a differential amplifier (or other device) indicating the ability of an amplifier to obtain the difference between two inputs while rejecting the components that do not differ from the signal (common mode). A high CMRR is important in applications where the signal of interest is represented by a small voltage fluctuation superimposed on a (possibly large) voltage offset, or when relevant information is contained in the voltage difference between two signals. The simplest mathematical definition of common-mode rejection ratio is: CMRR = 20 * log(differential gain / common mode gain).
CSV	Comma Separated Values

D

D/A	Digital to Analog
DAC	Digital to Analog Converter
DC	Direct Current
DDS	Direct Digital Synthesis
DHCP	Dynamic Host Configuration Protocol
DIO	Digital Input/Output
DNS	Domain Name Server
DSP	Digital Signal Processor
DUT	Device Under Test
Dynamic Reserve (DR)	The measure of a lock-in amplifier's capability to withstand the disturbing signals and noise at non-reference frequencies, while maintaining the specified measurement accuracy within the signal bandwidth.

E

XML	Extensible Markup Language. See Also XML.

F

FFT	Fast Fourier Transform
FIFO	First In First Out
FM	Frequency Modulation
Frequency Accuracy (FA)	Measure of an instrument's ability to faithfully indicate the correct frequency versus a traceable standard.
Frequency Modulated AFM (FM-AFM)	AFM mode where the frequency change between drive and measured signal encodes the topography or the measured AFM variable. See Also Atomic Force Microscope.
Frequency Response Analyzer (FRA)	Instrument capable to stimulate a device under test and plot the frequency response over a selectable frequency range with a fine granularity.

Frequency Sweeper — See Also Frequency Response Analyzer.

G

Gain Phase Meter — See Also Vector Network Analyzer.

GPIB — General Purpose Interface Bus

GUI — Graphical User Interface

I

I/O — Input / Output

Impedance Spectroscope (IS) — Instrument suited to stimulate a device under test and to measure the impedance (by means of a current measurement) at a selectable frequency and its amplitude and phase change over time. The output is both amplitude and phase information referred to the stimulus signal.

Input Amplitude Accuracy (IAA) — Measure of instrument's capability to faithfully indicate the signal amplitude at the input channel versus a traceable standard.

Input voltage noise (IVN) — Total noise generated by the instrument and referred to the signal input, thus expressed as additional source of noise for the measured signal.

IP — Internet Protocol

L

LAN — Local Area Network

LED — Light Emitting Diode

Lock-in Amplifier (LI, LIA) — Instrument suited for the acquisition of small signals in noisy environments, or quickly changing signal with good signal to noise ratio - lock-in amplifiers recover the signal of interest knowing the frequency of the signal by demodulation with the suited reference frequency - the result of the demodulation are amplitude and phase of the signal compared to the reference: these are value pairs in the complex plane (X, Y), (R, Θ).

M

Media Access Control address (MAC address) — Refers to the unique identifier assigned to network adapters for physical network communication.

Multi-frequency (MF) — Refers to the simultaneous measurement of signals modulated at arbitrary frequencies. The objective of multi-frequency is to increase the information that can be derived from a measurement which is particularly important for one-time, non-repeating events, and to increase the speed of a measurement since different frequencies do not have to be applied one after the other.
See Also Multi-harmonic.

Multi-harmonic (MH) — Refers to the simultaneous measurement of modulated signals at various harmonic frequencies. The objective of multi-frequency is to increase the

information that can be derived from a measurement which is particularly important for one-time, non-repeating events, and to increase the speed of a measurement since different frequencies do not have to be applied one after the other.
See Also Multi-frequency.

N

Noise Equivalent Power Bandwidth (NEPBW)	Effective bandwidth considering the area below the transfer function of a low-pass filter in the frequency spectrum. NEPBW is used when the amount of power within a certain bandwidth is important, such as noise measurements. This unit corresponds to a perfect filter with infinite steepness at the equivalent frequency. See Also Bandwidth.
Nyquist Frequency (NF)	For sampled analog signals, the Nyquist frequency corresponds to two times the highest frequency component that is being correctly represented after the signal conversion.

O

Output Amplitude Accuracy (OAA)	Measure of an instrument's ability to faithfully output a set voltage at a given frequency versus a traceable standard.
OV	Over Volt (signal input saturation and clipping of signal)

P

PC	Personal Computer
PD	Phase Detector
Phase-locked Loop (PLL)	Electronic circuit that serves to track and control a defined frequency. For this purpose a copy of the external signal is generated such that it is in phase with the original signal, but with usually better spectral characteristics. It can act as frequency stabilization, frequency multiplication, or as frequency recovery. In both analog and digital implementations it consists of a phase detector, a loop filter, a controller, and an oscillator.
Phase modulation AFM (PM-AFM)	AFM mode where the phase between drive and measured signal encodes the topography or the measured AFM variable. See Also Atomic Force Microscope.
PID	Proportional-Integral-Derivative
PL	Packet Loss (loss of packets of data between the instruments and the host computer)

R

RISC	Reduced Instruction Set Computer
Root Mean Square (RMS)	Statistical measure of the magnitude of a varying quantity. It is especially useful when variates are positive and negative, e.g., sinusoids, sawtooth, square waves. For a sine wave the following relation holds between the

amplitude and the RMS value: $U_{RMS} = U_{PK} / \sqrt{2} = U_{PK} / 1.41$. The RMS is also called quadratic mean.

RT — Real-time

S

Scalar Network Analyzer (SNA) — Instrument that measures the voltage of an analog input signal providing just the amplitude (gain) information.
See Also Spectrum Analyzer, Vector Network Analyzer.

SL — Sample Loss (loss of samples between the instrument and the host computer)

Spectrum Analyzer (SA) — Instrument that measures the voltage of an analog input signal providing just the amplitude (gain) information over a defined spectrum.
See Also Scalar Network Analyzer.

SSH — Secure Shell

T

TC — Time Constant

TCP/IP — Transmission Control Protocol / Internet Protocol

Thread — An independent sequence of instructions to be executed by a processor.

Total Harmonic Distortion (THD) — Measure of the non-linearity of signal channels (input and output)

TTL — Transistor to Transistor Logic level

U

UHF — Ultra-High Frequency

UHS — Ultra-High Stability

USB — Universal Serial Bus

V

VCO — Voltage Controlled Oscillator

Vector Network Analyzer (VNA) — Instrument that measures the network parameters of electrical networks, commonly expressed as s-parameters. For this purpose it measures the voltage of an input signal providing both amplitude (gain) and phase information. For this characteristic an older name was gain phase meter.
See Also Gain Phase Meter, Scalar Network Analyzer.

X

XML — Extensible Markup Language: Markup language that defines a set of rules for encoding documents in a format that is both human-readable and machine-readable.

Z

ZCtrl	Zurich Instruments Control bus
ZoomFFT	This technique performs FFT processing on demodulated samples, for instance after a lock-in amplifier. Since the resolution of an FFT depends on the number of point acquired and the spanned time (not the sample rate), it is possible to obtain very highly resolution spectral analysis.
ZSync	Zurich Instruments Synchronization bus

Index

Symbols

L

M

N

O

P

Q

R

S

T

U

V

W

Z