R&S®Pulse Sequencer
Pulse Signal Generation Software
User Manual

ROHDE & SCHWARZ

Make ideas real

This document describes the following software options:

- R&S®SMW-K300/-K301/-K302/-K304/-K306/-K307/-K308/-K309/-K501/-K502
  1413.8805.02, 1413.9776.02, 1413.8857.02, 1413.8957.02, 1413.9053.02, 1413.9118.02, 1414.1433.02, 1414.6706.02, 1413.9218.02, 1413.9260.02

- R&S®SMBV-K300/-K301/-K308
  1419.2744.02, 1419.2789.02, 1419.2973.02

- R&S®SMM-K300/-K301
  1441.1153.02, 1441.1201.02

- R&S®SGT-K300/-K301/-K308
  1419.7652.02, 1419.7700.02, 1419.7730.02

- R&S®SMBVB-K300/-K301/-K308
  1423.8414.02, 1423.8420.02, 1423.8437.02

This manual describes software version V2.8 and later of the R&S®Pulse Sequencer.

Contents

1 Welcome to the R&S Pulse Sequencer

The R&S Pulse Sequencer is an application that generates complex pulse signals by using predefined, configurable test scenarios with different complexity. You can simulate the signals of different emitter and receiver configurations, including antennas and scan types.

The signal can be played by the instruments listed below, for example to perform radar receiver tests.

- R&S SMW
- R&S SMBV
- R&S SMBV100B
- R&S SMM
- R&S SGT

The R&S Pulse Sequencer software is a standalone, PC-based application that creates waveform files or PDW files. This software is available for free download at https://www.rohde-schwarz.com/software/pulse-sequencer/.

Installation

You can find detailed installation instructions in the delivery of the option or in Chapter 2.2, "Installing the software", on page 20.

1.1 Key features

Pulse Sequencer standard features:
- ARB-based signal generation
- Multi-segment waveform sequencing
- Pulse shape definition with rise and fall time, droop, ripple, overshoot
- Modulation on pulse with all major formats like chirps, Barker codes, polyphase codes, PSKs, AM, FM
- Single pulse, pulse train generation with repetition count per pulse
- Inter-pulse modulation of amplitude, phase, frequency, etc. values from pulse to pulse
- Internal and external data sources for modulation
- Import of waveform files for sequencing with repetition count

Pulse Sequencer advanced features:
- ARB-based signal generation and multi-segment waveform sequencing
- Single pulse and pulse train generation with repetition count per pulse
- Powerful sequencing tool with loops, nested loops, subsequences, and overlays
- Antenna diagram definition and antenna scan definition

- Antenna diagrams like pencil beams, cosecant beams, Gaussian diagrams, user-defined antenna diagrams, phased array antenna diagrams
- Antenna scan types like helical scans, circular scans, conical scans
- Emitter definition by waveforms, antenna diagram, antenna scan, attitude information, EIRP, and carrier frequency
- Receiver definition by antenna diagram, antenna scan, and attitude information
- Calculation of signal under consideration of one-way free space propagation according to emitter and receiver location on the 2D map
- Import of R&S®WinIQSIM2™ or customer waveforms for interference generation on the 2D map

The emitter movements option R&S SMW-K304 extends the combination R&S SMW-K300/-K301/-K302/-K309 with the following features:

- Enhances the localized and direction finding scenario types by movement profiles for emitters and receivers
- Predefined line and arc movements for simple movement traces
- Waypoint generation from user-defined "Traces" in the GUI
- Waypoint import interface for complex movement traces
- WGS84 waypoint interface and import of NMEA waypoints
- Import of Google Earth and Google Maps `.kml` files
- East-North-Up (ENU) 2D vector trajectory interface (line, arc) for automatic waypoint generation
- Motion interface for dynamics input (velocity vector or velocity magnitude) in ENU and WGS84
- Predefined waypoint files for land vehicles, ships, aircraft and spacecraft
- User-definable and predefined vehicle description files for land vehicles, ships, aircraft and spacecraft
- Smoothing of waypoints using vehicle description files

Key features of the R&S SMW-K306 multiple-emitters option :

- Allows merging of multiple PDW lists in the "PDW list" (Collection) scenario type or multiple emitters in the "Emitters" (Collection) scenario into a single output file using a priority scheme for dropping

The R&S SMW-K302 option adds the following features for dynamic scenarios:

- A element that represents a real-world vehicle that can contain multiple emitters.
- The realization is similar to the DF receiver model.
- Three-dimensional placem"Platform"ent of emitters on the platform via 3-axis concept (X, Y and Z).
- A 2D and 3D preview of the emitter placement.
- Platforms are mapped to a certain inertia profile (car, ship or plane).
- Platforms have a tactical icon in dynamic scenarios.

The R&S SMW-K309 option adds the following map-related features:

- Use georeferenced maps as visual backgrounds for 2D and 3D map views.

- Import the following map formats:
    - GeoTIF
- Create trajectories using the "Trace" function.
  This function uses a series of user-selected points on the map (i.e. a user-defined trace) to generate waypoints, automatically.

The R&S SMW-K308 option Direction Finding extends the combination R&S SMW-K300/-K301 with the following features:

- Dedicated direction finding scenario and receiver element with multiple antennas and individual positioning
- Direction finding receiver definition with up to 20 antennas with individual positioning and pointing
- Generation of individual waveforms for each receive antenna

Extended Sequencing option R&S SMW-K501/-K502 extends the basic and advanced options with the following features:

- Real-time signal generation for unmodulated rectangular CW pulses and pulses with linear frequency modulation or Barker codes.
  Other pulse shapes are realized as sequencer-based ARB playback with min. calculation time and memory requirements
- Real-time playback of signals created with the R&S SMW-K300/-K301 options
- Dramatically decreased calculation times compared to simple ARB file creation
- Extended simulation time of complex scenarios

For description of the option, see R&S SMW-K501/-K502/-K503/-K504 Extended and Real Time Sequencing User Manaul.

1.2   Accessing the R&S Pulse Sequencer

To open the application

► On your PC, select "Start > R&S Pulse Sequencer > R&S Pulse Sequencer".

The software opens and per default displays the last opened workspace.

1.3   What's new

This manual describes software version V2.8 and later of the R&S®Pulse Sequencer.

Compared to the previous version, it provides the following new features:

- Added a dialog to show details about frequency calibration in calibrated setups, see Chapter 3.12, "Working with calibrated setups", on page 101.
- Added the sequence selection in a sequence collection scenario to the signal generation block, see "Sequence (Collection) Scenario Settings" on page 139.

- Added the function to configure a SCPI timeout for instrument control, see Chapter 23.1, "Setting general program settings", on page 551.
- Added RX information and velocity to the 2D live preview stats, see "2D Live Preview Stats" on page 425.
- Editorial changes.

1.4 Documentation overview

This section provides an overview of the R&S Pulse Sequencer user documentation. Unless specified otherwise, you find the documents on the R&S Pulse Sequencer product page at:

https://www.rohde-schwarz.com/manual/pulse-sequencer/

1.4.1 User manuals and help

Introduces the R&S Pulse Sequencer and describes how to start working with the product. Includes basic operations, typical examples, general information, and the description of all software modes and functions. It also provides a complete description of the remote control commands with programming examples.

The contents of the user manuals are available as help in the R&S Pulse Sequencer. The help offers quick, context-sensitive access to the complete information.

All user manuals are also available for download or for immediate display on the Internet.

1.4.2 Specifications and product brochures

The specifications document, also known as the data sheet, contains the technical specifications of the R&S Pulse Sequencer. It also lists the options and their order numbers as well as optional accessories.

The brochure provides an overview of the R&S Pulse Sequencer software and deals with the specific characteristics.

See https://www.rohde-schwarz.com/brochure-datasheet/pulse-sequencer/.

1.4.3 Release notes and open source acknowledgment (OSA)

The release notes list new features, improvements and known issues of the current firmware version, and describe the firmware installation.

The open source acknowledgment document provides verbatim license texts of the used open source software.

See https://www.rohde-schwarz.com/software/pulse-sequencer/.

1.4.4 Application notes, application cards, white papers, etc.

These documents deal with special applications or background information on particular topics.

See https://www.rohde-schwarz.com/application/pulse-sequencer/.

1.5 Scope

This user manual contains a description of the functionality that the software provides, including remote control operation.

Functions of the vector signal generators are also not described here; they are described in the user manual of the corresponding base unit. You find the latest version on the product page, for example at www.rohde-schwarz.com/manual/smw200a.

1.6 Notes on screenshots

When describing the functions of the product, we use sample screenshots. These screenshots are meant to illustrate as much as possible of the provided functions and possible interdependencies between parameters. The shown values may not represent realistic usage scenarios.

The screenshots usually show a fully equipped product, that is: with all options installed. Thus, some functions shown in the screenshots may not be available in your particular product configuration.

2 Getting started

This section describes the basic steps to be taken when starting up the R&S Pulse Sequencer for the first time.

2.1 Required options

The R&S Pulse Sequencer software is a stand-alone, PC-based application that creates waveform files. This software is available for free download on the R&S website. Waveforms generated with the software can be played by a subset of instruments, that must fulfill the minimum requirements listed below.

R&S SMW200A

R&S SMW200A base unit equipped with:

- Option standard or wideband baseband generator (R&S SMW-B9/-B10)
- Option frequency (e.g. R&S SMW-B1003/-B2003)
- Option standard or wideband baseband main module, two I/Q paths to RF (R&S SMW-B13T/-B13XT)
- Option pulse sequencer (R&S SMW-K300) per signal path

Additional non-mandatory options (require R&S SMW-B10/-B13T or R&S SMW-B9/-B13XT):

- Option multiple entities (R&S SMW-K76)
- Option advanced features for Pulse Sequencer (R&S SMW-K301) per signal path
- Option emitter movements R&S SMW-K304
  (requires option R&S SMW-B9)
- Option multiple emitters (R&S SMW-K306) per signal path
- Option emitter interleaving extended (R&S SMW-K307) per signal path
  (requires option R&S SMW-B9 and R&S SMW-K306)
- Option direction finding for Pulse Sequencer (R&S SMW-K308) per signal path
  (requires option R&S SMW-K301)
- Option dynamic radar platforms with multiple emitters (R&S SMW-K302)
- Option georeferenced maps as visual backgrounds for 2D and 3D map views
  (R&S SMW-K309)

- Option extended sequencer for R&S SMW-B10 (R&S SMW-K501) and for R&S SMW-B9 (R&S SMW-K502) per signal path
- Further options depending on the particular application
  For example, if waveform files generated with R&S®WinIQSIM2™ are used within the sequences, the corresponding R&S SMW-K2xx options

R&S SMBV100B

R&S SMBV100B base unit equipped with:

- Option frequency (R&S SMBV-B103/KB106)
- Option Pulse Sequencer (R&S SMBVB-K300)

Additional non-mandatory options:

- ARB memory extension (R&S SMBVB-K511/-K512/-K513)
  Option RF bandwidth extension (R&S SMBVB-K523/-K524)
- Option advanced features for Pulse Sequencer (R&S SMBVB-K301)
- Option direction finding for Pulse Sequencer (R&S SMBVB-K308)
- Further options depending on the particular application
  For example, if waveform files generated with R&S WinIQSIM2™ are used within the sequences, the corresponding R&S SMBVB-K2xx options

R&S SMM100A

R&S SMM base unit equipped with:

- Option wideband baseband generator R&S SMM-B9
- Option frequency (e.g. R&S SMM-B1006)

R&S SGT100A

R&S SGT100A base unit equipped with:

- Option frequency (R&S SGT-KB106)
- Option baseband generator (R&S SGT-K510)
- Option Pulse Sequencer (R&S SGT-K300)

Additional non-mandatory options:

- Option memory extension for ARB (R&S SGT-K51x)
- Option advanced features for Pulse Sequencer (R&S SGT-K301)
- Further options depending on the particular application
  For example, if waveform files generated with R&S WinIQSIM2™ are used within the sequences, the corresponding R&S SGT-K2xx options

R&S SMBV100A

R&S SMBV100A base unit equipped with:

- Option baseband generator (R&S SMBV-B51)
- Option frequency (R&S SMBV-B1xx)
- Option Pulse Sequencer (R&S SMBV-K300)

Additional non-mandatory options:

- Option RF bandwidth extension (R&S SMBV-K521/-K522)
- Option advanced features for Pulse Sequencer (R&S SMBV-K301)
- Further options depending on the particular application
  For example, if waveform files generated with R&S WinIQSIM2™ are used within the sequences, the corresponding R&S SMBV-K2xx options

2.2 Installing the software

Software updates and the release notes describing the improvements and modifications are available for download at the product page https://www.rohde-schwarz.com/software/pulse-sequencer/.

Software and hardware requirements

Minimum requirements:

- **Software**
  - Microsoft®Windows 10, 64-Bit operating system (x64-based or x86-based processor)
  - Microsoft®Windows 11, 64-Bit operating system (x64-based or x86-based processor)
- **Hardware**

Table 2-1: Hardware requirements

	Minimum requirements	**Recommended hardware**
AMD or Intel CPU	Dual Core, 2 GHz	Intel 8 Core i7 10700, AMD Ryzen series
RAM	≥ 2 GB	64 GB
Video	NVIDIA Quadro 128 MB or ATI Radeon	NVIDIA Quadro 128 MB or ATI Radeon
Video resolution	≥ 1280 x 1024 pixels	≥ 1920 x 1200 pixels
Rendering	OpenGL, shader model 3	OpenGL, shader model 3
Network	LAN 1 GB/s	LAN 1 GB/s

To install the software

Download the R&S Pulse Sequencer software form the Rohde & Schwarz website.

The R&S Pulse Sequencer software consists of the file
`PS-Install <major>.<minor>.<build>.<release>-x64.msi`.

1. Start the `PS-Install <major>.<minor>.<build>.<release>-x64.msi` file.

   The filename follows the naming conventions:
   - `<major>.<minor>` is the software version

- `<build>` is the build number.
  This is the day elapsed since 1 January 2000
- `<release>` indicates the release on the build date.
  This is 1/10 of the number of seconds elapsed since midnight.
- `x64` indicates that this is a 64-bit installation file.
  You can run the software version with the extension `x64` only on a 64-bit operating system, see "Software and hardware requirements" on page 20.

2. Depending on your user rights, select one of the following:

   a) Installation for all users (default)

      - Requires administrator rights
      - Is a per-machine installation
      - Project data and program data are common to all users
      - Users keep their individual settings

   b) Installation for current user only

      - Does not require administrator rights
      - Is a per-user installation
      - Installation in `%HomePath%`

3. Choose the setup type.

4. Follow the installation instructions.

   The installer performs the following actions:

   - If enabled, installs the R&S VISA software on the PC
   - Installs the R&S Pulse Sequencer software including an uninstall tool
   - Creates a shortcut on the desktop
   - Creates the folder structure listed in Table 2-2.

Table 2-2: Default file location (software installation for all users)

File type	File location
Program data	`%PROGRAMFILES(X86)%\Rohde-Schwarz\Pulse Sequencer` (`Path`)
Project data (Repository database) SDK files	`%PUBLIC%\Public\Documents\Rohde-Schwarz\ Pulse Sequencer` (`DataPath`)
Report files, startup log file, calculated data, volatile data	`%HOMEPATH%` (`HomePath` or `ReportPath`)
User settings (Workspace, etc.)	`%HOMEPATH%\AppData\Roaming\Rohde-Schwarz\ Pulse Sequencer`

See also:

- "To access the "Debug Information" dialog" on page 833
  For information on the version of the used VISA library and the list of all loaded modules.
- Chapter 23.3, "Defining storage locations", on page 557

For information on how to query and change data storage location.

2.3 Starting the R&S Pulse Sequencer for the first time

1. On your PC,

   a) Select "Start > R&S Pulse Sequencer > R&S Pulse Sequencer" or
   b) Double click the shortcut icon on the desktop.

   The "Startup Assistant" opens.
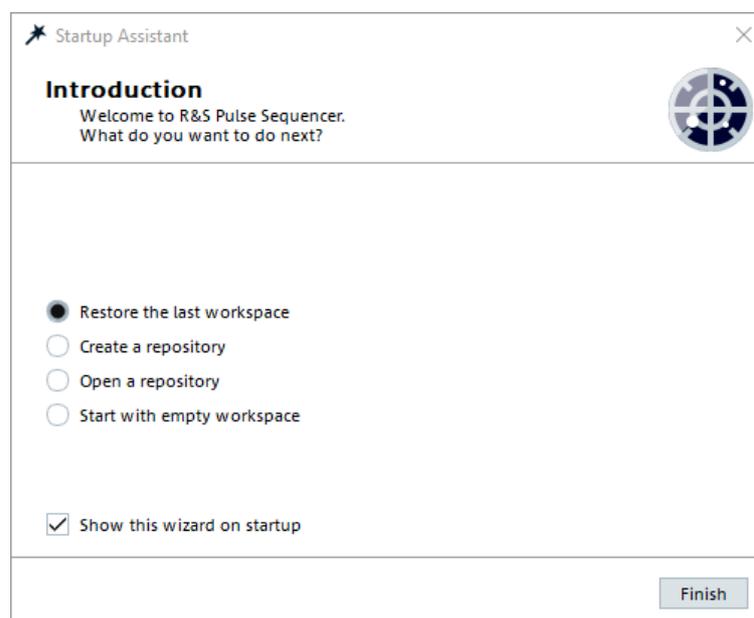
2. Select "Create a repository" > "Next".

3. Follow the instructions. Use the default settings when starting R&S Pulse Sequencer for the first time.

   A new repository ("New_<Year>-<Month>-<Day>T<Time>") is created that contains a simple scenario ("PTrain 01").
   Your workspace resembles the one shown in the following figure.
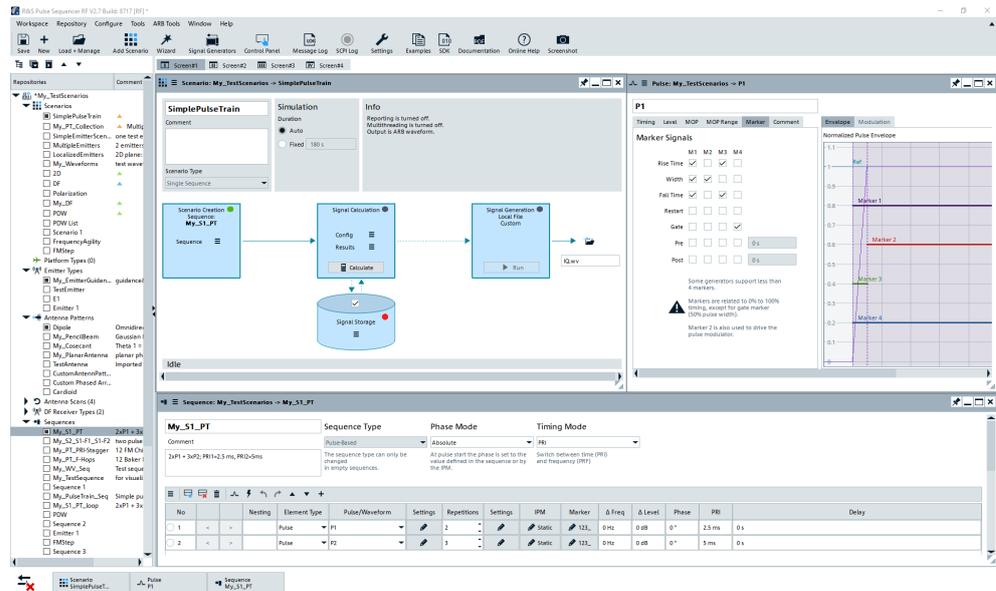
See also:
- Figure 2-1 (in Chapter 2.4, "Understanding the displayed information", on page 23)
- Chapter 2.5, "Software concept in brief", on page 29
- Chapter 2.7, "Trying out the software", on page 34

2.4 Understanding the displayed information

The following figure shows an example test scenario during software operation. The different information areas are labeled. They are explained in more detail in the following sections.
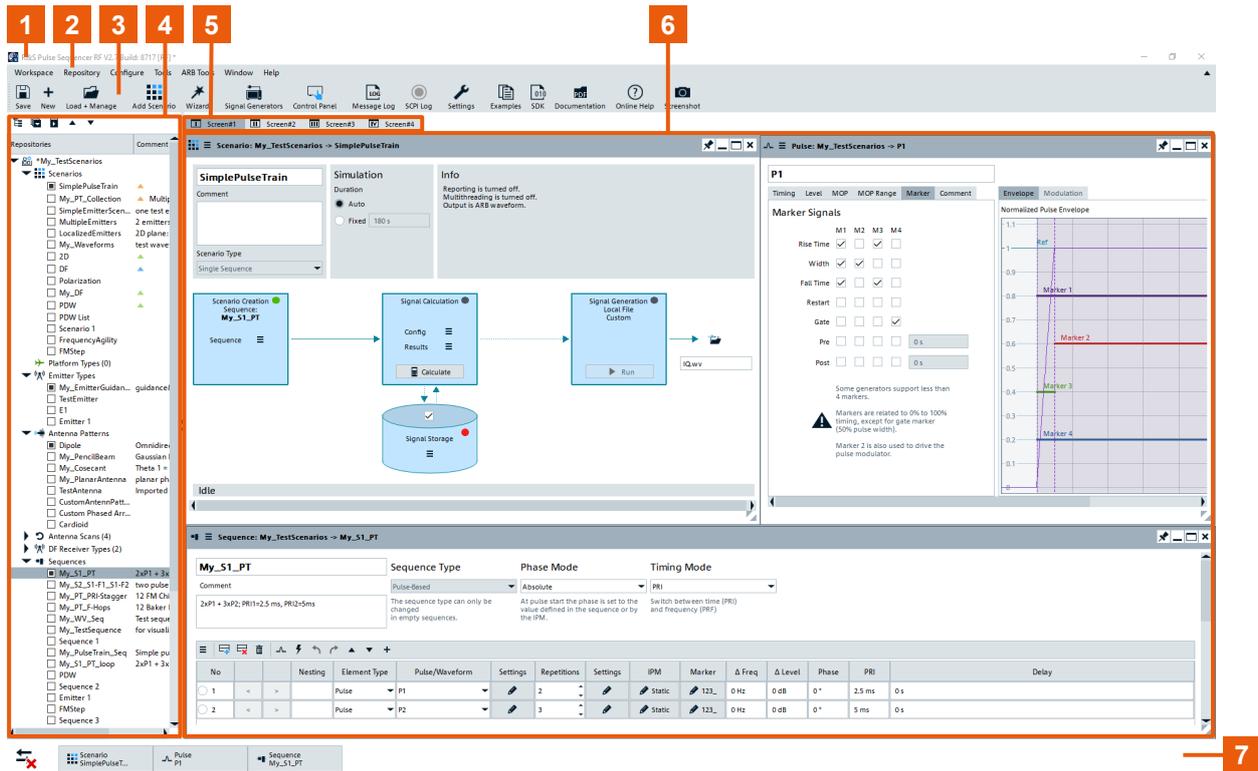
Figure 2-1: R&S Pulse Sequencer default workspace: understanding the displayed information

- 1 = Title bar
- 2 = Menu bar with standard functions, e.g. save/open file etc.
- 3 = Toolbar
- 4 = Project tree (repository tree)
- 5 = Screens for individual scenarios
- 6 = Work area
- 4+6 = Workspace
- 7 = Taskbar with indication of enabled remote control incl. details like IP address (if enabled), information on the required highest security level (if enabled) and access to active dialogs

The user interface of the R&S Pulse Sequencer offers an intuitive operating concept. Operation is similar to any operating system. All menus and dialogs use known elements, e.g., selection lists, checkboxes, and entry fields.

Menu bar

Table 2-3: Content of the menu bar

Workspace	Repository	Configure	Adjustment	Tools	ARB Tools	Window	Help
• Clear • Open • Save • Save As • Exit	• New • Load or Manage • Save • Import • Export • Edit	• Signal Generators • Remote Control • Colors • Settings	• Configure • Manage Data • Run level Adjustment	• Script Control Panel • Console • Wizard	• Composer	• Message Log • SCPI Log • MinSize All Windows • Cascade • Close All • Screen 1 .. 4	• Online Help • Documentation • Examples • SDK • Tutorials • Find connector • Debug Information • About

Toolbars

The R&S Pulse Sequencer offers two toolbars.

Standard functions can be performed via the icons in the main toolbar at the top of the screen. Click the arrow on the right-hand side of the menu bar to expand and collapse the toolbar.
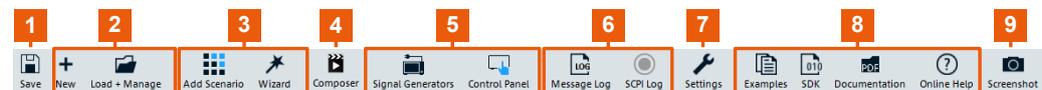
Figure 2-2: Standard functions in the main toolbar

1 = Save current workspace configuration
2 = Repository: Creates and loads or manages repository
3 = Project tree: Creates a scenario/Starts a wizard to create an emitter or pulse train in current repository
4 = Opens the "Waveform Composer"
5 = Generator: Opens the "Signal Generators" dialog/Opens the signal generator "control panel"
6 = Logs: Opens the "Message Log" window/"SCPI Log" window
7 = Opens the "Program Settings" dialog
8 = Help: Opens the "Example Files" folder/Opens the "SDK" folder/Opens the help documentation folder containing help documents/Displays context-sensitive help topic for currently selected element
9 = Hardcopy: Copies a screenshot of the active window to the clipboard

Functions relating to the "Project Tree" can be performed via the icons in the toolbar below the main toolbar.

Figure 2-3: Functions in the Project Tree toolbar

1 = Switches between tree view and table mode, see Chapter 4.3, "Repository view", on page 113
2 = Expand/collapse all nodes in the "Project Tree"
3 = Moves the elements within a node up/down in the "Project Tree"

Virtual screens

Using the R&S Pulse Sequencer you can handle several scenarios with even comprehensive settings at the same time. You can arrange the dialogs across multiple screens logically, so that you can easily switch between them. You can, for example, assign the subset of antenna dialogs to one screen and the pulse and modulation dialogs to another.

Repository tree

The repository tree shows the content of all currently loaded repositories. Repository elements are indicated with the assigned icons, their names, and the first comment line.

You can also switch the tree view to table mode, see Chapter 4.3, "Repository view", on page 113.

1 = Name, see "Name and comment" on page 26
2 = First comment line

Name and comment

Each repository element is identified with its unique name and optional description entered in the comment field. The description can contain several lines; new line is added by pressing [Shift+Enter].

See also "Dialog names" on page 27.
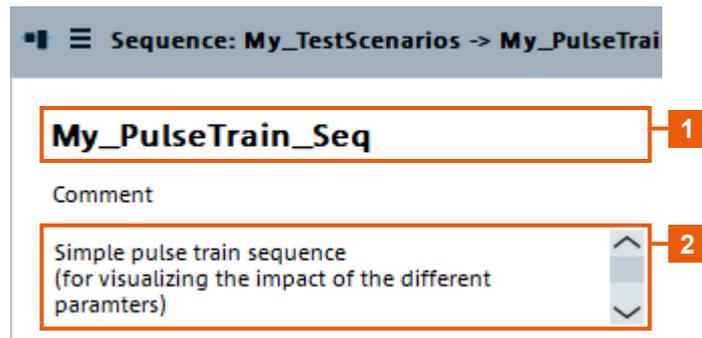
1 = A unique name to indicate the element in the repository
2 = Description; the first line is displayed in the repository tree

Context-sensitive menus

All user interface controls provide a context-sensitive menu. Context-sensitive menus are accessed with a right mouse click the control.
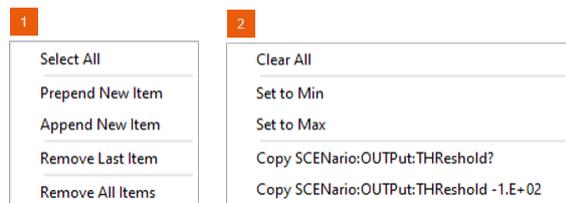
1 = Context menu of an icon
2 = Context menu of a parameter

Tooltips

Tooltips indicate the possible value range of a parameter or list overview information on current settings.
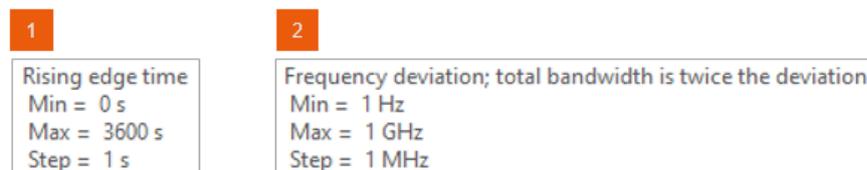
1 = Tooltip of a setting parameter, indicating the allowed value range
2 = Tooltip with detailed information on a setting

Dialog names

The name of a dialog indicates its content or the functionality this dialog covers. The used naming convention helps you to recognize which database element you are configuring, to which scenario it belongs and what its current name is.

"<Repository Element>: <Scenario Name> -> <Function Name>"

1 = Repository element
2 = Scenario name
3 = Function name

Undocking dialogs

If your workspace contains several simultaneously opened dialogs, you can undock dialogs and distribute them also outside the workspace, for example place them on a second monitor.

1 = Undocks the dialog

Interactive 3D displays

Several dialogs display the configured settings as 2D or as interactive 3D models or full 3D antenna radiation patterns.

Turning the mouse wheel zooms in and out on these dialogs. Moving the mouse while holding the left mouse key rotates the 3D view around its origin.
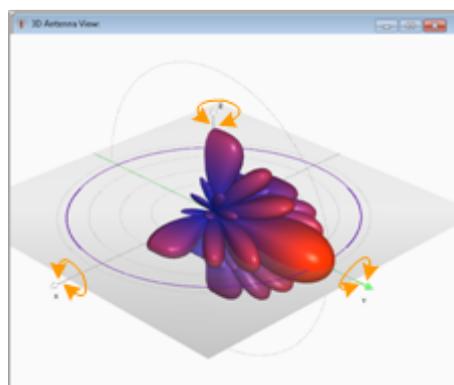
Interactive 2D map

Drag&drop elements into the 2D map or pan and zoom to change the displayed map area.

Keyboard shortcuts

Keyboard shortcuts are available for the most common functions in the software.

2.5 Software concept in brief

The following is brief introduction to the software concept and the way it handles files and settings.

Repository
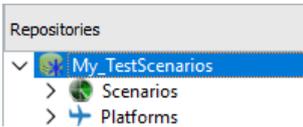
The R&S Pulse Sequencer organizes the data in repositories.

A repository is a **file-based database** located on the local hard drive or on a network storage drive.

Repositories are entirely managed by the R&S Pulse Sequencer. They can be password protected against unauthorized access. Repositories can be exported as archives and shared among multiple users if they are located on a network drive. Several users can have read permission to load and use the same repository simultaneously. However, only one user can have a write permission at a time.

For more information, see Chapter 4, "Organizing the project data in repositories", on page 106.

Scenario
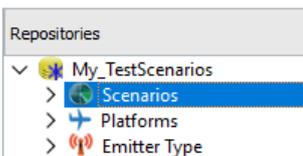
The scenario is the **top-level description** of the signals to be generated.

The type of the scenario determines which kind of signal is calculated and how this signal is processed. A more complex scenario combines several signals. Scenarios can also contain information about the time variation of the signals.

Starting a scenario creates the I/Q waveform, sends them to the ARBs and configures the signal generators.

For more information, see Chapter 5, "Selecting a suitable scenario and creating scenarios", on page 120.

Platform
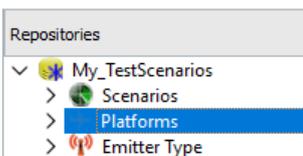
A platform is a **group of up to 8 emitters** assigned to a single vehicle. A platform can be static or moving. A platform is characterized by its icon, list of emitters, position and trajectory. The emitters on a platform can be individually configured.

For more information, see Chapter 12, "Creating platforms", on page 336.

Emitter
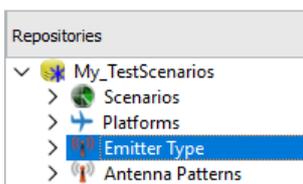
Emitters emulate **radar systems**. The emitter combines a sequence, an antenna scan, and an antenna pattern to one logical unit.

An emitter can work in more than one operating mode and switch between them. The *operating mode* is a description of the radar purpose, such as scanning, searching or tracking. For example, an airport surveillance radar can switch between different scan types to observe its airspace.

Each mode can have individual antenna and signal configuration. Multiple beams are possible within a mode.

For more information, see Chapter 16, "Creating complex 2D scenarios with receiver and TX items", on page 415.

Antenna

The antenna characterizes the **radiation pattern of the emitter**. Typical radar antennas like the phase array antennas can adapt their radiation pattern dynamically, depending on the operating mode the radar is working in. For example, an antenna can use a cosecant beam in scan mode and pencil beam in track mode.

Antennas can have various scan types. The scan type describes the movement of the antenna. For example, circular, sector and custom.

For more information, see Chapter 9, "Defining antenna patterns and antenna scans", on page 257.
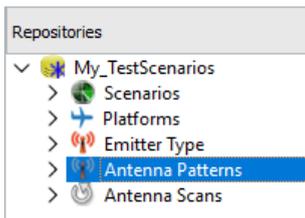
Receiver

Receivers are used to model **direction finding systems**. They comprise of up to 20 individual antennas with individual patterns and positioning relative to a direction finding system.

For more information, see Chapter 13, "Emulating receivers", on page 348.
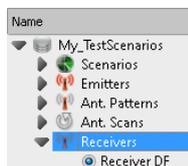
Sequence

A sequence describes how pulses are arranged to form a waveform. The most simple sequence comprises one pulse that is repeated with a constant PRI (pulse repetition interval). Typical sequences however are rather complex; they can contain pulse definitions, waveforms, and sequencing elements such as repetitions and loops.

The sequence is a **logical description**. It cannot directly be represented as an I/Q waveform.

For more information, see Chapter 7, "Building pulse sequences", on page 198.
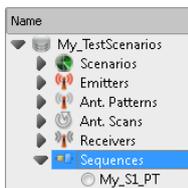
Pulse

In the R&S Pulse Sequencer, pulses are the **fundamental building block** of any signal. Pulses are described mathematically with their amplitude (envelope), the applied modulation on pulse (MOP) and the enabled marker signals. One single pulse description is however not sufficient to calculate (and process) a signal; pulses have to be organized in a sequence, and assigned to an emitter.

For more information, see Chapter 6, "Creating a pulse library", on page 148.
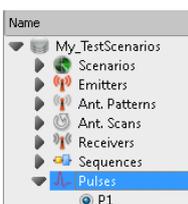
IPM (Inter-Pulse Modulation)

Inter-Pulse Modulation (IPM) profiles **modify a pulse parameter** (like level, pulse repetition interval, or frequency offset) from one pulse to another one. The IPM profiles are applied within a sequence.

For more information, see Chapter 8, "Defining and enabling inter-pulse modulation effects", on page 224.

Waveforms, interferers and imported signals
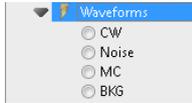
In addition to creating pulse signals you can also load waveform files and create waveform scenarios with them or even use them as interferes.

An interferer contains complete waveforms, that can be user-defined or created with the signal generation software R&S WinIQSIM2. Interferes can be mixed with the signal.

For more information, see Chapter 14, "Working with waveforms", on page 368.

Data source
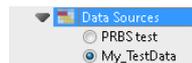
Data source defines a **numerical source for the symbols** used in the modulations applied on the pulse (MOP).

For more information, see Chapter 19, "Defining the data source", on page 523.

Plugins
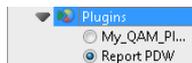
A plugin is an **extension** to the standard functions of the R&S Pulse Sequencer.

For example, a plugin can be a user-defined MOP or IPM profile.

For even more flexibility, plugins can contain variables.

If different values are assigned to the same variable in the different pulses, the applied MOP varies.

For more information, see "How to import and assign user defined plugins" on page 530.

2.6 Means of users interaction

This chapter provides an overview on how to work with the R&S Pulse Sequencer and describes the manual operation of the software and also the alternative ways of operation.

There are two ways to operate the R&S Pulse Sequencer:

- Manual operation:
  Run the software on your PC and use the mouse and/or keyboard.
- Remote control:
  Create programs to automate settings, tests and measurements. The software is controlled by a program, running on the same or on another computer.
  This way of operation is described Chapter 24, "Automation of R&S Pulse Sequencer", on page 568.

Basic operation concept

This section lists settings and functions that are common to the dialogs and are performed in the same manner throughout the software. You can use them alternatively or complementary.

As described in Chapter 2.5, "Software concept in brief", on page 29, the R&S Pulse Sequencer software is based on a relational database that organizes database elements within repositories. Database elements are indicated by their names; within one repository, element names must be unique (see "Name and comment" on page 26).

Creating elements

Once created, database elements can be used by other elements within the same repository. Elements can be created in both the bottom-up and the top-down direction, or in a combination of these two methods:

- *Bottom-up:* you can create your pulse and waveform library first, then build sequences, create scenarios, and use the created sequences or assign them to emitters.

- *Top-down:* create your scenario, create emitter, define the antenna pattern/scan, define the used sequence, and define the pulses and waveforms.
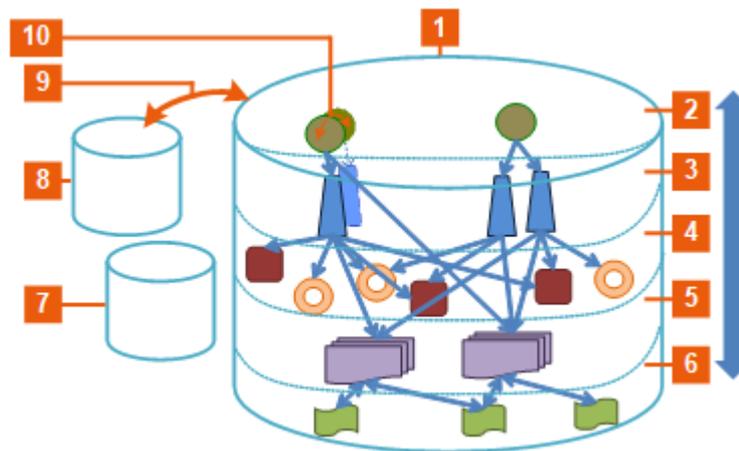

```
1, 7, 8 = Repository, one or more within the database
2        = Scenario level; scenarios describe one or more emitters and one or more sequences
3        = Emitter level; emitters are described among others with antenna patterns and antenna scans
4        = Antenna patterns and antenna scans
5        = Sequence level; sequences are built of one or more pulses or waveforms
6        = Pulse and waveform level
9, 10   = Creating elements at any level with clone and copy
```

For information on the naming conventions and description of the database elements, see Chapter 2.5, "Software concept in brief", on page 29.

Handling database elements

While creating database elements, consider also the following possibilities:

- Change
  Files describing elements are stored once but can be used several times. When one file is changed, its parent elements are also updated.

- Clone
  Elements can be cloned to create a deep copy of the database element.

- Delete
  Elements can be deleted if they are not referenced by other elements.

- Copy
  If two repositories are opened in the project tree, a drag and drop operation creates a copy of the selected element and all referenced elements.

Standard function in the context menus

Each dialog with settings belonging to a "higher-level" element provides standard functions to:

- Assign existing elements
- Create elements
- Edit the existing assigned element.

You can access these functions from the context-sensitive menus of the icons listed in Table 2-4.

Table 2-4: Overview of the used icons and their functions

Icon	Name	Function/Description
☰	<Function> Menu (the name changes depending on the situation)	Smart menu with several options, depending on the current element and situation: • New (Item) • Append item • Prepend item • Insert Item • Insert Item Before/After • Delete Item • Clear (Selection) • Edit • Select (from a list) • Rename • Import/Export
✏	More	Select Edit New
⌁ ⚡	Pulse, Waveform	Creates repository element
➡ ↺ ◧	Antenna Pattern, Antenna Scan, Sequence	Accesses a dialog with detailed settings
((A)) ⚡ ᴬ))	Emitter/Interferer, receiver	
3D 2D	3D, 2D diagram	
⊟₊ ⊟ₓ 🗑	Append item, Remove item, Delete	Appends or removes item, deletes all items
◀ ▶ ▲ ▼	Left/Right , Up/Down	Reorders items

Icon	Name	Function/Description
+	Copy and append	Copies the selected line of the table and appends it at the end of the table
✓	Select	Selects a table row or an item for editing
	Enable/Disable	Activates or deactivates settings
📷 🖨	Hardcopy, Print	Copies current screen or dialog to the clipboard; Prints current diagram
🔧	Name depends on contents	Opens a settings dialog. For example, "Visualization settings" or "Program settings".

Standard functions for elements handling

The following standard direct interaction functions are supported:
- Double-click
  Opens the selected element for editing
- Clone
  Creates a copy of the selected element
- Drag and Drop
  A drag and drop operation:
  – Creates a copy of the selected element, together with all referenced elements
  – Assigns an element to a 2D view
  – Imports a repository
  – Imports waveform files into the repository
- Mouse wheel
  Turning the mouse wheel zooms in and out.

Description of alternative methods

This manual describes the manual interaction with the software via the menus. Many elements that can be accessed form menus can also be accessed by clicking an icon or using the context menu. Alternative procedures are only described if they deviate from the standard operating procedures.

Throughout the manual, the term "select" refers to any of the described methods.

2.7 Trying out the software

This chapter introduces the most important functions and often used settings of the R&S Pulse Sequencer step by step. The complete description of all functions and their usage is provided in the corresponding main chapters of this user manual.

Prerequisite

The software is running and started up as described in Chapter 2.3, "Starting the R&S Pulse Sequencer for the first time", on page 22.

For the first signal generation tasks, you use the *on-demand Wizard* to generate an ARB signal, so you do not need any external instruments. More complex signal generation tasks, however, require an external signal generator equipped with the options listed in Chapter 2.1, "Required options", on page 18.

The software is manually operated. Try out the following:

2.7.1 Completing the scenario that was automatically created upon start up

We assume that you have started the software as described in Chapter 2.3, "Starting the R&S Pulse Sequencer for the first time", on page 22. Your workspace resembles the one shown on Figure 2-1 (see section Chapter 2.4, "Understanding the displayed information", on page 23).

Your repository contains:

● "P1": an unmodulated pulse with pulse width of 100 us and rise and fall times set to zero
● "PTrain 01": a sequence containing one single pulse, "P1". This pulse is processed once ("Rep. Count = 1") and has a pulse repetition interval "PRI = 200 us".

To create a sequence with 10 pulses, each having a PRI of 1 ms

1. In the repository tree, select "Scenario > Sequences > PTrain 01".

2. In the "Sequence Description" table, select:

   a) "Rep. Count = 10"

b) "PRI = 1 ms"
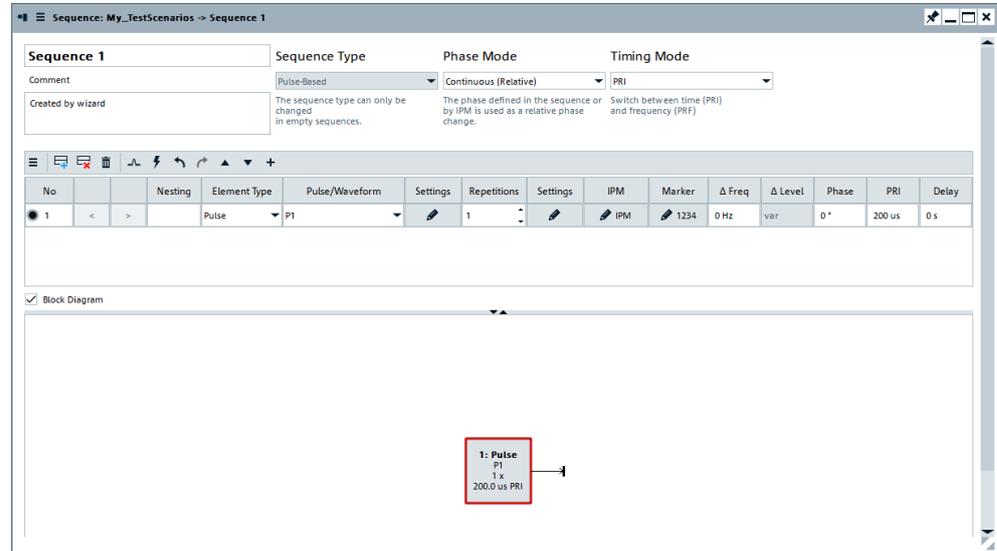
For information on the provided settings, see:
● Chapter 7.2, "Sequence settings", on page 200
● Chapter 6.2.2, "Pulse timing settings", on page 151

To assign the sequence to the scenario

When created by the "Wizard", the sequence is automatically assigned to the scenario. If not, proceed as follows:

1. In the repository tree, select "Scenario > PTrain 01".

   The "Scenario" dialog opens and displays a block diagram of the signal flow.

2. In the block diagram, select "Sequence" and select the icon to open the context menu.

3. Select "Select > PTrain 01".

   The status indicator in the "Sequence" block is green.
   The status indicator in the "Signal Calculation" block is still red.

2.7.2 Generating an ARB waveform file

The initial situation is the configuration described in "To assign the sequence to the scenario" on page 36.

To set the output target for the created signal

1. In the repository tree, select "Scenario > SimplePulseTrain".

   The "Scenario" dialog opens and displays a block diagram of the signal flow.

2. In the block diagram, select "Signal Generation" and select the icon to open the context menu.

3. Select "Config > Target > File" to save the generated signal to your PC.

   A list of the available Signal Generators appears.
   A check mark indicates the currently selected signal generator.

4. Select a signal generator from the list.

   The block name changes to indicate the selected signal generator.
   Note: If the previous selection was "Target > Generator", the target icon next to the block diagram changes to a yellow folder.

5. Edit the destination filename shown in the field next to the target icon.
   The default name is `IQ.wv`
   Change the name but do not change the file extension (e.g. `Gen1_Waveform.wv`).

6. In the context menu, select "Set Path...".

7. In the Windows Explorer, navigate to the directory in which the file will be saved.

   The block diagram shows the settings.
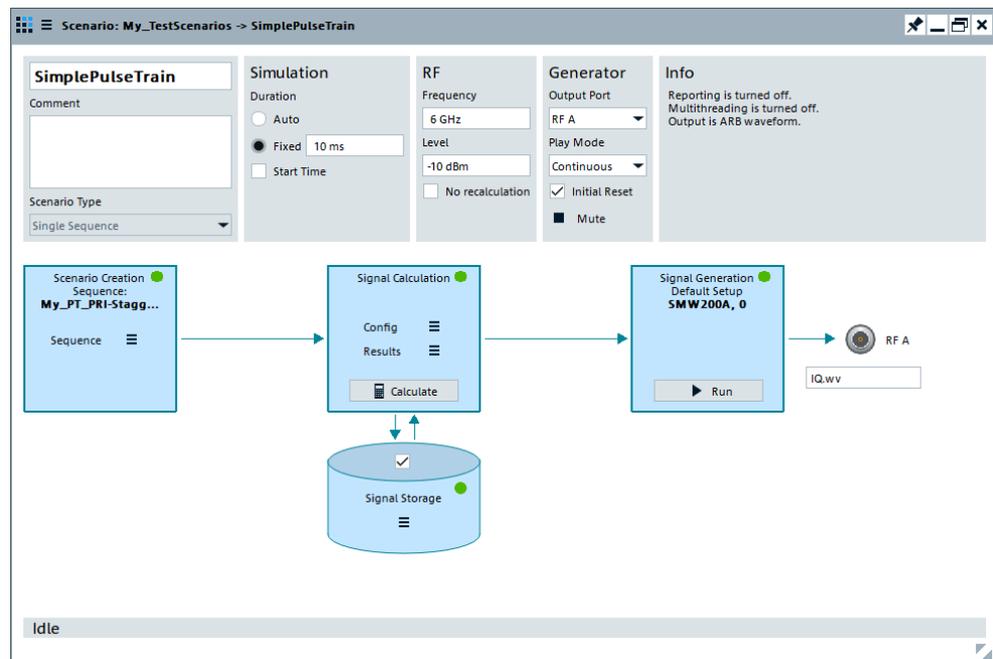
The "Info" area shows a status indication and displays information and warnings about the scenario. For example, if the generator does not support extended sequencing, the information displays here. The created signal is a multi-segment or ARB waveform.
For better visualization, change the default configuration so that an ARB waveform is generated.

8. Select "Signal Calculation > Config > Config".

9. In the "Signal Calculation Settings" dialog, select "Output > Create ARB waveform".

If all preconditions for signal generation are fulfilled, the "Calculate" button is active and you can start signal calculation.

To start the signal calculation

► In the "Scenario" dialog, select "Calculate".

The R&S Pulse Sequencer creates the ARB waveform and shows the current progress.
A green indicator in the "Signal Calculation" block confirms that the calculation is completed. The line between the "Signal Calculation" and the "Signal Generation" blocks confirms that the calculated signal can be transmitted to the selected target.

For information on the provided settings, see:
● Chapter 5.2, "Scenario settings", on page 121

2.7.3 Verifying the generated signal in the signal preview

It is often useful to check the generated signal visually, before you transfer it to the Signal Generators. The R&S Pulse Sequencer provides a built-in function, the "Signal Preview", to represent the generated signal in a graphical form.

The initial situation is the configuration described in "To start the signal calculation" on page 38.

To open the "Signal Preview" and visualize the generated signal

► In the "Scenario" dialog, select "Signal Calculation > Results > View" and select the name of the calculated waveform.
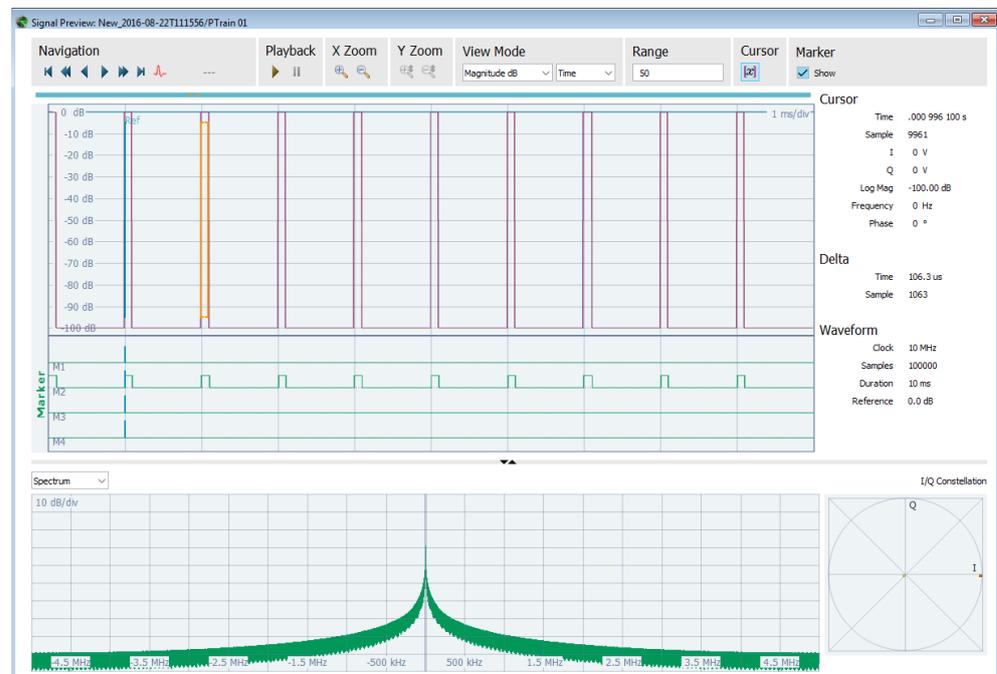
Figure 2-4: Signal Preview

The Signal Preview shows a sequence composed of 10 unmodulated pulses.

In its default view mode, the "Signal Preview" dialog displays the I and Q data as a function of time, the spectrum of the generated signal and I/Q the constellation diagram.

The displayed settings confirm that:

- Generated is a sequence of 10 pulses
- Each pulse has a PRI = 1 ms
- The waveform is 10 ms long, that is 10*PRI
- Each pulse has a pulse width of 100 us.

For information on the provided settings, see Chapter 15.1, "Waveform view and signal preview settings", on page 400.

For information on the provided settings, see Chapter 15.2, "Signal preview settings", on page 405.

2.7.4 Launching the built-in wizard

To help you get familiar with the software, the R&S Pulse Sequencer provides a built-in startup assistant, the "Wizard".

You can access this "Wizard":

- **On an application start**:
  That is, each time you start the software
- **On demand**:
  That is, any time during operation when you need a fast configuration of few settings.

To start the built-in wizard

► In the tool bar, select "Tools > Wizard".

The "Startup Assistant" opens.
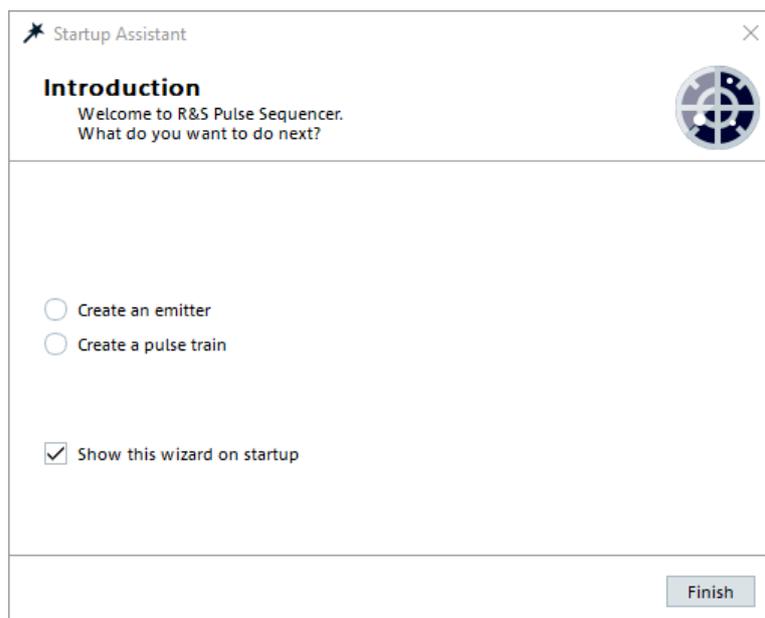
2.7.5 Using the wizard to create a complex scenario

The main application field of the R&S Pulse Sequencer is the generation of pulsed signals. This example uses the "Wizard" to introduce the way you can create and define antenna patterns, antenna scans, emitters, etc. and interact with the software.

To create a new single emitter scenario

1. Start the "Wizard", see Chapter 2.7.4, "Launching the built-in wizard", on page 39.

2. In the assistant dialog, select "Create Emitter".
   Click "Next".
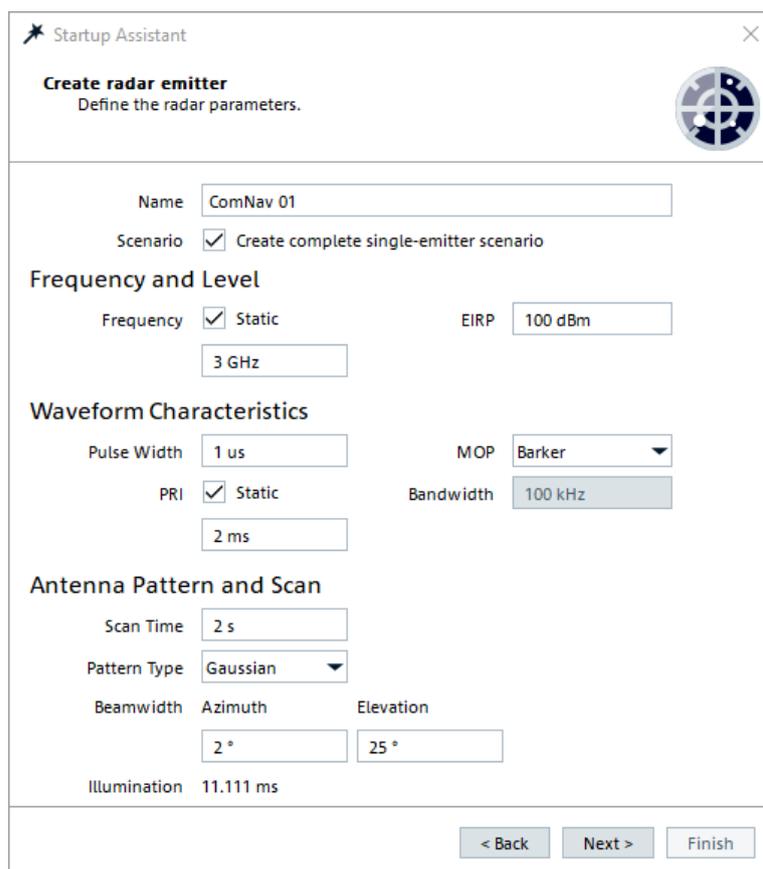
3. Use the default settings and select "Waveform Characteristics > MOP > Barker". Click "Next".

4. Click "Finish".
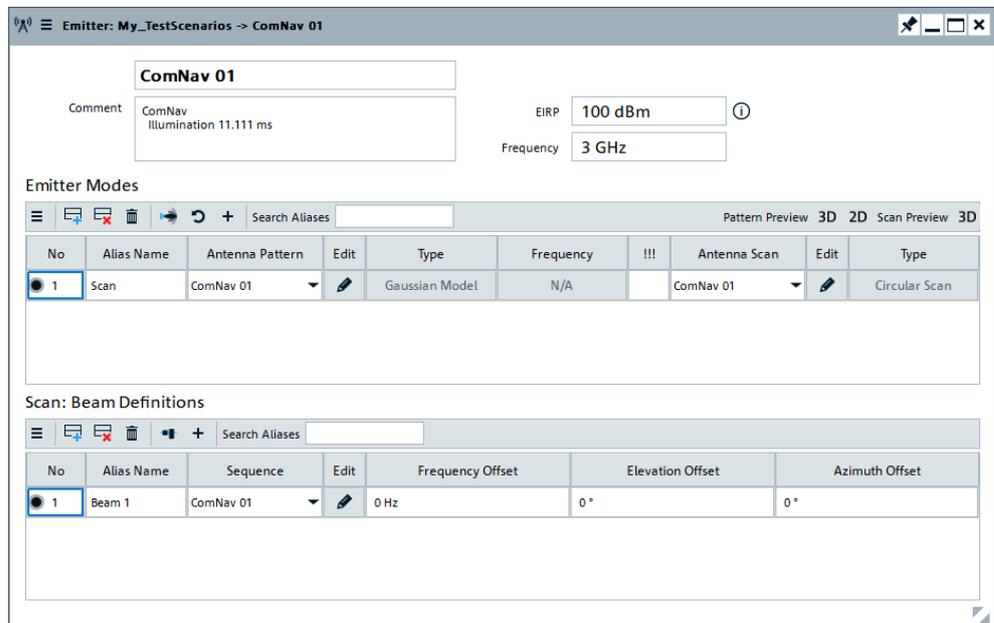
The software creates the scenario automatically with all required elements, e.g. one new emitter ("ComNav 01"), one new sequence ("ComNav 01"), new pulse ("ComNav 01"), etc.

For information on the provided settings, see:
- Chapter 10.1, "Emitter settings", on page 307
- Chapter 9.2, "Antenna pattern settings", on page 261
- Chapter 9.3, "Antenna scans settings", on page 280
- Chapter 6.2.5, "Modulation on pulse (MOP) settings", on page 162

To visualize the signal received by a static receiver

1. Open the automatically created emitter-based scenario ("ComNav 01").
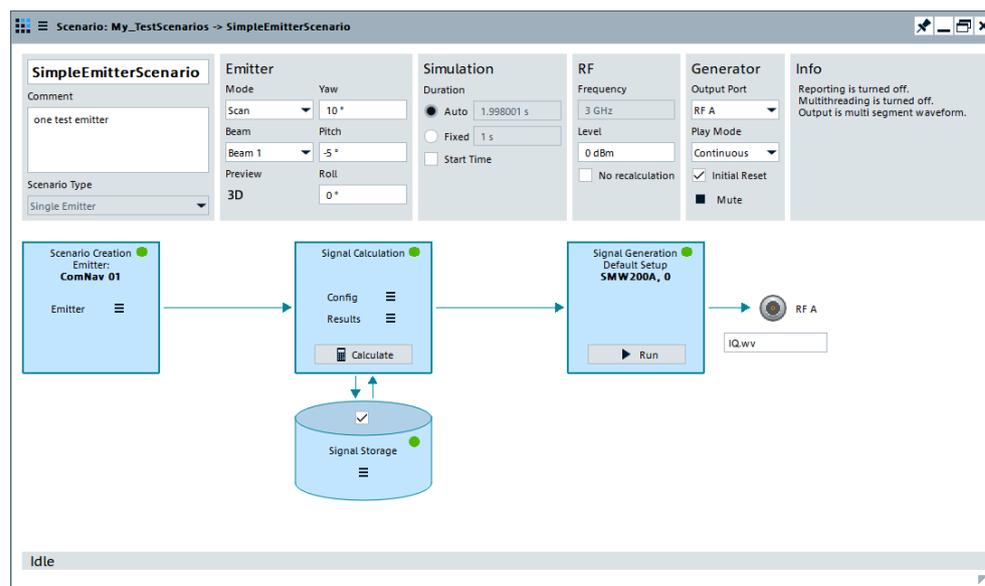
2. In the "Emitter" section on the block diagram, select the "3D" icon.
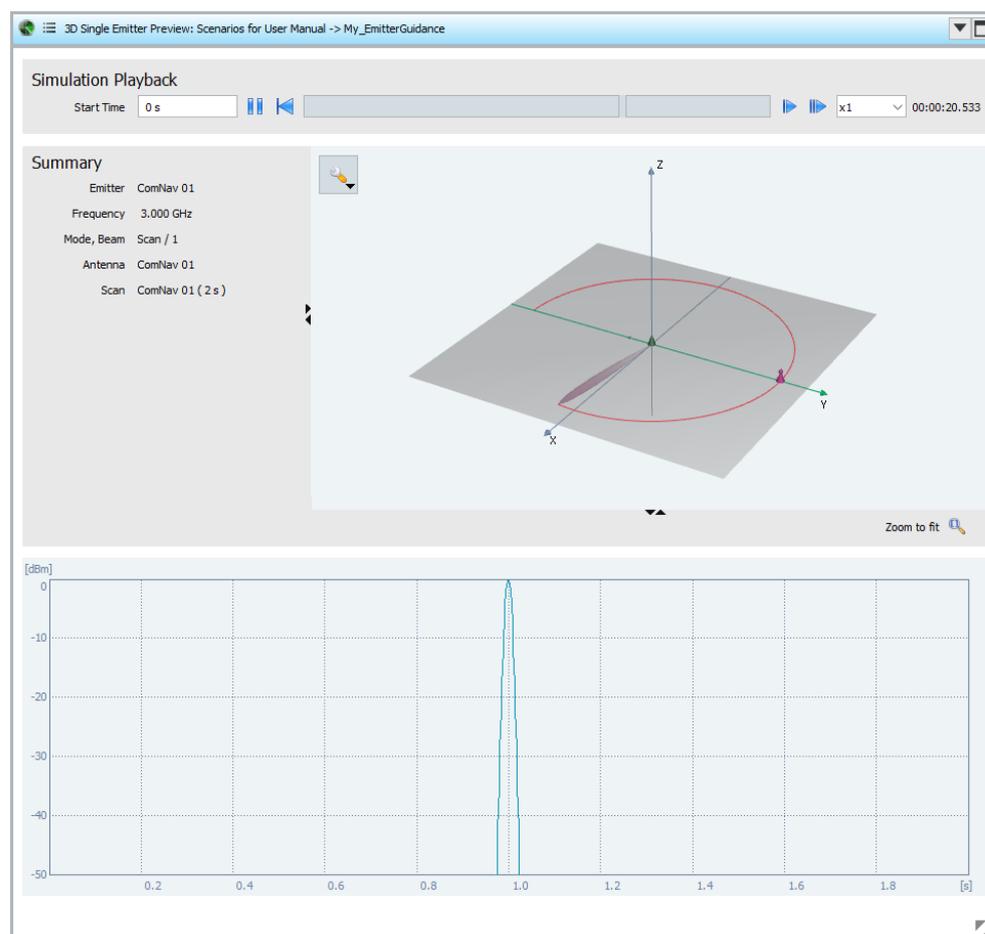
The "3D Single Emitter" dialog displays a live plot of the normalized signal power level at the receiver and an interactive 3D view of the emitter antenna pattern and scan.

3. Use the mouse wheel to zoom in and out.

4. To rotate the 3D view around its origin, left mouse click. Keep the key pressed and move the mouse.

   For information, see Chapter 16, "Creating complex 2D scenarios with receiver and TX items", on page 415

To create and verify the generated ARB waveform

1. In the "Scenario" dialog, define the storage location for the ARB files, see Chapter 2.7.2, "Generating an ARB waveform file", on page 36.
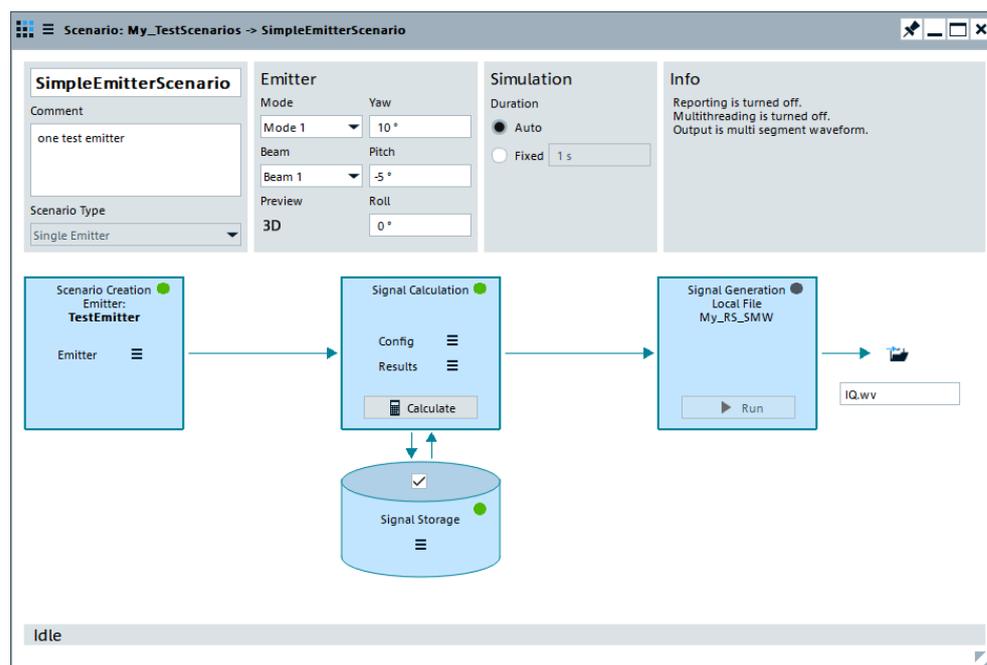
2. Select "Calculate" to start signal calculation.

3. Select "Signal Calculation > Results > View" and select the name of the calculated waveform.

4. In the "Signal Preview" dialog, select "View Mode > Phase".

5. Use the "Zoom In" button to change the scale on the y axis.

6. Select "Navigation > pulse Icon" to jump to the first pulse.
   Select the pulse icon again to scroll through the pulses.
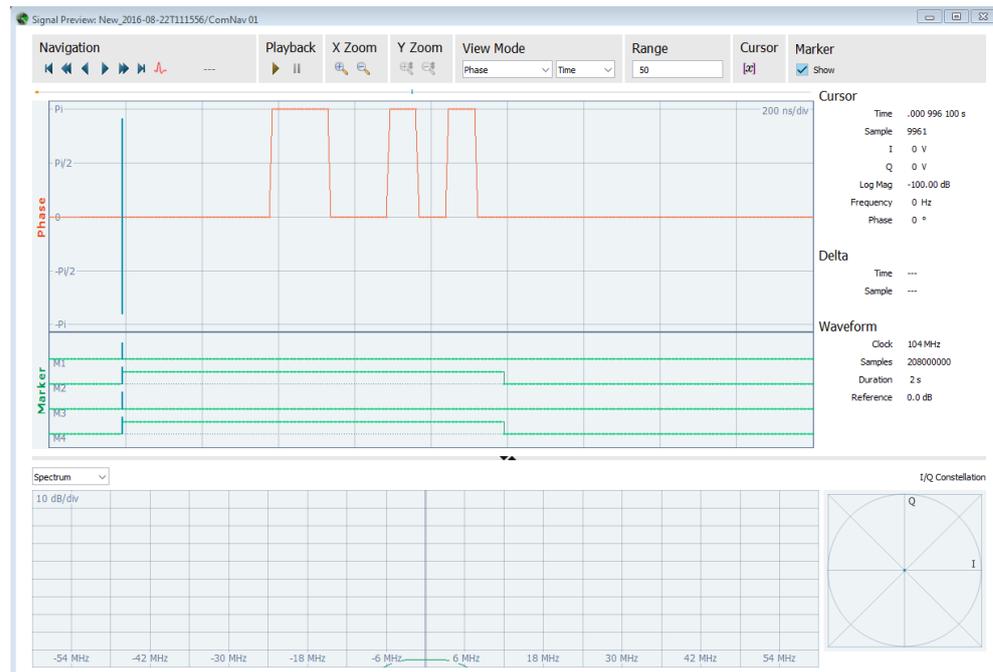
Figure 2-5: Signal Preview: Phase changes confirm the used Barker modulation (partial view, 10th pulse shown)

7. Compare the signal spectrum with the spectrum on Figure 2-4.

   You can retrieve more information about the waveform, for example observe the spectrogram, display a part of the signal in greater details, or enable and display marker traces.

   You find details in the following:
   ● Chapter 15.1, "Waveform view and signal preview settings", on page 400
   ● Chapter 21, "Defining and enabling marker signals", on page 534

2.7.6 Transferring the waveform to and playing it with a connected instrument

The example configurations are theoretical cases, because you rather do not use the software alone. Usually, the PC with the installed software would be connected via LAN to a vector signal generator.

Transferring of waveforms and playing them by a signal generator is beyond the scope of these first steps.

For description, see Chapter 17, "Playing the generated waveform files", on page 475.

2.7.7 Saving and recalling settings

Repositories are automatically saved when you exist the application. We recommend that you set a meaningful repository name before closing the application.

To rename the repository

Your workspace shows one automatically created repository, "New_yyyy_mm_tttt".

1. In the repository tree, select "Repository > New_yyyy_mm_tttt".

2. Change the name. Optionally, enter a comment.
   For example, enter "General > Name > K300 and K301 Tests" or a "Comment > Repository with test files".
   To enter a new line in the comment field, press [Shift+Enter].

3. Close the dialog.

To save the repository

A yellow data base icon ⬛ in the repository tree indicates that there are unsaved changes.

► In the menu bar, select "File > Save Repository > _Repository name_".

The repository is stored in the data path set during the program installation.

To exit the software

► Close the application software.

If the current repository contains unsaved changes, you are prompt to store or ignore them.
Note: Unsaved changes are lost.

For more information, see:
- Chapter 2.8.1, "Customizing your workspace", on page 48
- Chapter 4.4, "How to manage the project data", on page 115

2.7.8 Advanced features and examples

With these first steps, you have gained an impression of the provided functionality.

You find a comprehensive description of the full range of capabilities in the following sections:
- Chapter 6.3, "How to create a new pulse and adjust its settings", on page 190
- Chapter 7.3, "How to create sequences and use the control elements", on page 212
- Chapter 5.3, "How to select and create a test scenario", on page 145
- Chapter 3.3, "Configuring the test setup in the R&S Pulse Sequencer", on page 60
- Chapter 15.4, "Analyzing the calculated results", on page 411
- Chapter 17.5, "How to create and play the waveforms of simple scenarios", on page 489
- Chapter 22.2, "How to create test reports", on page 548

- Chapter 9.4, "How to create a library with antenna patterns and scans", on page 293
- Chapter 10.4, "How to create and configure emitters", on page 320
- Chapter 16.1, "How to create scenarios with receiver and TX items", on page 416
- Chapter 14.3, "How to create a waveform scenario and work with waveforms", on page 385
- Chapter 8.2, "How to create IPM profiles and vary pulse parameters", on page 245
- Chapter 21.2, "How to configure and visualize markers", on page 537
- Chapter 19.2, "How to configure the bit stream used by the MOP", on page 529
- "How to import and assign user defined plugins" on page 530

2.8 Customizing the software

You can change default application settings to adapt the user interface to your needs.

2.8.1 Customizing your workspace

A workspace comprises the repository tree on the left side with one or more loaded repositories, and the working area with one or more screens. You can create several workspaces with different repositories assigned to them. You can save and load a user-defined workspace and switch between the different workspaces.

To save a workspace

We assume that you have opened or created a repository and opened one or more dialogs in one of the screens.

1. In the menu bar, select "File > Save Workspace/Save Workspace As" or use the save icon in the toolbar.

2. In the explorer, enter a filename, e.g. `MyWorkspace.pswk` and click "Ok".

   Per default, the `*.pswk` file is stored in the application data directory.

To load a workspace

1. In the menu bar, select "File > Open Workspace".

   Existing default and user-defined workspaces are listed.

2. Select a workspace.

   The workspace opens. All dialogs and repositories associated with the workspace are displayed.

To close a workspace

You can close a workspace if you do not need it no longer or you need to switch to a different one.

► In the menu bar, select "File > Clear Workspace".

The workspace is closed.

Closing the workspace unloads all repositories and closes all dialogs.
You can open a workspace again, see

2.8.2 Changing colors and default configuration

You can change the default application settings, for example, change the display colors or change default settings of the R&S Pulse Sequencer.

To customize the display colors

1. In the menu bar, select "Configure > Colors".
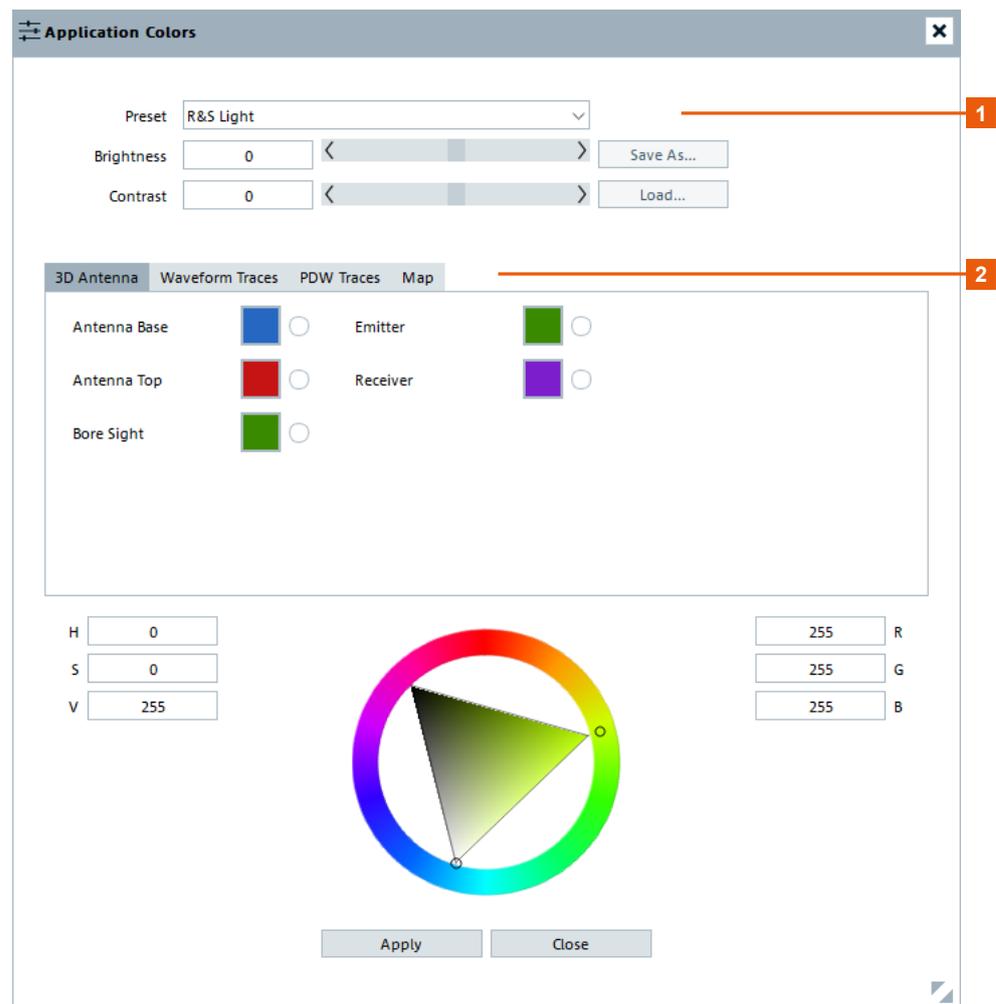
1 = Color scheme
2 = Assigns colors to the different display elements

2. Select a predefined color scheme and, if necessary, adjust the display colors individually.

3. Adjust the display colors of the individual elements, e.g select "Traces > Magnitude" and change the default color.

4. Select "Save As" to store your custom color profile in a file, e.g. `MyColors.col`.

To change the general program settings

1. In the menu bar, select "Configure > Settings".
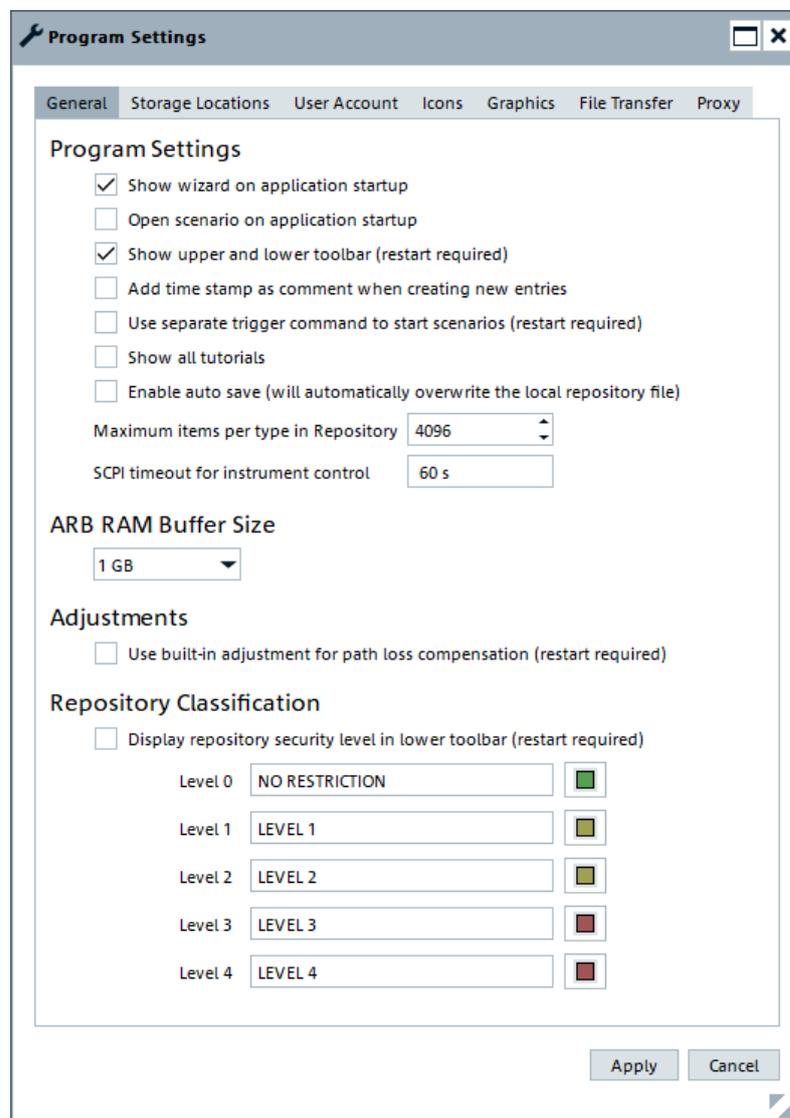
2. Change the settings, select the colors indicating the different security levels.
For information on the security levels, see "General Repository Settings"
on page 106.

3. Click "Apply".

To set the size and color scheme of the hardcopies

For documentation purposes, you can create a hardcopy or screenshot of the current screen or dialog. Hardcopies can be sent to the printer or stored to the clipboard.

1. To adjust the hardcopy size, in the menu bar, select "Configure > Settings > Graphics".
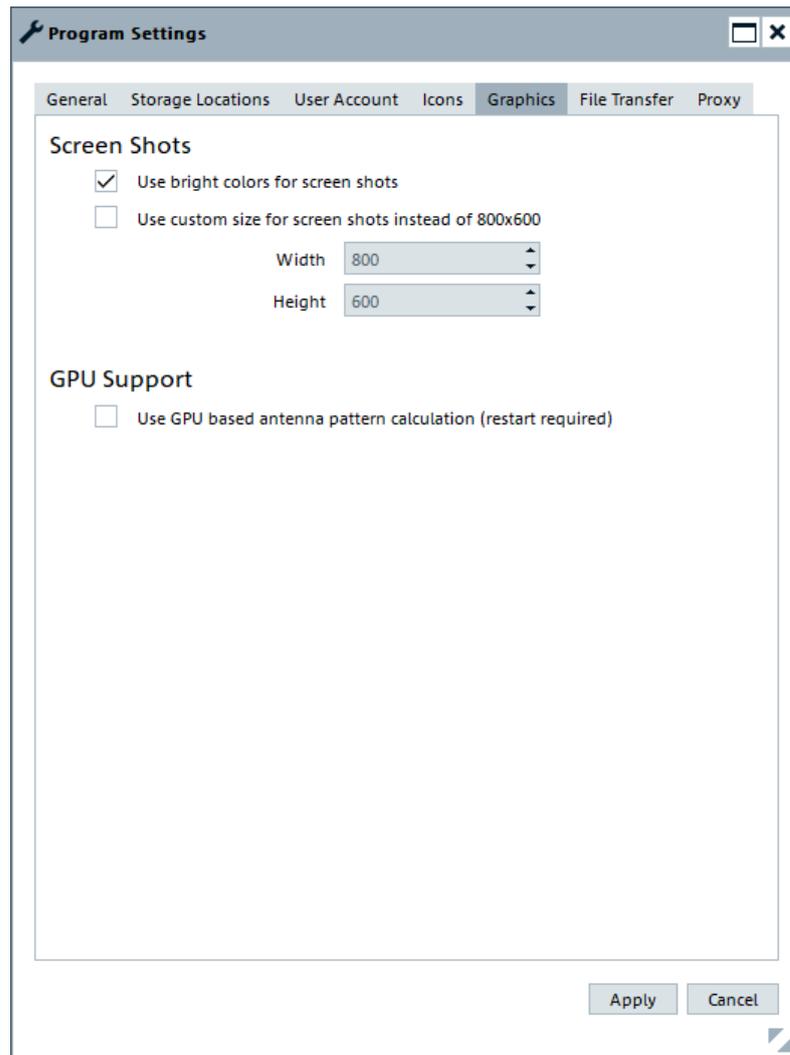
2. Select "Use bright colors for screenshots > On" e.g. for ink-saving printing.

   Brighter colors are used for all graphical screens instead of the ones set in the current color scheme.

3. Select "Use custom size for screenshots instead of 800x600 > On" and set the "Width" and "Height" in number of pixels.

4. Click "Apply".

The settings provided on the other tabs in the "Program Settings" dialog are explained in:

- Chapter 23, "Performing administration tasks", on page 551
- Chapter 17.3, "File transfer settings", on page 485

2.9 Getting information and help

If any questions or problems concerning the R&S Pulse Sequencer arise, an extensive online help system is provided in the software and can be consulted at any time.

The integrated help system provides the description of all functions. The help system is context-sensitive and provides information specifically for the current operation or setting to be performed. In addition, general topics provide an overview on complete tasks or function groups and background information.

Calling up Help

► To display the "Help" dialog for the currently focused screen element, e.g. a setting in an opened dialog, select the "Help" icon on the toolbar or press F1.

The "Help" dialog is displayed. A topic containing information about the focused element is displayed. If no context-specific help topic is available, a more general topic or the "Contents" tab is displayed.

The "Help" toolbar provides buttons with standard navigation functions. The "Help" window contains several tabs:
- "View" - shows the selected help topic
- "Contents" - contains a table of help contents
- "Index" - contains index entries to search for help topics
  The index is sorted alphabetically. You can browse the list, or search for entries in the list
- "Bookmarks" - contains custom bookmarks
- "Search" - provides text search

To close the Help window

► Select the "Close" icon in the upper right corner of the help window.

2.9.1 Tutorials

The R&S Pulse Sequencer offers tutorials on some of the software's functions and tools.

These tutorials are short animated clips on how to perform key actions.

There is also a tutorial of new features on startup.

Accessing the tutorials

By default, a tutorial is offered automatically the first time you access the associated function. For example, when you click "Map" in a 2D map scenario.

When a tutorial is offered, you can watch it or decline it.
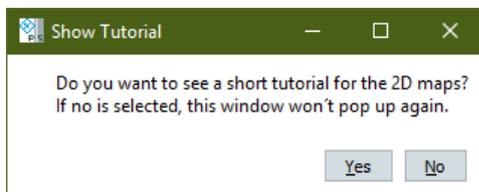
Figure 2-6: Show Tutorial message box

Click "Yes" to start the tutorial.

If you decline (i.e. by clicking "No"), the tutorial is not offered again.

To start a tutorial on demand

► In the menu, select "Help > Tutorial" and select on of the tutorials.

To re-enable all the tutorials

1. In the menu bar, select "Configure > Settings" to open the "Program Settings" dialog.

2. In the "Program Settings" dialog, select the "General" tab (see Chapter 23.1, "Setting general program settings", on page 551).

3. In the "General" tab, select the "Show all tutorials" checkbox.
   Tutorials appear when you access related functions.
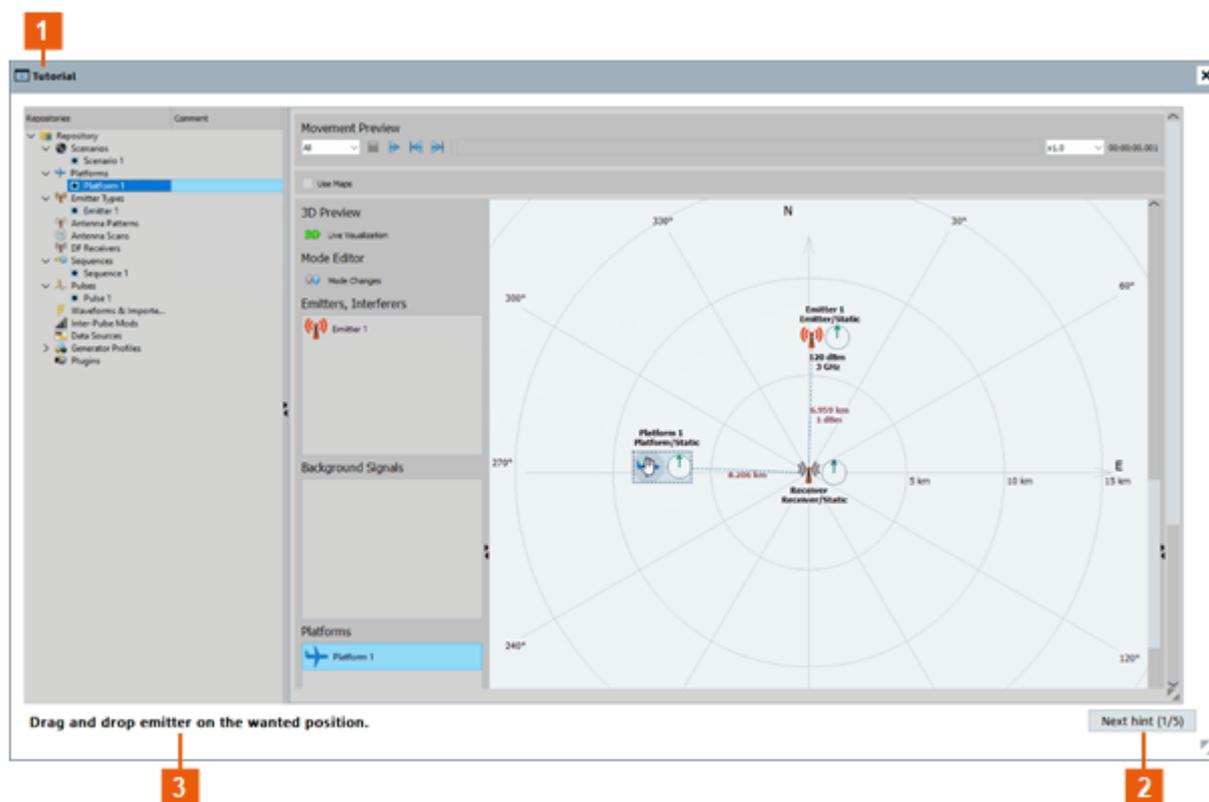
Figure 2-7: Tutorial window

1 = Topic of tutorial
2 = Start next tutorial
3 = Text describing current actions

(i)   It is not possible to interact with the user interface during the tutorial. Close the tutorial
      window to return to the user interface.

      The tutorials have no audio.

3 Preparing the system for use

The R&S Pulse Sequencer runs as a standalone software application on a PC. The software produces waveform or PDW files that can be used with a signal generator.

Initially, while becoming familiar with the waveform or PDW files or creating test signals, signal generators are not required. All configurations can be performed with the aid of virtual generators which are placeholders for physical instruments. Generators are defined in terms of ARB memory depth, RF frequency or signal bandwidth. Waveforms are then calculated to match the specific generators. Waveforms are stored on the local hard disk in the most suitable format for these generators.

Complex signal generation tasks require at least one signal generator. If a signal generator is equipped with the required options and assigned in the software, waveform files generated with the R&S Pulse Sequencer can be directly transmitted to it.

See Chapter 2.1, "Required options", on page 18 for an overview of the supported signal generators and a list of the required options.

Connect the test system first

Use of the R&S Pulse Sequencer does not require a connected external signal generator.

However, if the particular test setup is known, we recommend that you connect and cable the test system before you start the configurations in R&S Pulse Sequencer.

This section describes **the general steps** to be taken when preparing the test system for operation. It does not substitute the recommendations provided in the Getting Started manuals for the instruments.

3.1 Test setups

The R&S Pulse Sequencer generates complex pulsed signals for use with a signal generator. A typical setup includes a PC with installed software and at least one connected vector signal generator. Setups are fully controlled by the R&S Pulse Sequencer software. The software automatically derives all the required settings from the defined scenario.

This section shows examples with different complexity. We use these example setups to explain the required settings and configurations. The selected setups are representative examples and show the difference in terms of synchronization and time alignment.

Basic test setup

A simplified example of the basic setup is shown on Figure 3-1. The basic setup consists of a PC with installed R&S Pulse Sequencer software, a vector signal generator and a DUT (device under test).

Figure 3-1: Simplified test setup

Depending on the scenario, the generator in this setup can be an R&S SMW, an R&S SGT or an R&S SMBV.

Configuration of a direction finding scenario with a multichannel receiver

Direction finding receivers are used to identify the origin of a signal. They are equipped with multiple antennas. Each antenna has an individual position and covers a certain angular sector according to its antenna pattern. The signal received at each antenna differs based on the location of the signal. Thus, it is possible to identify the sector in which the signal source is located.

The figure on Figure 3-2 illustrates a typical test setup to stress a four-channel direction finding receiver (DUT).
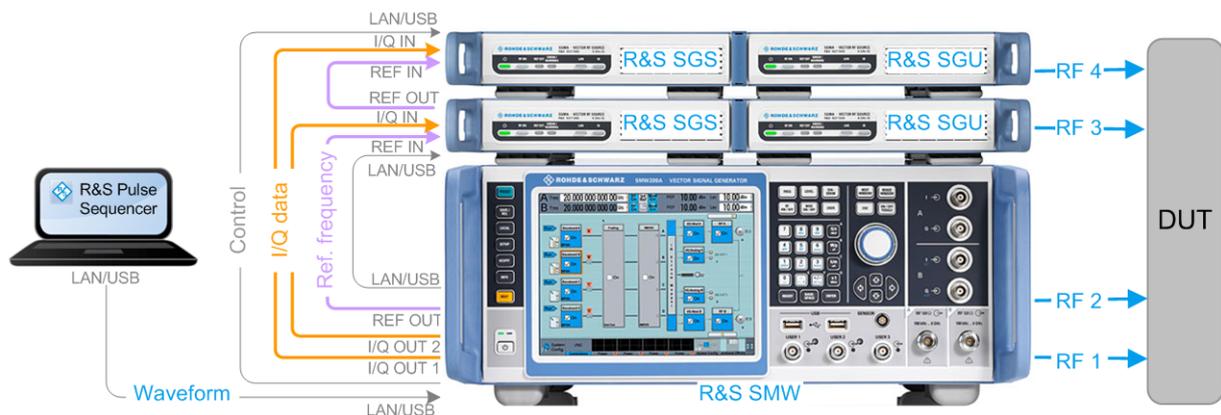
Figure 3-2: Direction finding setup for four individual RF channels up to 20 GHz

This setup requires four individual RF output signals, where typically each RF signal is then connected to the input port of the direction finding receiver. The signal generators in this setup are a two-path R&S SMW and two pairs of R&S SGS and R&S SGU instruments.

The R&S SMW is a microwave unit equipped (among other options) with R&S SMW-K76 and 2x R&S SMW-B120. The R&S SGU instruments are each equipped with the 20 GHz option, too.

The test setup for testing an eight-channel direction finding receiver with individual RF channels of up to 6 GHz includes an R&S SMW and six R&S SGT instruments. A nine-channel DF receiver can be tested with nine R&S SGT instruments.

3.2　About synchronization and time alignment

This description provides a brief introduction to the topic. It explains how the terms are used in the context of the software and list general tips for getting the best results.

ⓘ　Time alignment, synchronization, the generation of phase coherent signals and test system calibrations are topics that are beyond the scope of this description.

You find in-depth information in the Application Note 1GP108 "Generating Multiple Phase Coherent Signals – Aligned in Phase and Time".

Synchronization

The setup on Figure 3-2 is an example of a synchronized configuration. The term-synchronized setup refers to a setup where the reference frequency of one of the instruments (usually the primary instrument) is distributed to the other instruments.

Regarding the reference frequency, the instruments are connected in a **daisy chain**.

The four baseband signals in the setup on Figure 3-2 are generated by the R&S SMW and automatically aligned in time. The R&S SMW distributes the baseband clock to all basebands. Additional trigger signal or baseband clocks are not required.

Time alignment in primary/secondary configuration (R&S SMBV, R&S SGT and R&S SMW equipped with R&S SMW-B9 (wideband baseband))

The term primary/secondary setup refers to a setup where instruments share baseband clock and trigger signal. Primary/secondary configuration is supported by R&S SMW, R&S SMBV and R&S SGT instruments.

The example in Figure 3-3 shows four R&S SGT instruments, each generating one RF signal. To synchronize all baseband sources, the primary instrument provides its baseband clock and its trigger signal to the secondary instruments. The trigger signal is modulated on the baseband clock; an additional cable connection is not required.
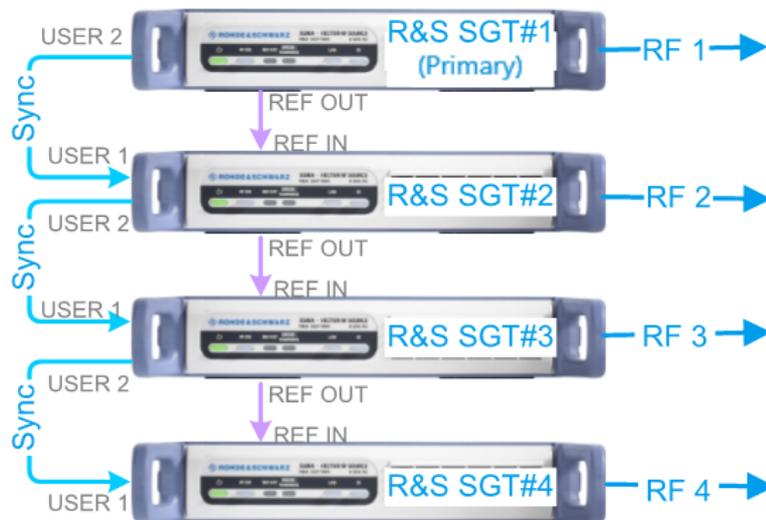
Figure 3-3: Time alignment of R&S SGT instruments

R&S SGT#1   = Primary instrument
R&S SGT#2 to R&S SGT#4 = Secondary instrument
Sync   = Baseband synchronization signal

Regarding the baseband clock signals, the instruments are connected in a **daisy chain**.

For details, see the user manual of the corresponding base unit.

Time alignment of several R&S SMW instruments equipped with R&S SMW-B10 (standard baseband)

The time alignment between several R&S SMW instruments equipped with R&S SMW-B10 follows a different approach. The example in Figure 3-4 shows the time alignment principle. The trigger signal of the primary instrument is distributed to the secondary instruments.

Regarding the trigger signal, the instruments are connected in a **star network**.
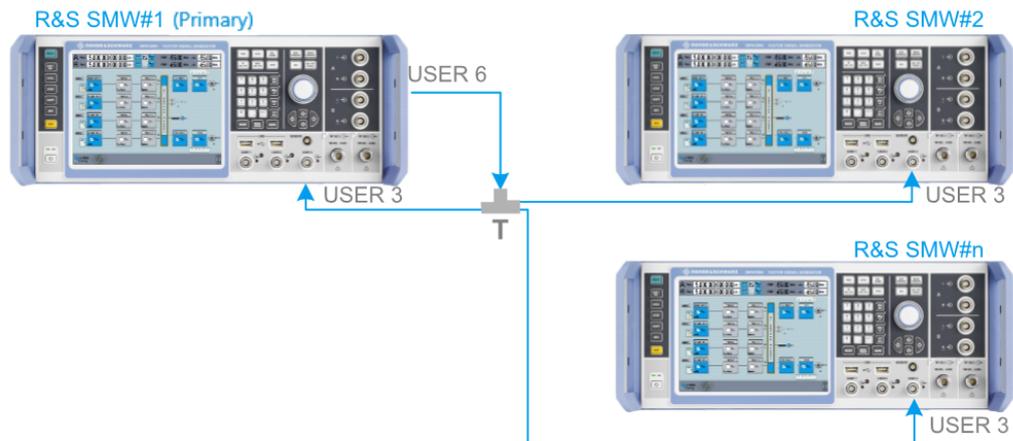
Figure 3-4: Time alignment of R&S SMW instruments

T   = T-connector
R&S SMW#1   = Primary instruments (trigger source)
R&S SMW#2, R&S SMW#n = N number of instruments

For details, see the user manual of the corresponding base unit.

Tips for getting the best results

Consider the following general rules:

● Always use the shortest cables possible.

● Use cables of the same length.

● Testing of direction finding receivers requires precisely synchronized, time aligned and phase coherent signals.

● If a direction finding setup consists of several instruments, all baseband signals have to use common trigger and baseband clock signals.
  See:
  – "Time alignment in primary/secondary configuration (R&S SMBV, R&S SGT and R&S SMW equipped with R&S SMW-B9 (wideband baseband))" on page 57
  – "Time alignment of several R&S SMW instruments equipped with R&S SMW-B10 (standard baseband)" on page 58

● In synchronized and primary/secondary test setups, use the same kind of signal generators.
  Do not mix R&S SMBV and R&S SGT in the same synchronization or primary/secondary chain.

● Always check the wiring diagram for the required cabling between the individual instruments.
  Ensure that all required cable connections are done.
  See "Wiring Diagram" on page 84.

3.3　Configuring the test setup in the R&S Pulse Sequencer

Waveforms generated by the R&S Pulse Sequencer can be transferred to and played by remotely connected signal generators.

The connection between the software and the signal generator is configured with the following main steps:

- Create and configure the **physical instruments**.
- Assign the **calculated signal to a generator**.
- Assign a **generator to the scenario**.

How to configure the test setup

To configure the test setup, follow the instructions:

1. "How to find connected instruments and configure them in the "Signal Generators" dialog" on page 60

2. Later in the workflow, you must also assign generators to the scenario and the calculated signals to the generators.
   See the instructions in:

   a) Chapter 17.5, "How to create and play the waveforms of simple scenarios", on page 489
   b) "To create, load and play the waveforms of a direction finding scenario automatically" on page 492

   See also:
   - "How to find out the full list of required options for a particular scenario" on page 63
   - "How to exchange test setups" on page 65

Related settings

For a concise description of the provided settings, see:

- Chapter 3.6, "Signal Generators settings", on page 75

How to find connected instruments and configure them in the "Signal Generators" dialog

1. Connect a signal generator to the LAN.

2. Connect the PC that the R&S Pulse Sequencer software is running on to the LAN.

3. In the R&S Pulse Sequencer, select "menu bar > Configure > Signal Generators".

   The "Default Setup" tab and the "Available Signal Generators" list are empty.
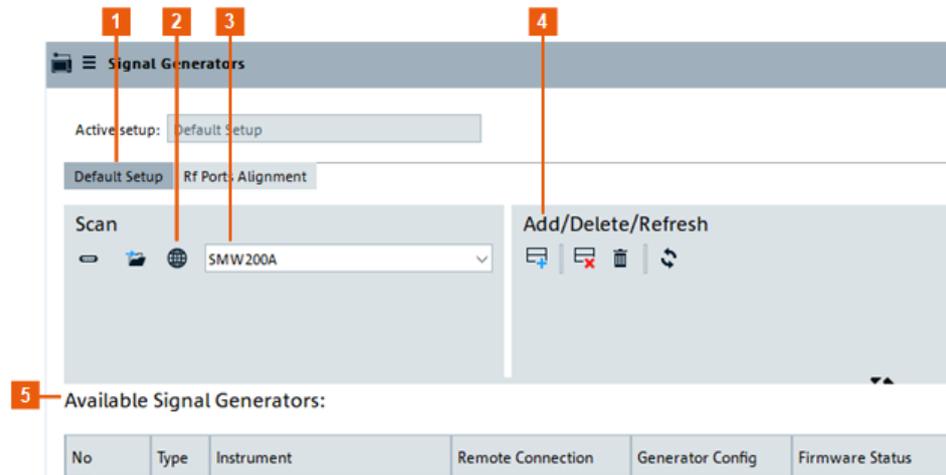
1 = Current setup
2 = Scan LAN
3 = Instrument model to limit and accelerate the scan process
4 = Add an instrument manually
5 = List of instruments in the current setup (empty on startup)

4. Select the "Default Setup" or the "RF Ports Alignment" tab.

5. To add an instrument, perform one of the following:
   - Execute a "Scan", e.g. in the LAN network, to search for connected instruments of a specified type.
   - In the "Add/Delete/Refresh" section, add an instrument manually.
     Enter the IP address, the computer name, or the complete VISA resource string of the connected instrument. Click "Ok".
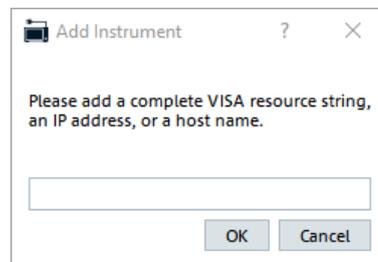
Note: The computer name syntax of R&S SMW instruments depends on the firmware version.

The two possible syntax forms are: `rssmw200a<serial_number>` and `smw200a-<serial_number>`.

The LAN is scanned for connected instruments.
If suitable instruments are found, the dialog lists them, together with information on their capabilities (see "Generator Configuration" section).
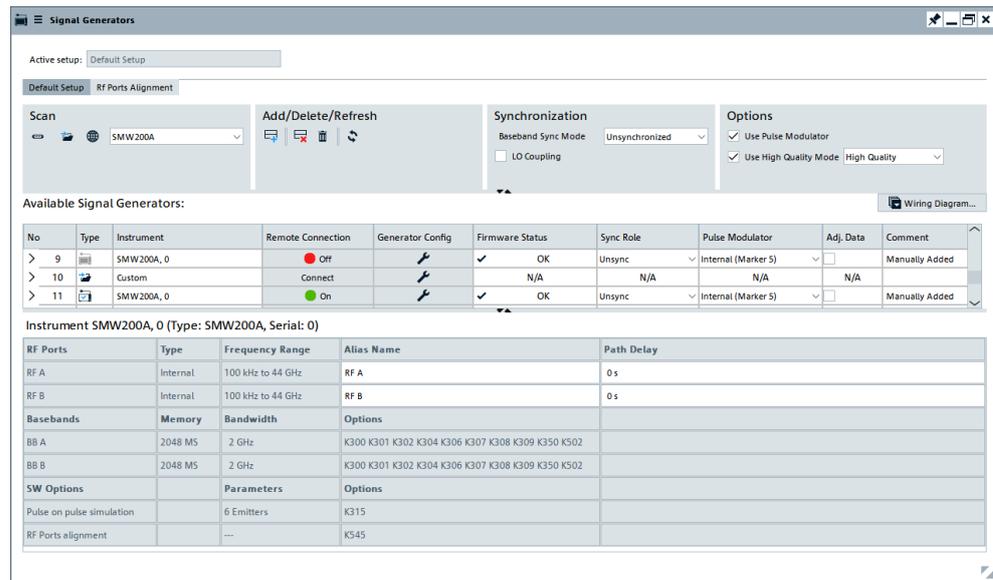
For a description of the provided settings, see Chapter 3.6, "Signal Generators settings", on page 75.

How to configure the primary/secondary order

Depending on your test setup and your test scenario, set the primary instrument and decide on synchronization requirements.

Open the "Signal Generators" dialog and proceed as follows:

1. Select the setup.

2. Enable synchronized setup. For example, select "Synchronization > Baseband Sync Mode = Common Trigger".

3. For each instrument in the list of instruments, select "Sync Role" > "Primary", "SEC x" or "Unsync".
    Per setup, there is exactly one primary instrument. The most recent selection overwrites the previous.
    Select "Unsync" for autonomous or unsynchronized instruments.

4. Set individual secondary numbers ("SEC x") according to the order of the connected secondary instruments.
    The indication "SEC x" influences the cabling, the trigger delay calculation and the system calibration.
    R&S Pulse Sequencer does not allow gaps in the numbering.
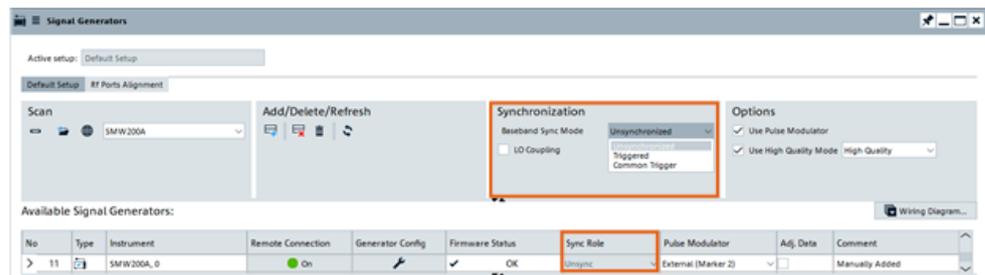
5. Observe the "Wiring Diagram" for correct cabling.

   The wiring diagram shows all the required cabling between the individual units.
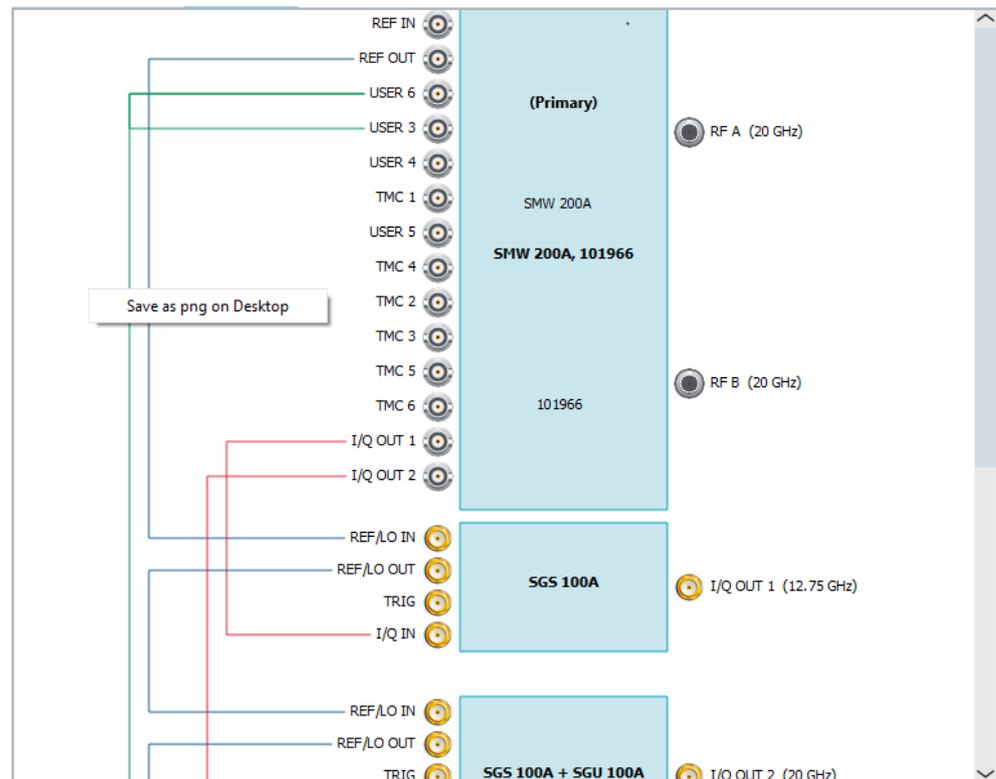
Tip: Save the wiring diagram in a file on the desktop.
In the "Wiring Diagram", right-click and select "Save as png on Desktop".

A printable overview image is created and stored that shows the whole wiring diagram.

6. Connect the instruments as required.

How to find out the full list of required options for a particular scenario

During the initial tests or if the signal generators are not purchased, you can configure virtual generators without access to physical instruments.

Try out the following:

1. In the menu bar, select "Configure" > "Signal Generators".

   The "Signal Generators" dialog opens.

2. If the "Available Signal Generators" list is empty, select "Add" and select a signal generator from the list.

    The new virtual generator appears in the list.
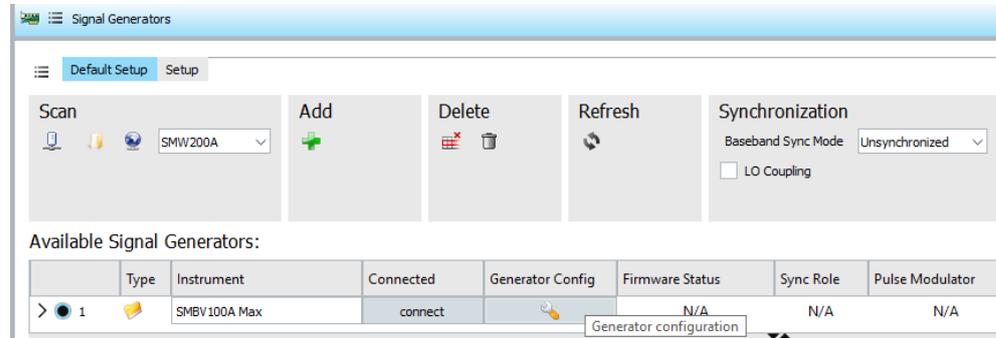
3. In the "Available Signal Generators" list, select a generator. Select the generator configuration icon.
    The "Generator Config" dialog opens.

4. In the "Generator Config" dialog, go to the "Pulse Sequencer Features".

5. Use the tooltip texts to find out more about each option.
    You can also select a checkbox and read the associated notes that appear in the "Capabilities" section of the dialog.

6. Select "Show Generator Config" to open the "Required Options" dialog. Here, you can see the list of required R&S Pulse Sequencer related software options.

7. Retrieve the required baseband and RF options in the same manner.
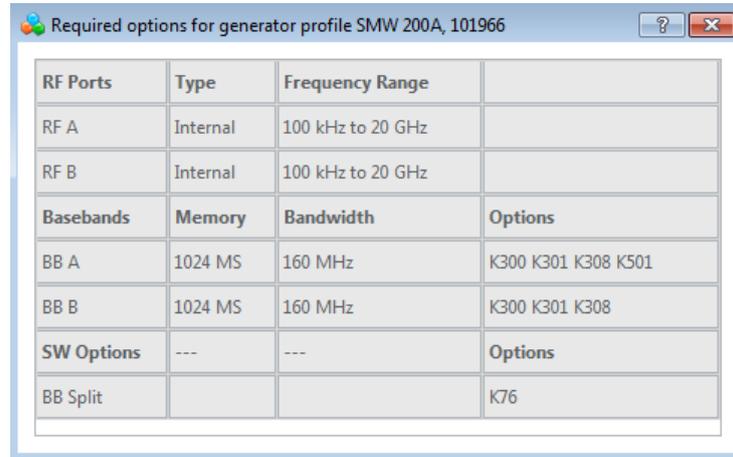
How to exchange test setups

If a similar test setup already exists, you can import its configuration, and then adjust it as required.

1. In the "Signal Generators" dialog, configure the setup.

2. Select the setup configuration that you want to share (export).
   The setup name is highlighted.

3. In the context menu, select "Export".
   Navigate to the directory, enter a filename and select "Save".

4. To load an existing setup, in the "Signal Generators" dialog select "Import" from the context menu.

5. Navigate to the directory with setup files, select the file and click "Open".
   The directory only shows files with the `*.ps_hwsetup` file extension.

   The selected file is loaded. The imported setup is appended in the list of setups.

3.4 Compensating time delays

Several test setups require precise timing between the generated signals. Timing alignment is crucial, for example, in direction finding scenarios and setups such as the one in Figure 3-2. Even if you connect and configure instruments as described in Configuring the test setup in the R&S Pulse Sequencer, time delay between the signals of the primary and the secondary instruments can appear.

This section explains in brief how to compensate for time delays in the generated signals. The process is manual. Time delays must be measured and compensated for each test setup separately.

Delay compensation principle

In the following, we assume three R&S SGT instruments connected in a synchronized setup (Figure 3-2). The instruments are configured and controlled by R&S Pulse Sequencer; the generated signals are fed to an oscilloscope. The oscilloscope runs in time measurement mode with fine time resolution. The trigger signal is the marker signal generated by R&S Pulse Sequencer and fed from the primary instrument to the oscilloscope.

Figure 3-5 shows schematically possible results displayed by the oscilloscope.
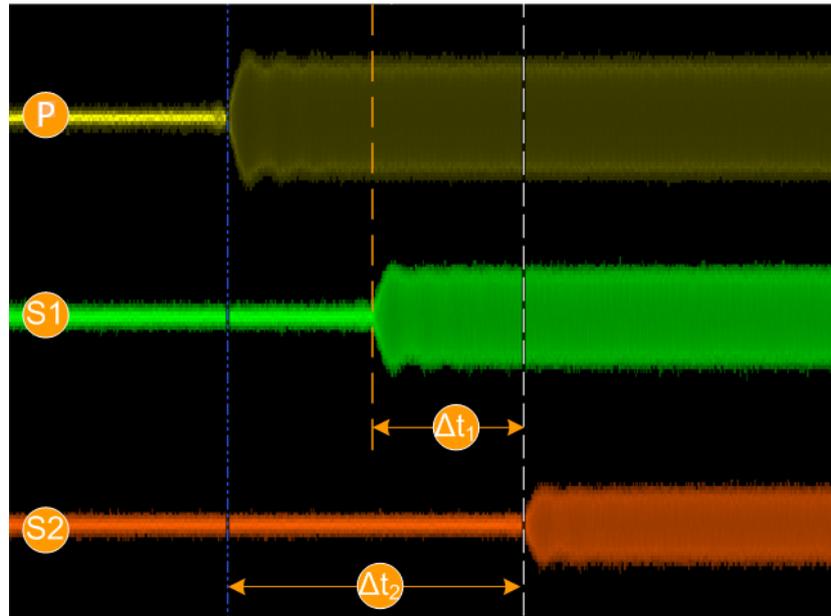
Figure 3-5: Time delays: Observing and measuring with an oscilloscope

P, S1, S2 = Signal traces for the primary and the secondary instruments
$\Delta t_1, \Delta t_2$ = Time delays

Although the setup is synchronized, due to the signal processing within the instruments and signal propagation time over the cables, the three signals are not time-aligned. Moreover, also level difference can be measured. The latter is due to path loss and can also be compensated, see Chapter 3.5, "Compensating path loss", on page 67.

The signal that arrives first is the signal of the primary instrument. It is impossible to speed up the signal of the secondary instruments. Therefore, we use the signal with the longest time delay as reference and add fixed delays to the other signals.

Measure the time difference (Δt_1 and Δt_2) between the instruments. Add these values in "Signal Generators" > Chapter 3.4, "Compensating time delays", on page 65.

Run the same measurement again and observe the signal timing with the oscilloscope. The tree signals are aligned in time.

Tips for getting the best results

- Use a synchronized setup.
  See Chapter 3.2, "About synchronization and time alignment", on page 57

- Create a test sequence composed of a repeating simple pulse with zero rise and fall times followed by a blank.
  For example, pulse with pulse width of 100 us, PRI = 200 us and 5 repetitions. Blank = 10 ms.

  See:
    - Chapter 6.3, "How to create a new pulse and adjust its settings", on page 190.
    - Chapter 7.3, "How to create sequences and use the control elements", on page 212.

- Use the first pulse right after the blank for time alignment.

- Use a high clock rate like 200 MHz.
  High clock rates result in steeper rising edges and thus help time alignment.

- Use an isotropic steady antenna (no scan) to eliminate the impact of antenna patterns and turns.

- Use emitters with the same power level.
  The latter is important for level calibrations, see Chapter 3.5, "Compensating path loss", on page 67.

3.5 Compensating path loss

The R&S Pulse Sequencer provides a build-in function to collect level calibration data using a R&S NRP power sensor per setup. Moreover, it saves and manages the collected files and applies the level correction values to the instruments in the setup.

Principle of level adjustment

Figure 3-6 shows an example of a simple test setup that we use to explain the path loss compensation principle. The instrument in this example is an R&S SMW but you can use any supported instrument.

Connect the R&S NRP power sensor to the input of the DUT. In this way the collected calibration data reflects the losses and delays caused by all cables, routers, or any other devices included in the test setup.

Figure 3-6: Path loss compensation (simplified setup)

1	= Symbolic representation of attenuator, combiner, or other component connected between the signal output and the DUT input
2	= R&S NRP power sensor
USB*	= In this example, R&S NRP power sensor is connected to the USB connector of the instrument; other connection possibilities exist
USB/LAN	= Physical connection between the R&S Pulse Sequencer and the instrument
RF 1, RF 2	= Two output signals; hence adjustment data is collected for the two signals separately

If level adjustment is enabled and started, R&S Pulse Sequencer detects R&S NRP power sensors connected to the instrument via USB automatically. Power sensors connected to the LAN can be configured in R&S Pulse Sequencer, too.

Upon level measurement start, the R&S Pulse Sequencer controls the instrument and sweeps the particular signal over the defined frequency range, so that the frequency response of the system is measured. Depending on where in the chain the sensor is connected, measured data includes cables, devices, etc. Once the measurement is completed, a dedicated plot indicates the frequency response of the signal. The plot is automatically scaled but you can use the marker function and retrieve exact level correction values at a particular frequency.

If current measurement results are inaccurate, you can repeat the measurement. In fact, measurement can be started as many times as required.

Adjustment data storage location

Throughout the software and this description, the calibration data is referred as adjustment data. You have several possibilities to handle the adjustment data. Measured adjustment data can be accepted or discarded, and once accepted, you can select the storage location for it.

Storage location can differ depending on, for example, security concerns (access rights to the adjustment database), network topology (bench-top operation vs. common setup), user access rights.

For details on the different possibilities and when to use which one of them, see Chapter 3.8.2, "Adjustment database storage location settings", on page 94.

Related settings

For a concise description of the provided settings, see:
- Chapter 3.8, "Adjustment configuration settings", on page 92
- Chapter 3.9, "Manage adjustment data settings", on page 97
- Chapter 3.10, "Run level adjustment settings", on page 99

3.5.1 Collecting level adjustment data

How to cable and configure the setup

► Connect the instruments incl. power sensors.
   Configure the R&S Pulse Sequencer as described in Chapter 3.3, "Configuring the test setup in the R&S Pulse Sequencer", on page 60.

How to identify and measure level differences

1. Generate a test signal, for example as described in "Tips for getting the best results" on page 66.

2. Feed the generated RF signals to an oscilloscope.

3. Feed also the marker signal generated by R&S Pulse Sequencer and output at the output connectors of the instrument to the trigger input of the oscilloscope.

4. Run the oscilloscope in time measurement mode with fine time resolution. Observe and measure the levels of the generated signals.

   If the signals are not of equal level, for example as shown on Figure 3-5, follow the procedures in this section.

How to enable the level adjustment function

1. In the menu bar, select "Configure > Settings > General > Adjustments > Use build-in adjustment procedure for path loss compensation > On".
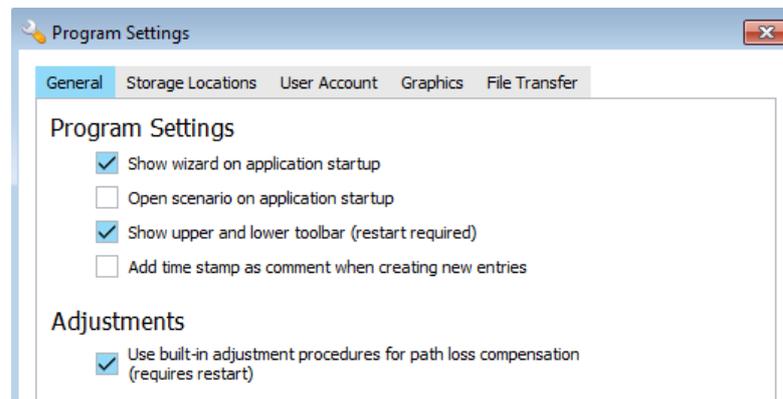
2. Restart the software.

   If successful, the new selection "Adjustment" appears in the menu bar.

3. In the menu bar, select "Adjustment > Configure".
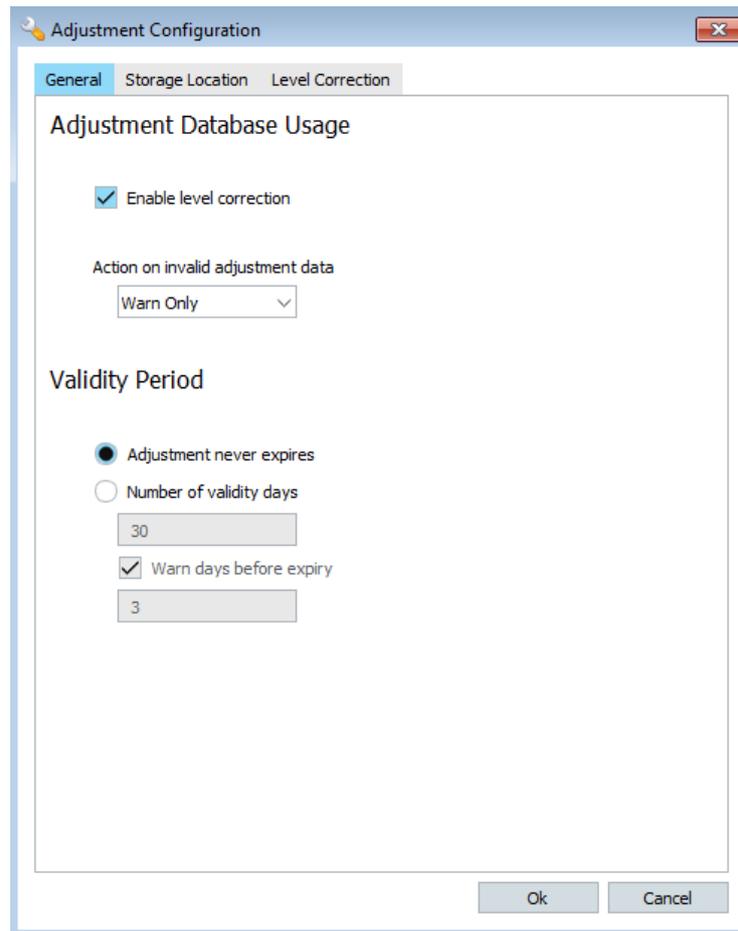
4. Select "General > Adjustment Database Usage > Enable Level Calibration > On".

5. Open the scenario dialog. Observe the indication.

A symbol indicates that level adjustment is enabled but there is no data collected or the collected data is incomplete.

How to control R&S NRP connect to R&S SMW via USB

1. Connect the R&S NRP sensor (RF signal) port to the DUT.

2. Connect the R&S NRP USB (control) port to the R&S SMW.

3. In the R&S SMW, map the sensor to the first power sensor channel:

   a) Select "RF > RF Measurements > NRP Sensor Mapping".
      R&S SMW detects and displays connected sensors automatically.
   b) If the list is empty, select "Scan".
   c) For the used sensor, set "Protocol > Legacy".

d) Set "Mapping = 1".
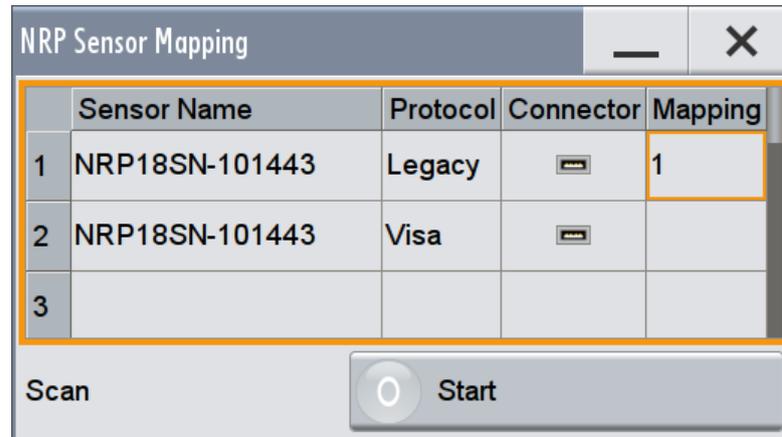
4. In the R&S Pulse Sequencer, select "Menu bar > Adjustment > Configure > Level Correction".
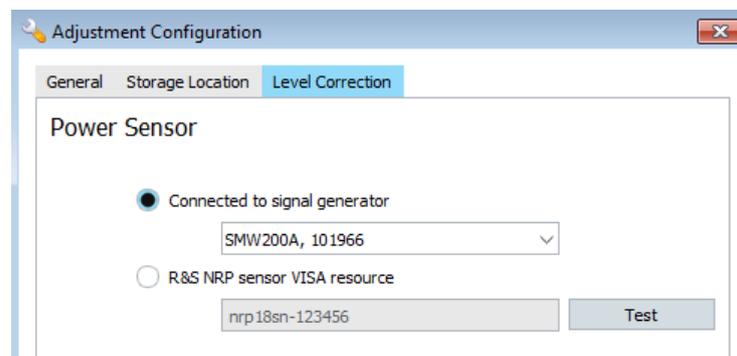
5. Select "Power Sensor > Connected to signal generator".
   Select the used instrument from the list.

6. Use the option "R&S NRP sensor VISA resource" if you connect the power sensor to LAN (instead via USB).

How to set level measurements limits

1. In the "Adjustment > Configure > Level Correction" dialog, set the "Target Noise Ratio" value as required.

2. Set the "Frequency Range" to be measured.

How to start level measurement

1. In the menu bar, select "Adjustment > Run Level Adjustment".

   The dialog indicates the selected instrument (see step 5) and the configured "Frequency Range".
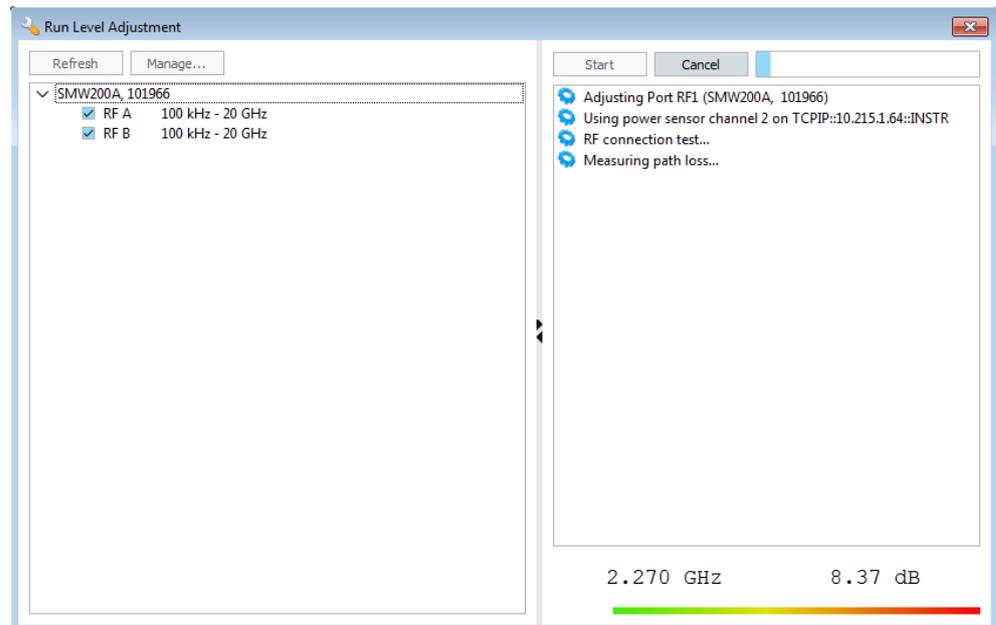
2. In the list of instruments, select the instrument and signal path for that you collect level adjustment data.

3. Select "Start".

4. Follow the displayed instructions.
   The software detects improper connection status, such as a power sensor connected to a wrong port, and displays a message.

5. Confirm that the power sensor is connected. Reconnect it, if necessary.

   The measurement is running. A progress bar indicates the progress status.
   Current frequency and correction values are also displayed.

6. Repeat the measurement for the other path.

   Successfully completed measurement is indicated by the message "Data collection completed. Adjustment database laded".

How to accept and apply adjustment data

Once collecting level adjustment data is competed, the "Manage Adjustment Data" dialog opens automatically.

If you have to open it manually, proceed as follows:

1. In the menu bar, select "Adjustment > Manage".

<host> = Indicates the computer on that the software runs
<current user> = Current user windows login name

2. If the measurement result complies with your expectation, select "Accept".

3. If the measured data is inaccurate, you can discard it ("Reject"), delete the current measurement or remove all saved adjustment data (empty current storage location).

4. Repeat the steps for all traces.

   Accepted adjustment data is saved in the database. The trace is indicated in green color.

5. If you repeat a measurement for that you have collected and saved adjustment data, you can choose to:

   ● Discard ("Reject") the newly measured data and retain the previously collected values.
   ● Replace the previous data and apply the new measurement ("Accept"). The previous file in the storage location is also overwritten.

   You recognize that adjustment data is collected and accepted by the status message "No data pending" indicated in the "Manage Adjustment Data" dialog. Accepted adjustment data is applied automatically to the generated signal.

How to define where adjustment data is saved

In this example, we select to save data on a particular instrument from the setup.

1. In the menu bar, select "Adjustment > Configure > Storage Location".

2. Select "Located on Hardware Setup".
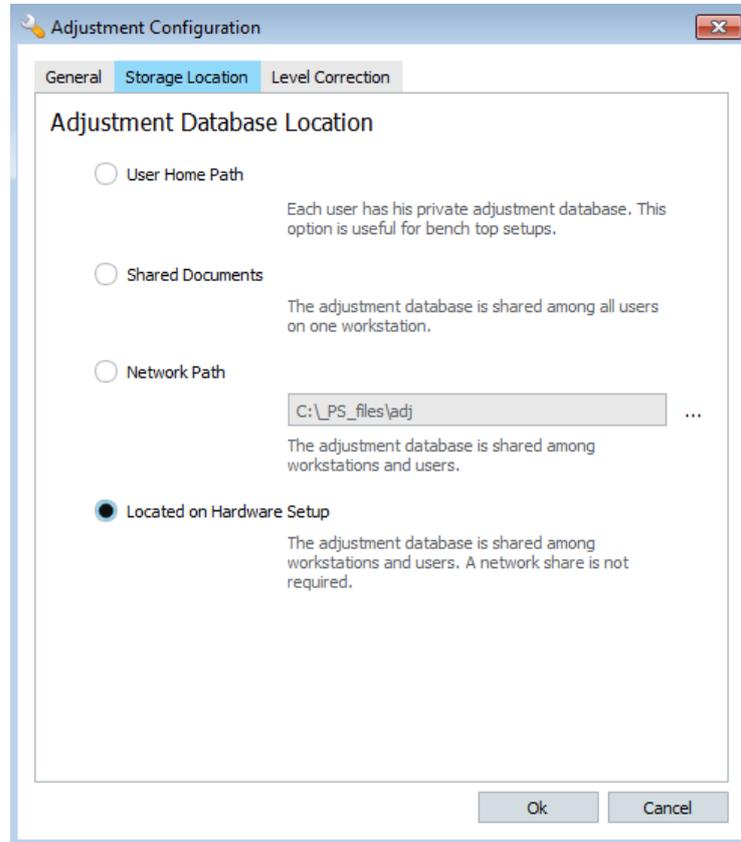
3. In the menu bar, select "Configure > Signal Generators".

4. Open the "Setup".

5. In the "List of Instruments", select the instrument on which you want to save the adjustment data.
   For this instrument, set "Adj. Data > On".

		Instrument	Supported	Create Profile	Generator Profile	Linked	Sync Role	Pulse Modulator	Adj. Data	Comment
>	● 1	SMW200A, 101966	✔ OK	=>	SMW 200A, 101966 ∨	✔ OK	Primary ∨	External (Marker 2) ∨	✔	Manually Added

The storage location for a setup can be on any instrument from this particular setup, regardless if it works as a primary or secondary instrument.
You can select exactly one instrument per setup at a time.

How to verify path loss compensation

1. Repeat the measurement described in

2. Compare the signal levels.

   If path loss is successfully compensated, the signal levels are equal.

3.6 Signal Generators settings

Access:

► Perform one of the following:
   - In the menu bar, select "Configure" > "Signal Generators"
   - In the block diagram of a scenario, select "Signal Generation" > "Config" > "Signal Generators".
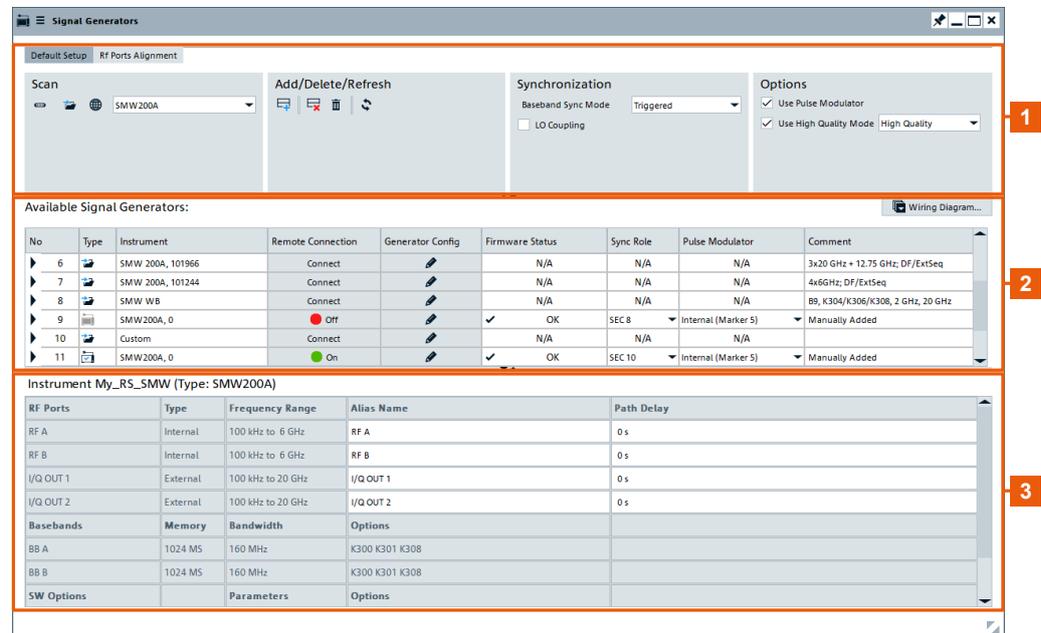
Figure 3-7: Signal Generators: understanding the displayed information

1 = Definition of settings in the active setup
2 = List of physical and virtual instruments
3 = Summary of selected generator

For step-by-step instructions, see Chapter 3.3, "Configuring the test setup in the R&S Pulse Sequencer", on page 60.

Settings

Default setup

The setup with the selected signal generator is active.

RF Ports Alignment

The RF ports alignment setup is active.

If an RF ports alignment setup is selected, the path and name of the imported setup file is displayed, see "RF Ports Alignment" on page 79.

Scan GPIB/USB/LAN, Any

Scans the selected local interface for connected instruments. To limit the scan time, select the instrument type, e.g. "SMW 200A".

The GPIB scan is limited to the first GPIB card that is installed (GPIB0).

SMW200A (BW 2GHz) Max

SMW200A (BW 160 MHz) Max

SMBV100A Max

SGT100A Max

SMBV100B Max

Connect

Manual Setup

Add

Adds an instrument manually.

"SMW200A (BW 2GHz) Max", "SMW200A (BW 160 MHz) Max", "SMBV100A Max", "SGT100A Max", "SMBV100B Max"

         Creates an instrument profile of the selected type.

"Connect"   Opens the "Add Instrument" dialog where you can define an instrument by its IP address/computer name or complete VISA resource string.

         If the instrument exists, it is added to the list of instruments.

         Note: The computer name syntax of R&S SMW instruments depends on the firmware version.

         The two possible syntax forms are: `rssmw200a<serial_number>` and `smw200a-<serial_number>`.

"Manual Setup"

         Opens the Generator config dialog dialog.

         You can define the instrument manually.

Remote command:

`INSTrument:ADD` on page 647

Delete, Delete All

Standard edit functions to handle the list of instruments.

Remote command:

`INSTrument:DELete` on page 594

`INSTrument:CLEar` on page 595

Refresh

Checks whether the listed instruments are connected or not and updates their connection status indication.

Synchronization

Groups the functions related to synchronous signal generation in multi-instrument setups.

Baseband Sync Mode ← Synchronization

Defines whether and which method the signal generator uses to synchronize the signals in the baseband domain.

Relevant in multi-instrument setups where the signals of the different emitters are generated in different paths and different signal generators.

"Unsynchronized"

Baseband signal generation starts upon signal generation start. The baseband generators are unsynchronized.

The indication in "Sync Role" confirms that the setup is unsynchronized.

"Triggered"   The instruments are connected in a primary/secondary chain. The role of the instruments is set with the parameter Sync Role.

Basebands are triggered by a general-purpose trigger signal. The time skew between the instruments is greater than in the "Common Trigger" mode and can change from trigger event to trigger event. See Figure 3-4.

"Common Trigger"

The instruments are connected in a primary/secondary chain and use the built-in multi-instrument mode of the signal generators.

If your instrument supports it, we recommend that you enable this mode. It ensures that trigger uncertainty is eliminated. With this mode, trigger delays between instruments remain constant. Hence, you can calibrate the setup, see Chapter 3.4, "Compensating time delays", on page 65.

For details, see the user manual of the corresponding base unit.

Remote command:
SETup:BBSYnc on page 651

LO Coupling ← Synchronization
Couples the LO signals.

Observe the Wiring Diagram for the required cabling.

For details, see the user manual of the corresponding base unit.

Remote command:
SETup:LOCPl:ENABle on page 651

Options
A Group of optional settings that require instruments equipped with specific options.

Use Pulse Modulator ← Options
Option: R&S SMW/SMBV/SGT-K22

The setting is useful if a signal with an improved pulse on/off ratio is required.

Enables the functionality in general that is for all instruments in the setup. Set the marker signal source per instrument with the parameter Pulse Modulator.

Remote command:
SETup:PMOD:ENABle on page 650

Use High Quality mode ← Options
Option: R&S SMW-B9

Sets the I/Q modulator of the signal generator to work in a high-quality mode that is sets the parameter "I/Q Modulator > Mode > High Quality".

For more information, see the user manual or the embedded help of signal generator.

High Quality   Enables compensation for I/Q skew and frequency response correction.
This mode generates a flat signal over a large bandwidth but requires a longer setting time and can lead to signal interruption.

High Quality Table

This mode provides optimization while maintaining settling time.
This mode is useful in the following cases:

- An R&S SZU is connected to an R&S SMW and the connection is active.
- In setups for the generation of phase-coherent signals.
- In DPD setups.

Remote command:
SETup:HIGHq:ENABle on page 649
SETup:HIGHq:MODE on page 649

RF Ports Alignment
Provides all settings and information for working with calibrated test setups.

See also Chapter 3.12, "Working with calibrated setups", on page 101.

No	Setup File	Freq. Range	View Cal. Data	Level Range	LO Coupling	Compatibility	Select Setup
1	/var/user/freq_switch_time_benchmark_lo_extern.rfsa	2 GHz to 18.8 GHz	👁	0 dBm to 0 dBm	On	🟢👁	Select Setup
2	/var/user/setupB.rfsa	5.99 GHz to 8.01 GHz	👁	-10 dBm to 0 dBm	On	🟡👁	Select Setup
3	/var/user/setupC.rfsa	975 MHz to 6.025 GHz	👁	-10 dBm to 0 dBm	On	🟢👁	Select Setup
4	/var/user/setupA.rfsa	2.975 GHz to 6.025 GHz	👁	-10 dBm to 0 dBm	On	🟢👁	Select Setup

Active Setup ← RF Ports Alignment
Displays the selected calibrated test setup.

Host Instrument ← RF Ports Alignment
IP, hostname or VISA of the primary instrument.

Connect/Refresh ← RF Ports Alignment
Loads or updates the calibration information in the "RF Ports Alignment" table.

No ← RF Ports Alignment
The first column indicates the currently selected setup file.

Setup File ← RF Ports Alignment
Name of the setup file of the calibrated setup, stored as *.rfsa file on the primary signal generator.

Frequency Range ← RF Ports Alignment
Frequency range in which the calibrated setup file is valid.

View Cal. Data ← RF Ports Alignment
Opens a dialog with detailed information about the calibrated frequency range of the setup.

See "To load and work with calibrated test setups" on page 101.

Level Range ← RF Ports Alignment

Level range in which the calibrated setup file is valid.

LO Coupling ← RF Ports Alignment

Displays if LO coupling is active in the setup.

Observe the Wiring Diagram for the required cabling.

Compatibility ← RF Ports Alignment

Displays the compatibility between the frequency and power ranges of the setup file and the frequency and power ranges of the scenario.

Click onto the field to open the "Signal Statistics" dialog.

Gray indicator

Signal not calculated.

Yellow indicator

Incompatibility between scenario and calibrated RF ports alignment setup.

Green indicator

Compatibility between scenario and calibrated RF ports alignment setup.

Remote command:
SETup:LIST? on page 646
SETup:COUNt? on page 592
SETup:SELect on page 593

Select Setup ← RF Ports Alignment

Selects the respective setup.

Available Signal Generators

Displays information on all instruments configured in the **current "Setup"**.

No ← Available Signal Generators

The first column indicates the currently selected instrument.

Remote command:
INSTrument:SELect on page 593

Type ← Available Signal Generators

The icon indicates whether the list entry is for a physical instrument (icon = signal generator) or a virtual instrument (icon = yellow folder).

Generators that are offline are still available as virtual instruments but the icon does not change here.

Instrument ← Available Signal Generators
Automatically retrieved instrument information, including instrument name, used remote control interface, serial number, firmware version and list of installed options.

Instruments that are currently not connected (not online) are displayed in gray.

Remote command:
INSTrument:LIST? on page 646
INSTrument:SELect on page 593
INSTrument:ONLine on page 647
INSTrument:NAME on page 647
INSTrument:PSEC on page 648
INSTrument:COUNt? on page 592
INSTrument:FIRMware? on page 647
*OPT? on page 579

Remote Connection ← Available Signal Generators
Displays and sets the connection status per instrument.
- **Green indicator**: instrument is connected.
- **Red indicator**: instrument is not connected.

Select the icon to change the status. Connecting/disconnecting a generator changes its availability for signal generation.

Physical instruments are available in "Scenario > Signal Generation > Config > Target > Generator".

Physical and virtual instruments are available in "Scenario > Signal Generation > Config > Target > File".

Remote command:
INSTrument:SELect on page 593
INSTrument:CONNect on page 646
INSTrument:ONLine on page 647

Generator Config ← Available Signal Generators
Opens the "Generator Config" dialog.

See Chapter 3.7, "Generator config dialog", on page 87.

Firmware Status ← Available Signal Generators
Visual indication of the result of the compatibility checks regarding the installed firmware version and option.

Compared are:
- Current installed firmware version vs last available firmware version
- Availability of at least one of the required pulse sequencer options

Consult the tooltip for more information.

Remote command:
INSTrument:SUPPorted? on page 648

Sync Role ← Available Signal Generators
Sets the primary/secondary order in synchronized setups.

Note: The correct primary/secondary order is critical, because several settings are set automatically. The R&S Pulse Sequencer sends SCPI commands to the primary and secondary instruments that adjust the reference frequency settings, connector direction, the mapped signals and trigger delays.

See also "How to configure the primary/secondary order" on page 62.

"Primary"   Primary instrument is the signal generator that is the source of reference frequency, the trigger and the baseband clock signals. Select the generator according to your particular setup.
There is exactly one primary instrument in a setup. The most recent selection overwrites all previous.

"SEC x"   Sets individual secondary numbers ("SEC x") according to the order that the secondary instruments are connected.
The indication "SEC x" influences the cabling and the trigger delay calculation.
Gaps in the numbering are not allowed.

"Unsynch"   Indicates an autonomous or an unsynchronized instrument.
The signal of this instrument is generated individually.
Signal delay is not considered and not compensated.

Remote command:
INSTrument:PSEC on page 648

Pulse Modulator ← Available Signal Generators
Sets the marker signal source, if "Options > Use Pulse Modulator > On".

If "Use Pulse Modulator > On" and "Pulse Modulator" ≠ "Off", the RF output of the signal generator is switched off during the pulse off time. This setting improves the pulse on/off ratio and extends the dynamic range of the signal to around 90 dB.

"Off"   Disables the function for the selected instrument.

"Internal"   Option: R&S SMW/SMBV/SGT-B9
The required marker signal is routed internally and embedded in the generated waveform.
Additional cabling is not required. Marker 2 (pulse and sequence) is also not required.

"External (Marker 2)"

Note:
This mode requires further configurations and dedicated cabling:
- Observe the Wiring Diagram for the required connection.
  Connect the cables are required.
- This mode uses the dedicated pulse and sequence marker M2 to mark the pulse width.
  - For the used pulses, set the M2 to mark the pulse width.
  - In the sequence, set the sequence marker M2 to use all pulse markers.
- In the "Waveform Generation" dialog:
  - Enable the dedicated marker "Use pulse modulator for improving on/off ratio"
  - Select a high clock rate, for example "Clock Rate = 200 MHz"[*)].

[*)] The processing of the waveforms and the marker signals depends on the baseband generator in the processing instrument. The following applies, if R&S SMW is used:
- Option: R&S SMW-B10:
  Marker signals are output at the ARB clock rate and are not resampled. High clock rate results in a smaller jitter between the I/Q samples and the marker signals.
  A good choice for the parameter is "Waveform Generation > Clock Rate = 200 MHz".
- Option: R&S SMW-B9:
  Marker signals are resampled, irrespectively of the used clock rate.
  The jitter between the I/Q samples and the marker signals is always 3.3 ns.

Remote command:
INSTrument:SELect on page 593
INSTrument:PMOD on page 650

Adj. Data ← Available Signal Generators
Available if use of adjustment data is enabled in "Configure" > "Settings" > "General" > "Adjustments"

Defines the instrument that holds the adjustment data for a particular hardware setup, if the storage location for the adjustment files is set to "Hardware Setup".

For details, see "How to define where adjustment data is saved" on page 74.

Remote command:
INSTrument:SELect on page 593
INSTrument:ADB:STATe on page 649

Comment ← Available Signal Generators
Additional information.

Double-click the "Comment" field to change the displayed information.

Generator Summary

A side-to-side overview of the selected generator.

Instrument SMW200A, 103417 (Type: SMW200A, Serial: 103417)

RF Ports	Type	Frequency Range	Alias Name		Path Delay
RF A	Internal	100 kHz to 20 GHz	RF A		0 s
RF B	Internal	100 kHz to 20 GHz	RF B		0 s
Basebands	**Memory**	**Bandwidth**	**Options**		
BB A	2048 MS	2 GHz	K300 K301 K302 K304 K306 K307 K308 K309 K350 K502		
BB B	2048 MS	2 GHz	K300 K301 K302 K304 K306 K307 K308 K309 K350 K502		
SW Options		**Parameters**	**Options**		
Pulse on pulse simulation		6 Emitters	K315		
RF Ports alignment		---	K545		

Remote command:

INSTrument:CAPabilities on page 646

RF Port ← Generator Summary

Hardware interface name.

Type ← Generator Summary

Indicates whether an external instrument is required.

Frequency Range ← Generator Summary

Supported max. frequency per interface.

Alias Name ← Generator Summary

Per default, interfaces are referred with the names indicated as "RF Port" but you can enter a meaningful alias name instead.

Path Delay ← Generator Summary

Adds a delay, for example to compensate for path loss.

For details, see Chapter 3.5, "Compensating path loss", on page 67.

Basebands ← Generator Summary

Baseband block name.

Memory ← Generator Summary

Installed or required memory.

Bandwidth ← Generator Summary

Installed or required baseband bandwidth.

Options, SW Options ← Generator Summary

Installed or required options.

Wiring Diagram

Visualizes the required cabling connections between the signal generators for the **current "Setup"** and the selected **primary/secondary order ("Primary/Secondary")**

The diagram shows all signal generators, incl. external RF instruments like connected R&S SGT, R&S SGS and R&S SGU generators.

Note: Set the primary/secondary order according to your setup. Observe the wiring diagram.

Ensure that all required cable connections are done.

See also Figure 3-2.
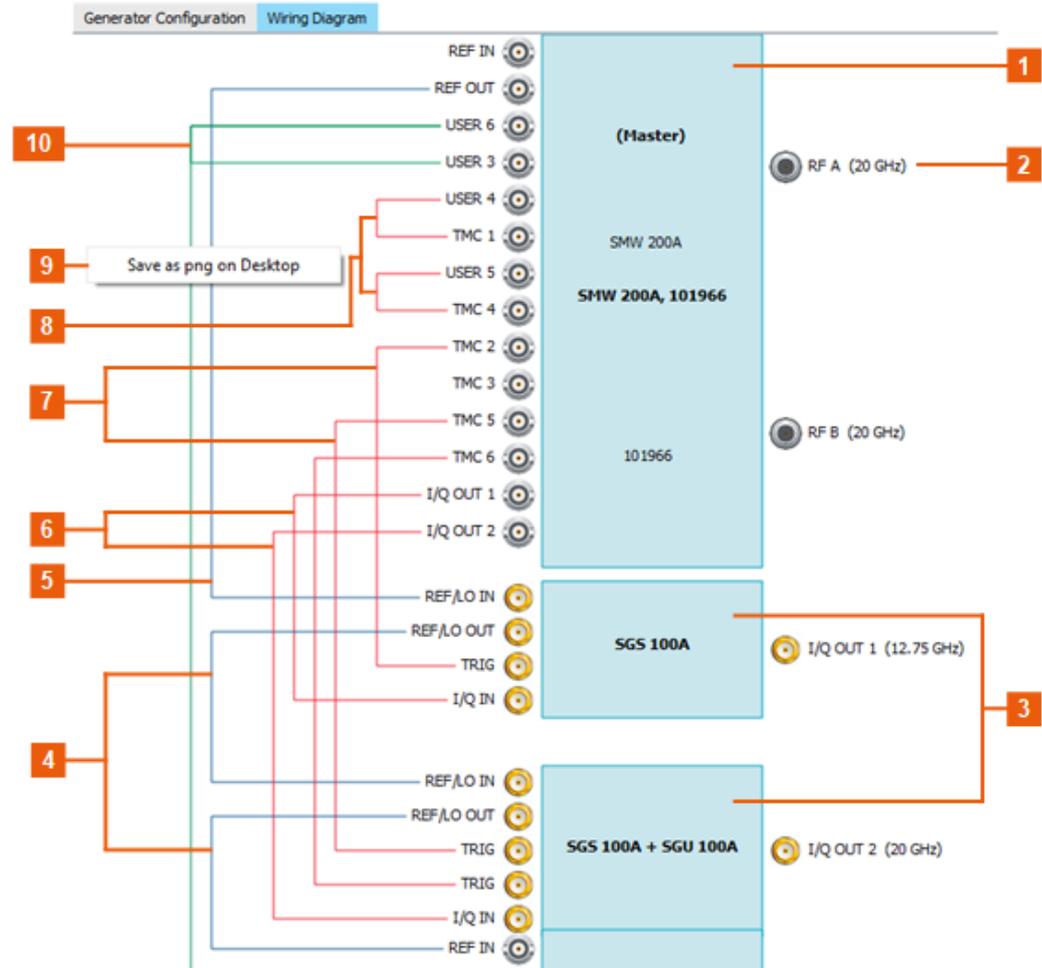
Figure 3-8: Wiring diagram (detail): understanding the displayed information

1   = Primary signal generator ("Primary/Secondary = Primary")
2   = Connector indication; select it to show its position on the instrument
3   = External RF instruments, connected to and controlled from the R&S SMW
4   = Reference frequency, distributed as a daisy chain
5   = Reference frequency signal of the primary instrument, fed in to the external RF instrument
6   = I/Q data connection to the external RF instruments (R&S SGS + R&S SGU)
7   = Pulse width marker signal, fed in to the secondary instrument
8   = Pulse width marker signal fed as input signal for the pulse modulator
9   = Saves the wiring diagram in a printable file and stores it on the desktop
10 = Trigger signal of the primary instrument, fed in to the first secondary instrument

The diagram is interactive:
● Mouse over to display additional information and hints, like max. RF, and assigned signal.
● Select a connector to show its position on the instrument.
● Right-click to access the context menu.

Find Connector Dialog

Several dialogs contain icons representing the physical connectors on an instrument.

In the menu bar, select "Help" > "Find Connector".

If you select a connector icon in the wiring diagram, the "Find Connector" dialog opens and shows you exactly where the connector is on the instrument.

Figure 3-9 shows a typical example of the "Find connector" dialog. The "External RF" settings contain several connector icons. In this example, "BBMM2" is highlighted.
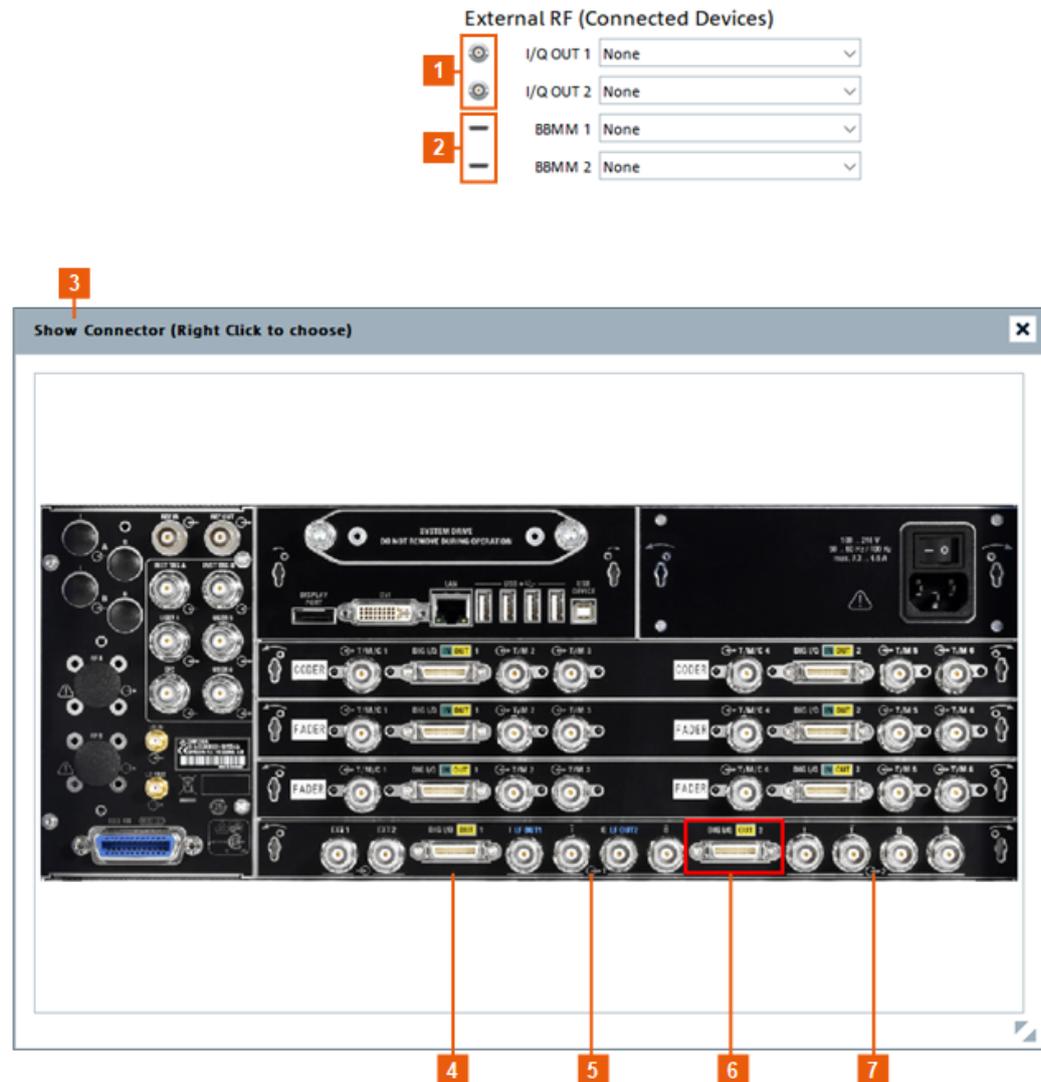
Figure 3-9: Example of a back panel in the Find connector dialog

1 = Icons to show I/Q out connectors in the "Find connector" dialog
2 = Icons to show BBMM connectors in the "Find connector" dialog
3 = "Find Connector" dialog showing external connectors
4 = "BBMM1" connector
5 = "I/Q out 1" connectors
6 = "BBMM2" connector (highlighted)
7 = "I/Q out 2" connectors

This dialog also has a right-click context menu.

Use this menu to access images of other devices and find specific connectors.

Each device listed in the context menu has a submenu of the connectors available on that device.
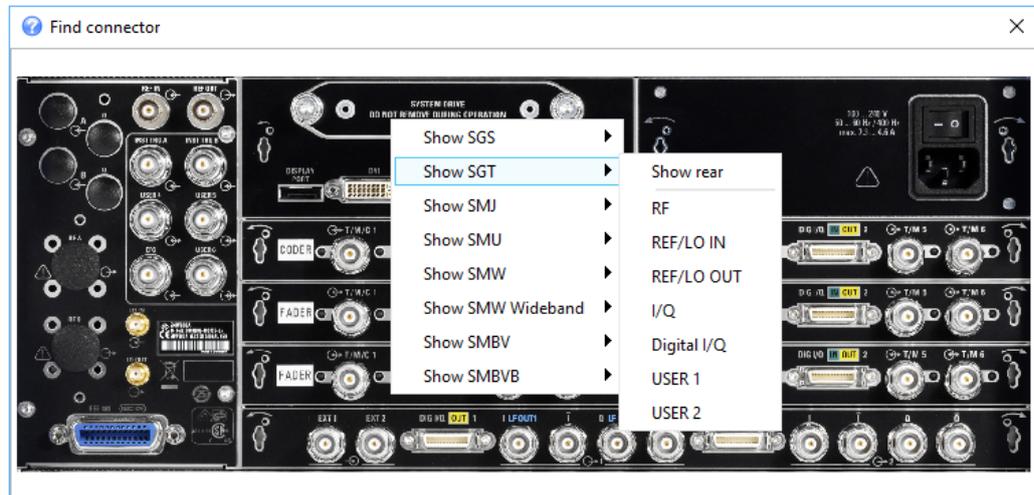
3.7 Generator config dialog

Access:

1. In the menu bar, select "Configure" > "Signal Generators"

2. In the "Generator Config" column, select the tool icon of an instrument.

You can edit the parameters in the "Generator Config" for virtual instruments. The parameters for physical generators that are online are displayed grayed out. Disconnect the generator to make changes. See "Remote Connection" on page 81.
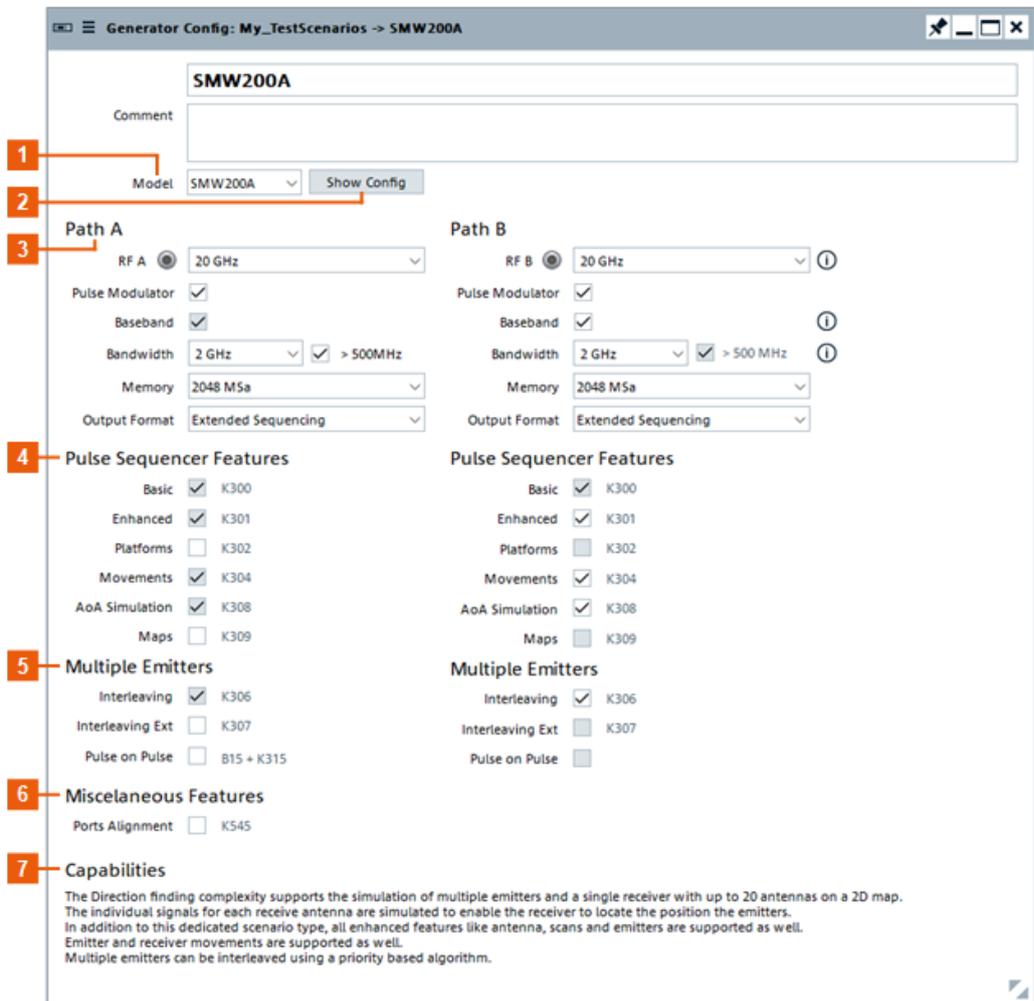
Figure 3-10: Generators and required options: understanding the displayed information

1 = Generator type
2 = Displays a list of the required options for the selections in the dialog
3 = Settings for the two generator paths (A and B)
4 = Options required to play signals generated with the R&S Pulse Sequencer, per signal path
5 = Options for pulse-on-pulse simulation and interleaving of multiple signals into a single output file, per signal path
6 = Displays optionally installed features
7 = Notes describing the capabilities of selected options.

For step-by-step instructions, see Chapter 3.3, "Configuring the test setup in the R&S Pulse Sequencer", on page 60.

Settings:

Name

Enter the name of the generator.

Remote command:

GENerator:NAME on page 589

GENerator:CREate on page 588

GENerator:CATalog? on page 587

GENerator:SELect on page 588

GENerator:REMove on page 590

Comment

Enter a short description.

Remote command:

GENerator:COMMent on page 589

Model

Selects the type of the signal generator from the list of supported instruments.

The hardware characteristics of the selected generator influence the way the created waveform file is processed. The available ARB bandwidth, for example, limits the waveform length.

Remote command:

GENerator:TYPE on page 646

Show Config

Displays an overview of the required options.

RF Ports	Type	Frequency Range	
RF A	Internal	100 kHz to 20 GHz	
RF B	Internal	100 kHz to 44 GHz	
Basebands	**Memory**	**Bandwidth**	**Options**
BB A	2048 MS	2 GHz	K300 K301 K302 K304 K306 K307 K308 K309 K502
BB B	2048 MS	2 GHz	K300 K301 K302 K304 K306 K307 K308 K309 K502
SW Options		**Parameters**	**Options**
Pulse on pulse simulation		6 Emitters	K315
RF Ports alignment		---	K545

Required options for generator config SMW200A, 102372

Remote command:

GENerator:CAPabilities on page 646

Path

Groups the features for each RF path.

"RF A" or "RF B" output connector

Select the connector icon to access the "Find Connector" dialog. The physical location of the current connector on the front/rear panel of the instrument is shown.

"RF A" or "RF B" maximum frequency

Sets the maximum frequency for the path.

"Pulse Modulator"

When selected, the RF output of the signal generator is switched off during the pulse off time. Switching off the RF output improves the pulse on/off ratio and extends the dynamic range of the signal.
Not available for "RF B" unless enabled for "RF A".

"Baseband"

Toggles the baseband for "RF B".
Note: When you select baseband for "RF B", the "Pulse Sequencer" settings appear to enable you to select the required options for "RF B".

"Bandwidth"   Sets the baseband bandwidth.

"> 500 MHz"

If selected, indicates that this generator is a wideband instrument. Not available for "RF B" unless enabled for "RF A".

"Memory"

Sets the ARB memory.

"Output Format"

Switches between waveform file format and the format for the extended sequencer.

"External RF (Connected Devices)"

To find a particular connector, select the connector option (see "Find Connector Dialog" on page 86).

Select the type of connected external device.

These settings are not available for wideband baseband generators ("> 500 MHz" > "On").
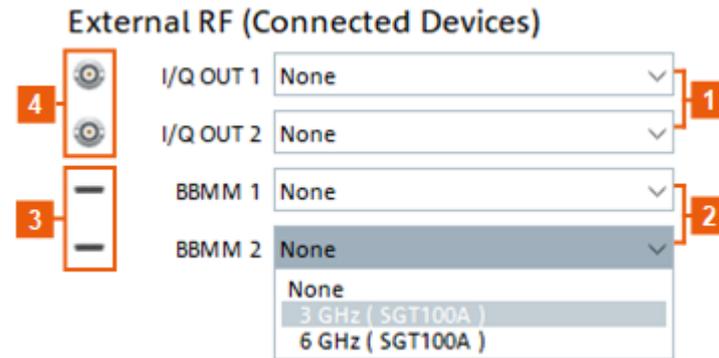
1 = Drop-down lists for selecting the devices connected to "I/Q out 1" and "I/Q out 2"
2 = Drop-down lists for selecting the devices connected to "BBMM1" and "BBMM2"
3 = Icons show BBMM connectors the "Find connector" dialog
4 = Icons show I/Q out connectors in the "Find connector" dialog

Remote command:
GENerator:CAPabilities on page 646
INSTrument:CAPabilities on page 646

Pulse Sequencer Features

Selects the options for both baseband paths.

The "Basic" option for baseband A is selected by default.

Use the tooltip text to obtain a short description of each option (as shown for "Maps" in Figure 3-11).
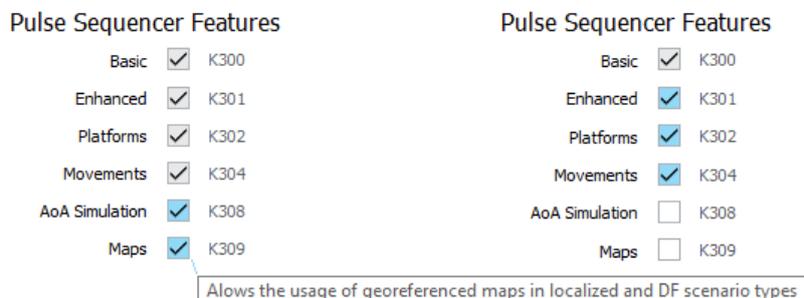
Figure 3-11: Example with selected options and tooltip text

Note: You cannot select an option in baseband B unless it is already selected for baseband A.

You cannot deselect an option in baseband A unless it is already deselected for base-band B.

Remote command:
GENerator:CAPabilities on page 646

Multiple Emitters
Settings for multiple emitters and the use of interleaving or pulse-on-pulse simulation.

Use the tooltip text to obtain a short description of each option.

Note: You cannot select an option in baseband B unless it is already selected for base-band A.
You cannot deselect an option in baseband A unless it is already deselected for base-band B.

Available options depend on the selected signal generator model.

"Emitter Interleaving"
> Enables the calculation of multi-emitter scenarios that contain several emitters. Activates a priority-based interleaving algorithm so that mul-tiple signals can be merged into a single output file by dropping pul-ses which overlap in time.

"Emitter Interleaving Ext"
> Increases the number of interleaved emitters to 512 per baseband.

"Pulse on Pulse"
> Allows additional permanent emitters for pulse-on-pule simulation.

Remote command:
INSTrument:CAPabilities on page 646

Miscellaneous Features
Displays available features of also installed options, e.g. ports alignment.

Capabilities
Lists the features of the selected options.

As options are selected via the checkboxes, the associated notes appear in the "Capa-bilities" list.

3.8 Adjustment configuration settings

Access:

► Enable level adjustments.
See "How to enable the level adjustment function" on page 69.

For step-by-step instructions, see Chapter 3.5.1, "Collecting level adjustment data", on page 68.
See also Chapter 3.5, "Compensating path loss", on page 67.

Settings:

3.8.1 General settings

Access:

1. Enable level adjustments.
   See "How to enable the level adjustment function" on page 69.

2. In the menu bar, select "Adjustment > Configure".
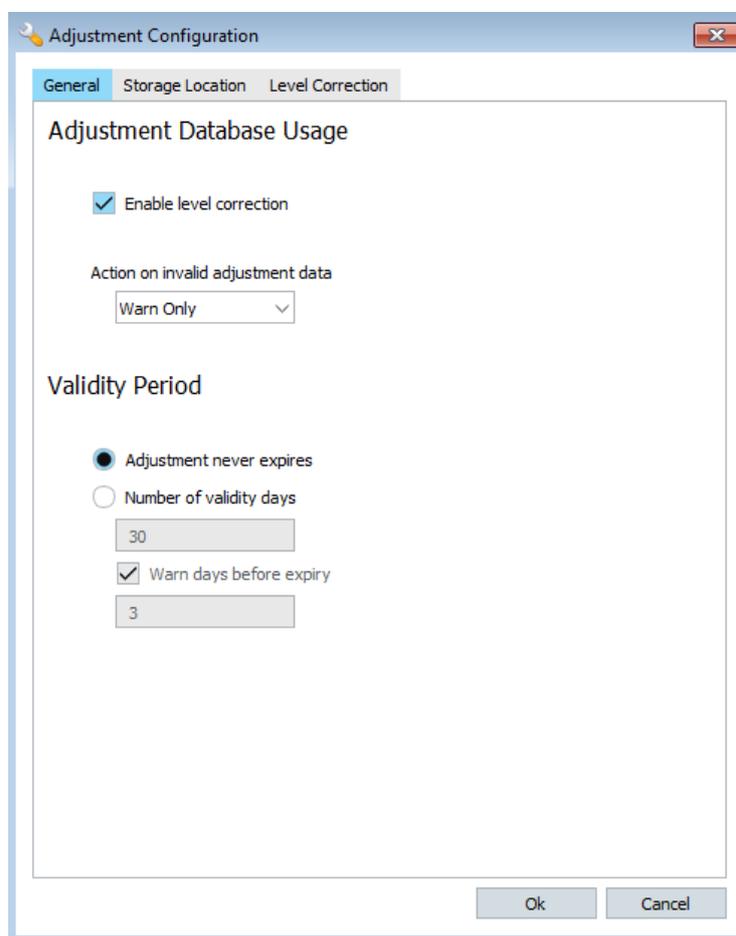
The "General" tab comprises global configuration settings.

Settings:

Adjustment Database Usage

Defines whether collected data is used to compensate path loss and how the software handles invalid adjustment data.

Adjustment data is invalid if its creation date exceeds the expiration period set with the parameter Validity Period.

Validity Period

Sets how long collected data is regarded as valid ("Adjustment never expires" or "Number of validity days").

You can enable a notification service that warms you a defined number of days before the validity period expires.

3.8.2 Adjustment database storage location settings

Access:

1. Enable level adjustments.
   See "How to enable the level adjustment function" on page 69.

2. In the menu bar, select "Adjustment > Configure > Storage Location".

   With the provided settings, you define where the collected adjustment data is held.
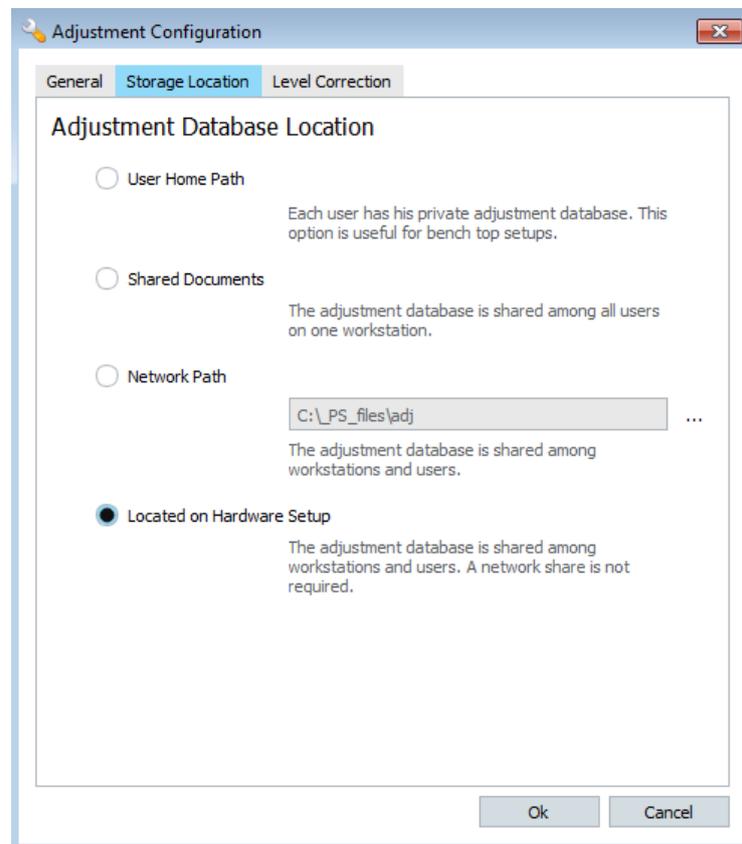
Settings:

User Home Path

Saves the database in a **private directory (`%HomePath%`)**. Adjustment data is available for a particular user only.

See also Table 23-2.

Shared Documents

Saves the database in a **local shared directory on a PC**. Adjustment data is available to users with access rights to this particular PC.

See also Table 23-2.

Network Path

Saves the database in a **network share**. Adjustment data is shared among users with access rights to the network.

To specify the location, enter the file "Path" or select the "More" icon and navigate to the network location.

See also Table 23-2.

Located on Hardware Setup

Saves the database on a **particular instrument** within the hardware setup the adjustment data applies for. To define the instrument for a setup, set "Signal Generators" > Adj. Data > "On".

See also "How to define where adjustment data is saved" on page 74.

3.8.3 Level correction settings

Access:

1. Enable level adjustments.
   See "How to enable the level adjustment function" on page 69.

2. In the menu bar, select "Adjustment > Configure > Level Correction".

With the provided settings, you define how the adjustment data is collected.
For step-by-step instructions, see Chapter 3.5.1, "Collecting level adjustment data", on page 68.
See also Chapter 3.5, "Compensating path loss", on page 67.

Settings:

Power Sensor
Defines which power sensor is used and how the R&S Pulse Sequencer detects it.

"Connected to signal generator"
Use this option if the power sensor is connected via **USB** to the signal generator.
Listed are all detected signal generators.
To disable this option, select "Not Assigned".

"R&S NRP sensor VISA string"

        Use this option if the power sensor is connected via **LAN**.
        As for any other instrument, R&S Pulse Sequencer requires the VISA resource string to establish the remote connection to the power sensor.

"Test"   Starts a short connection test procedure in that the software uses the VISA resource string, opens a remote connection, sends the `*IDN?` query to the power sensor and proves for the response.

Target Noise Ratio

Sets the fixed noise filter of the power sensor.

For more information, see the corresponding R&S NRP user manual.

Frequency Range

Sets the frequency range for that the level measurement is performed.

Select "Minimum" and "Maximum" frequency values that are within the frequency range of the instrument. Depending on your application, you can select a portion of the supported frequency range.

Set the "Increment" value to adjust the measurement's resolution. Smaller steps increase the resolution but also the measurement time.

3.9 Manage adjustment data settings

Access:

1. Enable level adjustments.
   See "How to enable the level adjustment function" on page 69.

2. In the menu bar, select "Adjustment > Manage".

For step-by-step instructions, see Chapter 3.5.1, "Collecting level adjustment data", on page 68.

See also Chapter 3.5, "Compensating path loss", on page 67.

Settings:

Use this dialog for
Indicates the parameter that the corrections are applied for.

Generator, Path
Selects the instrument (and for instruments equipped with more than one path also the path) for that the adjustment data is displayed.

The instrument is indicated the assigned generator name and its "Primary/Secondary" assignment (see Available Signal Generators).

Accept, Reject
Applies or discards the displayed adjustment data.

Trace color indicates the following:
- Red: unaccepted adjustment data

- Green: accepted adjustment data

If the measurement is repeated, the display shows both traces. In this case:
- "Accept" applies the new adjustment data; current data is overwritten
- "Rejects" retains current data; new adjustment data is lost.

Remote command:
See also `ADJustment:RELoad` on page 644.

Clear
Deletes current adjustment data from the storage location.

Remove
Empties (deletes all) the current storage location.

3.10 Run level adjustment settings

Access:

1. Enable level adjustments.
   See "How to enable the level adjustment function" on page 69.

2. In the menu bar, select "Adjustment > Run Level Adjustment".
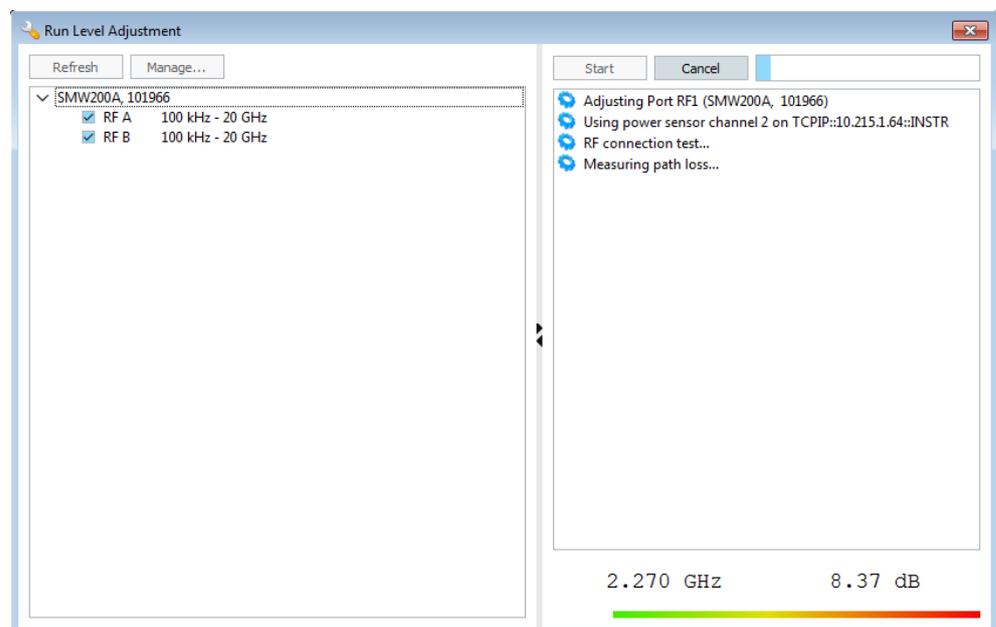
3. From the list of instruments, select the instruments and path and set "Start".

   The dialog indicates current measurement status like used frequency, process progress, or progress messages.
   For step-by-step instructions, see Chapter 3.5.1, "Collecting level adjustment data", on page 68.
   See also Chapter 3.5, "Compensating path loss", on page 67.

Settings:

Refresh
Updates the dialog.

Manage
Opens the Manage Adjustment Data dialog.

Starts
Triggers the measurement.

Cancel
Aborts current measurement.

3.11 Finding required connectors on instruments

In particular in a setup with several instruments, it is useful if you can get a better overview on the location of connectors on a particular instrument.

To display connectors and their location on instruments

1. In the menu bar, select "Help > Find Connector > Show Instrument".
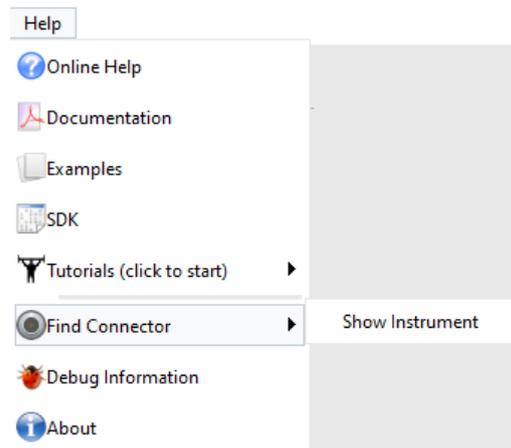
2. Right click the instrument image and select the instrument type that you want to display the connector location for.

3. Select a particular connector.

   A blinking square indicates the location of the connector on the front or rear panel of the instrument.

Several further dialogs also provide the find connector function, see "Find Connector Dialog" on page 86.

3.12 Working with calibrated setups

Requires the option R&S SMW-K545 on the signal generators used in the test setup.

Working with calibrated setups requires RF ports alignment in the test setup. RF ports alignment refers to test setups where the DUT gets several RF signals which have to be coherent. The RF signals come from several R&S SMW signal generators (multi-instrument setup). Due to external cabling and slight deviations between multiple signal generators, the resulting RF signals at the output ports of the instruments can differ in phase, time delay and power in dependency to the signal frequency and power. RF ports alignment corrects such signal deviations. Consequently, the DUT receives coherent RF signals.

For detailed information on RF ports alignment, see the R&S SMW-K545 user manual.

Access:

1. In the menu bar, select "Configure > Signal Generators".

2. Select the tab "RF Ports Alignment".

   Here you can select calibrated setups stored as *.rfsa files on the primary signal generator in a test setup.

To load and work with calibrated test setups

1. In the field "Host Instrument", enter the IP address of the primary signal generator in your test setup.

   The table under "RF Ports Alignment" lists all available setups.

   The table under "Available Signal Generators" lists all signal generators in the setup.

Note:
- If there is only one signal generator in the setup, the outputs "RF A" and "RF B" are calibrated against each other.
- If there are more signal generators in the setup, the secondary instruments are calibrated against the primary instrument. Open the "Wiring Diagram" for information on the connection of the instruments. See also "Wiring Diagram" on page 84.

2. Assign the signal to the signal generator.
   See Chapter 17.2, "Assign signal source or receiver signals to generator", on page 481.

3. Select "Signal Calculation > Calculate" to calculate the signal.

   During signal calculation, the pulse sequencer software compares the frequency and level limits of the scenario with the calibration limits of the selected RF ports alignment setup. The compatibility is indicated in the "RF Ports Alignment" block.
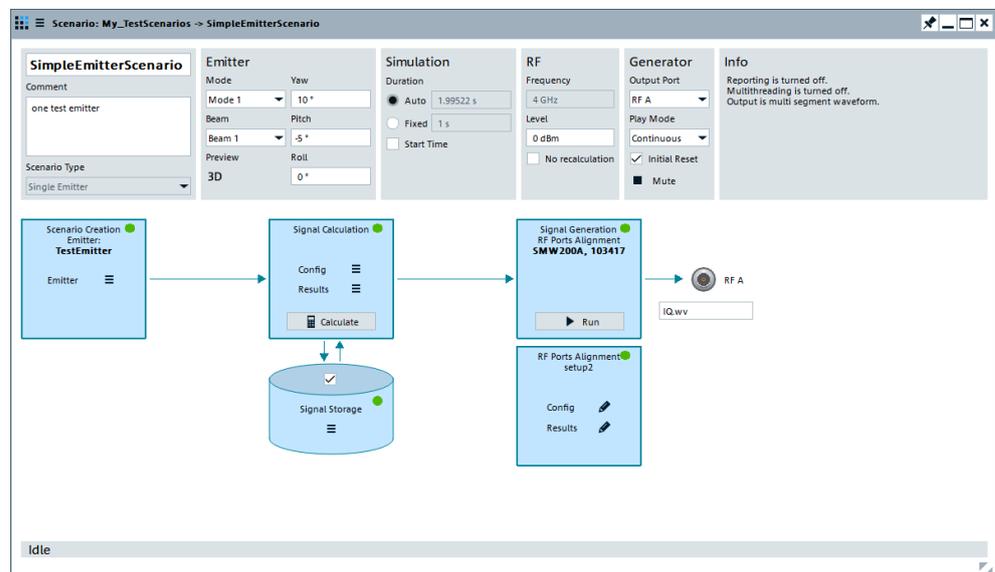
4. Check the results of the signal calculation.

   - **Green indicator**: Compatibility between scenario and RF ports alignment setup.
   - **Yellow indicator**: Incompatibility between scenario and RF ports alignment setup.

5. Select "RF Ports Alignment > Results".

   The "Signal Statistics" dialog displays the comparison and compatibility between the scenario and the selected RF ports alignment setup.
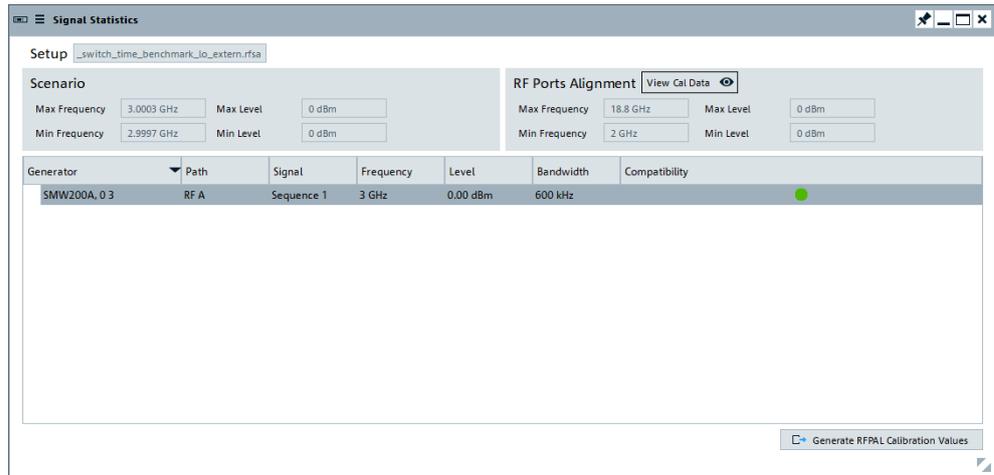
If the scenario contains signals that fall outside the frequency and level range of the calibration, you can export a list of frequency and level values from the Pulse Sequencer Software. You can import these files into the RFPAL Software to calibrate the setup in the required frequency and level ranges.

6. Click "View Cal. Data" to open the dialog to check the frequency calibration.
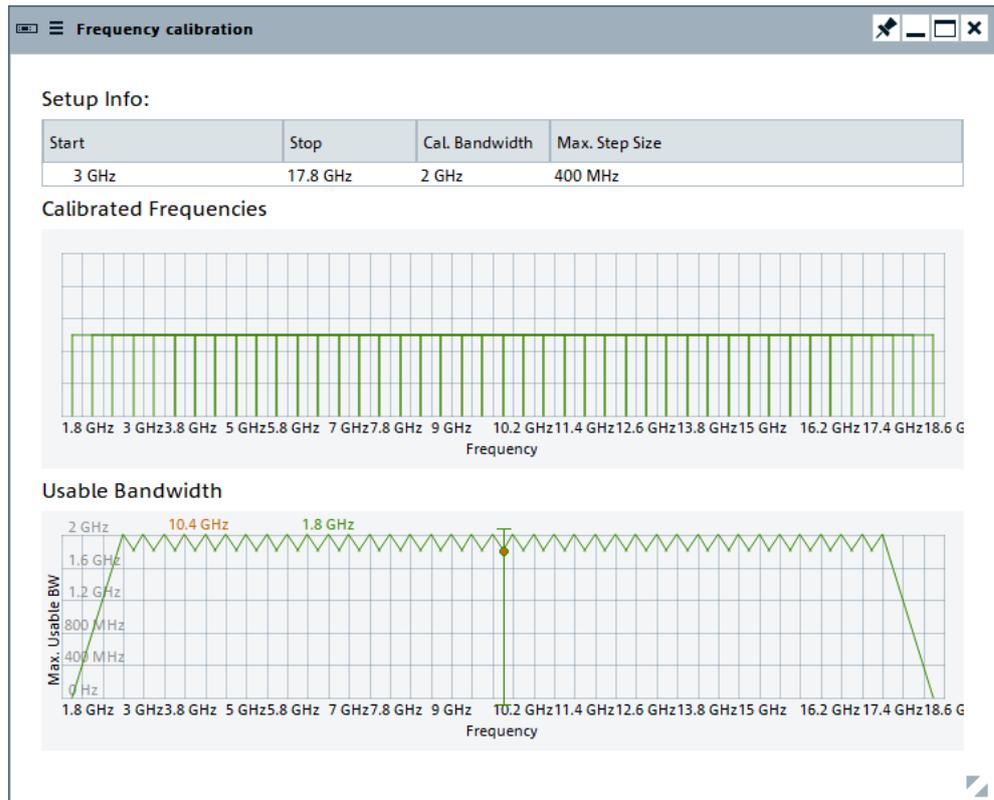
a) Check the section "Setup Info" for information about the selected calibrated setup.

b) Check the section "Calibrated Frequencies" for the calibrated frequencies in the selected setup.
Right-click in the diagram to select "Lines" for "Frequency" in the diagram via the context menu. Position the lines at the desired values for evaluation.

c) Check the section "Usable Bandwidth" for the usable bandwith dependent on the frequency in the selected setup.
Place the marker at the desired frequency to measure the usable bandwith at a certain frequency.
Right-click in the diagram to select "Lines" for "Bandwidth" and "Frequency" in the diagram via the context menu. Position the lines at the desired values. For example, define with the "Frequency" lines a certain frequency range and use the "Bandwidth" lines to measure the delta of the usable bandwith in the selected frequency band.

7. To export the calibration values of as `*.csv` file:

a) Click "Generate RFPAL Calibration Values".
A dialog opens.

b) Select the "Target Path".

c) Enter the name for the "Frequencies File".

d) Enter the name for the "Levels File".

e) Click "Generate".
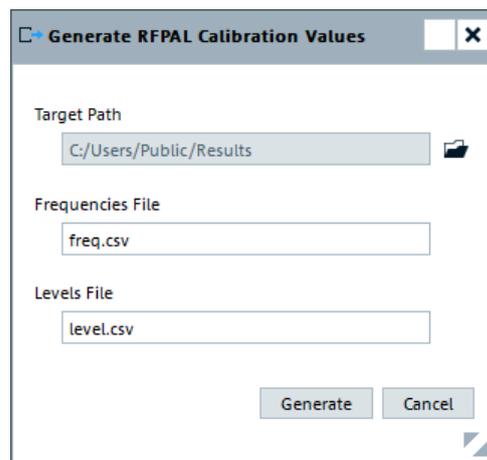
The generated files are stored in the selected folder.

Example:

Incompatibility between scenario and RF ports alignment setup

Differences in the frequency or level ranges between scenario and RF ports alignment setup cause incompatibilities. The yellow indicator in the "RF Ports Alignment" block indicates incompatibility.

To check the incompatibility:

1. Select "RF Ports Alignment > Results".

    The "Signal Statistics" dialog opens.
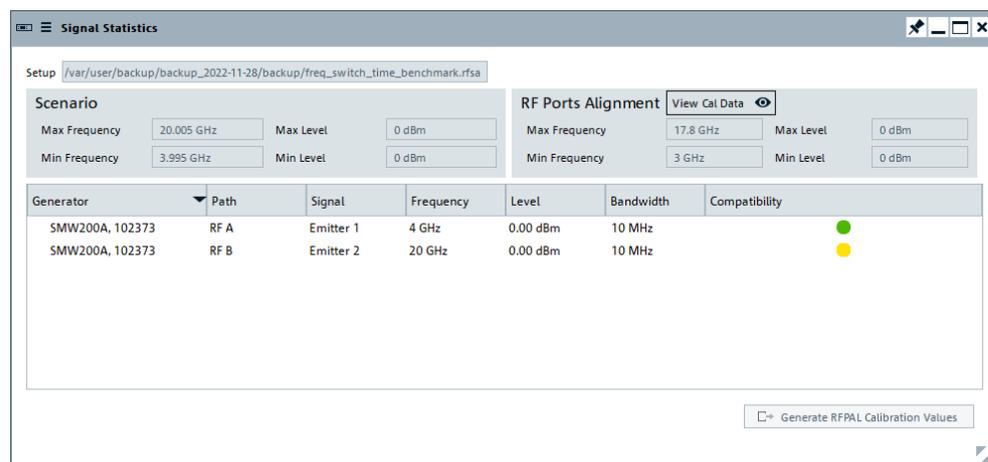
2. To correct the incompatibility in this example, do one of the following:

    a) Change the frequency setting for Emitter 2 to a value within the calibration range given in the RF ports alignment setup, e.g. 16 GHz.
    b) In the menu "Configure > Signal Generators" > "RF Ports Alignment", select a valid RF ports alignment setup from the listed setups ("Compatibility": green indicator).

4 Organizing the project data in repositories

This section explains the basic repository concept and informs you briefly how the software database is built.

An introduction to the concept of elements handling is provided in "Creating elements" on page 32.

4.1 Repository settings

Access:

►   In the project tree, double-click the repository name.

The repository tree shows the content of the currently loaded repositories.

Settings:

General Repository Settings
Provides general information related to repository management.

The "General" tab.

Info ← General Repository Settings

Provides information like the name of the author and the date of creation.

"Name"   When a new repository is created, it is named automatically but you can rename it afterwards.
The automatically assigned names follow the syntax:
`New_<yyyy-mm-dd><T><hhmmss>`, where `<yyyy-mm-dd>` and `T<hhmmss>` indicate the current date and time respectively.

"Author"
Indicates the author.

"Created"
Automatically assigned date and time indication.

"Version"   Repository version.

Remote command:
REPository:CATalog? on page 587
REPository:CREate on page 588
REPository:SELect on page 588
REPository:REMove on page 590
REPository:AUTHor on page 722

REPository:DATE on page 723
REPository:VERSion on page 724

Classification ← General Repository Settings
Classification information is assigned to each repository to set the security level.

"Level" "Level 0" is the lowest level that means no restrictions and "Level 4" is the highest one. Elements belonging to a repository with higher level cannot be copied to a repository with lower one.
If more than one repository is loaded, the workspace displays the highest required classification level, see Chapter 2.4, "Understanding the displayed information", on page 23.
See also "To change the general program settings" on page 50.

"Disable copy operations to lower level repositories"
If enabled, you cannot copy repository elements for repositories with higher classification level to repositories with lower one.

Remote command:
REPository:SECurity on page 724

Functional Range ← General Repository Settings
The functional range defines whether basic or advanced features are used in the repository. The complexity level also reflects the option concept of the signal generators.

For example, if "Functional Range > Advanced settings ... (K300/K301)" is used, the signal generator has to be equipped with the option Advanced Features for R&S Pulse Sequencer, e.g. R&S SMW-K301.

See also:
● Chapter 1, "Welcome to the R&S Pulse Sequencer", on page 13
● Chapter 2.1, "Required options", on page 18

Remote command:
REPository:COMPlexity on page 723

Antenna Cross Polarization > Attenuation ← General Repository Settings
Sets the attenuation level used to calculate the cross-polarized antenna patterns, see "Polarization" on page 265.

Remote command:
REPository:XPOL:ATTenuation on page 724

Comment
Enter a short description of the repository.

Remote command:
REPository:COMMent on page 590

Users
Provides settings related to user management.

For detailed information, see Chapter 23.2, "User administration", on page 553.

Storage

This tab provides settings necessary to configure the default storage folder for the repositories.
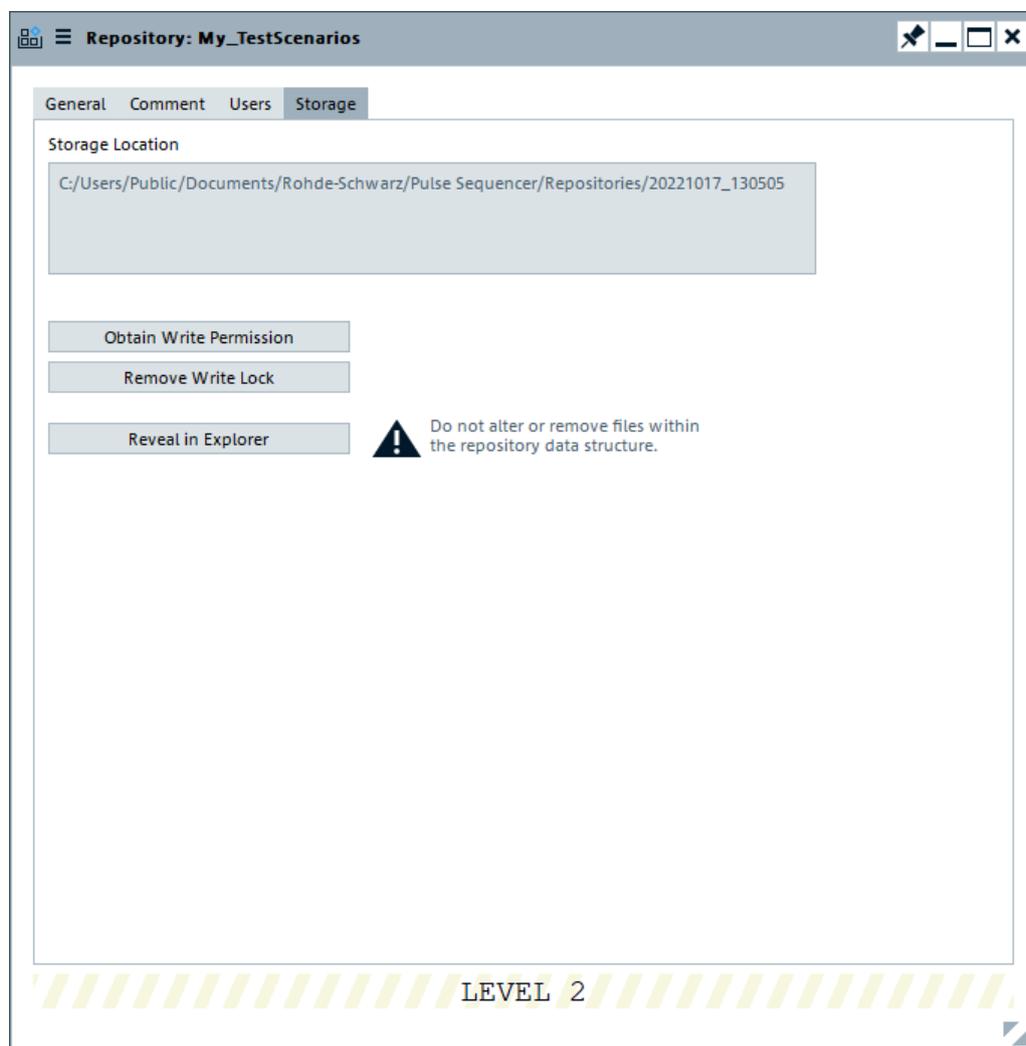
"Path"   Displays the location (network or local folder) the repository is stored in.

Remote command:
REPository:PATH? on page 723

"Obtain Write Permission"

Repositories allow the simultaneous access from several users. If a user with write permission has opened a repository, this write permission is exclusive. Another user will be granted with the write permission only after the first user had closed the repository or removed it from its workspace.

You can request an exclusive write permission only on a repository that is not opened by another user with write permission.

See also:
- "Discovered Repositories on the Mass Storage" on page 111
- "To obtain write permission on a repository" on page 119

"Remove Write Lock"

The R&S Pulse Sequencer locks any improperly or unexpectedly closed repository and indicates this situation with a red lock symbol 🔒 in the *"Repository Tree"*. A locked repository can be unlocked only from the same PC and by the same user with write permission that had opened the repository before.
See "To remove write lock that results from a previously crashed session" on page 834.

"Reveal in Explorer"

You can open the repository file structure in the Windows Explorer.

Note: Do not change the repository content by changing the file structure in the Windows Explorer.
To move or share repositories with other users, always use the "Export/Import Repository Archive" function.

See:
- "To create and export a repository archive" on page 117
- "To import a repository archive via the menu bar" on page 118

Remote command:
REPository:FILename? on page 723
REPository:SAVE on page 724

4.2 Repository manager settings

Access:

► In the menu bar, select "File > Load and Manage Repository".
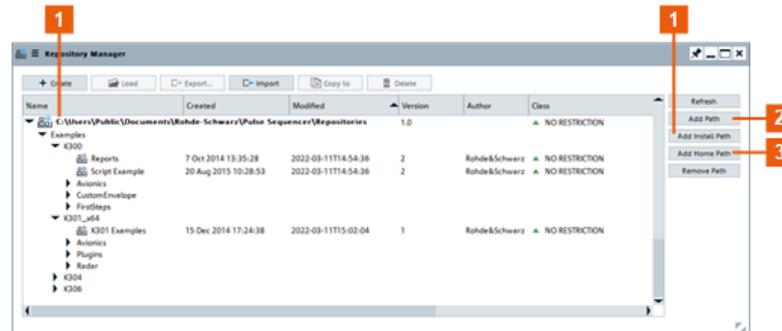
Figure 4-1: Repository Manager: understanding the displayed information

1 = User-defined path for saving and possibly sharing repositories alternatively to the DataPath and the HomePath
2 = Default repository path (DataPath), see also Table 2-2
3 = User home path (HomePath)

The "Repository Manager" comprises the setting for managing the repository files and archives on the mass storage.

Settings:

Discovered Repositories on the Mass Storage
Lists all found directories and the repository files that they contain.

You can change the column width and order.

A lock symbol on the database icon in the "Repository Manager" indicates one of the following situations:
- Repository is opened by another user with explicit write access.
- Insufficient user rights for the particular repository.
- Insufficient user rights on the file system (directory) the repository is stored in.

Try to:
- Wait until the repository has been closed and try to obtain exclusive write permission, see "To obtain write permission on a repository" on page 119.
- Change your current user or use a user with different role, see "To change the default user used to access repositories" on page 556.
- Log in as user with sufficient (write) access rights on the file system.

Remote command:

REPManager:CATalog? on page 725

Create

Adds a repository in the selected directory.

See also Chapter 23.3, "Defining storage locations", on page 557.

Load

Loads and displays the selected repository in the workspace.

See "To load a repository" on page 117.

Remote command:

REPManager:LOAD on page 725

Export

Exports the selected repository to an archive file.

See "To create and export a repository archive" on page 117.

Remote command:

REPManager:EXPort on page 726

Import

Imports a repository from a directory.

Remote command:

REPManager:IMPort on page 727

Copy to

Copies the selected repository to a new location.

The R&S Pulse Sequencer automatically copies all relevant files and database elements.

Delete

Deletes the entire repository from the permanent mass storage, if:
- The current user has write permission, see Table 23-1.
- The repository is not opened by another user with write permission.

Remote command:

REPManager:DELete on page 726

Refresh

Refreshes the list of found repository files.

Add Path

Opens the "Select Folder" dialog that is similar to the Windows Explorer.

Navigate to the required directory and load it. Found repository files are displayed in the list.

See also "To set the storage location for repositories" on page 559.

Remote command:
`REPManager:PATH:ADD` on page 727
`REPManager:CATalog?` on page 725

Add Install Path, Add Home Path

Adds and loads the directory automatically. Found repository files are displayed in the "Discovered Repositories on the Mass Storage" on page 111.

Where:

- "Add Install Path" opens the `DataPath` directory.
  Per default, this directory is the same displayed in "Repository Manager" (see Figure 4-1).
- "Add Home Path" opens the `HomePath` directory.

For information on the default file paths, see Table 2-2.

See also Chapter 23.3, "Defining storage locations", on page 557.

Remote command:
`REPManager:PATH:ADD` on page 727
`REPManager:CATalog?` on page 725

Remove Path

Removes the selected path. The repository files within this directory are also removed from the list.

Remote command:
`REPManager:PATH:DELete` on page 727

4.3 Repository view

By default, the repository view is set to the tree view. In the tree view, every item of the repository element, e.g. "Scenarios", is displayed.

See "Repository tree" on page 26.

You can switch the tree view to a list view, which displays the number of items in the repository element in brackets. If you select a repository element in the list view, the "Item Overview" dialog opens and displays all items of the selected repository element.

Access:

1. In the project tree toolbar, click ⬚.
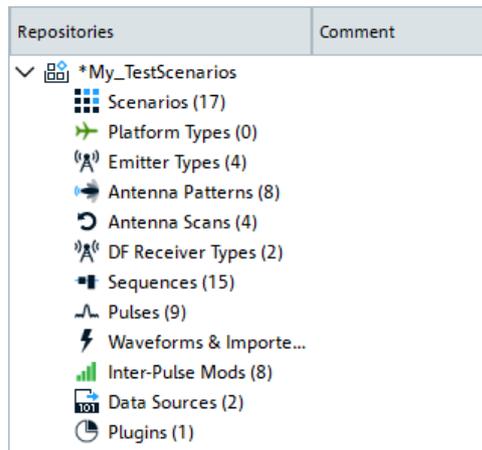
2. Click the repository element to open the item overview.

Settings:

Repository

Displays the name of the repository.

Items
Displays the name of the selected repository element related to the items.

Search Alias
Filters the list according to the string entered in the dialog.
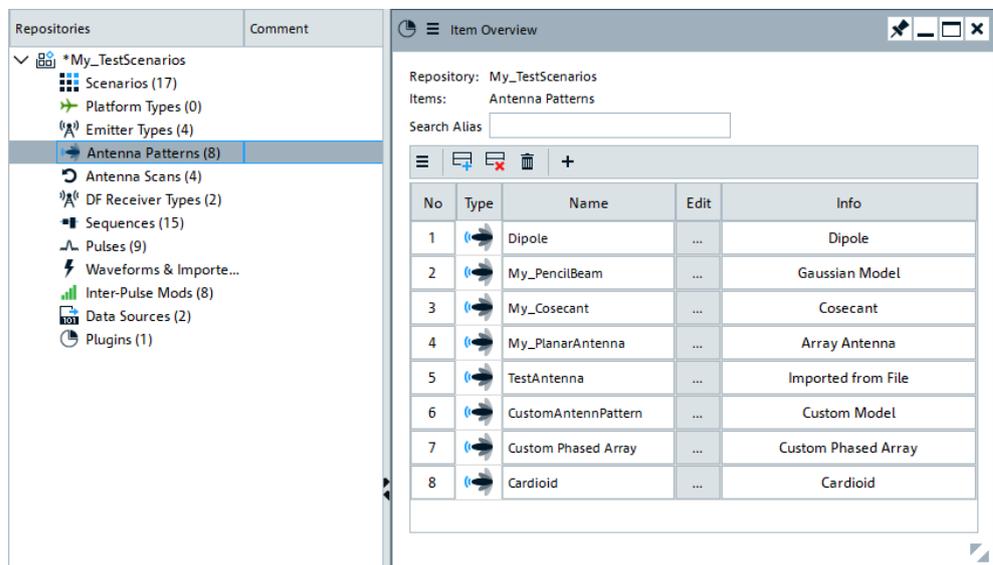
Edit, Append, Remove, Remove all, Copy and append
Standard functions for items handling, see "Standard function in the context menus" on page 33.

Item overview table
The table shows the following information:

No ← Item overview table
Displays the number of the item.

Type ← Item overview table
Displays the type of the item, e.g. "Antenna Patterns".

Name ← Item overview table
Displays the name of the item.

Edit ← Item overview table
Opens the dialog to edit the selected item.

Info ← Item overview table
Displays important parameters of the item.

4.4 How to manage the project data

See:

To create a repository

1. In the menu bar, select "File > New Repository".

   A new empty repository is created and added to the project tree.
   Repositories are named automatically. The repository names follow the syntax:
   `New_<yyyy-mm-dd><T><hhmmss>`, where `<yyyy-mm-dd>` and `T<hhmmss>` indi-
   cate the current date and time respectively.

2. In the project tree, double-click the repository name.

   The "Repository" dialog opens.

3. Change the settings. For example, change the automatically assigned "Name" and "Comment".

4. Set the "Classification > Level >" (see "Classification" on page 108).

5. Set the complexity level, for example, "Functional Range > Direction Finding" (see "Functional Range" on page 108).

6. Close the dialog.

To delete a repository

You can delete repositories that you no longer need, if this repository is not opened by another user with write permission.

1. Use a user with write access rights, e.g. "Admin" or "Creator" (see Table 23-1).

2. In the menu bar, select "File > Load and Manage Repository".

3. From the listed files, select a repository file, e.g. "My_TestScenarios".

4. Select "Delete".

To remove a repository from the workspace

► In the project tree, select the repository e.g. "My_TestScenarios". In the context menu, select "Unload from Workspace".

The repository is removed from the workspace but not deleted. You can open it again, see "To load a repository" on page 117.

To load a repository

1. In the menu bar, select "File > Load and Manage Repository".

2. If necessary, for example by loading and exchanging repositories with another user, change the directory in that the software searches for repository files.

3. From the listed files, select a repository file, e.g. "K301 Examples".

4. Click "Load".

The repository is opened and displayed in the project tree.

To create and export a repository archive

We assume that you have created a repository and configured at least one scenario.

1. Perform one of the following:

   a) In the repository tree, select the repository.
      In the context menu, select "Export".
   b) In the menu bar, select "File > Export Repository".
      From the listed files, select a not used repository file.
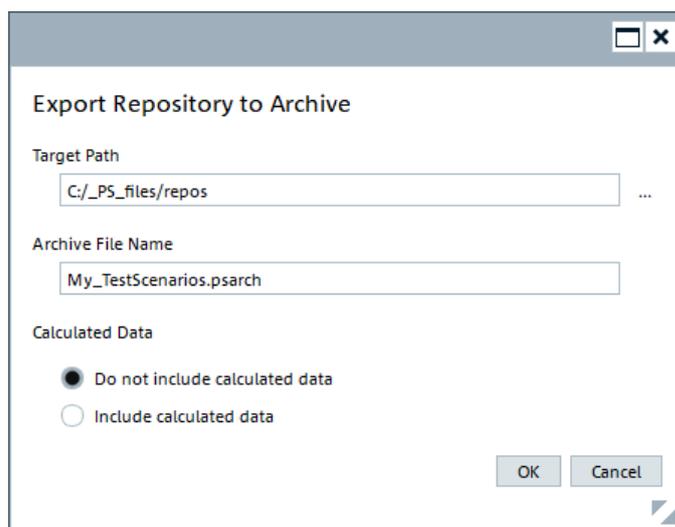
2. In the "Export" dialog, select "Target Path" to change the directory in which the repository archive file is stored.

3. Per default, the software saves repository archives as `Export.psarch` file. Select "Archive File Name" to change the filename.

4. Define whether the archive file includes the calculated data or not. Including calculated date increases the size of the archive.

5. Click "Ok".

   The repository archive is stored in a file with the selected filename. The file extension `*.psarch` is assigned automatically.

To import a repository archive per drag&drop

1. In your file explorer, navigate to the network directory with the repository archive.

2. Select a repository file, drag&drop it in the R&S Pulse Sequencer.

   The repository is opened and displayed in the project tree.

To import a repository archive via the menu bar

1. In the menu bar, select "File > Import Repository".

2. Navigate to the network directory. Select a `*.psarch` file.

   The "Load and Manage Repository" dialog opens.

3. Select a repository file and click "Load".

   The repository is opened and displayed in the project tree.

To share a repository with other users

To move or share repositories, use the "Export/Import Repository Archive" function. Do not access the repositories from your Windows Explorer.

1. See "To create and export a repository archive" on page 117.

2. See "To import a repository archive via the menu bar" on page 118.

To obtain write permission on a repository

Repositories can be opened with executive write permission by one user at a time.

If you have sufficient user rights, you can request an exclusive write permission on a repository that is not opened by another user with write permission.

1. In the project tree, double-click the repository name.

2. In the "Repository" dialog, select "Storage".

3. Select "Obtain Write Permission".

5 Selecting a suitable scenario and creating scenarios

This section explains the provided scenario types and helps you select the scenario most fitting to your current task.

5.1 Overview of the available scenarios and their complexity

The R&S Pulse Sequencer supports six scenarios with different complexity. Select the scenario most fitting your test case:

- **Single sequence**
  The single sequence scenario is used to create sequences from individual pulses. Pulse parameters include the pulse envelope, modulation on pulse (MOP), carrier frequency, and level. The sequence editor defines the order in which the pulses are generated and also adds control elements such as loops and fillers. Inter-pulse modulation profiles can be applied to vary pulse parameters.

- **Sequences (collection)**
  The sequences (collection) scenario is used to create multiple sequences which can be selected in arbitrary order.
  Pulse parameters include the pulse envelope, modulation on pulse (MOP), carrier frequency, and level. The sequence editor defines the order in which the pulses are generated and also adds control elements such as loops and fillers. Inter-pulse modulation profiles can be applied to vary pulse parameters.

- **Waveform sequence**
  The waveform sequence combines individual waveforms to one joint waveform or to one multi-segment waveform. The sequence editor defines the order of the individual waveforms and adds control structures such as loops and fillers.

- **Single emitter**
  The emitter scenario combines a pulse train with an antenna pattern and a particular antenna scan type. An emitter comprises several modes. Each mode is defined by an antenna pattern and an antenna scan, and multiple beams that contain the pulse train.

- **Emitters (collection)**
  The emitter collection scenario provides a choice of multiple emitters that can be selected manually. An emitter comprises several modes. Each mode is defined by an antenna pattern and an antenna scan, and multiple beams that contain the pulse train.
  Multiple emitters can be interleaved into multiple output files using groups and a priority-based dropping algorithm.
  If pulses are overlapping, the pulses with the lower priority are dropped.

- **Localized emitters**
  The localized emitters scenario combines multiple emitters and one receiver on a 2D map. An emitter comprises several modes. Each mode is defined by an

antenna pattern and an antenna scan, and multiple beams that contain the pulse train. The receiver is defined by a single antenna pattern and scan.
Multiple emitters can be interleaved into multiple output files using groups and a priority-based dropping algorithm.
If pulses are overlapping, the pulses with the lower priority are dropped.

- **Direction finding**
  The direction finding scenario simulates multiple emitters and a single receiver with up to 10 antennas on a 2D map.
  The individual signals for each receive antenna are simulated to enable the receiver to locate the position of a signal source.
  Multiple emitters can be interleaved into multiple output files using groups and a priority-based dropping algorithm.
  If pulses are overlapping, the pulses with the lower priority are dropped.

- **PDW list (collection)**
  The PDW list collection finding scenario provides a choice of multiple PDW lists that can be selected manually. This scenario generates a Rohde & Schwarz proprietary PDW-based signal from a list of imported PDWs in custom format.
  Imported PDW lists are read from a waveform object.
  Multiple emitters can be interleaved into multiple output files using groups and a priority-based dropping algorithm.
  If PDWs are overlapping, the PDWs with the lower priority are dropped.
  See Chapter 18, "Working with PDWs", on page 500.

5.2 Scenario settings

Access:

► Select "Repository Tree > Scenario".

The "Scenario" dialog provides the settings, necessary to configure the signal processing.
Settings include: settings to select the signal source, to create the ARB waveform, to select the generator, and to transfer the waveform. The dialog also provides basic instrument control functions that are required for waveform playback and signal routing.
The dialog also shows an interactive block diagram of the main processing blocks, together with their current settings and state.
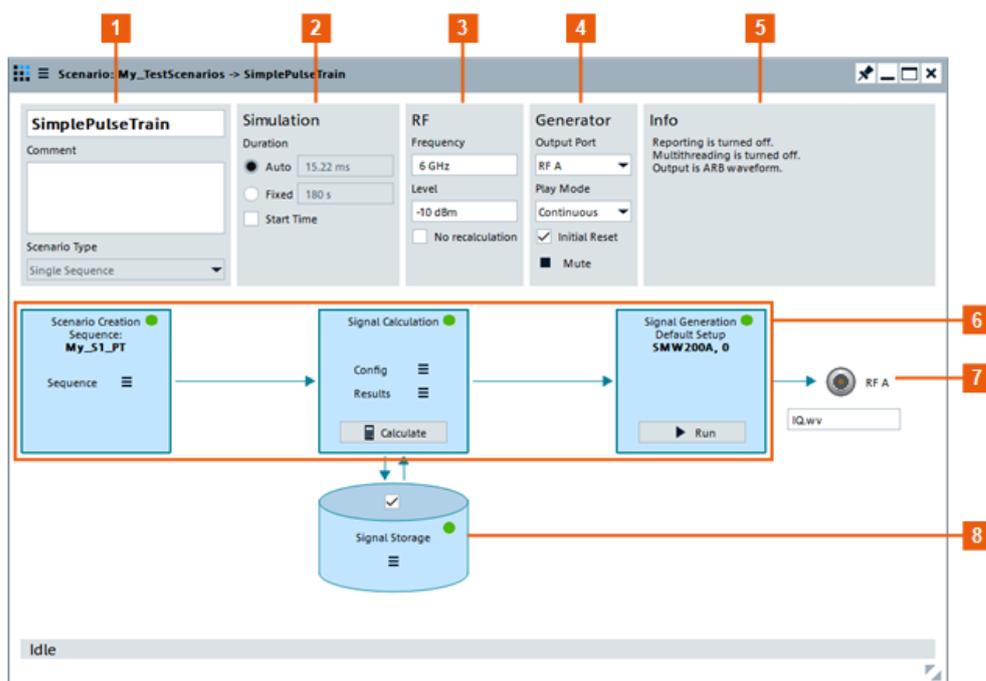
Figure 5-1: Scenario dialog: Understanding the displayed information

1	=	Scenario overview
2	=	Simulation duration settings (see "Clock, Duration" on page 479)
3	=	Main RF parameters
4	=	Functions for basic instrument control
5	=	Info: report generation, scenario setup, warnings about missing but required assignments or configurations
6	=	Block diagram of the processing chain (see "Scenario creation" on page 129, Signal Calculation, Signal Generation, Signal Calculation
7	=	Selects and indicates the signal path that plays the generated signal (Output Port). If "Warning UNCAL" displays, adjustment data is missing, see Chapter 3.5, "Compensating path loss", on page 67.
8	=	Represents the network storage the generated ARB waveform is stored in
Green LED	=	Indicates suitable source and instrument or stored local waveform
Yellow LED	=	Indicates insufficient instrument capabilities
Red LED	=	Indicates that there is no source, no generator or no waveform assigned
Dotted line	=	Interrupted signal flow, meaning there is no calculated waveform or no generator selected

The provided settings depend on the selected scenario type and on the target for signal generation (ARB file or generator).

5.2.1 Common scenario settings

The following settings are common to all scenario types:

Scenario Name

Enter a unique name to identify your current scenario in the repository.

Remote command:

SCENario:CREate on page 588

SCENario:CATalog? on page 587

SCENario:NAME on page 589

SCENario:SELect on page 588

SCENario:REMove on page 590

Comment

Enter a short description of the scenario. The first line is displayed in the repository tree.

Remote command:
SCENario:COMMent on page 590

Scenario Type
Displays the scenario type.

Remote command:
SCENario:TYPE on page 741

Simulation
Contains settings relating to the scenario duration and interleaving in appropriate scenarios.

Interleaving ← Simulation
Option: R&S SMW-K306/-K307

Enables interleaving in scenarios that support it.

For a description of interleaving, see:
- "About the interleaving" on page 326
- "Interleaving" on page 133.

Remote command:
SCENario:LOCalized:INTerleaving on page 742
SCENario:DF:INTerleaving on page 742
SCENario:CEMit:INTerleaving on page 742
SCENario:CPDW:INTerleaving on page 742

Frequency Agility ← Interleaving ← Simulation
Requires R&S SMW with firmware version 5.xx.xxx and higher. To observe the installed firmware version, select Signal Generators > "Available Signal Generators" > select the instrument > Instrument and observe the details.

In "Emitters (Collection)", "Localized Emitters" and "Direction Finding" scenarios, enables frequency agility in interleaving.

If enabled, special CNTRL PDWs are created that enable the controlled signal generator to change its center frequency. The process is called frequency agility and the time that it takes for the generator to switch its center frequency is referred to as frequency running.

During the frequency tuning, time pulses are dropped. For details on frequency tuning, see the R&S SMW datasheet.

Frequency agility is useful to simulate emitters that transmit subsequently and on different frequencies. Suitable scenario is the emitter (collection) scenario.
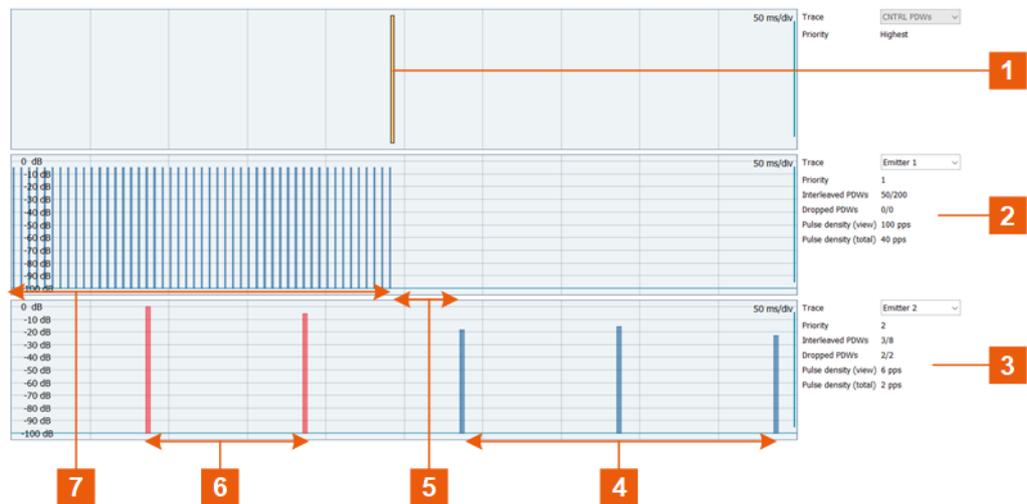
Figure 5-2: Analyze Interleaving: Understanding the displayed information (zoom in to show the transition time)

1 = CNTRL PDWs trace, CNTRL PDW generated when the frequency transition from f1 to f2 is possible
2 = Trace of emitter #1
3 = Trace of emitter #2
4 = Emitter#2 transmitts on frequency f2
5 = Transition from frequency f1 to f2
6 = Dropped PDWs from emitter #2
4 = Emitter#1 transmitts on frequency f1

One additional CNTRL PDW is generated automatically at the end of the signal to switch the frequency back to the frequency in the first track. This reverse frequency switch is especially required in scenarios with "Target = Generator" and "Generator > Play Mode = Continuous". Zoom out the "Analyze Interleaving" display to observe the last CNTRL PDW.

Remote command:
SCENario:DF:INTerleaving:FREQagility on page 742
SCENario:LOCalized:INTerleaving:FREQagility on page 742
SCENario:CEMit:INTerleaving:FREQagility on page 742

Duration ← Simulation
Sets the simulation duration and works like the "Duration" settings in the "Signal Calculation > Config > Config > Duration" dialog. If you change the settings here, they update automatically in the "Signal Calculation > Config > Config > Duration" dialog.

The generated signal can:
- Last a specified duration.
  Short sequences are repeated, longer once are truncated.
- Have an automatically determined content and duration, that best fits the current scenario and configuration. After the signal calculation, the duration is displayed.

Short sequences result in waveform files with small size.
Short sequences are repeated, longer sequences are truncated.

See also Chapter 17.1, "Signal calculation settings", on page 476.

Remote command:
SCENario:OUTPut:DURation:MODE on page 761
SCENario:OUTPut:DURation:TIME on page 761
SCENario:OUTPut:DURation:AUTO? on page 762

Start Time ← Simulation
Activate "Start Time" to define a time-based trigger.

"Simulation Time"

Set the date and time when the simulation is started.
To set the start of the simulation to the current date and time, click ⟳.
If you set a simulation time earlier than the current time, time-based triggering is not possible.

"Time Reference"

Set the time reference in synchronization to the common time reference of the signal generator in the test setup..

- "Generator Time"
  Use this setting if the signal generator works with its internal clock as timebase for triggering.
- "External 1pps"
  Use this setting if the signal generator works with an external time source as timebase for triggering.

"Time Source"

Select a signal generator as time source for time-based triggering.
You can select a certain signal generator as time source in a multi instrument setup, where no primary and secondary instrument is defined.

Remote command:
SCENario:TRIGger:TIME:STATe on page 762
SCENario:TRIGger:TIME:DATE on page 763
SCENario:TRIGger:TIME:TIME on page 763
SCENario:TRIGger:TIME:REFerence on page 763
SCENario:TRIGger:TIME:SOURce on page 763

Start preview after trigger ← Simulation

Available if a generator is selected: "Signal Calculation > Config > Target > Generator".

Opens the "2D Movement Preview" automatically after you select "Signal Generation > Run".

This preview shows the current position and attitude of the emitters in sync with the RF signals played back from the triggered signal generators.

Remote command:
SCENario:TRIGger on page 767

RF

Comprises RF settings:

Frequency ← RF

Sets the carrier frequency of the instrument on which the generated waveform is modulated on.

Emitters use their current operating frequency, see "EIRP" on page 308.

Remote command:
SCENario:OUTPut:FREQuency on page 764

Level ← RF

Sets the reference level.

This value is used by the calculation of the pulse envelope. The reference level corresponds to the "Ref." line, displayed on the "Pulse Envelope Graph" as a relative ratio. For example, "Ref. = 1" corresponds to 100% * "Level".

For unattenuated pulses, the "Level" sets the pulse top power.

Example: Pulse top power of an unattenuated pulse

If:
- "Level = -30 dBm"
- "Pulse > Level > Attenuation > Top Power = 0 dB"

Then the pulse top power would be -30 dB (at the RF output).

Example: Pulse top power of an attenuated pulse

If:
- "Level = -30 dBm"
- "Pulse > Level > Attenuation Top Power = 10 dB"

Then the pulse top power would be -40 dB (at the RF output).

If:
- "Level = -30 dBm"
- "Pulse > Level > Attenuation > Top Power = 0 dB"
- Δ Level = 10 dB

Then the pulse top power would be -20 dB (at the RF output).

Remote command:
SCENario:OUTPut:LEVel on page 764

No Recalculation ← RF
If enabled, prevents waveform recalculation if the RF frequency is changed.

Remote command:
SCENario:OUTPut:SUPRess:ENABle on page 764

Generator
Comprises settings related to the processing instrument.

Output Port ← Generator
Selects the signal path that plays the generated signal.

The available values depend on the connected instrument. Listed are all available output connectors, where the green color indicates the active (mapped) outputs.

Example:
Let the test setup use a two-path R&S SMW with the connected R&S SGS and R&S SGS+R&S SGU.

The generated signal can be routed to both RF outputs and to the IQ OUT connectors to that the R&S SGS instruments are connected.
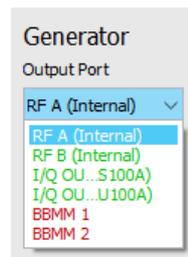
The indication in the block diagram at the output of the last processing block ("Signal Generation") confirms the selection.

Remote command:
SCENario:GENerator:PATH on page 745

Play Mode ← Generator
Defines the way the generated sequence is processed: played back once ("Single Shot") or looped ("Continuous").

Remote command:
SCENario:OUTPut:RUNMode on page 765

Initial Reset ← Generator
Sends an initial reset command (*RST) to the connected instrument to set this instrument to a definite state.

Remote command:
SCENario:OUTPut:RESet:ENABle on page 764

Mute all / Unmute all ← Generator
Mutes or unmutes the RF outputs of all generators.

Info

Displays status information on the current scenario setup:

- Kind of generated waveform file, single segment or MSW
- Report generation status, type and file location
  To enable report generation, select "Signal Calculation > Config > Config > Report-ing > Reporting > Enable"
  See also "Reporting" on page 481 and Chapter 22, "Creating reports and docu-menting measurement results", on page 542.
- Warnings about incomplete or incompatible settings.
  For troubleshooting, open the settings dialog directly by clicking ⟨Goto first error⟩.

Scenario creation

Groups and accesses settings related to the signal source.

Emitter/Emitters ← Scenario creation

Access settings related to emitters.

"Emitter"   Available in single emitter scenarios.
                Opens a context menu that enables you to edit, add or clear an emit-ter. You can also select an available emitter for the scenario from the menu.

"Emitters"   Available in multiple emitter scenarios.
                Opens the "Emitters (Collection)" dialog where you can change the settings for multiple emitters.
                For more details, see single or multiple emitters.

Map ← Scenario creation

Opens the 2D map in map-based scenarios.

See Chapter 16.2, "2D map settings", on page 424.

PDW lists ← Scenario creation

Opens the "PDW Lists (Collection)" dialog where you can edit, add and remove lists for the scenario.

See Chapter 18, "Working with PDWs", on page 500

Sequence/Sequences ← Scenario creation

Access settings related to the pulse sequence.

"Sequence"   Opens a context menu that enables you to edit, add or clear a sequence. You can also select an available sequence for the scenario from the menu.
                Available in "SimplePulseTrain" and "PDW" scenarios.

"Sequences"   Opens the "Sequences (Collection)" dialog where you can change the settings for multiple sequences.
                Available in "PT Collection" scenarios.
                For more details, see single or multiple pulse sequences, a waveform sequence.

Signal Calculation

Groups and accesses settings related to the waveform generation and the calculated output signal.

Config ← Signal Calculation

In general, the software calculates a waveform most suitable for the target currently selected in the "Signal Generation" block.

For example, the waveform is an ARB file or a multi-segment waveform.

If you prefer to change these settings, however, select "Config".
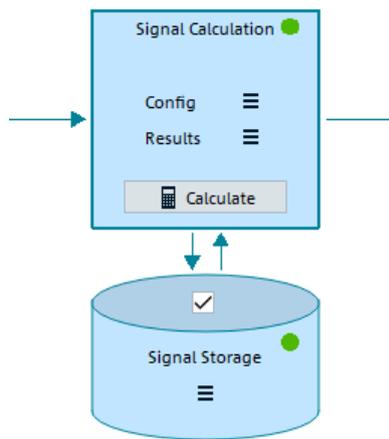
"Output Format"

Accesses a dialog where you can change the settings influencing the waveform generation, see Chapter 17.1, "Signal calculation settings", on page 476.

"Open Report Folder"

Opens the report folder in Windows Explorer, see Chapter 22, "Creating reports and documenting measurement results", on page 542.

"Target"　　　　　Allows you to select your target, either "File" or "Generator", from a list.

"Set Path"

Requires "Target > File".
Defines the storage place for the generated ARB file.

"Open Path in Explorer"

Requires "Target > File".
Accesses the storage folder in the Windows Explorer.

"Clear Target"

Removes the selected target and sets the selection to "None".

"Sanitize"

> Requires "Target > Generator".
> Removes uploaded waveforms in the current scenario from the hard disk of the signal generator.
>
> * "This Scenario"
> * "Entire Repository"
> * "All Pulse Sequencer"
>
> Remote command:
> SCENario:SANitize on page 767

"Signal Generators"

> Accesses the "Signal Generators" dialog, see Chapter 3.6, "Signal Generators settings", on page 75.

Remote command:

SCENario:OUTPut:PATH on page 764

SCENario:STARt on page 743

SCENario:TRIGger on page 767

SCENario:STOP on page 743

SYSTem:PROGress? on page 797

SCENario:STATe? on page 743

Assign ← Signal Calculation

In a "Scenario Type = Localized Emitters/Emitters (Collection)/Direction Finding/PDW List (Collection)", accesses the dialog for mapping of signals to Signal Generators.

See Chapter 17.2, "Assign signal source or receiver signals to generator", on page 481.

Remote command:

SCENario:OUTPut:TARGet on page 765

SCENario:GENerator on page 745

SCENario:GENerator:PATH on page 745

Results ← Signal Calculation

To observe the calculation results, select "Results".

Depending on your user rights, you have read-only or full access to the repository. Users granted with full access rights are allowed to perform all the following functions:

"View"

> Selects a generated file and visualizes it.
> See Chapter 15, "Visualizing and analyzing signals", on page 400.
>
> Remote command:
> SCENario:VOLatile:SEL on page 767
> SCENario:VOLatile:VIEW on page 767

"Save calculated results"

Exports and stores the waveform files in the repository, i.e. the waveform is archived in the repository.

You can access archived waveforms anytime later, e.g. after software restart.

Remote command:
SCENario:CACHe:VOLatile:RELease on page 744

"Load calculated results"

Loads and play files from this storage; stored waveform files are played back without being recalculated.

Remote command:
SCENario:CACHe:VOLatile:RESTore on page 744

"Clear calculated results"

Removes the calculated results from the scenario but not from the storage. Change the required settings and calculate again.

Remote command:
SCENario:CACHe:VOLatile:CLEar on page 745
SCENario:CACHe:REPository:CLEar on page 745

"View memory usage"

Opens the "Memory usage" dialog where you can see the overview of current memory usage for each generator.

"View signal statistics"

Opens the "Signal Statistics" dialog where you get information about the signal parameters, e.g. frequency, level and bandwidth. The results are displayed in a table.

Calculate ← Signal Calculation

Select to start the signal calculation.

The calculation and generation process depend on whether a separate trigger for starting the scenario is used or not. To enable separate trigger, set "Menu bar > Configure > Settings > General > Program Settings > Use separate trigger command to start scenarios > On".

Starts signal calculation, if the prerequisites are fulfilled.

The software informs you about the estimated file size. A "Busy" indication in the status bar and detailed progress information indicates that the calculation is in progress.

Using triggers:
● If common trigger is used and a signal generator is selected, the waveform is transferred and stored in its volatile memory.
If common trigger is used and a destination is selected, the signal is transferred and stored in the repository.
Signal generation is not started.
● With separate trigger, trigger this process step with the "Load" function.

"Stop"   Aborts the current calculation process.

Remote command:

SCENario:CALCulate on page 742

SCENario:STOP on page 743

SCENario:TRIGger on page 767

Signal Storage ← Signal Calculation

Changes the storage settings.

"Signal Storage" > "Enable"

        Enables file storage in the repository when selected.

        Remote command:

        SCENario:CACHe:REPository:ENABle on page 744

"Signal Storage" > "Clear"

        Deletes files from the repository.

        Remote command:

        SCENario:CACHe:VOLatile:CLEar on page 745

        SCENario:CACHe:REPository:CLEar on page 745

Remote command:

SCENario:CACHe:REPository:VALid? on page 744

Interleaving

Option: R&S SMW-K306/-K307

Displayed in "PDW Lists (Collection)", "Emitters (Collection)", "Localized Emitters" and "Direction Finding" scenarios, if "Simulation > Interleaving > On". See "Simulation" on page 124.

Groups and accesses settings related to the interleaving process and the analysis of interleaved and dropped PDWs.

Enables you to combine the emitters or PDW lists into multiple output files using groups, "Merge" mode, and a priority-based dropping algorithm, "Drop" mode.

The interleaving is a separate optional post-processing step in the signal calculation. If this step is required, it can be triggered separately (see "Simulation" on page 124).
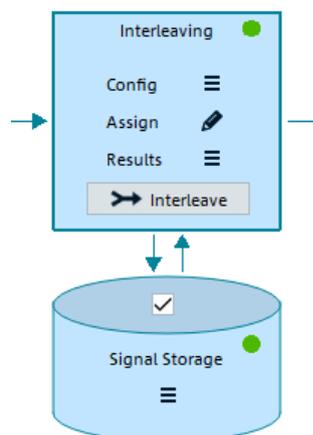
See also:
- "Emitters (Collection) scenario settings" on page 140

- "Localized emitters scenario settings" on page 142
- "Direction finding scenario settings" on page 143
- "PDW list scenario settings" on page 144

Config ← Interleaving

Allows you to configure the interleaving settings.

See:

- Chapter 11.1, "Emitters (Collection) and localized emitters interleaving settings", on page 327
- Chapter 18.6, "PDW list (Collection) interleaving settings", on page 516

"Interleaving Groups"

Accesses a dialog where you can change the settings influencing the waveform generation, see Chapter 17.1, "Signal calculation settings", on page 476.

"Target"　　　　Allows you to select your target, either "File" or "Generator".

"Set Path"

Requires "Target > File".

Defines the storage place for the generated ARB file.

"Open Path in Explorer"

Requires "Target > File".

Accesses the storage folder in the Windows Explorer.

"Sanitize"

Requires "Target > Generator".

Removes uploaded waveforms in the current scenario from the hard disk of the signal generator.

- "This Scenario"
- "Entire Repository"
- "All Pulse Sequencer"

Remote command:

SCENario:SANitize on page 767

"Signal Generators"

Accesses the "Signal Generators" dialog, see Chapter 3.6, "Signal Generators settings", on page 75.

Assign ← Interleaving

In a "Scenario Type = Localized Emitters/Emitters (Collection)/Direction Finding/PDW List (Collection)", accesses the dialog for mapping of signals to Signal Generators.

See Chapter 17.2, "Assign signal source or receiver signals to generator", on page 481.

Results ← Interleaving

To observe the statistical information on the interleaved and dropped PDWs or pulses, select "Results".

Depending on your user rights, you have read-only or full access to the repository. Users granted with full access rights are allowed to perform all the following functions:

"View" > "Analyze"

Opens the "Analyze Interleaving" dialog.
See Chapter 18.9, "Analyze interleaving", on page 521.

"View" > "Interleaved/Dropped PDWs"

Opens the "Dropped PDWs" dialog presents a visualization of the results.
See Chapter 18.8, "Signal preview with interleaved and dropping statistics", on page 520.

"Save calculated results"

Exports and saves the output signal files in the repository, i.e. the files are archived in the repository.
You can access archived files anytime later, e.g. after software restart.

Remote command:
SCENario:CACHe:REPository:ENABle:INTerleave
on page 744

"Load calculated results"

Loads and play files from this storage; stored waveform files are played back without being recalculated.

Remote command:
SCENario:CACHe:VOLatile:RESTore on page 744

"Clear calculated results"

Removes the calculation results and enables you to edit the scenario settings and recalculate.

Remote command:
SCENario:CACHe:VOLatile:CLEar on page 745
SCENario:CACHe:REPository:CLEar on page 745

"View signal statistics"

Opens the "Signal Statistics" dialog where you get information about the signal parameters, e.g. frequency, level and bandwidth. The results are displayed in a table.

"View memory usage"

Opens the "Memory usage" dialog where you can see the overview of current memory usage for each generator.

Interleave ← Interleaving
Option: R&S SMW-K306

Select to start interleaving.

Available in "PDW List (Collection)" and "Emitter (Collection)" scenarios, if "Interleaving > On" is used.

Triggers the calculation of a single output file that comprises the individual PDWs or pulses, where overlapping PDWs or pulses are dropped, based on a defined priority.

The "Interleaving progress" dialog visualizes the number of PDWs or pulses that are included in the output file and the percentage of other that are dropped.

See "Interleaving" on page 133.

Remote command:
SCENario:INTerleave on page 743

Signal Storage ← Interleaving
Changes the storage settings.

"Storage" > "Enable"
   Enables file storage in the repository.

   Remote command:
   SCENario:CACHe:REPository:ENABle on page 744

"Storage" > "Clear"
   Deletes files from the repository.

   Remote command:
   SCENario:CACHe:VOLatile:CLEar on page 745
   SCENario:CACHe:REPository:CLEar on page 745

Signal Generation
Contains settings related to the signal output.
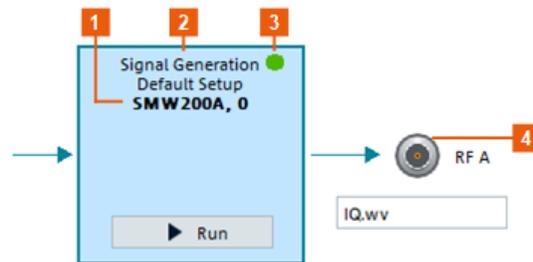
1 = Generator, as selected with "Config > Target > Signal Generators"
2 = Current setup name, as selected in the dialog "menu bar > Configure > Signal Generators", see Chapter 3.6, "Signal Generators settings", on page 75
3 = Status indication
4 = Processing signal path, as selected with Output Port

Run/Stop ← Signal Generation
Provides the following functions:

"Run"
   Runs the configured scenario.
   Available if common trigger is used. See "Calculate" on page 132.
   If a signal generator is selected, loads the waveform in the ARB and starts signal generation.

"Stop"
   Aborts the current calculation process.

Remote command:
SCENario:STARt on page 743
SCENario:STOP on page 743

Load ← Signal Generation
Available, if a separate trigger for starting the scenario is used.

If a signal generator is selected, the waveform is transferred and stored in its volatile memory. Signal generation is not started.

Remote command:

SCENario:STARt on page 743

SCENario:STOP on page 743

Trigger ← Signal Generation

Available, if a separate trigger for starting the scenario is used.

If a signal generator is selected, starts signal generation.

Signal routing and RF settings are configured as defined with:
- Frequency and Level
- Initial Reset, Output Port and Play Mode

Remote command:

SCENario:TRIGger on page 767

SYSTem:PROGress? on page 797

SCENario:STATe? on page 743

UNCAL

Warning symbol indicates that the built-in adjustment function for path loss compensation is activated but there is no adjustment data collected.

See Chapter 3.5, "Compensating path loss", on page 67.

5.2.2 Scenario-specific settings

Single sequence and Waveform scenario settings

For an example of a single sequence scenario, see Figure 5-1.

These scenarios generate an ARB signal from a selected sequence or a waveform.

Use the context menu in the "Scenario Creation" block to select and configure the transmitted sequence or waveform.

See:
- Chapter 7, "Building pulse sequences", on page 198
- Chapter 14, "Working with waveforms", on page 368
- Chapter 17.5, "How to create and play the waveforms of simple scenarios", on page 489
- Chapter 5.2.1, "Common scenario settings", on page 122

Remote command:

SCENario:SEQuence on page 745

SCENario:SEQuence:CLEar on page 746

Sequence (Collection) Scenario Settings
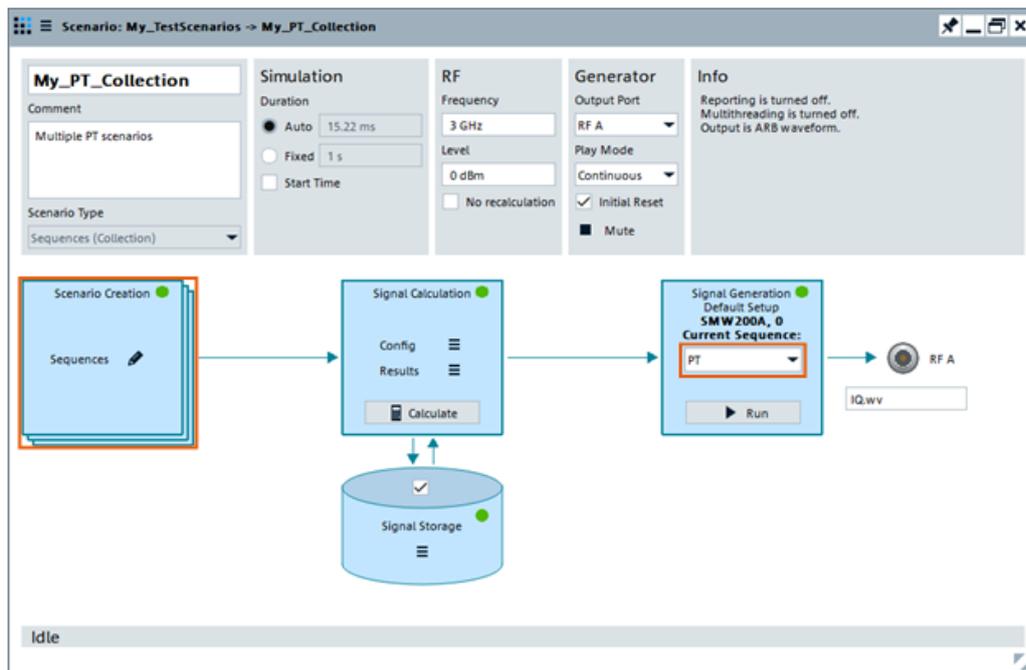
A scenario with multiple sequences requires a list of sequences. You can switch between these sequences using "Scenario Creation" menu and select the one to be transmitted.

The "Current Sequence" field in the "Signal Generation" block indicates the alias name of the sequence currently selected.

See:
- Chapter 7.2.6, "Lists with multiple sequences", on page 210
- Chapter 5.2.1, "Common scenario settings", on page 122

Remote command:
SCENario:CSEQuence on page 755
SCENario:CSEQuence:CURRent on page 746

Single emitter scenario settings
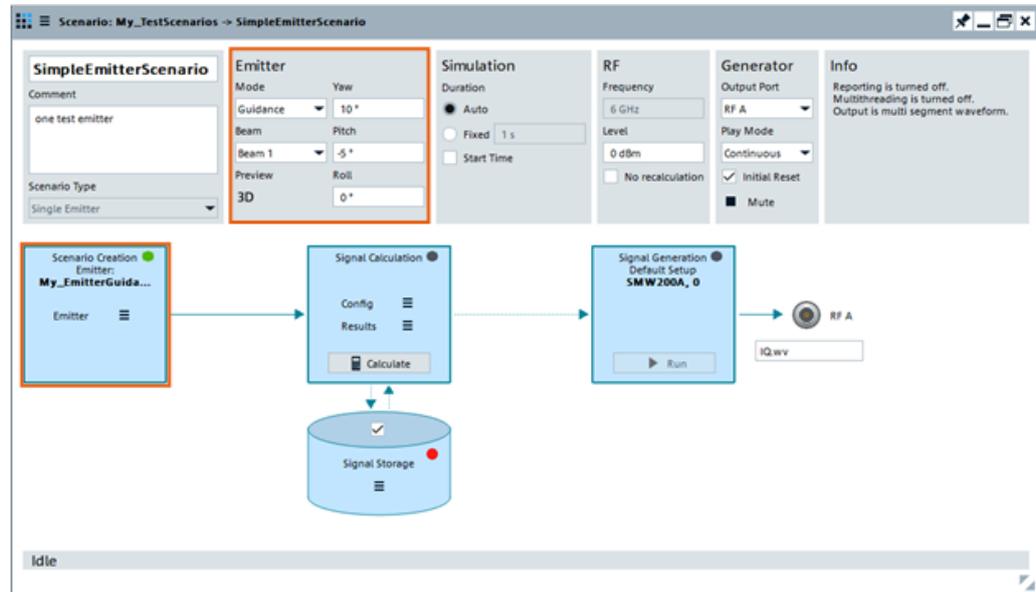
Use the context menu in the "Scenario Creation" block to select and configure the emitter.

You can switch between the emitter modes, define the current beam, set the attitude, and visualize the signal received by a static receiver on a 3D preview.

In this scenario, the "Frequency" is a read-only parameter that displays the Frequency value of the current emitter (as selected in the "Emitter" dialog).

See:
- "Emitter attitude" on page 305
- Chapter 10.4, "How to create and configure emitters", on page 320
- Chapter 10.1, "Emitter settings", on page 307
- Chapter 5.2.1, "Common scenario settings", on page 122

Remote command:
SCENario:EMITter on page 747
SCENario:EMITter:MODE on page 748
SCENario:EMITter:MODE:BEAM on page 748
SCENario:EMITter:CLEar on page 596
SCENario:EMITter:DIRection:PITCh on page 747
SCENario:EMITter:DIRection:YAW on page 746
SCENario:EMITter:DIRection:ROLL on page 747

Emitters (Collection) scenario settings
You can use "Emitter (Collection)" scenario to test the receiver's ability to detect the signal from different static emitters. In this dedicated scenario, you can configure several emitters and switch between them sequentially. You can also configure the receiver and change its position in the scanning beam of the emitters.

In addition to processing the emitters sequentially, if the selected signal generator supports **interleaving**, you can also combine them into multiple output files using groups and a priority-based dropping algorithm. During the interleaving process, overlapping pulses with lower priority are dropped. You can configure the priority of the individual emitters.

If "Simulation > Interleaving > On", the block diagram shows the dedicated Interleaving block.

See:
- Chapter 11, "Combining the signals of multiple emitters", on page 325
- "Interleaving" on page 133
- "To configure complex scenarios with several emitters" on page 322
- Chapter 10.3, "Emitters (Collection) settings", on page 312
- "To create, load and play the waveforms of a direction finding scenario automatically" on page 492

Remote command:
SCENario:CEMit:INTerleaving on page 742

Localized emitters scenario settings
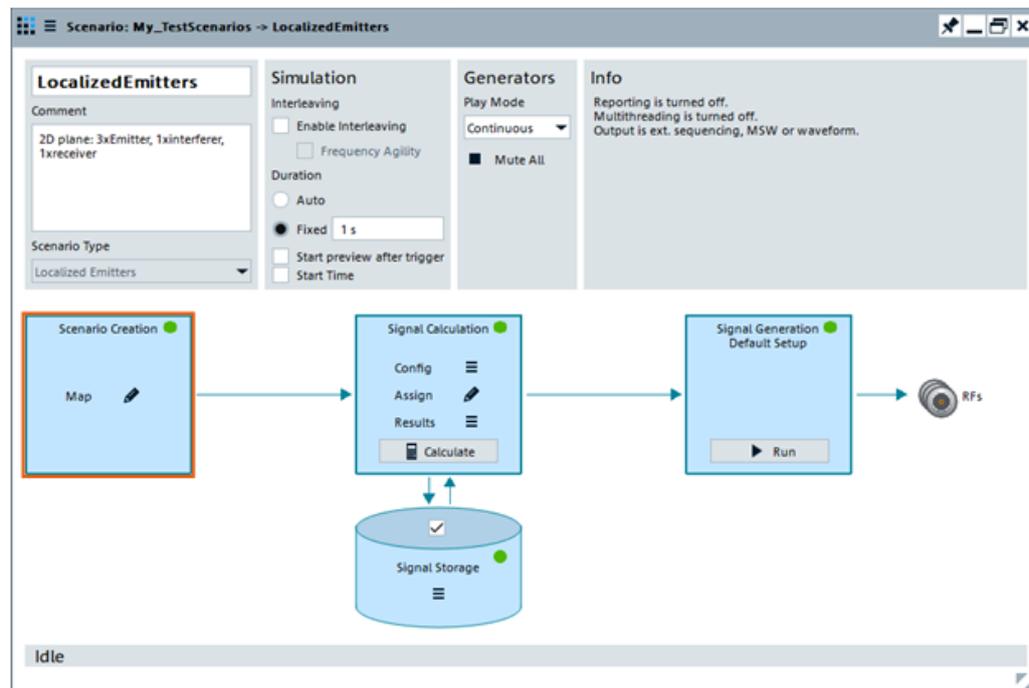
You can use the "Localized Emitter" scenario to test the receiver's ability to detect the sum signal from different static emitters/interferers. In this dedicated scenario, you configure the signal of one or more emitters/interferers that would be received by a receiver with defined characteristics. You can also configure the receiver and change its position in the scanning beam of the emitters.

Select "Scenario Creation > Map" to display the 2D view of the receiver and currently configured emitters/interferers, together with their main characteristics.

If "Simulation > Start preview after trigger > On", the "2D Movement Preview" opens automatically when you select "Run" in the "Signal Generation" block (see "Start preview after trigger" on page 127).

In this preview, you see the current position and attitude of the emitters in sync with the RF signals played back from the triggered signal generators.

This feature is only available after a signal generator has been set via "Signal Generation > Config".

In addition to processing the emitters sequentially, if the selected signal generator supports interleaving, you can also combine them into multiple output files using groups and a priority-based dropping algorithm. During the interleaving process, overlapping pulses with lower priority are dropped. You can configure the priority of the individual emitters.

If "Simulation > Interleaving > On", the block diagram shows the dedicated Interleaving block.

See:
- Chapter 16.1, "How to create scenarios with receiver and TX items", on page 416
- Chapter 17.2, "Assign signal source or receiver signals to generator", on page 481
- Chapter 5.2.1, "Common scenario settings", on page 122

When you create ARB files ("Local File > Target > File"), select "Local File > Open Path in Explorer" to access the generated waveforms and the `Setup_Info.txt` file. Follow the instructions in this `Setup_Info.txt` file.

See also "To create, transfer and play the waveforms of scenarios manually" on page 496.

Remote command:
`SCENario:LOCalized:INTerleaving` on page 742

Direction finding scenario settings

The "Direction Finding" scenario simulates multiple emitters and a single receiver with up to 20 antennas on a 2D map. This scenario is suitable to test direction finding systems and to detect interfering signals or transmissions in unauthorized bands.

The generated files are the individual signals of each receive antenna.
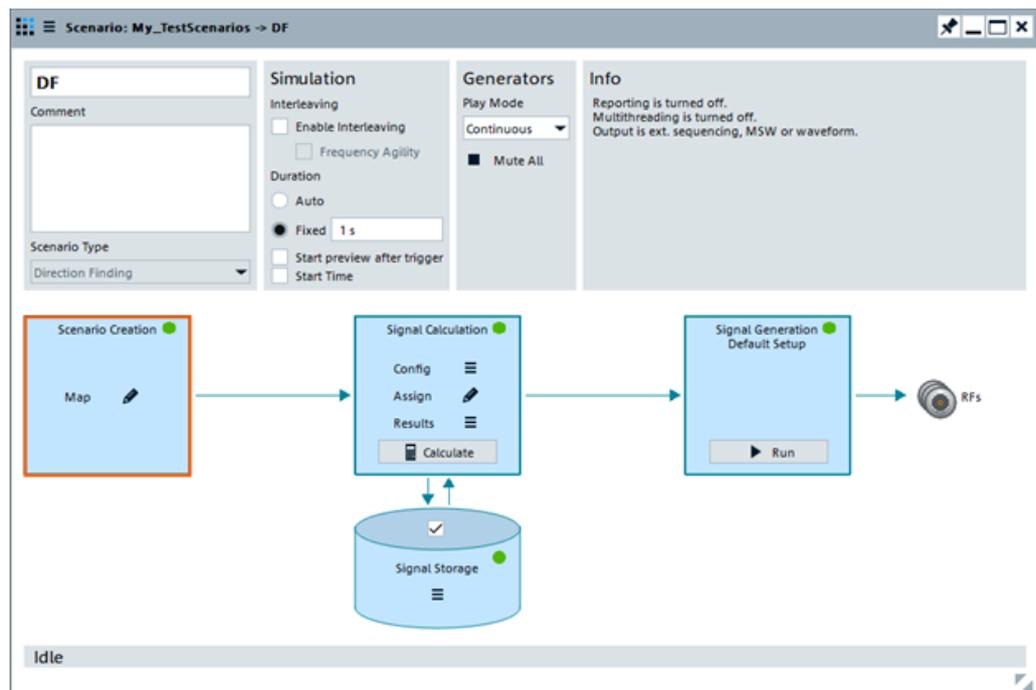
Select "Scenario Creation > Map" to display the 2D view of the receiver and the configured emitters/interferers, together with their main characteristics.

If "Simulation > Start preview after trigger > On", the "2D Movement Preview" opens automatically when you select "Signal Generation > Run" (see "Start preview after trigger" on page 127). This feature is only available after you set a signal generator via "Signal Generation > Config".

In this preview, you see the current position and attitude of the emitters in sync with the RF signals played back from the triggered signal generators.

In addition to processing the emitters sequentially, if the selected emitters/interferers supports interleaving, you can also combine them into multiple output files using groups and a priority-based dropping algorithm. During the interleaving process, overlapping pulses with lower priority are dropped. You can configure the priority of the individual emitters.

If "Simulation > Interleaving > On", the block diagram shows the dedicated Interleaving block.

See:
- Chapter 13.2, "Receiver settings", on page 352
- Chapter 16.1, "How to create scenarios with receiver and TX items", on page 416
- Chapter 17.2, "Assign signal source or receiver signals to generator", on page 481

Remote command:
SCENario:DF:INTerleaving on page 742

PDW list scenario settings

The "PDW List (Collection)" scenario is dedicated for importing custom PDW lists from legacy test cases and importing them into the application.

The generated files are compatible with the file format that can be loaded in and processed by R&S SMW equipped with the option Extended Sequencer (R&S SMW-K501/K502).

The advantage of using this scenario type is the size of the output files and duration of the generated signal. In comparison, you can also use other scenario types but in this case the R&S Pulse Sequencer creates a waveform. Waveforms describing long simulation time are usually large.

For details on the file format, see R&S SMW-K501/-K502/-K503/-K504 Extended and Real-Time Sequencing, Real-Time Control Interface user manual.

Moreover, you can use the "PDW List (Collection)" scenario to configure several PDW lists and switch between them sequentially. You can also combine them into multiple output files using groups and a priority-based dropping algorithm. During the interleaving process, overlapping PDWs with lower priority are dropped, where you define the priority of the individual PDW lists.

If "Simulation > Interleaving > On", the block diagram shows the dedicated Interleaving block.
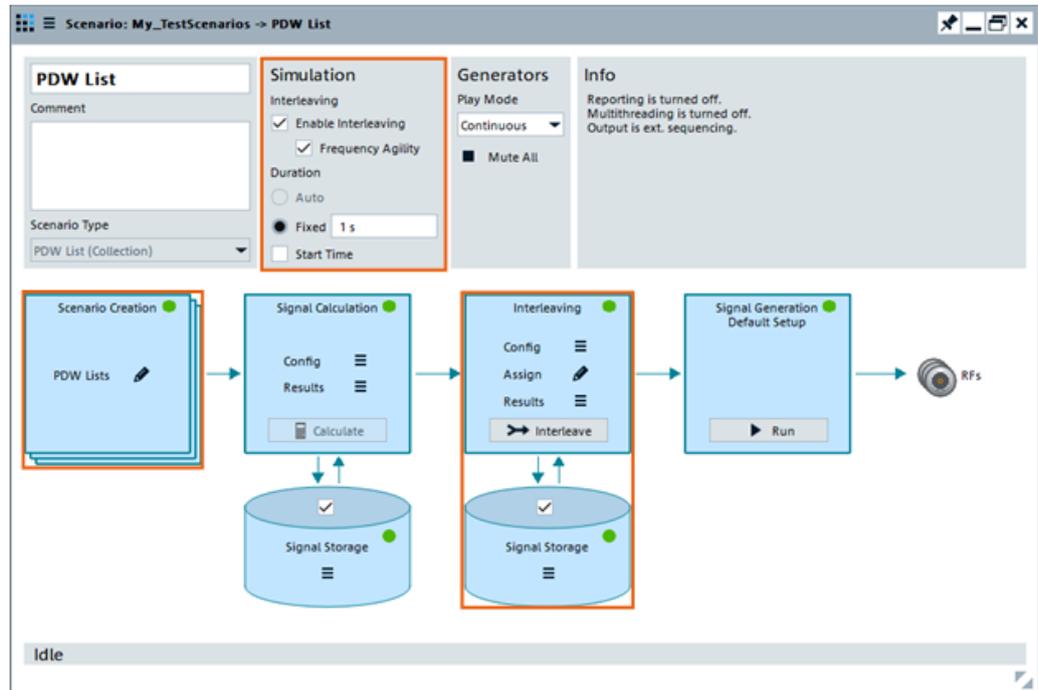
See:
- Chapter 18, "Working with PDWs", on page 500

Remote command:
SCENario:CPDW:CURRent on page 746
SCENario:CPDW:INTerleaving on page 742

5.3 How to select and create a test scenario

The overview in Table 5-1 helps you decide which scenario type is suitable for a certain test case.

Table 5-1: Typical test cases with their suitable scenario types

Test case	Scenario type
Low-level receiver tests Receiver tests with IPM profiles and modulation on pulse	Single sequence Sequences (collection) Waveform sequence
Receiver stress tests, where the receiver antenna is an omnidirectional antenna. The emitted signal follows the amplitude changes of antenna patterns and scans Simulation of the signal at the receiver antenna output The receiver power is normalized to maximum	Single emitter Emitters (collection)

Test case	Scenario type
Advanced receiver tests, like reception of multiple signals at the same time. The received signals can be the signals of emitters/interferers also in the presence of the background noise of many other emitters Emitters/Interferers and receiver can be moving objects The emitter and the receiver antenna patterns and antenna scans are considered Receiver power is calculated according to free space loss between emitter and receiver	Localized emitters
Receiver tests for direction finding applications Detection of interferers in dense electro-magnetic environments	Direction finding
Import of custom PDW list Merging of multiple PDW lists into a single output file using a priority scheme for dropping	PDW list
Merging of multiple emitters into multiple output files using groups and a priority-based dropping algorithm.	Emitters (collection) Localized emitters Direction finding

To create a single pulse train scenario

1. To access the scenario settings dialog, perform one of the following:

   a) Select "Repository Tree > Scenario > New".

   b) In the "Toolbar", select the scenario icon ⊞.

   A scenario wizard opens that helps you to select a predefined scenario configuration.
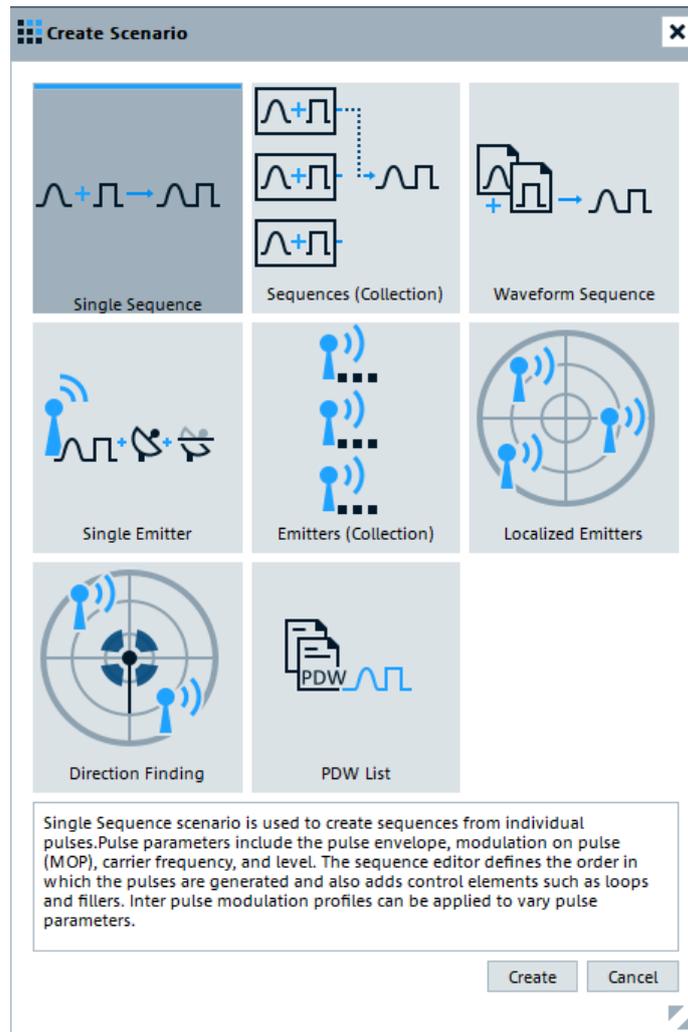
2. Select "Scenario Type > Single Sequence".
   Click "Create"

   Created is a new scenario with default name `Scenario <n>`, where `n` is a number starting at one.
   The provided settings depend on the selected scenario type.
   See Chapter 5.2, "Scenario settings", on page 121 for description.

3. Add information to describe and identify the scenario, like a name or a comment.
   Use the standard functions in the context menus to select or create sequences, select the signal generator, etc.
   See Table 2-4.

   Perform further configurations, see:
   ● Chapter 7.3, "How to create sequences and use the control elements", on page 212
   ● Chapter 15.4, "Analyzing the calculated results", on page 411

6 Creating a pulse library

This section provides background information on pulse generation and pulse characteristics, explains the provided settings and how to use them to create a pulse library.

6.1 Basics on pulse signals and pulse generation

Some background knowledge on basic terms and principles used in pulse generation is provided here for a better understanding of the required configuration settings.

The pulse parameters to be configured are based primarily on the IEEE 181 Standard 181–2003. For detailed descriptions refer to the standard documentation ("IEEE Standard on Transitions, Pulses, and Related Waveforms", from the IEEE Instrumentation and Measurement (I&M) Society, 7 July 2003).

Basic terms

The definition of the pulse parameters depends on the used timing profile (see Standard Timing Profile) and distinguishes between:

- **Power** 10/50/90
- **Voltage** 10/50/90
- (Power/Voltage) 10/100

Where the name of the timing profile follows the naming convention:

- Power/Voltage: indicates whether power or voltage-based reference levels are used
- Indication of the used reference levels, i.e.:
  - 10/50/90 means that three reference levels are used: the lower (10% nominally), the medial (50% nominally) and the upper (90% nominally)
  - 10/100 indicates that two levels are used: the base (0%) and the top (100%) level

A pulsed signal is a signal whose carrier power is modulated by two states: On and off.

If the top power level of a pulse is not constant, this effect is called an amplitude **droop**.

The graphic on Figure 6-1 illustrates the main pulse parameters and characteristic values when a **Power** 10/50/90 profile is used. If voltage profile is used, the definition of the pulse parameters is identical, but the parameters are related to **voltage**.
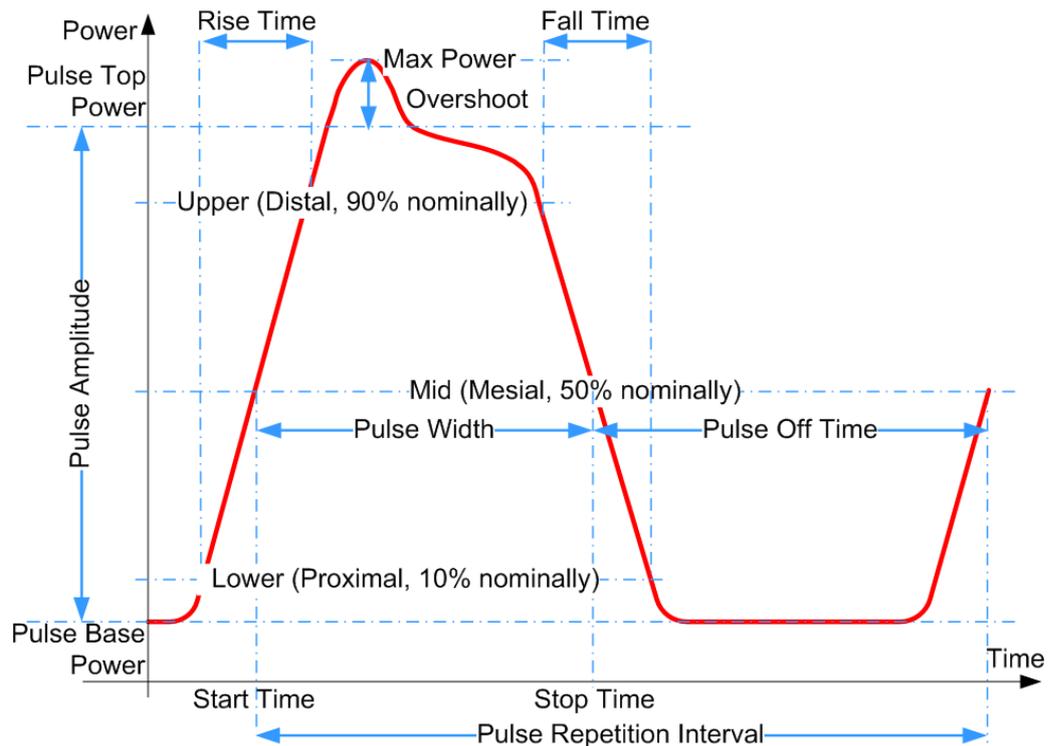
Figure 6-1: Definition of the main pulse parameters and characteristic values ("Timing Profile > Power 10/50/90")

Pulse parameters ("Timing Profile > 10/50/90")

The following definitions apply:

- **Rise time, fall time** (also known as transition duration of the first and the second transitions)
  This is the difference between two reference levels (10% and 90% nominally) of the same transition.

- **Pulse width** (also known as pulse on time or pulse duration)
  This is the difference between the first and the second transition of one pulse, i.e. the duration the signal remains above the 50% reference level.

- **Pulse off time** (also known as pulse separation)
  The duration between the 50% reference level of the second transition of one pulse and the 50% reference level of the first transition of the subsequent pulse in the same pulse train.

- **Pulse repetition interval** (also known as pulse period)

  Is defined as the duration of one complete cycle consisting of:

  – The pulse width (i.e. pulse on time or pulse duration)

  – The pulse off time (pulse separation)

Pulse parameters ("Timing Profile > 0/100")

Note that the pulse parameters differ, if a "Timing Profile > 0/100" is used.

The following definitions apply:

- **Rise time, fall time**
  This is the difference between the 0% and 100% reference levels of the same transition, i.e. between the base and the top level.
- **Pulse width** (pulse on time)
  The duration the signal power remains at the top level (100% reference level).
- **Pulse off time**
  The duration the signal power remains at the base level (0% reference level).
- **Pulse repetition interval**

  Is defined as the duration of one complete cycle consisting of:
  - The rise time
  - The pulse width
  - The fall time
  - The pulse off time

For description of the related settings, see:

- Chapter 6.2.3, "Pulse level settings", on page 155
- Chapter 6.2.4, "Pulse envelope settings", on page 159

6.2 Pulse settings

Access:

► Select "Repository Tree > Pulse > New".

Pulses are characterized by the following settings:

6.2.1 Pulse name and comment
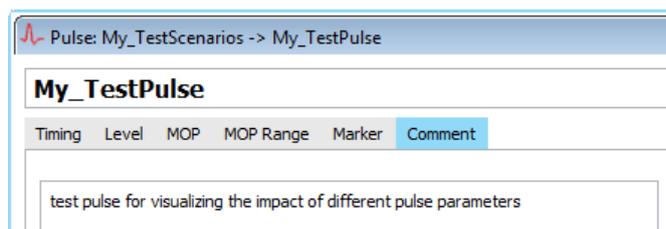
Settings:

Pulse Name

Displays the name of the current pulse.

Remote command:
PULSe:CREate on page 588
PULSe:NAME on page 589
PULSe:CATalog? on page 587
PULSe:SELect on page 588
PULSe:REMove on page 590

Comment

General pulse information.

Remote command:
PULSe:COMMent on page 590

6.2.2 Pulse timing settings

Access:

► Select "Pulse > Timing".
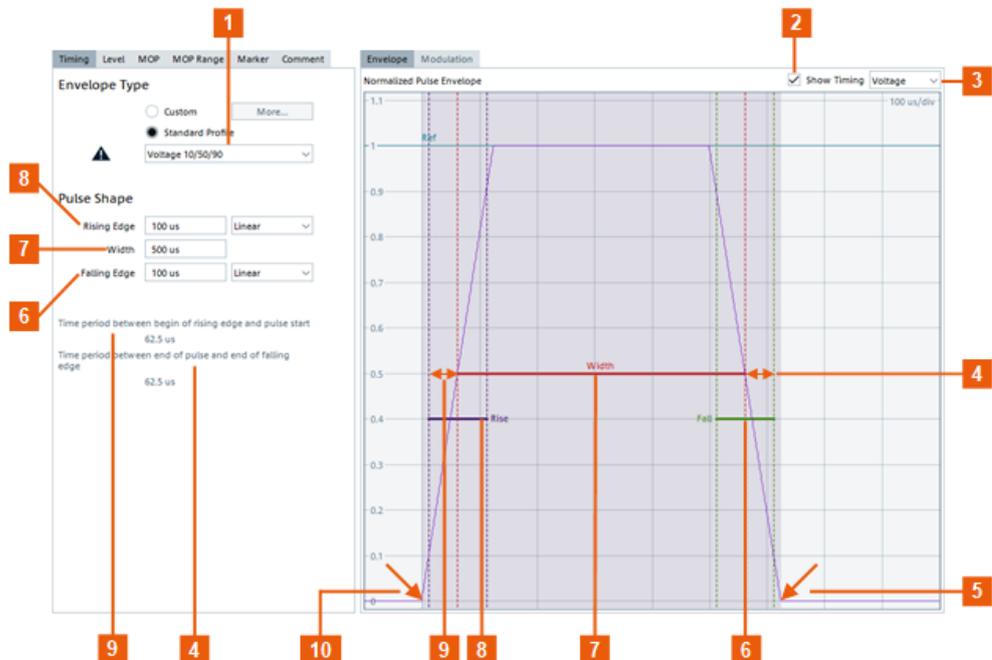
Figure 6-2: Pulse timing: understanding the displayed information

1   = "Standard Profile > Voltage 10/50/90"
2   = "Show Timing > On" indicates the pulse timing parameters on the envelope graph
3   = "Pulse Envelope Units > Voltage"
4   = "Time period between end of pulse and the end of falling edge = 62.5 us". I.e. the time it takes
            the voltage to fall from 50% to 0% of the top level
5, 10 = Beginning of the rising and falling edge
6   = "Falling Edge = 100 us", i.e. the time it takes the voltage to fall from 90% to 10% of the top level
7   = "Width = 500 us", i.e. the duration the voltage is above 50% of the top level
8   = "Rising Edge = 100 us", i.e. the time it takes the voltage to rise from 10% to 90% of the top level
9   = "Time period between beginning of the rising edge and pulse start = 62.5 us". I.e. the time it
            takes the voltage to rise from 0% to 50% of the top level

Settings:

Use the following parameters to define the pulse in the time domain:

Custom Envelope
Applies an envelope shaping according to a custom envelope function.

See Chapter 6.2.4, "Pulse envelope settings", on page 159.

Remote command:
PULSe:CUSTom on page 698
PULSe:ENVelope:MODE on page 700

Standard Timing Profile
Enables and selects a predefined standard envelope profile.

Select from a voltage-based "Voltage 10/50/90", a power-based "Power 10/50/90" or a
0-100% "0/100" timing profile.

For brief introduction to the timing profiles and their impact on the pulse timing, see
Chapter 6.1, "Basics on pulse signals and pulse generation", on page 148.

See Figure 6-3 for illustration of the pulse width, pulse rise and fall time, and period
between the beginning of the rising edge and the pulse start. The figure shows these
parameters for each of the envelope profiles.

Enable "Show Timing" to visualize the timing parameters. Note that the "Envelope
Graphs" use different "Normalized Pulse Envelope" settings.
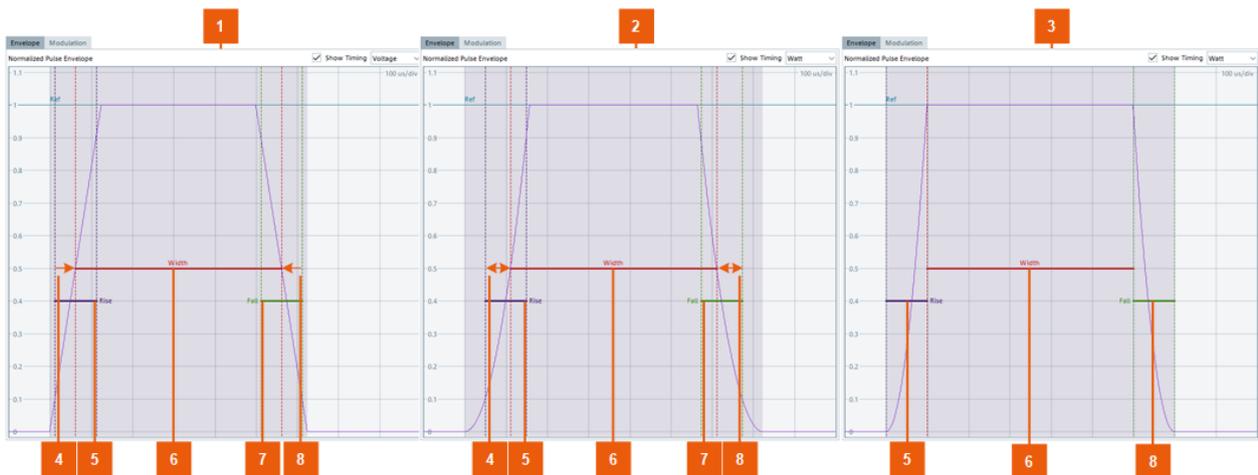
Figure 6-3: Pulse timing parameters and the envelope profiles (linear rising and falling slopes)

1 = "Voltage 10/50/90" profile
2 = "Power 10/50/90" profile
3 = "0/100" profile
4 = "Time period between beginning of the rising edge and pulse start" (with"0/100" profile this time is 0 us)
5 = "Rising Edge = 100 us"
6 = "Width = 500 us"
7 = "Falling Edge = 100 us"
8 = "Time period between end of pulse and the end of falling edge" (with"0/100" profile this time is 0 us)

Remote command:
PULSe:TIME:REFerence on page 714

Pulse Shape Settings

These parameters define the pulse shape:
- Rising/Falling edge
- Width
- Rising/Falling slope

The current pulse shape is displayed on the Envelope graph. To visualize the timing parameters on the graph, select "Envelope > Show Timing > On".

The parameter definition depends on the selected Standard Timing Profile, see:
- "Pulse parameters ("Timing Profile > 10/50/90")" on page 149
- "Pulse parameters ("Timing Profile > 0/100")" on page 149

The total pulse duration and the pulse off time are calculated automatically from the selected pulse width, rise and fall time, and the PRI (see PRI/PRF).

"Rising/Falling Edge"

Transition time of the rising/falling edge.

Remote command:
PULSe:TIME:RISE on page 714
PULSe:TIME:FALL on page 714

"Rising Slope Type"

Form of the rising/falling slope.

Available are linear, cosine, raised and root cosine. You can also use arbitrary envelope data or import you custom shapes, see Chapter 20, "Defining complex modulation schemes and IPM profiles", on page 530.

Remote command:
PULSe:TYPE:RISE on page 714
PULSe:TYPE:FALL on page 714

"Width"

Pulse duration.

Remote command:
PULSe:TIME:WIDTh on page 714

Time period between beginning of rising edge and pulse start / Time period between end of pulse and end of falling edge
Display information on the time periods during which the RF power is attenuated or suppressed but per default there is modulation or data content present.

Note: Avoiding data truncation.
To avoid truncation of data during the rising time, add guard bits to the data source.

Per default, modulation is already present during the rising and falling phases.

You can restrict the pulse area on which the modulation is applied, see "Restrict MOP Range" on page 185.

6.2.3 Pulse level settings

Access:

► Select "Pulse > Level".
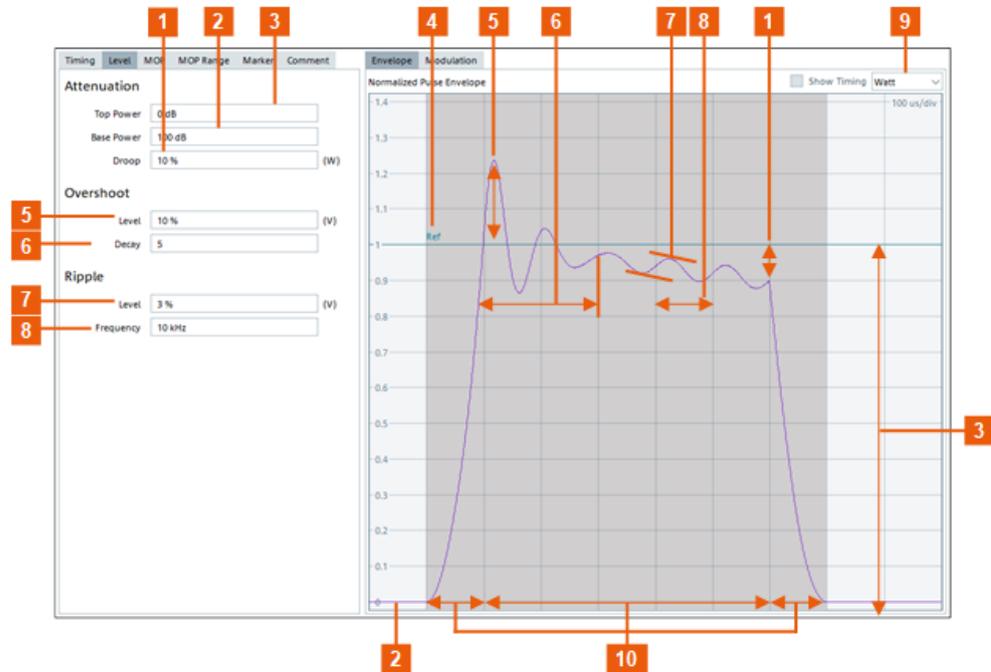
Figure 6-4: Pulse level: understanding the displayed information

1 = "Droop", i.e. amplitude change from the beginning to the end of the pulse ON time

2 = "Base Power", i.e. high attenuation; typically 100 dB

3 = Pulse top power corresponds to 1000 mW ("Pulse Envelope = Watt"), i.e. little or no attenuation ("Attenuation Top Power = 0 dB")

4 = Reference value used by the calculation of the pulse level parameters: "Ref. = 0 dB (1 Watt)"; the value is set with the parameter Level

5 = Overshoot level, in volt (diagram uses Watts)

6 = "Decay", i.e. damping constant that influences the time it takes the overshoot to decay

7 = Amplitude of the oscillating signal, in volt (diagram uses Watts)

8 = Signal oscillating with the selected frequency ("Frequency = 10 kHz", i.e. the oscillation period is 100 us)

9 = Pulse amplitude units (Y-axis units)

10 = Pulse timing: "Profile > Standard 0/100", "Rising Edge = Falling Edge = 100 us", "Width = 500 us" (see Standard Timing Profile)

Settings:

Use the following parameters to define the power/amplitude pulse parameters:

Attenuation

Simulates changes in the pulse amplitude.

"Top Power"	The pulse top power is used as a reference (100%) to determine other parameter values such as the rising/falling thresholds. Typically, the "Top Power" uses a little or no attenuation, such as 0 dB. See also Figure 6-1.
"Base Power"	The pulse base power is used as a reference (0%) to determine other parameter values such as the rising/falling thresholds. Typically, the "Base Power" uses a high attenuation, such as 100 dB. You can change this value if you use a "0/100" profile. The difference between the "Top Power" and the "Base Power" is the pulse amplitude.
"Droop"	Droop is a measure for the amplitude change from the beginning to the end of the pulse. The percentage ratio values are calculated in %W. A positive value decreases the amplitude.

Figure 6-5: Illustration of the effect of the amplitude droop (Profile = 0/100)

1 = "Droop = 10% (W)"

Remote command:
PULSe:LEVel:ON on page 700
PULSe:LEVel:OFF on page 700
PULSe:LEVel:DROop on page 700

Overshoot

Simulates that the signal exceeds the pulse "Top Power".
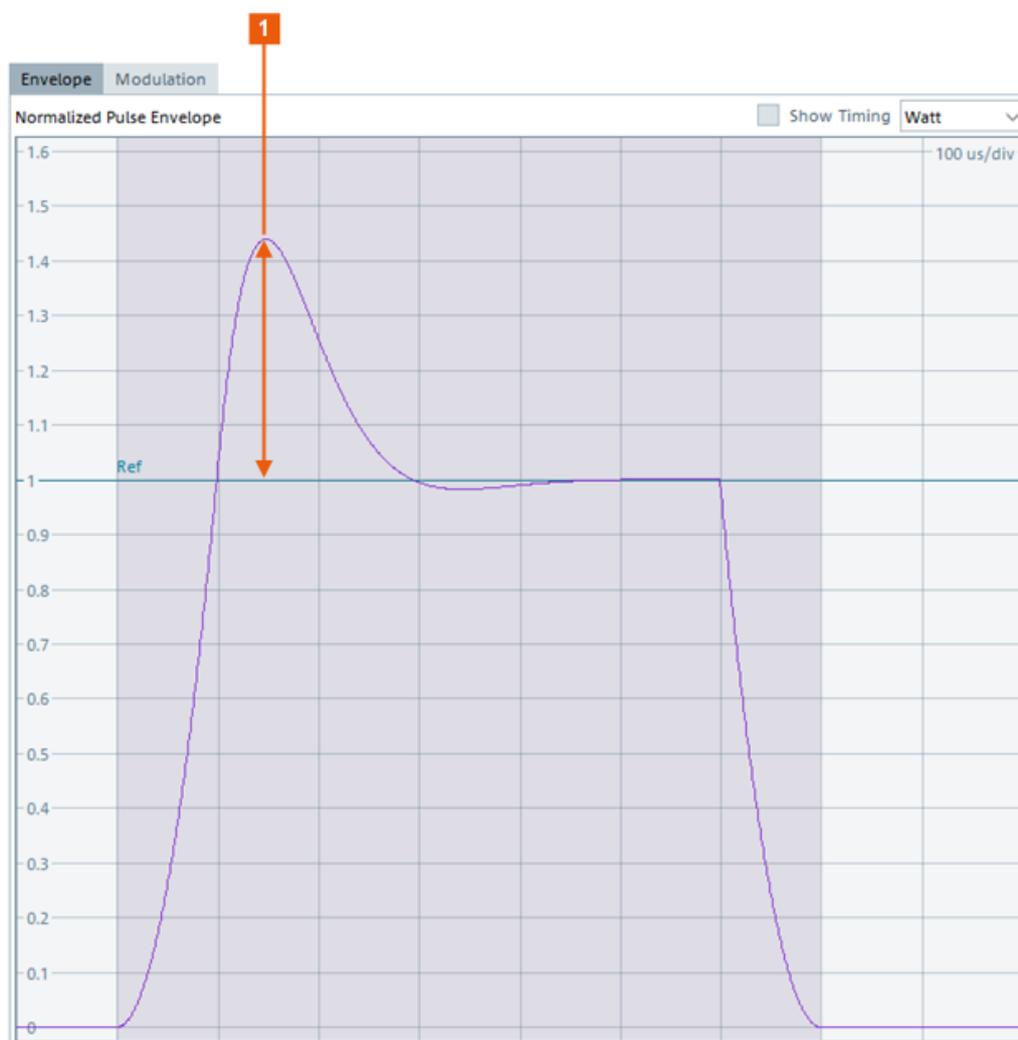
Figure 6-6: Illustration of the effect of the overshoot parameters (Profile = 0/100)

1 = "Overshoot = 20%" (value in percentage related to volts)

"Level"	The height of the local maximum (Max. power) after a rising edge, divided by the pulse amplitude, see Figure 6-1.
"Decay"	Sets the normalized decay constant. It influences the time it takes the overshoot to decay. "Decay = 1" is the aperiodic border case. The bigger the value the longer the decay time.

Remote command:
PULSe:OVERshoot on page 713
PULSe:OVERshoot:DECay on page 713

Ripple

Simulates the unwanted ringing effect, i.e. that the signal oscillates around the power level. This oscillation is defined with a frequency and level.
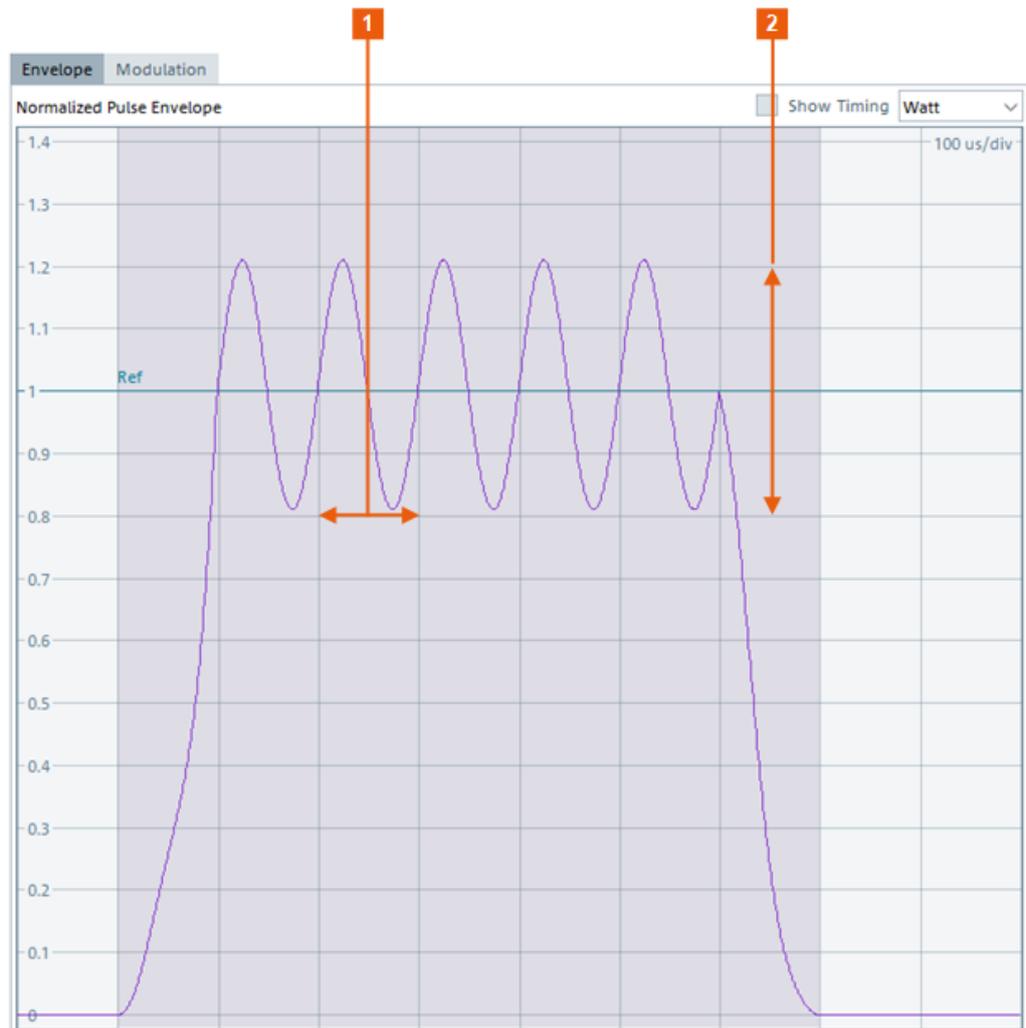
Figure 6-7: Illustration of the effect of the ripple parameters (Profile = 0/100)

1 = "Level = 20%" (value in percentage related to volts)
2 = "Frequency = 10 kHz", i.e. the oscillation period is 100 us

"Level"	The ripple is calculated as the difference between the maximum and minimum deviation from the pulse top reference, within a user specified interval.
	The percentage ratio values are calculated in %W, if the "Pulse Envelope" is defined in W, otherwise in %V.
"Frequency"	Sets with which frequency the ripple oscillates.

Remote command:
`PULSe:RIPPle` on page 713
`PULSe:RIPPle:FREQuency` on page 713

6.2.4 Pulse envelope settings

Instead of defining a pulse by its rise, on and fall time you can use your custom envelope shape.
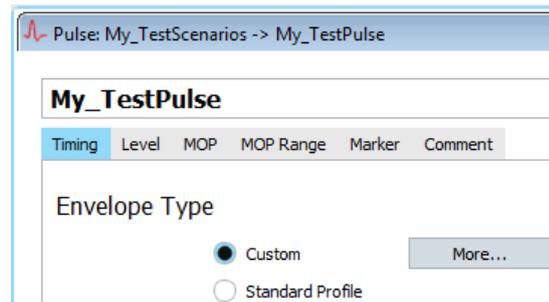
Custom envelopes are described in one of the following ways:

- As an equation, see "Envelope Definition as an Equation" on page 159
- As a function of imported I/Q data, see "Envelope definition as a function of imported data" on page 160

Both ways affect the level versus time and can be used with any kind of modulation (MOP). Custom envelopes are multiplied with the existing pulse shape as defined with the timing parameters. The software maps the custom envelope to the pulse shape.

Settings:

Envelope Definition as an Equation
Envelope is described by an equation, as selected with the parameter "Preset".
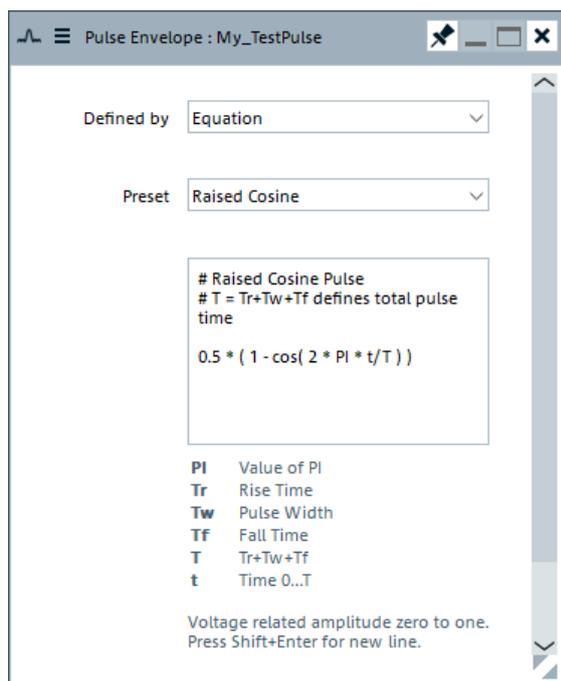
The software provides a selection of envelope shapes. Upon selection, the dialog displays the used equation and information on the used parameters and variables.

For description of the provided envelope functions, see "Equations and parameters used to define custom pulse envelopes" on page 861.

See also:
- "To generate a pulse with raised cosine envelope shape" on page 194
- "To define your custom envelope shape with an equation" on page 195

Remote command:
PULSe:ENVelope:EQUation on page 700

Envelope definition as a function of imported data
You can define an envelope function, store it in a file, and import and load files describing envelope functions.
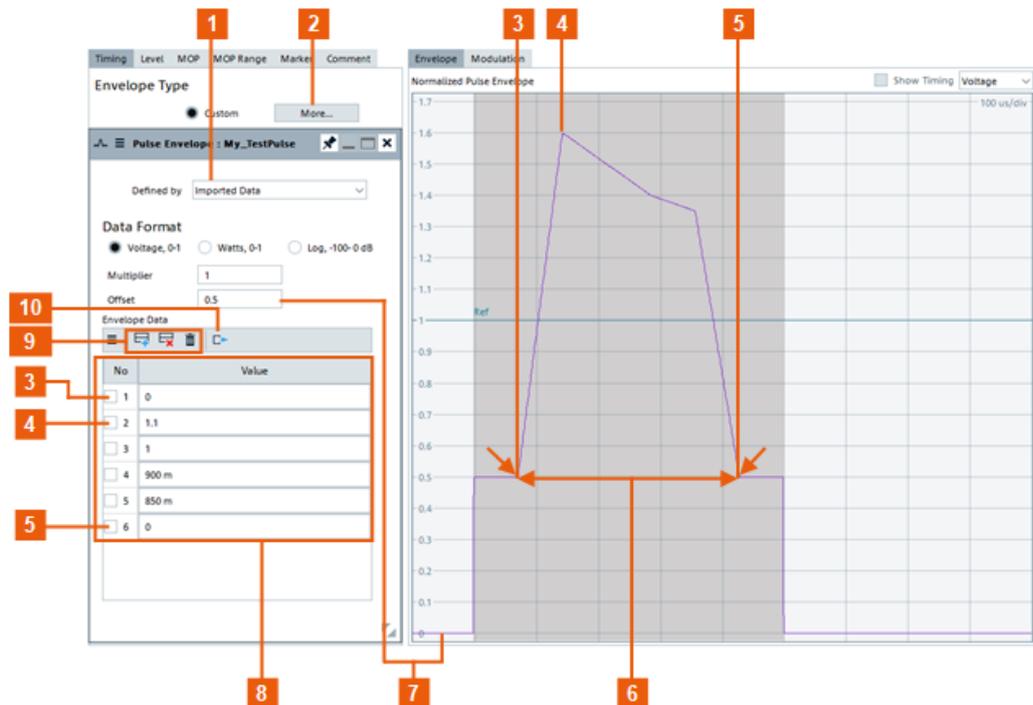
Figure 6-8: Custom Envelope > Use Imported Data: understanding the displayed information

1 = "Use Imported Data"
2 = "Custom Envelope"
3, 4, 5 = "Value", exemplary values on the y-axis
6 = The duration between the beginning of the rising edge and the end of the falling edge is 100 us + 500 us + 100 us = 700 us. I.e. the x-axis values of the six items are distributed equidistant, at each 100 us
7 = "Offset = 0.5"
8 = "Envelope Data" table, i.e. "Value" is the value on the y-axis
9 = Icons that access standard functions
10 = "Import data from file" access a dialog for loading of custom envelopes

"Envelope Data Table"

An envelope curve consists of two or more items. Each item is a line, defined by the y-value of end point; the start point is the end point of the previous item.

The x-axis values are calculated automatically so that the items are distributed equidistant between the beginning of the rising edge and the end of the falling edge.

"Data Format"   Selects the scaling used by the Envelope graph.

"Multiplier"   Applies a multiplier to the Y-values.

"Offset"   Offsets the envelope.

[icons]   Icons that access standard functions, like adding or deleting a row (see also Table 2-4)

Use the functions in the context-sensitive menu to append new item or reorder the existing item.

⚐"Import data from file"

Standard file management function to load custom envelopes from and as an ASCII file with `*.txt` or `*.csv` file extension.

When the file is loaded, the "Data Import" dialog assists you to select the used a delimiter and separator to interpret the file content.
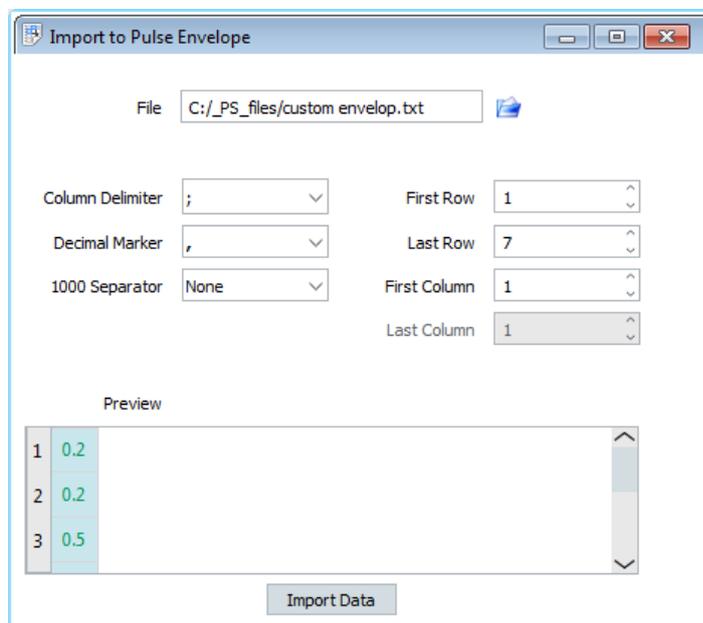
The provided settings are self-explanatory.

Remote command:

PULSe:ENVelope:DATA:ITEM:ADD on page 591
PULSe:ENVelope:DATA:ITEM:SELect on page 593
PULSe:ENVelope:DATA:ITEM:DELete on page 594
PULSe:ENVelope:DATA:ITEM:VALue on page 698
PULSe:ENVelope:DATA:UNIT on page 699
PULSe:ENVelope:DATA:MULTiplier on page 699
PULSe:ENVelope:DATA:OFFSet on page 699
PULSe:ENVelope:DATA:LOAD on page 698
PULSe:ENVelope:DATA:CLEar on page 596

6.2.5 Modulation on pulse (MOP) settings

Access:

1. Select "Pulse > MOP".

2. Select "Enable Modulation on Pulse (MOP)".

Figure 6-9: MOP: understanding the displayed information

1 = Enables that the pulse signal is modulated
2 = "Comment"
3 = "MOP Type" selects the modulation scheme
4 = Modulation parameters, depending on the selected "MOP Type"
5 = "Data source and Filter" settings (if required by the selected "MOP Type")

Further available settings depend on the selected modulation.

- AM and ASK modulation
- FM and FSK modulation
- Chirp modulation
- Phase modulation
- Vector modulation

- Noise
- Plug-ins

The Modulation graph displays the characteristics of the currently used MOP.

Some MOP types require a data source. For description of the related settings, see Chapter 6.2.5.9, "Data source, coding, filter", on page 184.

6.2.5.1 Common MOP parameters

Enable

Enables/disables that the pulse signal is modulated. If the MOP is disabled, the software generates an envelope signal based on the pulse level and timing parameters.

Remote command:
PULSe:MOP:ENABle on page 707

Comment

Optional description.

Remote command:
PULSe:MOP:COMMent on page 590

MOP Type

Selects a modulation scheme.

Further available settings depend on the selected modulation.

Remote command:
PULSe:MOP:TYPE on page 710

6.2.5.2 AM and ASK modulation

The amplitude modulation (AM) and amplitude shift key modulation (ASK) are modulation schemes that vary the amplitude of the signal.

Settings:

AM

Enables an AM with a single tone.

"Type"   Modulation method, like a double-sideband ("Standard"), a low side-band ("LSB"), an upper sideband ("USB"), or a double sideband with suppressed carrier ("LSB+USB") amplitude modulation.

"Frequency"   Modulation frequency.

"Mod. Depth"

                         Depth of the modulation signal in percent, i.e. the peak change in the RF amplitude from its unmodulated value to the amplitude of the unmodulated carrier.

Remote command:

PULSe:MOP:AM:TYPE on page 701

PULSe:MOP:AM:FREQuency on page 700

PULSe:MOP:AM:MDEPth on page 701

AM Step

Defines a modulation as a sequence of discrete states (table rows), each described with its "Duration" and "Level".

Use the standard "Append", "Remove Last", or "Delete All" functions to add or remove a row. The provided settings are self-explanatory.
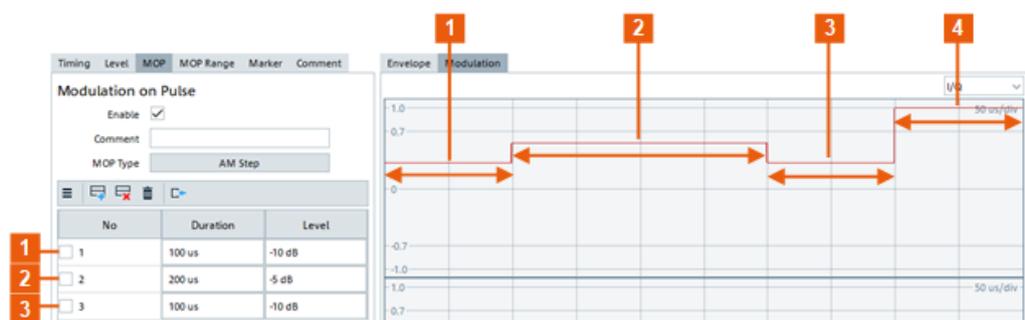
Figure 6-10: MOP Type > AM Step: understanding the displayed information (Pulse Width = 500 us, Restrict MOP to = Pulse Width)

1, 2, 3 = Three discrete states with different attenuation values and a total duration of 400 us
4   = The last 100 us are not attenuated

If the total time of the defined states is shorter than the duration of the pulse area on which the MOP is applied, an attenuation of 0 dB is assumed for the remaining time.

Remote command:
PULSe:MOP:AMSTep:ADD on page 591
PULSe:MOP:AMSTep:COUNt? on page 592
PULSe:MOP:AMSTep:SELect on page 593
PULSe:MOP:AMSTep:INSert on page 594
PULSe:MOP:AMSTep:LEVel on page 701
PULSe:MOP:AMSTep:DURation on page 701
PULSe:MOP:AMSTep:CLEar on page 596
PULSe:MOP:AMSTep:DELete on page 594

ASK

Enables an ASK where the amplitude of the RF carrier is attenuated for a symbol value of zero and remains at full level for symbol values of one. The level of attenuation is specified as depth in percent.

My_TestPulse

| Timing | Level | MOP | MOP Range | Marker | Comment |

Modulation on Pulse

Enable ✓

Comment

MOP Type: ASK

Mod. Depth: 90 %

Symbol Rate: 500 kHz

☐ Inverted

"Mod. Depth"

Depth of the modulation signal in percent.
$m = (Amplitude_{max} - Amplitude_{min}) / (Amplitude_{max} + Amplitude_{min})$

"Symbol Rate"   Symbol rate of the modulating signal.

"Inverted"   Inverts the amplitude-shift keying (ASK) modulation.
Meaning the RF carrier is attenuated for a symbol value of one.

"Data Source, Coding, Filter"

See:
- Chapter 19.1, "Data sources settings", on page 526
- "Coding" on page 184
- "Filter" on page 185

Remote command:
PULSe:MOP:ASK:MDEPth on page 702
PULSe:MOP:ASK:INVert on page 702
PULSe:MOP:ASK:SRATe on page 702

6.2.5.3 FM and FSK modulation

The frequency modulation (FM) and frequency shift keying (FSK) modulation are modulation schemes that vary the frequency of the signal.

Settings:

FM

Enables an FM with a single tone.

"Frequency"   Modulation frequency

"Deviation"   FM frequency deviation

Remote command:

PULSe:MOP:FM:FREQuency on page 710

PULSe:MOP:FM:DEViation on page 710

FM Step

The FM step modulation is a common pulse compression technique. It defines a modulation as a sequence of discrete states (table rows), each described with its "Duration" and "Frequency".

Use the standard "Append", "Remove Last", or "Delete All" functions to add or remove a row. The provided settings are self-explanatory.

Example: Sequence composed of four pulses with "MOP = FM Step"

- Configure an FM step with f_{start} = -50 MHz, f_{end} = 50 MHz, Step = 25 MHz, #Steps = 5.
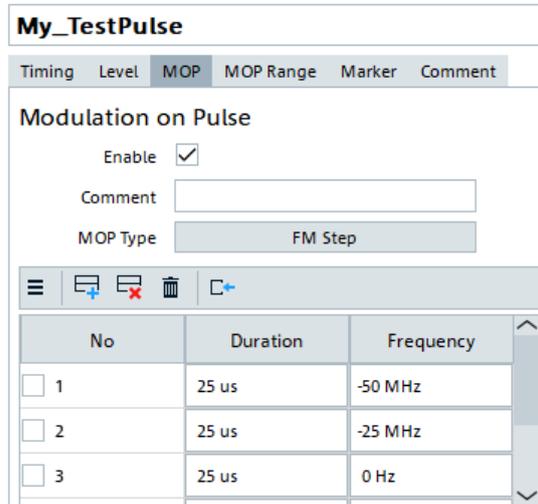
- Build a sequence and use the "Signal Preview" dialog to visualize the signal.
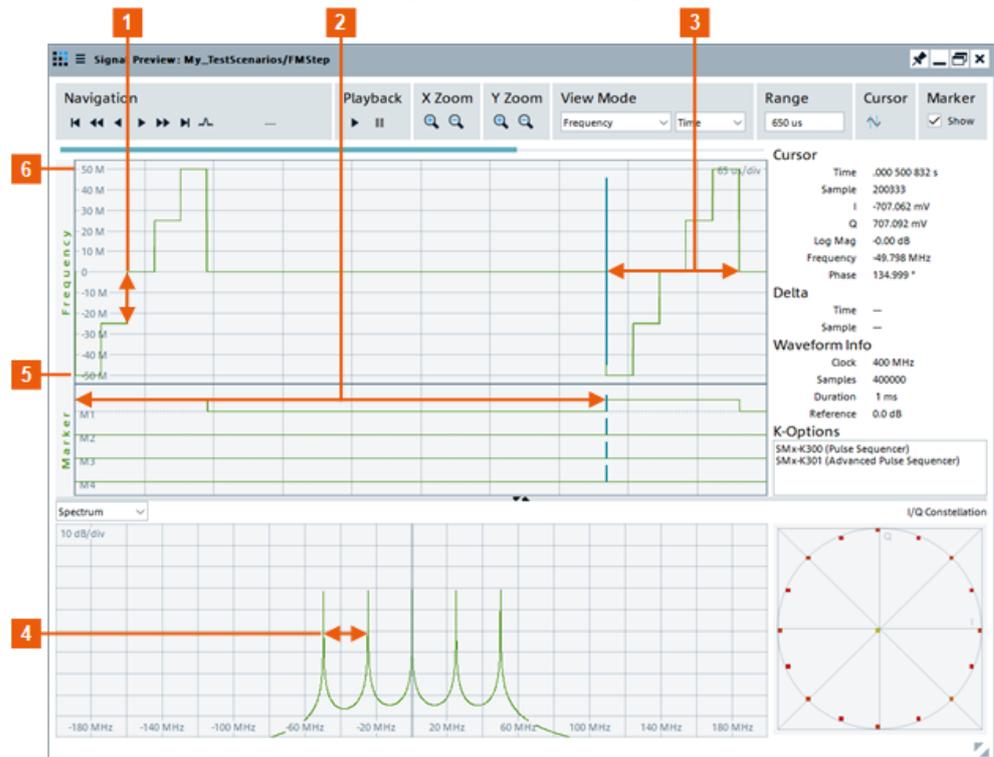
1 = Step = 25 MHz
2 = PRI = 500 us
3 = Pulse width = 125 us
4 = Frequency spacing = 25 MHz
5, 6 = Start and stop frequency

Remote command:

PULSe:MOP:FMSTep:ADD on page 591

PULSe:MOP:FMSTep:COUNt? on page 592

PULSe:MOP:FMSTep:SELect on page 593

PULSe:MOP:FMSTep:INSert on page 594

PULSe:MOP:FMSTep:FREQuency on page 702

PULSe:MOP:FMSTep:DURation on page 701

PULSe:MOP:FMSTep:CLEar on page 596

PULSe:MOP:FMSTep:DELete on page 594

FSK, 4FSK, 8FSK, 16FSK, 32FSK, 64FSK

Enables an FSK. High symbols set the frequency to "+Deviation" and low symbols to "–Deviation".
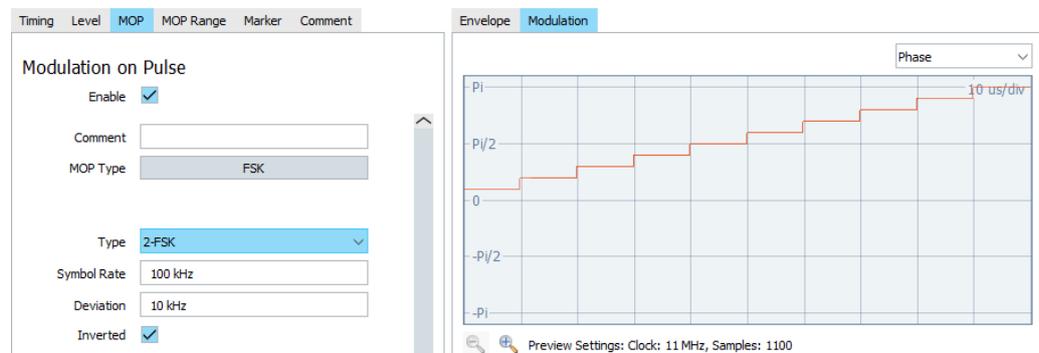
"Type"	Selects the FSK modulation type.
"Symbol Rate"	Symbol rate of the modulating signal.
"Deviation"	FSK frequency deviation
"Inverted"	Inverts the FSK modulation.
	Observe the effect in the Modulation graph.

"Data Source, Coding, Filter"

See:

- Chapter 19.1, "Data sources settings", on page 526
- "Coding" on page 184
- "Filter" on page 185

Remote command:

PULSe:MOP:FSK:TYPE on page 705

PULSe:MOP:FSK:DEViation on page 710

PULSe:MOP:FSK:SRATe on page 711

PULSe:MOP:FSK:INVert on page 702

PULSe:MOP:4FSK:DEViation on page 710

PULSe:MOP:4FSK:SRATe on page 711

PULSe:MOP:8FSK:DEViation on page 710

PULSe:MOP:8FSK:SRATe on page 710

MSK

Enables an MSK.

"Symbol Rate"   Symbol rate of the modulating signal.

"Inverted"   Inverts the MSK modulation.
Observe the effect in the Modulation graph.

"Data Source, Coding, Filter"
See:
- Chapter 19.1, "Data sources settings", on page 526
- "Coding" on page 184
- "Filter" on page 185

Remote command:
`PULSe:MOP:MSK:SRATe` on page 702
`PULSe:MOP:MSK:INVert` on page 702

6.2.5.4 Chirp modulation

A chirp is a signal in which the frequency varies over the time. The chirp sweeps the RF signal across a set frequency range.

The chirp modulation scheme is used for pulse compression.

Settings:

Linear Chirp
The linear chirp varies the frequency linearly over time.

"Type"   Defines the frequency variation:
- Linearly increasing ("Up")
- Linearly decreasing ("Down")
- Following a full sine wave ("Sine")
- Ascending and then descending ("Triangular")

"Deviation"   Chirp deviation.
The "Total Deviation" value updates automatically if you change the "Deviation".

Remote command:
PULSe:MOP:CHIRp:TYPE on page 705
PULSe:MOP:CHIRp:DEViation on page 705

Piecewise Linear Chirp
Define the chirp as a sequence of chirp intervals with user-definable length and frequency deviation.
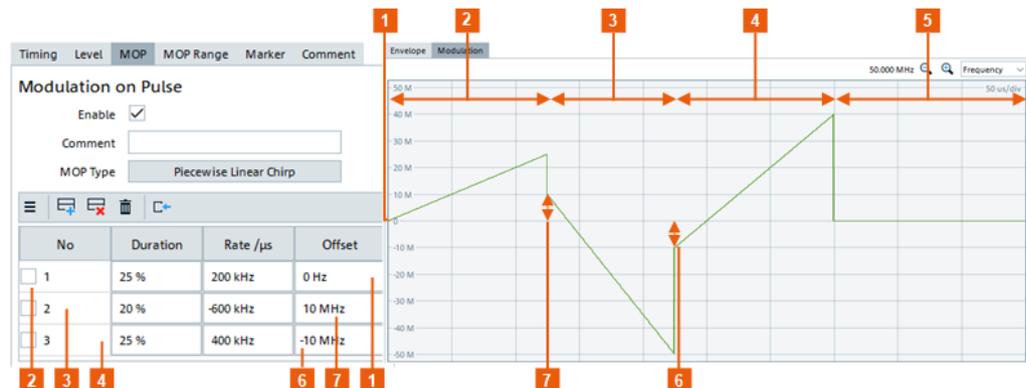
Figure 6-11: Piecewise Linear Chirp: understanding the displayed information

1   = Offset = 0 Hz; the start chirp frequency is 0 Hz
2, 3, 4 = Three chirp intervals with total duration of 70% of the whole time of which the MOP is applied[*]
5   = In the remaining 30% of the chirp time, the frequency is kept constant
6   = Negative offset; the start chirp frequency in the third interval is -10 MHz
7   = Positive offset; the start chirp frequency in the second interval is 10 MHz
*   = Theoretical example, in practice you rather vary the frequency during the whole time

"Duration"   Set the length of each chirp interval as a percentage of the whole duration to which the MOP is applied.
For example, if a pulse width of 100 us and rising/falling edge of 0 us is used and the MOP is applied without restrictions, the whole duration is 100 us.
If the sum of all intervals is less than 100%, constant frequency is used in the remaining part.
If the sum of the defined intervals exceeds 100%, the parts outside the 100% are ignored.

Remote command:
PULSe:MOP:PIECewise:DURation on page 705

"Rate/µs"   Set the chirp rate in Hz/µs.

Example:

If the MOP is applied on 100 us and a chirp with "Rate = 400 KHz/us" is applied for 25 us ([Duration = 25%]), then the frequency deviation during this interval is 10 MHz.

Remote command:
PULSe:MOP:PIECewise:RATE on page 705

"Offset"   Offsets the start frequency of the chirp in the selected interval. Meaning within the interval, the chirp is shifted with the defined value.

Remote command:
PULSe:MOP:PIECewise:OFFSet on page 706

Remote command:
PULSe:MOP:PIECewise:ADD on page 591
PULSe:MOP:PIECewise:COUNt? on page 592
PULSe:MOP:PIECewise:INSert on page 594
PULSe:MOP:PIECewise:CLEar on page 596
PULSe:MOP:PIECewise:DELete on page 595
PULSe:MOP:PIECewise:SELect on page 593
PULSe:MOP:PIECewise:DURation on page 705
PULSe:MOP:PIECewise:RATE on page 705
PULSe:MOP:PIECewise:OFFSet on page 706

Custom Chirp
The custom chirp modulation is defined as a sequence of discrete "Frequency" stages.
The duration of each stage depends on the pulse duration to which the MOP is applied and the number of defined frequency stages.

"New/Select, Append, Insert, Delete, Clear"   Icons that access standard functions, like add or delete row (see also Table 2-4).
Use the functions in the context-sensitive menu to append new item or reorder the existing item.

"Import data from file"

Standard file management function to load a custom chirp from and as an ASCII file with `*.txt` or `*.csv` file extension.

When a file is loaded, the "Data Import" dialog assists you to select the used delimiter and separator to interpret the file content.
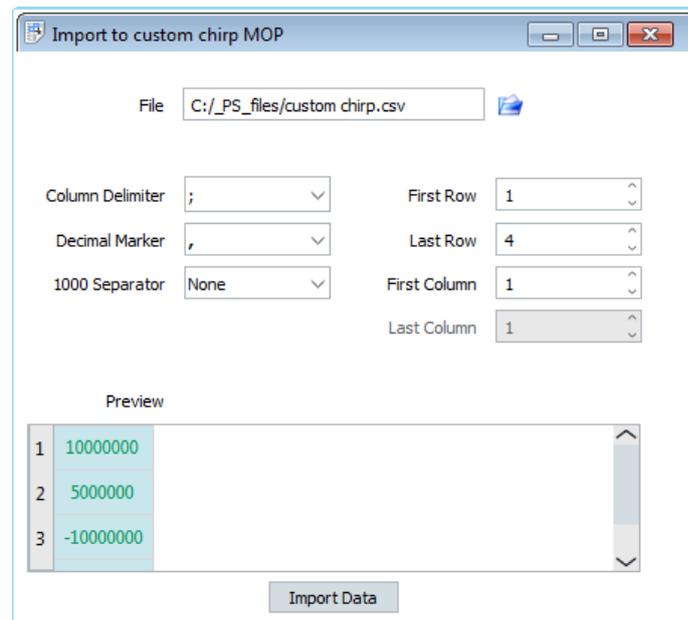
The provided settings are self-explanatory.

Remote command:
PULSe:MOP:CCHirp:ADD on page 591
PULSe:MOP:CCHirp:INSert on page 594
PULSe:MOP:CCHirp:COUNt? on page 592
PULSe:MOP:CCHirp:SELect on page 593
PULSe:MOP:CCHirp:FREQuency on page 706
PULSe:MOP:CCHirp:DELete on page 595
PULSe:MOP:CCHirp:CLEar on page 596

Equation

Define your custom chirp mathematically as a function of time, e.g. *1e6*(-1+2*t/T)*.

Select "Update Preview" to visualize the signal.

See also Chapter B, "Formula syntax", on page 860.

Remote command:
PULSe:MOP:NLCHirp:EQUation on page 711

Polynomial Chirp

Define the chirp as a polynomial with one or more "Term"/"Coefficient" value pairs.

The instantaneous frequency versus time *f(t)* is calculated according to the equation:

$f(t) = \sum (a_n * t^n)$, where:

- *n = 1 .. 32* is the term
- $a_n = - 10^{22}$ to 10^{22} are the coefficients.

Figure 6-12: Polynomial Chirp: understanding the displayed information

1 = First term n = 0
2 = Second term n = 2 (the coefficient a_1 is not used)
3 = Coefficient a_0 = 3e6 = 3 MHz
4 = Coefficient a_2 = 2.5e14

The polynomial chirp defined with the settings on Figure 6-12 is:

$f(t) = a_0 + a_2 \cdot t^2 = 3e6 + 2.5e14 \cdot t^2$.

Remote command:
`PULSe:MOP:PCHirp:ADD` on page 591
`PULSe:MOP:PCHirp:COUNt?` on page 592
`PULSe:MOP:PCHirp:SELect` on page 593
`PULSe:MOP:PCHirp:INSert` on page 594
`PULSe:MOP:PCHirp:TERM` on page 711
`PULSe:MOP:PCHirp:COEFficient` on page 711
`PULSe:MOP:PCHirp:CLEar` on page 596
`PULSe:MOP:PCHirp:DELete` on page 595

6.2.5.5 Phase modulation

Provided are the following phase modulation technics:

Barker

Phase modulation with Barker codes results in signals with low autocorrelation properties.
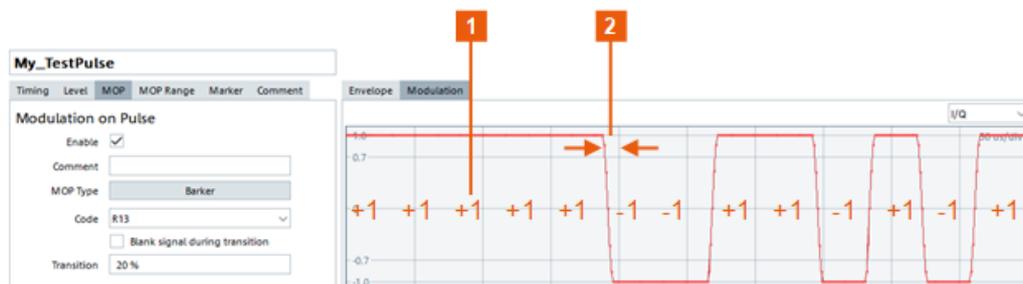
Figure 6-13: MOP Type > Barker: understanding the displayed information

1 = Barker sequence "Code = R13" (not displayed in the user interface but added for better explanation)
2 = Transition time for "Transition = 20%" ("Blank signal during transition > Off")

Barker codes are used for pulse compression and are defined by:

"Code"   A Barker code is a finite sequence of N values of +1 and −1; R13, for example, is the '+1+1+1+1+1-1-1+1+1-1+1-1+1' sequence.

"Blank signal during transition"

If "Transition > 0%", blanks out the signal during the transition time. Observe the effect on the modulation envelope, the displayed spectrum, and on the I/Q constellation diagram.
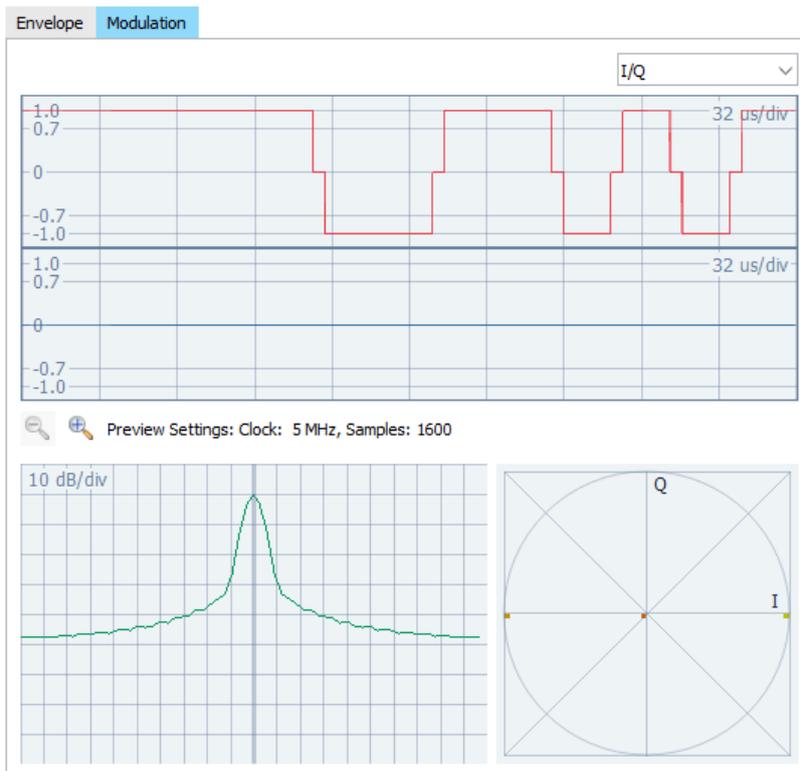
Figure 6-14: Effect of the Blank signal during transition = On (MOP Type = Barker, Code = R13, Transition = 20%)

"Transition"   Sets the transition duration for constant envelope operation.
A "Transition = 0%" is the border case, where the change for +1 to -1 or conversely is performed immediately. The transition skips (switches back and forth on a direct line) between the constellation points.
With a "Transition > 0%", the two constellation points are still exactly opposite, but the transition follows a circular arc. The values on the transition curve are uniformly distributed.
Observe the effect of a "Transition > 0%" on the modulation envelope, the displayed spectrum, and on the I/Q constellation diagram. The greater the value, the slower the transition, the longer the transition time.

"Transition = 0%"	"Transition = 20%"
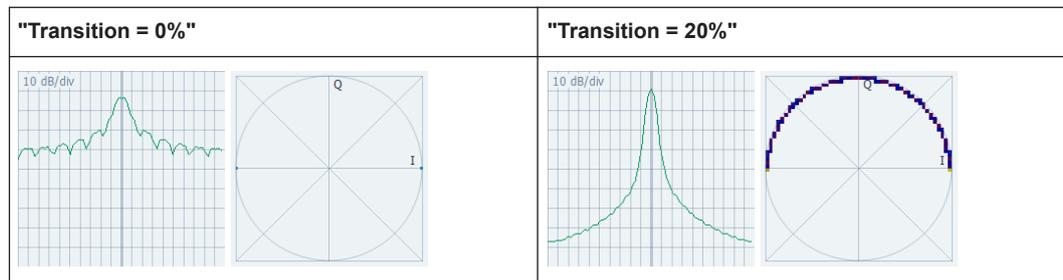
Remote command:
PULSe:MOP:BARKer:CODE on page 703
PULSe:MOP:BARKer:TTIMe on page 703
PULSe:MOP:BARKer:BLANk on page 703

Poly Phase

A polyphase code modulation is suitable for pulse compression and is used in low probability of intercept (LPI) radars.

"Type"   Selection of polyphase compression codes: the default Frank code, and four modified versions of the Frank code, the P1, P2, P3 and P4 codes.

"Length M"   Sets the code order, i.e. the number of elements.

Remote command:
PULSe:MOP:POLY:TYPE on page 712
PULSe:MOP:POLY:LENGth on page 712

Custom Phase

Defines a sequence of phase values. The phase changes are distributed evenly.

Use the standard "Append", "Remove Last", or "Delete All" functions to add or remove a row. The provided settings are self-explanatory.
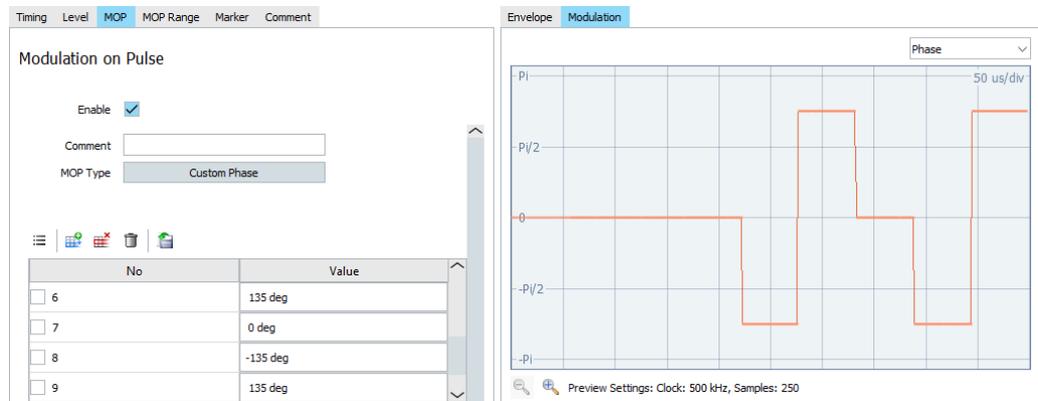
Remote command:
PULSe:MOP:PLISt:ADD on page 591
PULSe:MOP:PLISt:COUNt? on page 592
PULSe:MOP:PLISt:SELect on page 593
PULSe:MOP:PLISt:INSert on page 594
PULSe:MOP:PLISt:VALue on page 712
PULSe:MOP:PLISt:CLEar on page 596
PULSe:MOP:PLISt:DELete on page 595

BPSK

A binary phase shift keying (BPSK) modulation applies a phase shift with a selected offset ("Phase") at each constellation point.

"Type"   Selects the BPSK modulation method, like a standard BPSK modulation or a BPSK modulation with a constant envelope (C-BPSK).

"Symbol Rate"   Symbol rate of the modulating signal, user defined or automatically estimated.

"Phase"   Phase shift between the constellation point.

"Transition"

> These parameters are important, when a BPSK modulation with a constant envelope is used.
> A "Transition = 0%" is a border case, where the change between two constellation points is performed immediately. The transition skips (switches back and forth on a direct line) between the constellation points.
> The greater the value, the slower the transition. With a "Transition > 0%", the two constellation points are still exactly opposite, but the transition follows a circular arc.
>
> The "Transition Function" defines how the values are distributed on the transition curve.
> - "Linear": all values are uniformly distributed in the transition
> - "Cosine": there are fewer values in the middle of the transition and accumulated values near to the two end constellation points.

"Data Source, Coding, Filter"

> See:
> - Chapter 19.1, "Data sources settings", on page 526
> - "Coding" on page 184
> - "Filter" on page 185

Remote command:

PULSe:MOP:BPSK:TYPE on page 704

PULSe:MOP:BPSK:SRATe on page 704

PULSe:MOP:BPSK:SRATe:AUTO on page 704

PULSe:MOP:BPSK:PHASe on page 703

PULSe:MOP:BPSK:TTYPe on page 704

PULSe:MOP:BPSK:TTIMe on page 704

QPSK

A quadrature phase shift keying (QPSK) modulation applies a phase shift at each constellation point.

MOP Type	QPSK

Type	Standard
Symbol Rate	100 kHz

"Type"   Selects the QPSK modulation method.

- "Standard"
  A regular QPSK modulation.
- "OQPSK, DQPSK"
  An offset QPSK ("OQPSK"), or a differential QPSK "DQPSK" modulation.
- "SOQPSK-A, SOQPSK-B, SOQPSK-TG"
  Three variants of the shaped-offset QPSK ("SOQPSK") modulation.
  For the aeronautical telemetry "SOQPSK-TG" modulation, you can enable differential encoding according to the telemetry standard IRIG 106-04.

Remote command:
`PULSe:MOP:QPSK:TYPE` on page 712
`PULSe:MOP:QPSK:SOQPsk:IRIG` on page 712

"Symbol Rate"

Symbol rate of the modulating signal.

Remote command:
`PULSe:MOP:QPSK:SRATe` on page 712

8PSK
An 8 phased shift keying modulation applies a phase shift at each constellation point.

"Symbol Rate"

Symbol rate of the modulating signal.

Remote command:
`PULSe:MOP:8PSK:SRATe` on page 710

6.2.5.6   Vector modulation

Provided are the following QAM modulations:

16QAM, 32QAM, 64QAM, 128QAM, 256QAM
A quadrature amplitude modulation QAM applies an amplitude shift at each constellation point.

"Type"   Selects the QAM type.

"Symbol Rate"

                   Symbol rate of the modulating signal.

                   Remote command:
                   PULSe:MOP:QAM:TYPE on page 706
                   PULSe:MOP:QAM:SRATe on page 711

6.2.5.7 Noise

Generates an AWGN noise signal with selected "Bandwidth".

The noise is superimposed during all pulse phases.

Remote command:

PULSe:MOP:NOISe:BWIDth on page 711

6.2.5.8 Plug-ins

Custom modulation schemes are used in the same way as the internal modulation schemes.

See Chapter 20, "Defining complex modulation schemes and IPM profiles", on page 530.

Settings:

Plug-in,
Selects and loads a user-defined modulation from a file.

The modulation must exist in the "Plug-in" library.

Use the standard context menu functions to:
* Reset the plug-in variables
* Access the load plug-in settings, see Chapter 20, "Defining complex modulation schemes and IPM profiles", on page 530

Remote command:
PULSe:MOP:PLUGin:NAME on page 589

Plug-in Variables
Resets all variables to the preset values defined in the plug-in.

Remote command:
PULSe:MOP:PLUGin:VARiable:CATalog? on page 689
PULSe:MOP:PLUGin:VARiable:SELect on page 689
PULSe:MOP:PLUGin:VARiable:VALue on page 689

Data Source, Coding, Filter, Comment

See:

- Chapter 19.1, "Data sources settings", on page 526
- "Coding" on page 184
- "Filter" on page 185

6.2.5.9   Data source, coding, filter

Data Source and Filter

Data Source	PRBS test
	☑ Reset Data Source at End of Pulse
Coding	Gray
Filter	Cosine
Symbols	10
Roll Off	1

Settings:

The following settings are available for a subset of the provided MOP types:

Data Source

Selects the data source for the custom modulation.

Create the data source first. See Chapter 19, "Defining the data source", on page 523 for description of the provided settings.

"Restart Data Source at the End of Pulse"	Per default, data is retrieved cyclically. With this parameter, you can activate data source restart at the end of each pulse.

Remote command:

PULSe:MOP:DATA:DSRC on page 707

PULSe:MOP:DATA:DSRC:RESet on page 707

Coding

Coding is a technique used to improve the signal properties and signal reception. In general, the coding schemes are applied before the modulation. The modulation symbols are coded directly before I and Q values or frequency shifts are assigned.

This software supports the differential and gray coding and the combination of both.

For more information, see Chapter E, "Common coding algorithms", on page 893.

Remote command:

PULSe:MOP:DATA:CODing on page 707

Filter

Filters are applied to shape the signal before it is modulated on the RF. The selected filter type and shape affect the output signal, especially while generating broadband signals.

This software supports the following filter types: "Rectangular", "Gauss", "Gauss FSK", "Cosine", "Root Cosine", "Lowpass", "SOQPSK" and "Rectangular (R&S SMW)".

Depending on the selected filter type, one or more filter parameters are provided. You can adjust the filter characteristics to select steeper edges, to customize the transition bandwidth, or to adjust the filter length.

For more information, see Chapter F, "Supported filter and impact of the filter parameters", on page 894.

Remote command:
PULSe:MOP:FILTer:TYPE on page 709
PULSe:MOP:FILTer:BT on page 708
PULSe:MOP:FILTer:BWIDth on page 709
PULSe:MOP:FILTer:LENGth on page 709
PULSe:MOP:FILTer:ROLLoff on page 709

6.2.6 MOP range

Access:

► Select "Pulse > MOP Range".

Restrict MOP Range

Customizes (restricts) the pulse area on that the MOP is applied.

With its default settings ("Configure > Colors > Preset > R&S Standard"), the software indicates (highlights) the pulse area on that the MOP is applied. See Figure 6-15 and compare the colors used to indicate the pulse (gray) and the pulse area on that MOP is applied (dark gray).

See also Chapter 2.8.2, "Changing colors and default configuration", on page 49.
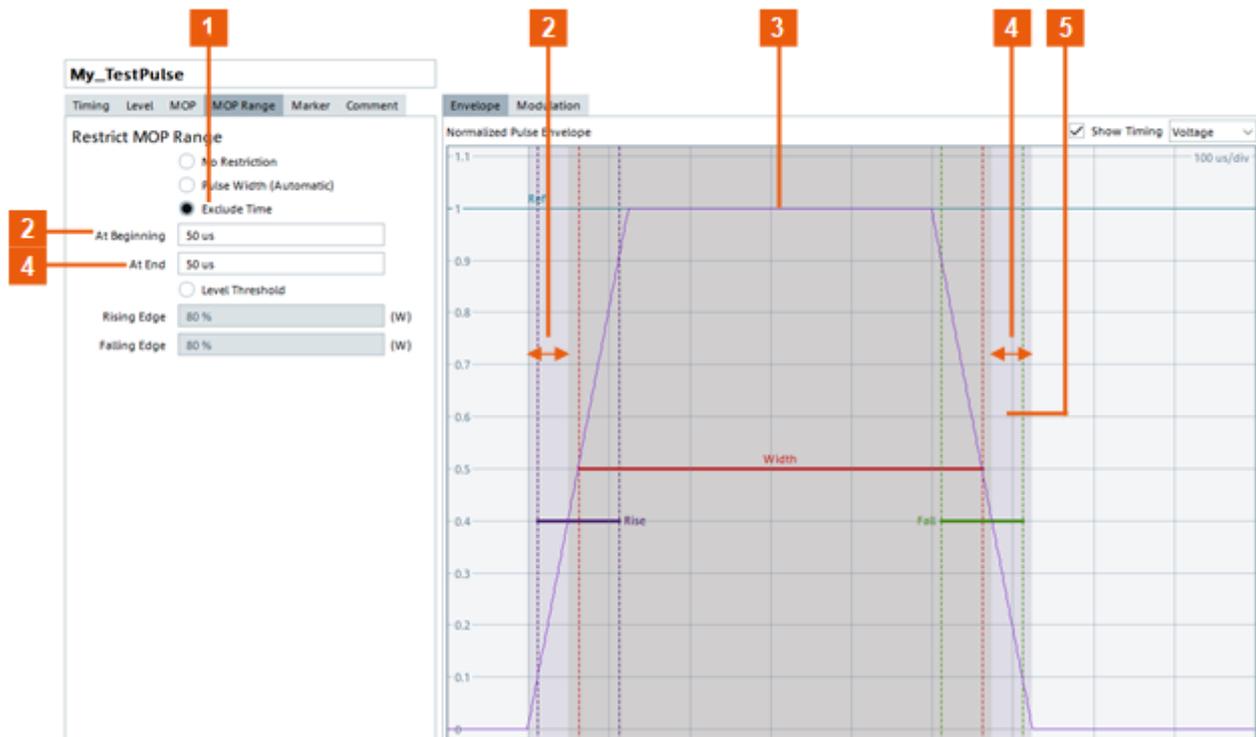
Figure 6-15: Restrict modulation to a certain area of the pulse: understanding the displayed information ("Timing Profile > Voltage 10/50/90")

1 = "Restrict modulation to a certain area of the pulse > Exclude Time"
2, 4 = Excluded are 50 us from the beginning of the rising edge and at the end of the falling edge. Note that the pulse area the MOP is applied on is wider than the pulse width
3 = Dark gray color: indicates the pulse area on that the MOP is applied
5 = Gray color: indicates the pulse duration

"No restrictions"
    The MOP is applied from the beginning of the rising edge until the end of the falling edge.

"Pulse Width (Automatic)"
    The MOP is applied during the time specified as pulse width time ("Pulse > Timing > Width").

"Exclude time"   Defines a time span to be excluded at the beginning of the rising edge and at the end of the falling edge.

"Level Threshold"
    The MOP is applied during the time the signal level is greater than the specified level limits.

Remote command:
PULSe:MOP:EXCLude:ENABle on page 707
PULSe:MOP:EXCLude:MODE on page 708
PULSe:MOP:EXCLude:TIME:STARt on page 708
PULSe:MOP:EXCLude:TIME:STOP on page 708
PULSe:MOP:EXCLude:LEVel:STARt on page 708
PULSe:MOP:EXCLude:LEVel:STOP on page 708

6.2.7 Envelope and Modulation graphs

Envelope graph

Displays the current pulse shape depending on the following:

- The selected Standard Timing Profile
- The Pulse Shape Settings settings
- Pulse envelope settings.

For example, see the envelope graph on Figure 6-4.

"Show Timing" Set "Show Timing > On" to visualize the pulse width, rise and fall time on the graph.
This visualization is useful if you are not familiar with the definition of the pulse parameters.
For a detailed description of the different profiles you can use, see also Chapter 6.1, "Basics on pulse signals and pulse generation", on page 148.

"Normalized Pulse Envelope > Pulse envelope unit"
Changes the used units.
The options "Voltage", "Watt" and three logarithmic scales with different resolution ("10 dB/Div", "5 dB/Div" and "1 dB/Div") are available.
To display the effect of large overshoots or ripples for example, use the "5 dB/Div" or "1 dB/Div" scale.

"Temporary delta information"
Drag the mouse cursor on the graph.
The delta information is displayed temporarily above the line.

Example: Measuring the pulse width with the help of delta marker
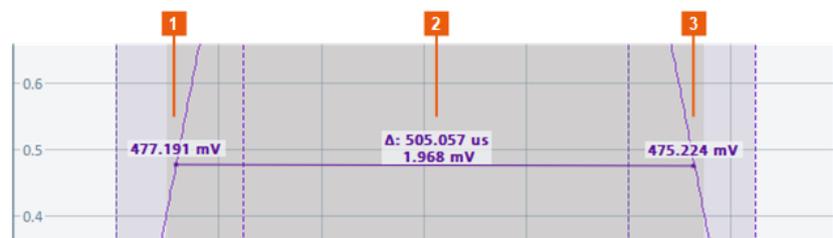
1 = Start y value [units depending on the selected "Pulse Envelope Units"]
2 = Delta information: upper value is the Δx [s], lower value is the Δ y [units depending on the selected "Pulse Envelope Units"]
3 = End y value [units depending on the selected "Pulse Envelope Units"]

Remote command:
PULSe:PREView:MODE on page 715

Modulation graph

Displays the current modulation characteristics depending on the selected MOP range.

The MOP is visualized on three diagrams: the time domain view, the spectrum, and the I/Q constellation diagrams.

Figure 6-16: Modulation graph: Understanding the displayed information (MOP Type > Polyphase code, Type = P2, Length M = 10)

1 = Zoom in/out on the x axis; y axis zoom is supported for the frequency display
2 = Indicated is the "Frequency" variation
3 = Time domain view
4 = FFT preview settings
5 = I/Q constellation diagram
6 = Spectrum

Time domain display ← Modulation graph

Use the "Modulation" parameter to change the displayed modulation characteristics.

Available are:

"I/Q diagram"   Displays the I and Q signal versus time. Both signals use a linear scale in the range between -1.0 and +1.0.
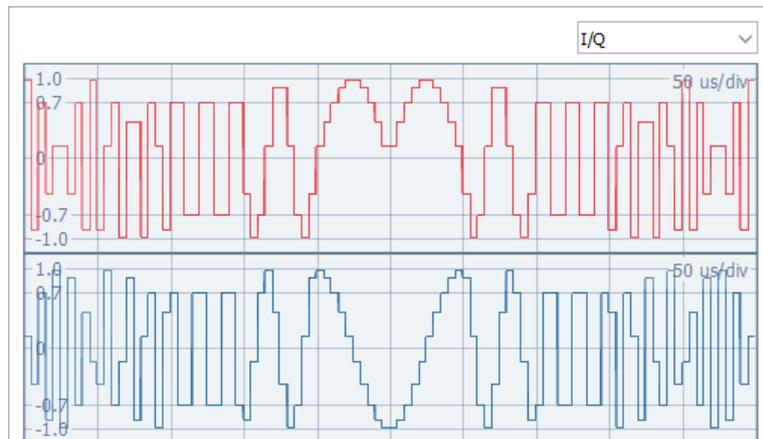
"Phase diagram"

Displays the phase angle of the signal versus time. The diagram shows the phase changes in the range -π and +π.
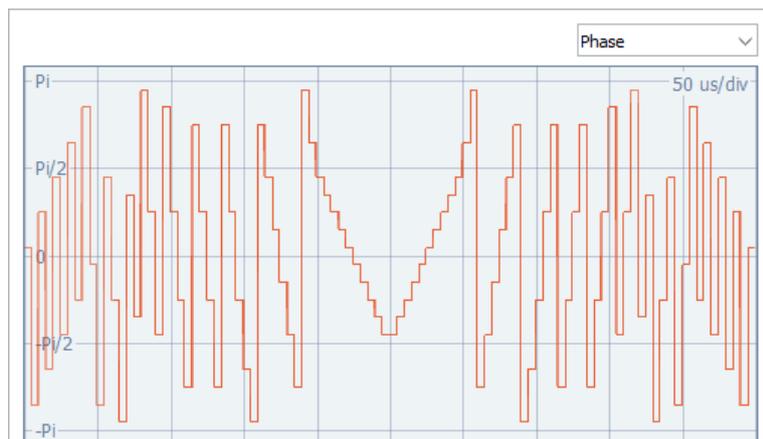
A phase change is equivalent to a rotation at a constant radius in the constellation diagram.

"Frequency diagram"

Displays the instantaneous signal frequency versus time.

Remote command:
PULSe:PREView:MOP on page 715

FFT preview settings ← Modulation graph
Displays information about the FFT preview and allows to set parameters:

"Max. Samples"

Displays the maximum value of samples possible.

"Zoom in/Zoom out"

Changes the resolution on the x and y axis.

"Clock, Samples"

Displays clock frequency and number of samples.

"Oversampling"

> Sets the oversampling factor in the modulation preview.

"FFT Size"   Sets the FFT size in the modulation preview.

"Envelope"   Activates the envelope in the modulation preview.

I/Q constellation diagram ← Modulation graph
Displays the probability at which I/Q points occur.

The color scale is logarithmic; the most probable points are indicated with bright yellow color.

Spectrum ← Modulation graph
Displays the FFT spectrum. The FFT uses a logarithmic scale between 0 dB and -100 dB.

6.3 How to create a new pulse and adjust its settings

Pulses are the fundamental building elements of any sequence and therefore have to be created before you can create a sequence.

See:

- "To create a pulse" on page 190
- "To define the timing parameters of a pulse" on page 191
- "To define the level parameters of a pulse" on page 192
- "To define and apply modulation on the pulses (MOP)" on page 193
- "To generate a pulse with raised cosine envelope shape" on page 194
- "To define your custom envelope shape with an equation" on page 195
- "To visualize the pulse characteristics" on page 197

To create a pulse

1. Perform one of the following:

   a) Select "Repository Tree > Pulse > New".
   b) In the "Sequence" dialog, select the pulse icon.

   A new pulse with default settings is created and added to the repository tree.
   New pulses are named `Pulse <n>` where n is a number starting at one.
   You can add information to describe and identify the pulse, like a name or a comment.

2. Change the pulse name, e.g. "P1".

   A pulse is described by its timing and level parameters, applied modulation on pulse (MOP) and enabled marker signals.
   For background information on the pulse parameters, see Chapter 6.1, "Basics on pulse signals and pulse generation", on page 148.

   For description of the required settings, see:
   - Chapter 6.2.2, "Pulse timing settings", on page 151

To define the timing parameters of a pulse

1. In the repository tree, select "Pulse > PulseName e.g. P1 > Timing".

2. Select "Standard Profile > On".

3. Select "Profile > Voltage 10/50/90".

4. Set the timing-related parameters, such as:

   - Delay
   - Rise/fall
   - On/off time
   - The shape of the rising and falling slopes

5. To configure the pulse repetition interval (PRI), use the parameter PRI/PRF.

6. Use the "Envelope" graph to visualize the current envelope profile.
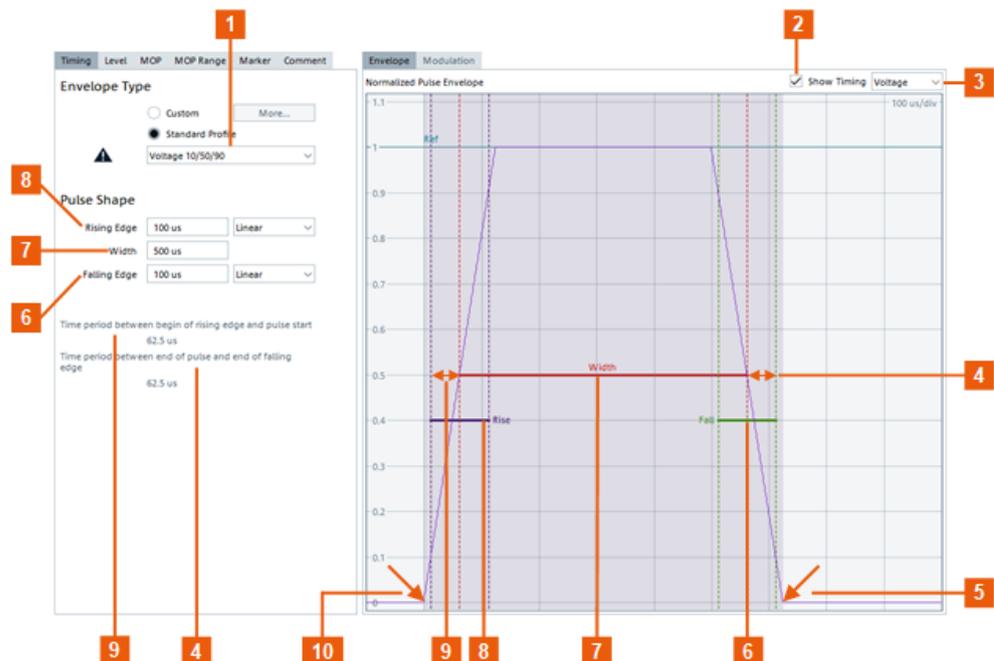
Figure 6-17: Pulse timing: understanding the displayed information

1 　　= "Standard Profile > Voltage 10/50/90"
2 　　= "Show Timing > On" indicates the pulse timing parameters on the envelope graph
3 　　= "Pulse Envelope Units > Voltage"
4 　　= "Time period between end of pulse and the end of falling edge = 62.5 us". I.e. the time it takes
　　　　the voltage to fall from 50% to 0% of the top level
5, 10 = Beginning of the rising and falling edge
6 　　= "Falling Edge = 100 us", i.e. the time it takes the voltage to fall from 90% to 10% of the top level

7	= "Width = 500 us", i.e. the duration the voltage is above 50% of the top level
8	= "Rising Edge = 100 us", i.e. the time it takes the voltage to rise from 10% to 90% of the top level
9	= "Time period between beginning of the rising edge and pulse start = 62.5 us". I.e. the time it takes the voltage to rise from 0% to 50% of the top level

You can also import a custom shape or I/Q data, see:
- "To generate a pulse with raised cosine envelope shape" on page 194
- "To define your custom envelope shape with an equation" on page 195

Per default, the pulses within a pulse sequence that is built by repeating the same pulse, have constant on time and PRI.

You can define IPM profiles to vary the pulse parameters on a pulse-to-pulse basis. See Chapter 8.2, "How to create IPM profiles and vary pulse parameters", on page 245.

To define the level parameters of a pulse

Pulse level settings control the RF output power level during all pulse phases. The level is determined by two main settings, the "Top Power" and the "Base Power".

To access the level settings:

1. In the repository tree, select "Pulse > PulseName > Level".

   The tab comprises the pulse level settings, required to simulate a non-ideal pulse. The major reasons for non-ideal pulse shapes are the overshoot and ripple.

2. Change the "Level" settings.
   Use the Envelope graph to visualize the pulse characteristics.

   The "Envelope Graph" confirms that the envelope overshoots the "Top Power", experiences damped oscillations and amplitude droop and settles to the final value.
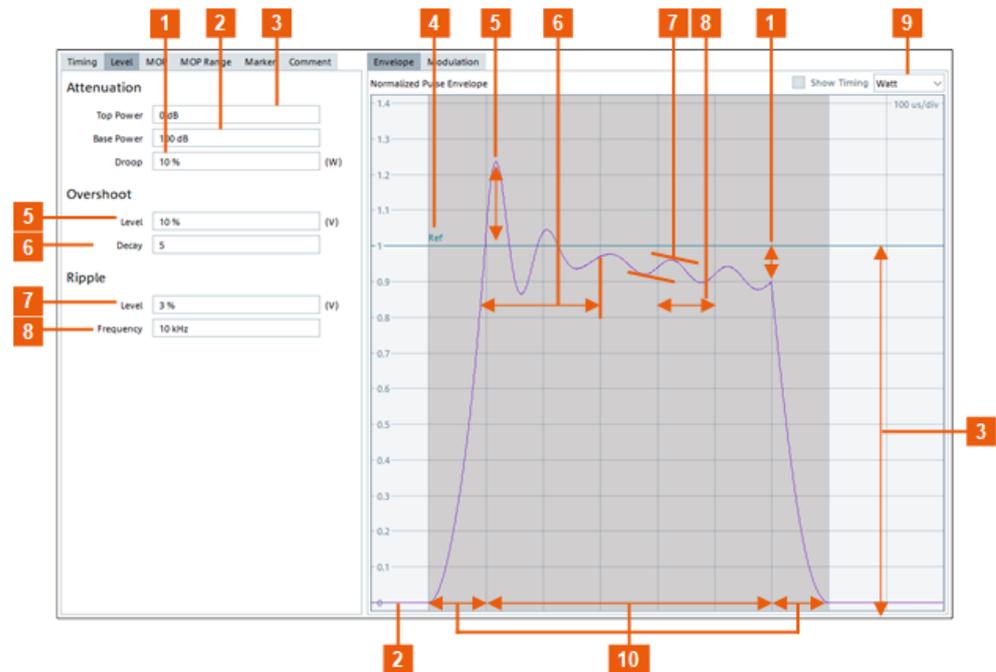
Figure 6-18: Pulse Level: understanding the displayed information

1 = "Droop", i.e. amplitude change from the beginning to the end of the pulse ON time
2 = "Base Power", i.e. high attenuation; typically 100 dB
3 = Pulse top power corresponds to 1000 mW ("Pulse Envelope = Watt"), i.e. little or no attenuation ("Attenuation Top Power = 0 dB")
4 = Reference value used by the calculation of the pulse level parameters: "Ref. = 0 dB (1 Watt)"; the value is set with the parameter Level
5 = Overshoot level, in volt (diagram uses Watts)
6 = "Decay", i.e. damping constant that influences the time it takes the overshoot to decay
7 = Amplitude of the oscillating signal, in volt (diagram uses Watts)
8 = Signal oscillating with the selected frequency ("Frequency = 10 kHz", i.e. the oscillation period is 100 us)
9 = Pulse amplitude units (Y-axis units)
10 = Pulse timing: "Profile > Standard 0/100", "Rising Edge = Falling Edge = 100 us", "Width = 500 us" (see Standard Timing Profile)

To define and apply modulation on the pulses (MOP)

The "MOP" settings define the intra-pulse modulation, i.e. the modulation applied on the pulse rise, on, and off time. The R&S Pulse Sequencer provides a wide range of commonly used modulation schemes, such as AM, FM, or Chirp. You can also add your custom pulse content, Chapter 20, "Defining complex modulation schemes and IPM profiles", on page 530.

Some "MOP" require data sources that are used with a modulation scheme. A data source for the symbols of the modulation has to be created first.

See Chapter 19, "Defining the data source", on page 523 for description of the provided settings.

To access the "MOP" settings:

1. In the repository tree, select "Pulse > PulseName > MOP".

The available modulation settings depend on the selected "MOP Type". See Chapter 6.2.5, "Modulation on pulse (MOP) settings", on page 162.

2. Select for example "MOP > MOP Type > Chirp > Linear Chirp".
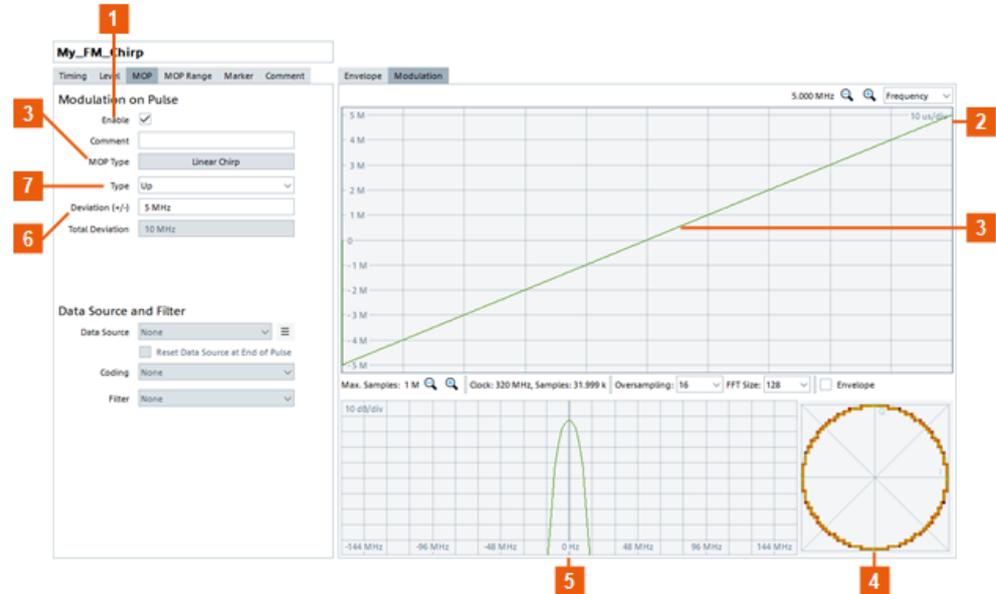
Figure 6-19: Modulation on pulse (MOP): understanding the displayed information

1 = "Enable Modulation on Pulse (MOP) > On", i.e. MOP is applied
2 = Frequency diagram
3 = Enabled is a liner chirp modulation; the diagram confirms the frequency variation
4 = Constellation diagram
5 = Spectrum
6 = Frequency deviation = 5 MHz, i.e. the frequency changes in the range - 5 MHz to 5 MHz
7 = "Type > Up", i.e. the frequency is increasing linearly

For a description of the provided settings, see Chapter 6.2.6, "MOP range", on page 185.

3. Select "MOP > Enable > On".

4. Use the Restrict MOP Range settings to determine the exact pulse part to which the modulation is applied, e.g.:

   a) Select "Level Threshold > On".
   b) Select "Timing > Rising Edge = Falling Edge = 25%".

   Use the "Signal Preview" dialog to observe the effect of the enabled modulation, see Chapter 15, "Visualizing and analyzing signals", on page 400.

To generate a pulse with raised cosine envelope shape

1. Select "Pulse > Timing"

2. Select "More" to customize the envelope.

3. Select "Use Equation".

   Envelope is described by an equation, as selected with the parameter "Preset".

4. Select "Preset > Raised Cosine"
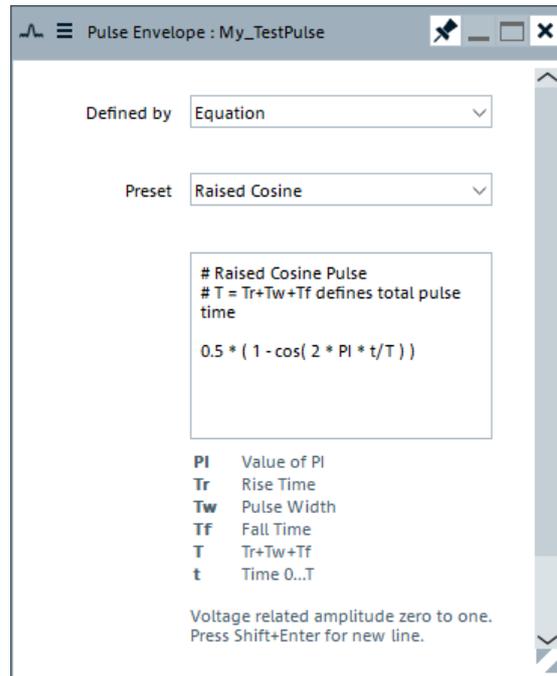
The dialog displays the used equation.

For overview of the available envelope shapes, see "Equations and parameters used to define custom pulse envelopes" on page 861.

5. Select "Pulse > Timing> Custom Envelope > On" to apply the envelope.

The Envelope graph displays the resulting envelope.

To define your custom envelope shape with an equation

This example explains how to enable an envelope shape that follows the shape on the following figure.
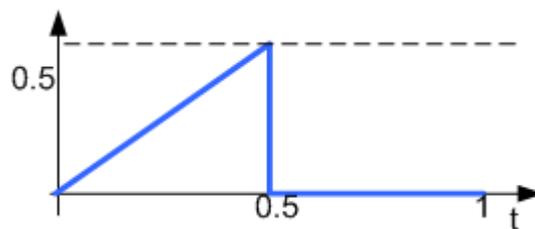
Equations are related to voltage. Allowed are amplitude values 0 to 1.

1. Select "Pulse > Timing > More". Select "Use Equation".

2. Select "Preset > User". Enter the equation:
   ```
   t/T < 0.5 ? t/T : 0
   ```
   See also Chapter B, "Formula syntax", on page 860.

3. Select "Envelope > Normalized Pulse Envelope > Voltage". Compare the resulting shape.
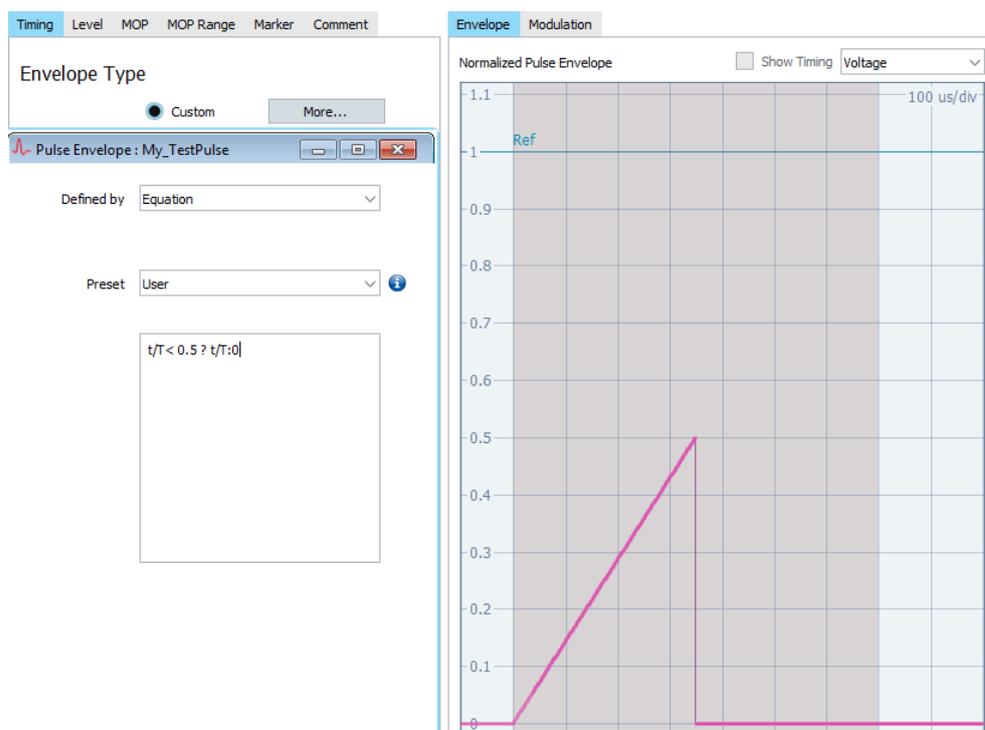
To generate an envelope shape in a table form

1. Select "Pulse > Timing > More".
   Select "Use Imported Data".

2. Select "Import Data from File". Navigate to the file describing the envelope and load it.
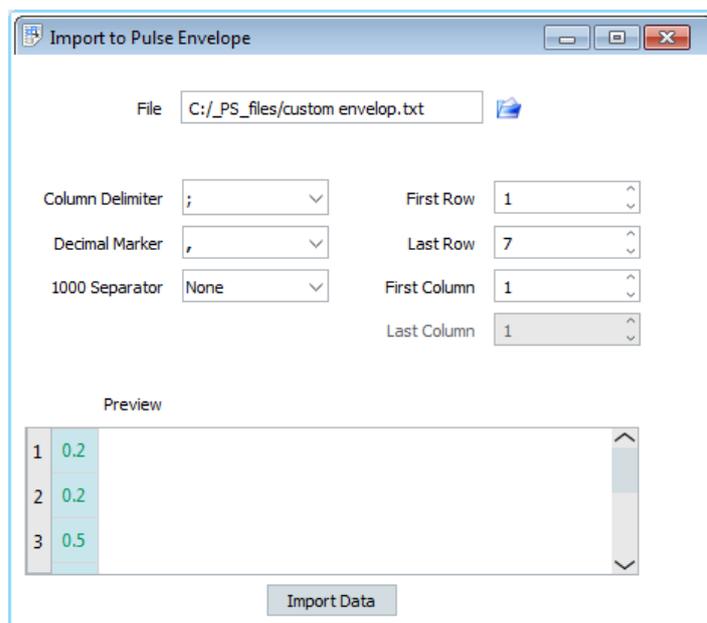
To visualize the pulse characteristics

1. Use the "Modulation Graph" to visualize the configured modulation schemes.
   See "Modulation graph" on page 188.

2. Use the "Envelope Graph" to display the envelope shape.
   See Figure 6-17.

3. Set "Show Timing > On" to visualize the pulse width, rise and fall time on the
   graph.

7 Building pulse sequences

This section explains the sequencing principles and signal processing, like creating loops and defining overlaying signals.

For information on sequences containing waveforms, see Chapter 14, "Working with waveforms", on page 368.

7.1 About the sequencing principles

Sequences are described in table form where the rows are processed sequentially and appended to the final waveform, see Figure 7-1.

Figure 7-1: Default segments processing: One segment after the other

Each row is repeated once or as defined with the repetition count parameter (see "Repetition" on page 204).

See also:

● "To define and enable pulse repetition" on page 213
● Chapter 7.2.2, "Pulse repetition settings", on page 206

Complex sequences can also include the following:

● **Loop**
A loop spans one or more rows (pulses) and processes these rows in the same order several times. Nested loops are complex loops that span rows and loops. The loop duration is defined by the loop repetition number. The loop repetition number is a fixed value or a randomly selected value within a value range, defined with its minimum, maximum and step values.
See also:
  – "To define and apply a loop" on page 214
  – Chapter 7.2.3, "Loop settings", on page 207
● **Overlay**
An overlay processes rows in parallel. The row processing start at the same time and lasts up to the overlay duration is. Longer segments are truncated.
Overlay is a method that changes the default behavior (Figure 7-1) and adds several pulses (segments) to a common time line; that is segments are stacked on top of each other. An overlay group spans several segments of any kind, like regular pulses and fillers.
Segments are added to the existing data starting at the beginning of the first segment. If the new segment is shorter than the overlay length, the remainder is left blank. Longer segments are truncated, see Figure 7-2.
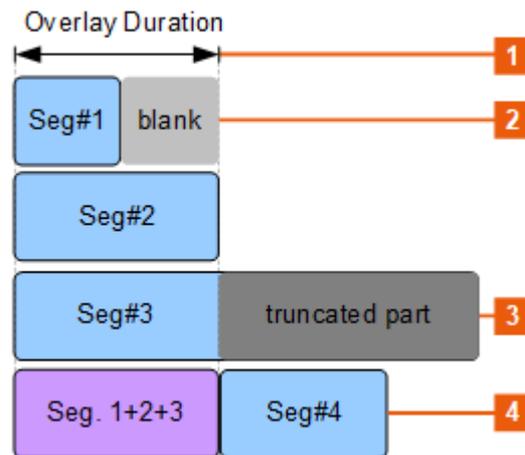
Figure 7-2: Segments processing in overlay mode: Segments are added

1 = Overlay Duration: sets the length of the resulting sum segment
2 = Short segment: remainder is left blank
3 = Long segment: segment is truncated
4 = Resulting sum segment (Seg#1+Seg#2+Seg#3): Seg#4 is processed right after the overlay duration elapses

See also:

– "To define and enable overlaying segments" on page 217

– "To include a subsequence in an existing overlay" on page 219

– Chapter 7.2.4, "Overlay settings", on page 208

- **Subsequence**
  A subsequence defines a sequential sequence of segments within an overlay.

  See also:

  – "To include a subsequence in an existing overlay" on page 219

- **Filler**
  Fillers are unmodulated signals used to fill in certain period in a sequence.

  Signals can be of type:

  – Continuous wave (CW)

  – Blank

  There are two ways to set the duration of a filler: by defining the time duration to be filled in (total duration) or by defining of a certain moment of time (time synchronization).
  The filler duration can be a fixed value or defined as an equation. If the filler is configured within a loop, the equation uses the loop variables.

  See also:

  – "To create and add a CW segment" on page 216

  – Chapter 7.2.5, "Fillers settings", on page 209

7.2 Sequence settings

Access:

► In the repository tree, select "Sequence > New".
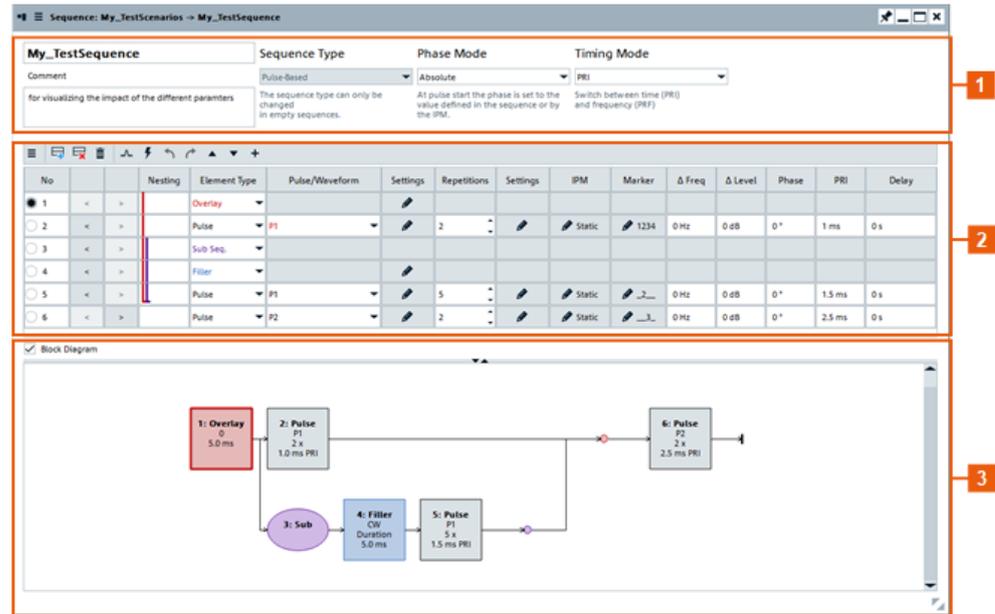
Figure 7-3: Sequence Settings

1 = Sequence name, comment, type, and phase and timing mode
2 = Sequence description table, where each row represents one item
3 = Interactive graphical representation of the current sequence, displayed if "Block Diagram > On"

For step-by-step descriptions, see Chapter 7.3, "How to create sequences and use the control elements", on page 212.

Settings:

7.2.1 Sequence description settings

Sequence Name, Comment, Type, Sequence Description Table, Block Diagram

A sequence is defined with its unique name, an optional description and with information on the sequence content, waveforms, or pulses.

A sequence is defined in table form where each row represents one item.

If enabled, an interactive block diagram displays the current sequence configuration.
- Left-click to select the item (see "Select, No." on page 203).
- Right-click to access the context menu (see "New, Insert, Append, Remove, Clear Items, Undo/Redo" on page 203).
- Double click an item works like the "Settings (...)" icon in the "Sequence Description Table" and accesses the settings of the item directly.

Remote command:

SEQuence:CREate on page 588

SEQuence:CATalog? on page 587

SEQuence:SELect on page 588

SEQuence:NAME on page 589

SEQuence:COMMent on page 590

SEQuence:TYPE on page 789

SEQuence:REMove on page 590

Phase Mode

Defines how the phase is set at each pulse start.

"Absolute"   The pulse start phase is set to a value defined in the sequence description table (Phase) or by the IPM.
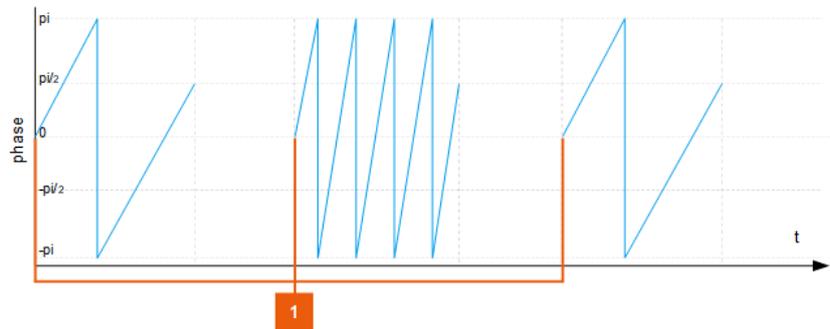
Figure 7-4: "Phase Mode = Absolute" and "Sequence Description (table) > Phase = 0 deg" (IPM not defined)

1 = Start phase of all pulses is 0 deg

"Continuous (Relative)"

The phase defined in the sequence description table (Phase) or by IPM is used as a relative phase change. The phase of the last valid sample of a pulse is held; the specified phase value is applied as a phase offset on it.
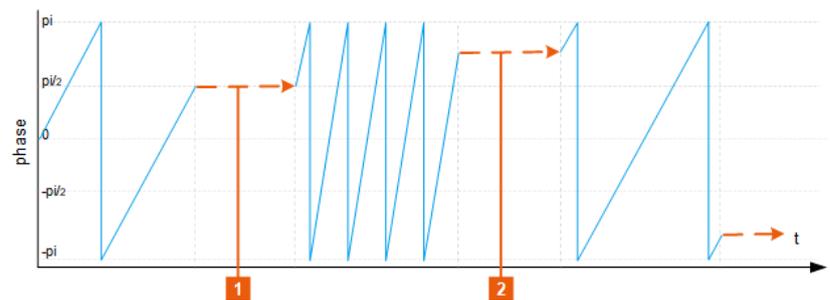
Figure 7-5: "Phase Mode = Continuous (Relative)" and "Sequence Description (table) > Phase = 0 deg" (IPM not defined)

1, 2 = With "Phase = 0 deg", the phase offset is also 0 deg. The phase of last sample of a pulse is also the start phase of the next one

"Memory"   If there are frequency steps, the phase is set to a state as if the frequency has never changed.

Figure 7-6: "Phase Mode = Memory" and "Sequence Description (table) > Phase = 0 deg" (IPM not defined)

1, 2 = Alternating switching between the outputs of frequency synthesizers (two signals in the example). At the beginning of each pulse, the phase is set to a state the frequency would have had if it had been continuously running

Remote command:
SEQuence:PHASe:MODE on page 789

Timing Mode
Switches between time (PRI) and frequency (PRF), see PRI/PRF.

Remote command:
SEQuence:TIME:MODE on page 789

New, Insert, Append, Remove, Clear Items, Undo/Redo
Standard functions for items handling, see "Standard function in the context menus" on page 33.

Select a row, for example, to delete it or to insert a row before it.

Remote command:
SEQuence:ITEM:ADD on page 591
SEQuence:ITEM:CLEar on page 596
SEQuence:ITEM:DELete on page 595

Select, No.
Select an item to perform any row-based actions.

Items are indicated by consecutive number; also used for indication in remote control.

Remote command:
SEQuence:ITEM:SELect on page 593
SEQuence:ITEM:COUNt? on page 592

Nesting
Indents table rows to include them in a loop, to assign them to an overlay or build parallel branches.

See "To define and enable overlaying segments" on page 217.

Remote command:
SEQuence:ITEM:INDent on page 791

Element Type
Defines the type of content of the current item. Available are:

"Pulse/Wave-   To select a pulse/waveform, use the Pulse/Waveform.
form"

"Filler"   Creates dummy data, like a blank segment, or a CW signal.
                  See Chapter 7.2.5, "Fillers settings", on page 209.

"Loop"   Creates a loop
                  See Chapter 7.2.3, "Loop settings", on page 207.

"Overlay"   An overlay processes sequence items in parallel.
                  See Chapter 7.2.4, "Overlay settings", on page 208.

"Sub. Seq"   In overlay, includes a sequence in a sequence, i.e. creates a parallel
                  branch containing a sequence of items.
                  See To include a subsequence in an existing overlay.

Remote command:
SEQuence:ITEM:TYPE on page 795

Pulse/Waveform
Lists all available pulses/waveforms.

- To access the settings of an existing pulse/waveform, select "... (Settings)".
- To create a new pulse/waveform, select the icon.
  See also Chapter 6, "Creating a pulse library", on page 148.

Remote command:
SEQuence:ITEM:PULSe on page 793
SEQuence:ITEM:WAVeform on page 793

Repetition
Sets the number of times the item is repeated. For more complex repetition patterns,
select "... (more)".

See Chapter 7.2.2, "Pulse repetition settings", on page 206.

Remote command:
SEQuence:ITEM:REP:COUNt:FIXed on page 794
SEQuence:ITEM:LOOP:COUNt:FIXed on page 791

Settings
Accesses the "Rep: Line Item" dialog with further repetition settings.

IPM
Enables and defines an inter-pulse modulation.

See Chapter 8, "Defining and enabling inter-pulse modulation effects", on page 224.

Marker
Defines the active markers per pulse phase, see Chapter 21, "Defining and enabling
marker signals", on page 534.

Δ Freq.

Enables a frequency offset relative to the reference frequency.

Frequency-dependent parameter values are defined relative to this value.

See also "To define and apply a loop" on page 214.

Remote command:
SEQuence:ITEM:FREQuency:OFFSet on page 790

Δ Level

Defines a level offset per row.

A negative level offset simulates a pulse attenuation. For example, a "Level = -3 dB" means that the pulses of the current row are processed with power level that is the half of the power.

Level-dependent parameter values are defined relative to this value.

See also "To define and apply a loop" on page 214.

Remote command:
SEQuence:ITEM:LEVel:OFFSet on page 791

Phase

Sets a phase offset at the beginning of the row. Phase-dependent parameter values are defined relative to this value.

For example, a "Phase = 30 deg" shifts the phase in the first pulse of the current row; enabled modulation would further change the phase. The phase offset is not compensated at the row end.

See also "Phase Mode" on page 201.

Remote command:
SEQuence:ITEM:PHASe:OFFSet on page 793

PRI/PRF

The column name changes depending on the selected Timing Mode.

Sets the pulse repetition interval (PRI), i.e. the pulse period or the pulse repetition frequency (PRF).

See Figure 6-1).

Use the "PRI" parameter to define pulses with constant total pulse cycle duration and enabled timing variations like jitter for example.

Remote command:
SEQuence:ITEM:PRI on page 793
SEQuence:ITEM:PRF on page 793

Delay

Enables a start delay.

A delay within an overlay shifts the start time of the overlay elements relative to each other. Time-dependent parameter values are defined relative to this value.

See also "To define and apply a loop" on page 214.

Remote command:
SEQuence:ITEM:PDELay on page 793

7.2.2 Pulse repetition settings

Per default, a table row in the sequence description table, i.e. a pulse or a waveform, is processed once. To repeat an item several times, enable a repetition.

For a step-by-step instruction, see "To define and enable pulse repetition" on page 213.

Repetition Number
Defines how many times a sequence item, for example a pulse or loop, is repeated.

The repetition number can be:
- A fixed value
- A randomly selected value within a value range, defined with its minimum, maximum and step values ("Randomly Selected)")
- Defined to fit in a selected duration ("Auto Set by Duration"), where the repetition number can be rounded up or down

Remote command:
SEQuence:ITEM:REP:TYPE on page 795
SEQuence:ITEM:REP:COUNt:MINimum on page 794
SEQuence:ITEM:REP:COUNt:MAXimum on page 794
SEQuence:ITEM:REP:COUNt:STEP on page 795
SEQuence:ITEM:REP:COUNt:DURation on page 794
SEQuence:ITEM:REP:COUNt:ROUNding on page 795

Repetition Variables
You can define repetition variables that are used by the pulse calculation within one row.

"Prefix"   Indicates the repetition the variable belongs to. If empty, no variables are used.
                  The variable prefix is a case-sensitive entry.

Remote command:
SEQuence:ITEM:REP:VARiable on page 795

7.2.3 Loop settings

Per default, table rows in the sequence description table are processed sequentially. To process a subset of pulses or waveforms several times, create a loop.
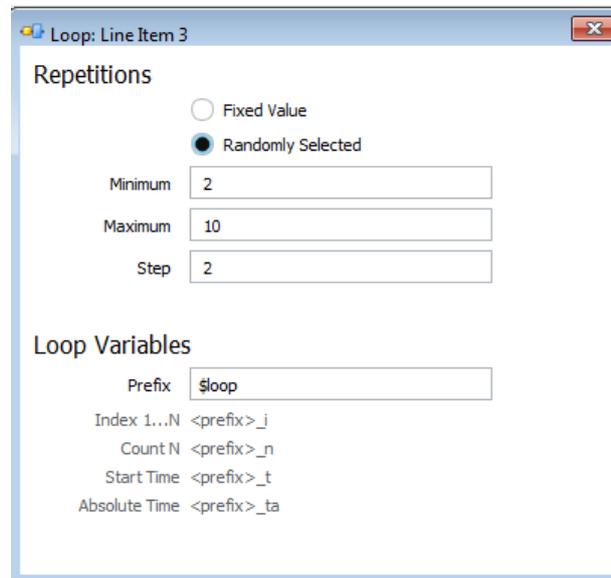
For a step-by-step instruction, see "To define and apply a loop" on page 214.

Loop Repetition
The loop repetition number is a fixed value or a randomly selected value within a value range, defined with its minimum, maximum and step values.

Remote command:
SEQuence:ITEM:LOOP:TYPE on page 792
SEQuence:ITEM:LOOP:COUNt:MINimum on page 791
SEQuence:ITEM:LOOP:COUNt:MAXimum on page 791
SEQuence:ITEM:LOOP:COUNt:STEP on page 792

Loop Variables
You can define loop variables that are used by the pulse calculation within the loop. When you use a variable, always indicate the prefix and the extension.

"Prefix"   Indicates the loop to that the variable belongs. If empty, no loop variables are used.
                  The variable prefix is a case-sensitive entry.

"Loop Index 1 .. N"

"<prefix>_i" indicates the current repetition number and is a value in the range 1 .. N, where N is the repetition number.

"Loop Count N"

"<prefix>_n" indicates the repetition number, fixed or randomly selected as defined with Loop Repetition.

"Start Time"   "<prefix>_t" indicates the start time of each subsequent loop

"Absolute Time"

"<prefix>_ta" indicates the absolute time

Remote command:
SEQuence:ITEM:LOOP:VARiable on page 792

7.2.4 Overlay settings

An overlay processes rows in parallel. The row processing starts at the same time and lasts up to the overlay duration is. Longer items are truncated.
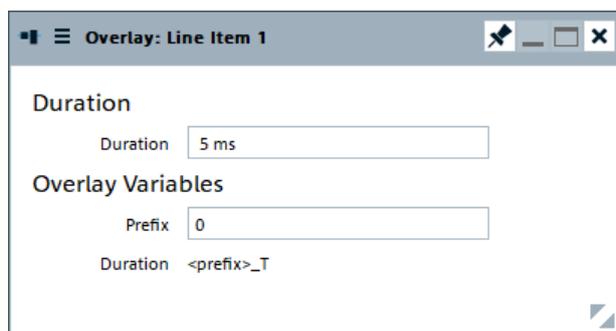
Overlay Duration = Sets the length of the resulting sum item

For a step-by-step instruction, see "To define and enable overlaying segments" on page 217.

Duration
Sets the duration of the sum item.

Remote command:
SEQuence:ITEM:OVL:WTIMe on page 792

Overlay Variables
You can define overlay variables that are used by the pulse calculation.

"Prefix"   Indicates the overlay to that the variable belongs. If empty, no overlay variables are used.
The variable prefix is a case-sensitive entry.

"Duration"   "<prefix>_t" indicates the overlay duration.

Remote command:
SEQuence:ITEM:OVL:VARiable on page 792

7.2.5 Fillers settings

You can enable and add fillers.
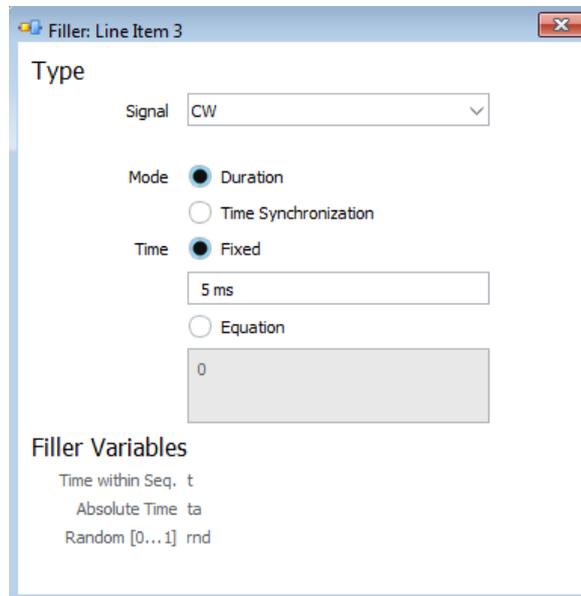
For a step-by-step instruction, see "To create and add a CW segment" on page 216.

Signal

Selects the type of signal to be generated for filling in the dummy segment.

"Blank"   A blank segment with the selected duration is appended to the sequence.

"CW"   A segment containing a continuous wave (CW), i.e. unmodulated sig-
              nal, is used.

"Hold"   Holds the last available pulse sample.

Remote command:
SEQuence:ITEM:FILLer:SIGNal on page 789

Mode

Filling segments are commonly used for synchronization purposes. This parameter defines the way the filler duration is determined:

"Duration"   Enables a dummy signal for a given period of time.

"Time Synchronization"

              In this mode, signal data is generated up to the defined point in time.
              If the selected time has already elapsed, an error is generated and
              the calculation is stopped.
              Use this mode if a signal has to start at a defined point in time and
              the length of the previous signals vary or is not known.

Remote command:
SEQuence:ITEM:FILLer:MODE on page 789

Time

Defines the timing of the filler.

"Fixed"   Uses a fixed time

"Equation"   Defines the time point as an equation.

Remote command:
SEQuence:ITEM:FILLer:TIME on page 790
SEQuence:ITEM:FILLer:TIME:FIXed on page 790
SEQuence:ITEM:FILLer:TIME:EQUation on page 790

7.2.6 Lists with multiple sequences

Access:

1. In a "Sequences (Collection)" scenario, select "Sequences > Edit".
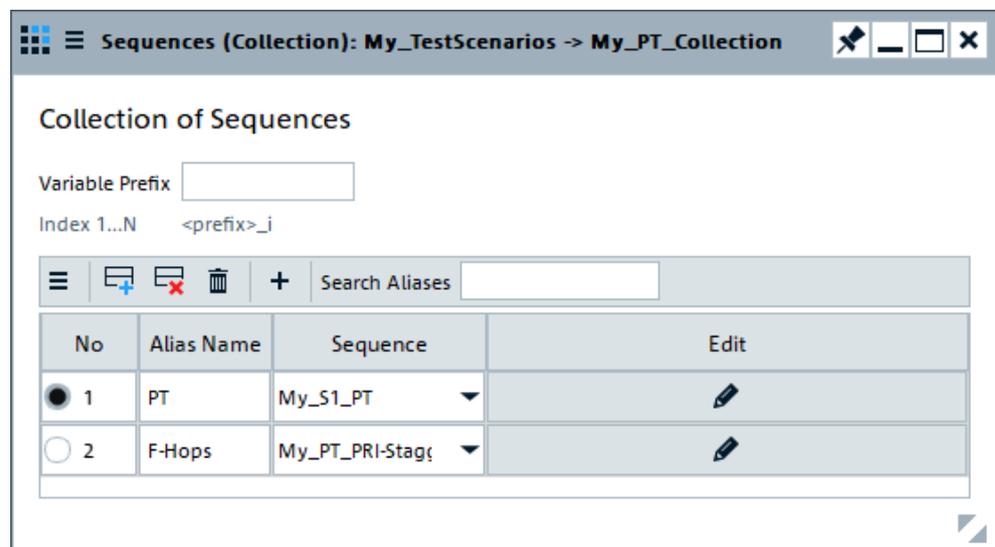
   Switching between multiple sequences is possible in the pulse train sequential scenario.

2. Select "Edit > ..." to access the properties of the individual sequence.
   See Chapter 7.2, "Sequence settings", on page 200.

Variables

You can define collection variables and use them, e.g., in the formula editors, in the reports, etc.

"Prefix"   Indicates the sequence to that the variable belongs. If empty, no vari-
                      ables are used.
   The variable prefix is a case-sensitive entry.

"Index 1 .. N"

                      "<prefix>_I" indicates the current number and is a value in the range
                      1 to N.
                      Enter the `<prefix>` only; the extension `_I` is appended automati-
                      cally.

"Count N"

                      "<prefix>_n" is the number of variables.
                      Enter the `<prefix>` only; the extension `_I` is appended automati-
                      cally.

Remote command:
SCENario:CSEQuence:VARiable on page 755

List of sequences
Displays a list of sequences.

To add a sequence, select the "Data Source" icon and select "Insert After".

To reorder sequences (items in the list), use the "Up/Down" functions.

Use the standard "Append", "Remove Last", "Delete All" or "Copy and append" func-
tions to add or remove a sequence. The provided settings are self-explanatory.

Remote command:
SCENario:CSEQuence:ADD on page 591
SCENario:CSEQuence:SELect on page 592
SCENario:CSEQuence:CURRent on page 746
SCENario:CSEQuence:CLEar on page 596
SCENario:CSEQuence:DELete on page 595

Alias Name ← List of sequences
Enters a name.

Remote command:
SCENario:CSEQuence:ALIas on page 746

Sequence ← List of sequences
Select a sequence from the list of available sequences.

Remote command:
SCENario:CSEQuence on page 755

Edit ← List of sequences
Opens the "Sequence" dialog for the selected sequence, see Chapter 7.2, "Sequence
settings", on page 200.

7.3 How to create sequences and use the control elements

See:

- "To create a simple sequence" on page 212
- "To create and add a CW segment" on page 216
- "To define and enable pulse repetition" on page 213
- "To define and enable overlaying segments" on page 217
- "To include a subsequence in an existing overlay" on page 219
- "To define and apply a loop" on page 214
- "To create a scenario with several pulse train sequences" on page 220

To create a simple sequence

This example explains how to create a simple sequence with two pulses, P1 and P2.
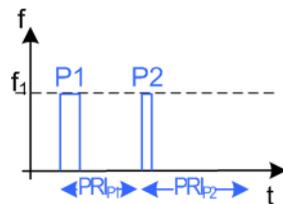
We assume that the required pulses have been created, see Chapter 6.3, "How to create a new pulse and adjust its settings", on page 190.

1. In the repository tree, select "Sequence > New".

2. In the "Sequence" dialog, enter a "Name" and a "Comment", e.g. `My_S1_PT`.

3. For each sequence item, perform the following steps:

   a) Select the "Append New Item" icon.
   b) In the "Sequence Description" table, select "Type > Pulse". Select an existing pulse, e.g. "Pulse > P1".
   c) If the list is empty, select the pulse icon to create a pulse first.
   d) See Chapter 6.3, "How to create a new pulse and adjust its settings", on page 190
   e) Set the PRI_{P1}, e.g. "PRI = 2.5 ms".

4. Enable "Show Block Diagram".

1 = Pulse with different PRIs, i.e items with different duration
2 = Sequence with two items

The items are processed once and one after the other (see also Figure 7-1). You can change the default processing, see:

- "To define and enable pulse repetition" on page 213
- "To define and enable overlaying segments" on page 217
- "To define and apply a loop" on page 214

To define and enable pulse repetition

This example explains how to repeat pulses several times, for example to configure a pulse sequence as illustrated on Figure 7-7.
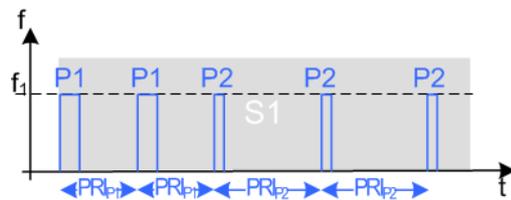
Figure 7-7: Example of pulse sequence composed of 2xP1 and 3xP2, where the pulses use different PRIs

PRI_{P1} = 2.5 ms
PRI_{P2} = 5 ms

We assume that a sequence with two items has been created, see "To create a simple sequence" on page 212.

1. Open this sequence.

2. In the "Sequence Description" table, for the first pulse, select "Rep. Count = 2"

3. For the second item, set "Rep. Count = 3".

The pulse sequence S1 consists of 5 pulses and has a PRI_{S1} = 20 ms.

Use the"Waveform View" display to visualize the resulting waveform, see Chapter 15, "Visualizing and analyzing signals", on page 400.

To define and apply a loop

This example explains how to create a loop with pulses and to repeat it several times, for example to configure a pulse sequence as illustrated on Figure 7-8.
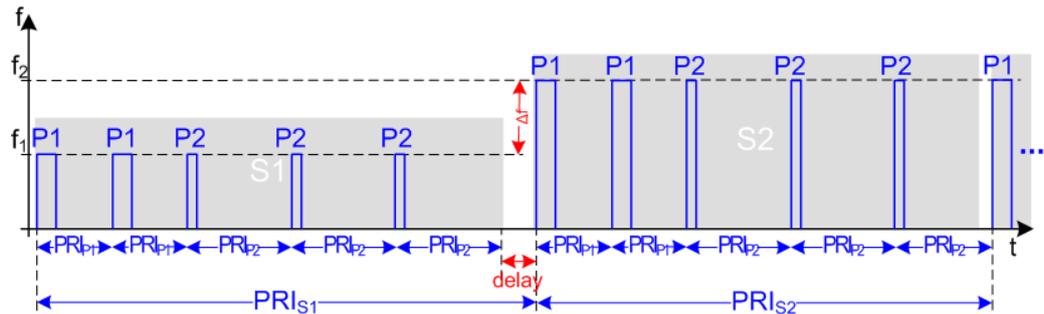
Figure 7-8: Example of pulse sequence composed of two subsequences, S1 and S2

S1, S2 = Both sequences follow the same pulse pattern (2xP1 and 3xP2)
PRI_{S1} = PRI_{S2} + "Delay"
f_2 = f_1 + "ΔFreq"

We assume that a sequence with four items has been created, as shown on the following figure.

1.   In the "Sequence Description" table:

a) Select the third item. Select "Insert Item Before Selection"
b) Select "Element Type > Loop"
c) Select the "... (more)" icon and enable a random loop repetition.
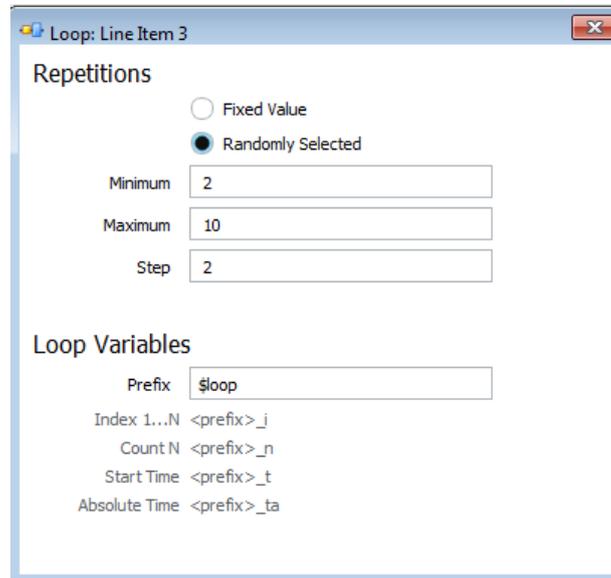
The loop is repeated 2, 4, 6, 8 or 10 times.

d) To include the last two items (Line items 3 and 5) in the loop, select the "Increase indentation level >" icon.

The sequence graph in the "Sequence Description" confirms that the two items are included in the loop. The loop is repeated a random number of times.

2. To enable a different PRI for the looped items, select the first pulse in the loop and set "Delay = 2 ms".

$PRI_{S1} = PRI_{S2} + 2$ ms = 22 ms.

3. For the pulses in the loop, set the "Δ Freq = 500 KHz" and "Δ Level = -10 dB".

4. Assign the sequence `My_S2_S1-F1_S1-F2` to a pulse train scenario.
   Start the calculation. Select "Results > View".

   The "Data View" confirms that the waveform comprises two subsequences with different PRI, where the second one is repeated four times.
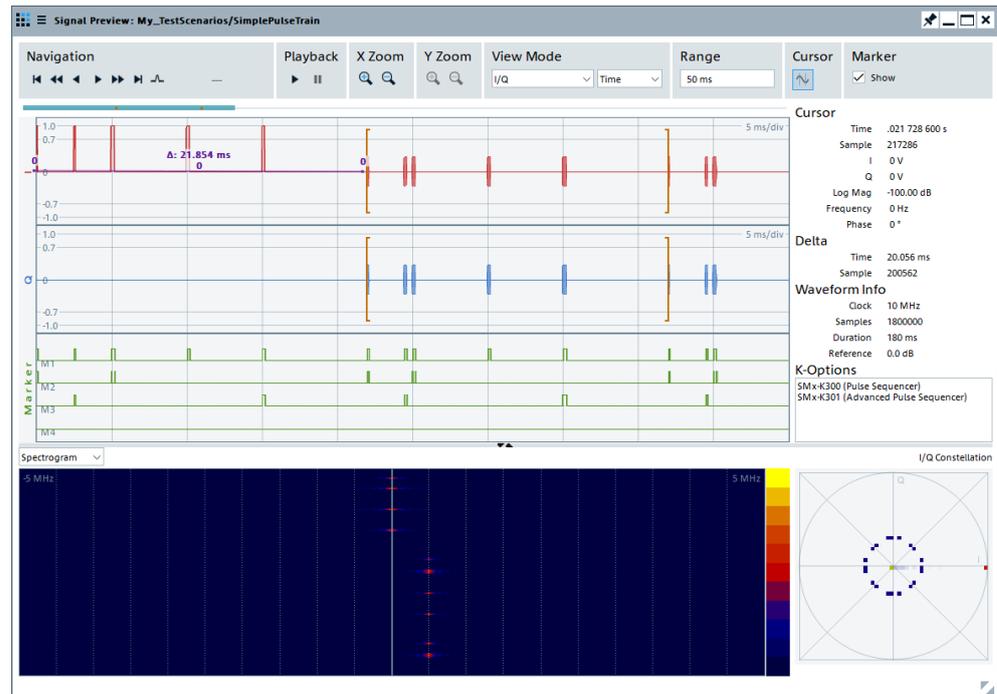
Figure 7-9: "Signal Preview": Example of a sequence

Select "View Mode > Frequency" and "View Mode > Average Power" to observe the frequency and level values.
For more information, see Figure 15-1.
For more information, see Chapter 15.2, "Signal preview settings", on page 405.

To create and add a CW segment

1. Open an existing sequence.

2. In the "Sequence Description" table, add an item. Select "Type > Filler" > "... (more)".

3. In the "Configure Filler" dialog:

   a) Select "Signal > CW"
   b) Select "Mode > Duration"

c) Select "Time > Fixed = 5 ms"
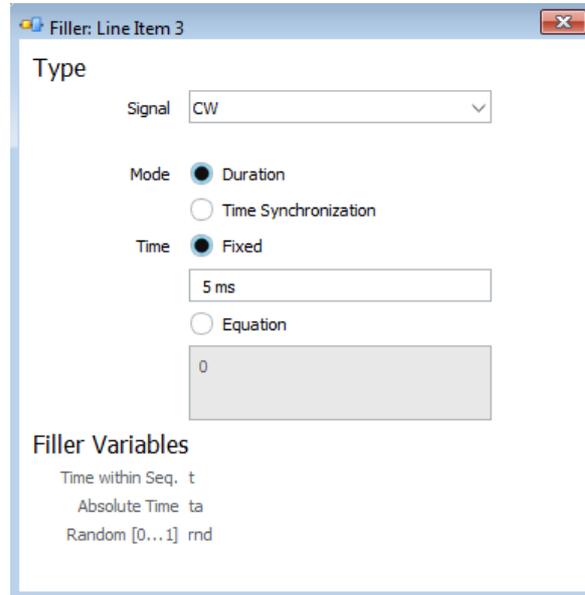
To define and enable overlaying segments

This example explains how to overlay three items (see Figure 7-2).

We assume that a sequence with four items has been created.

1. In the "Sequence Description" table of this sequence:

   a) Select the first item.
   b) Select the "New/Select" icon > "Insert Item Before Selection"
   c) For the appended (first item), select "Type > Overlay" > "Settings".

2. In the "Configure Overlay" dialog, select "Overlay > Duration = 5 ms".
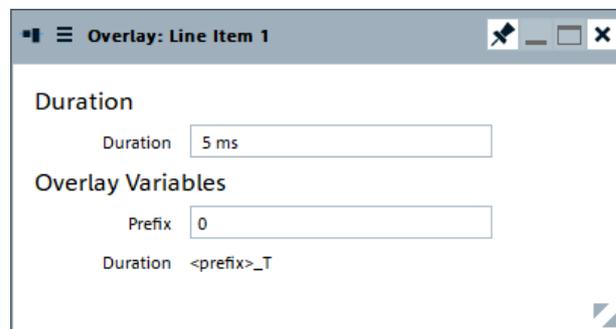
The block diagram confirms that an overlay is created. There are, however, no segments added to it.
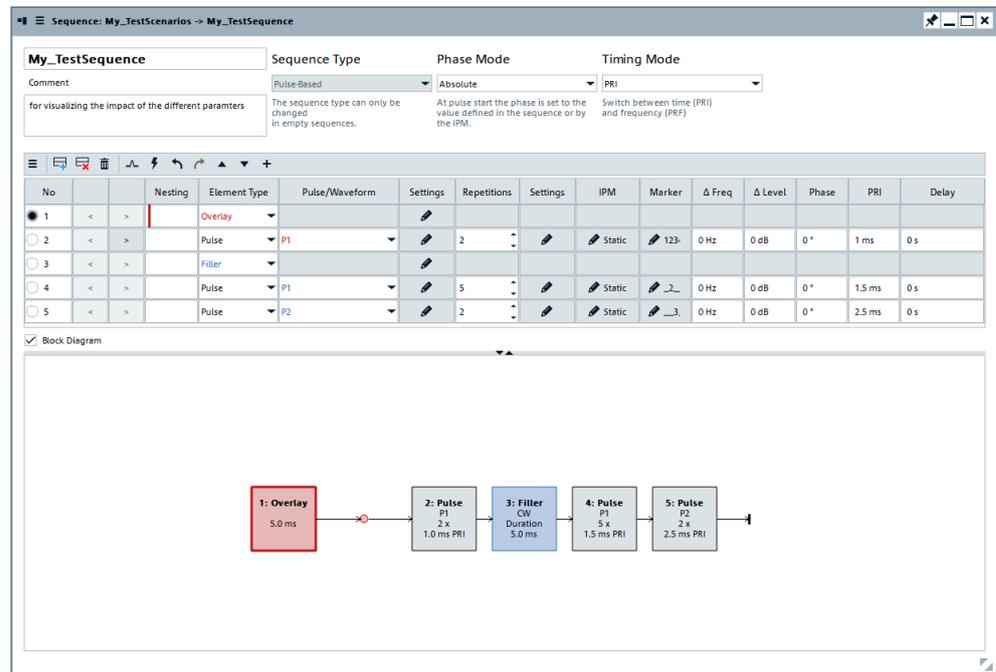
3. To include the first three items (Line items 2, 3 and 4) in the overlay, select the "Indent" icon.
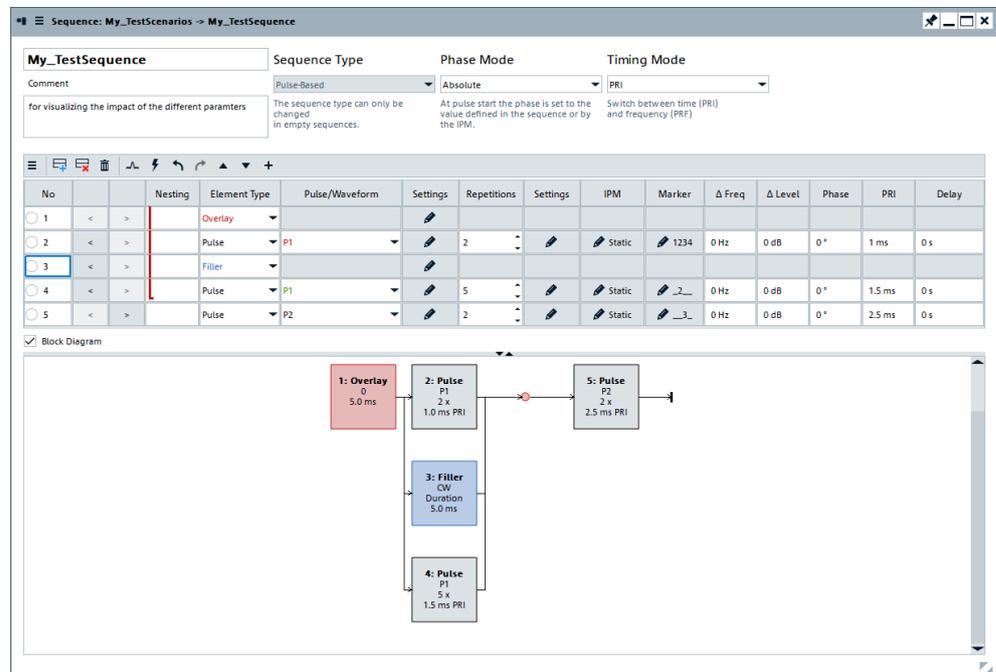
The block diagram confirms that the three items are overlaid. The fourth item is processed after elapsing of the overlay duration.

4. To observe the effect of an enabled overlay, open the "Data View" dialog.

See also Chapter 15.4, "Analyzing the calculated results", on page 411.

1	= The first P1 (short segment) and the first P2 (long segment) are superimposed
2	= Overlay "Duration = 5 ms"
3	= Generated pulse sequence = sum segment followed by the item outside of the overlay
4	= Outside segment: 2xP2 (appended right after the overlay)
5, 6, 7	= Marker traces ("M1 = M2 = M3 = All"): indicate the pulse On time of each of the pulse
8	= Overlay
9	= Long segment: 4x pulse P1 with "PRI = 1.5 ms" (segment is truncated)
10	= Short segment: 2x pulse P1 with "PRI = 1 ms" (remainder is left blank)

To include a subsequence in an existing overlay

1. Open the sequence you have created with "To define and enable overlaying seg-ments" on page 217.

2. In the "Sequence Description" table, insert a new item after the second one. Select "Type > Sub Sequence".

3. To build a parallel branch with the items 4 and 5, use the "Indent" icon.

The block diagram confirms that the items are overlaid. The second parallel branch consists of two subsequent processed pulses; the fourth item will be processed after elapsing of the overlay duration.

To create a scenario with several pulse train sequences

If your test situation requires you to test the receiver's ability to detect different pulse sequences, you can use "Sequences (Collection)" scenario. In this dedicated scenario, you can configure several pulse train sequences and switch between them sequentially.

The following example creates a complex scenario with two pulse train sequences to explain the principle.

1. Select "Repository Tree > Scenario > New".

2. In the "Scenario Wizard", select "Scenario Type > Sequences (Collection)".
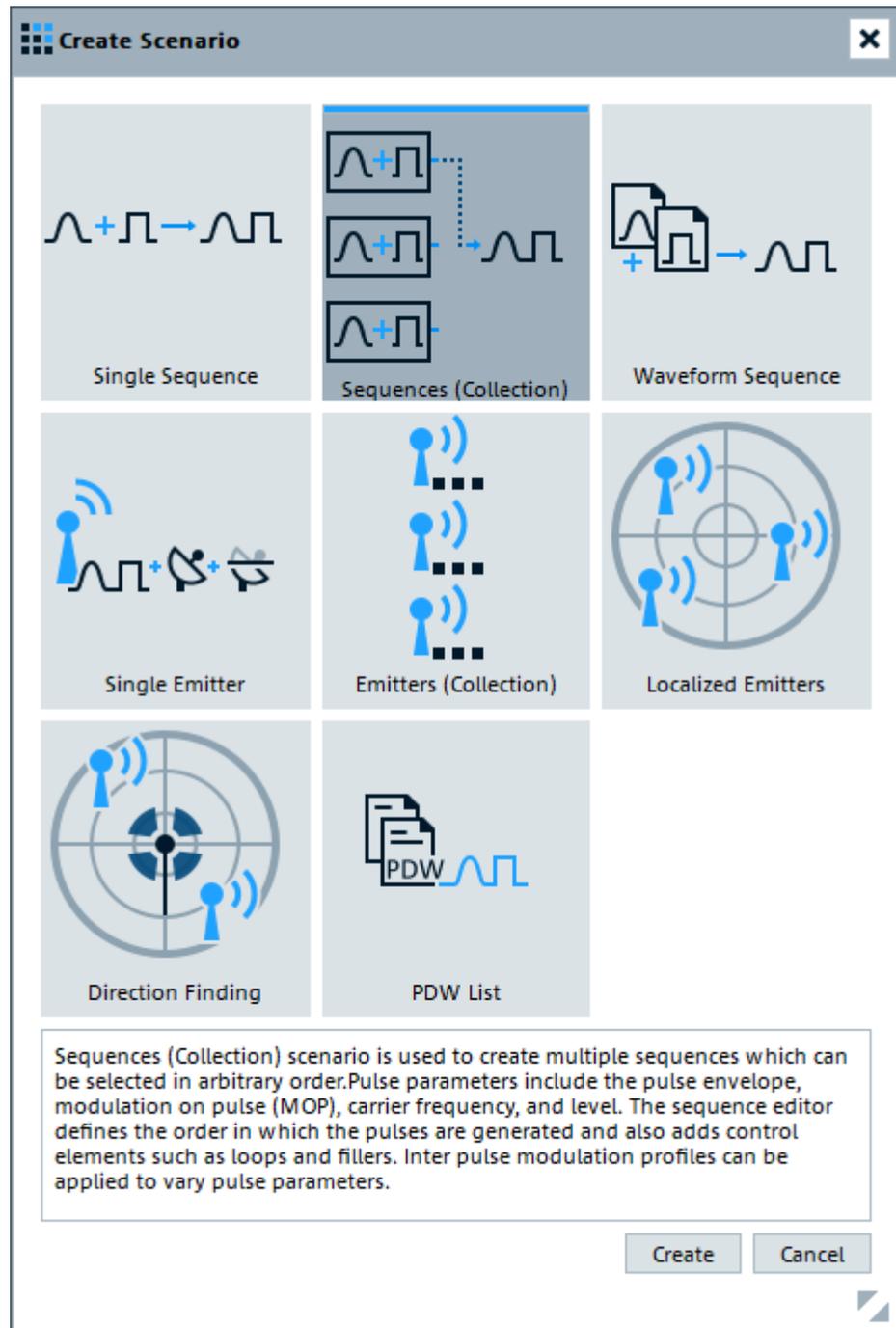
3. Click "Create".

4. Enter a scenario name and a description.

5. In the block diagram, select "Sequences > Edit".

    The "Sequences (Collection)" dialog opens.

6. Use the standard "New" function to insert the first sequence in the list.

    You can select from the list of available sequences or create a new one.

See also "To create a simple sequence" on page 212.

7. In the list of sequences, select a "Sequence". Use the standard "New", "Append/ Remove" and "Up/Down" functions to add items to the list or reorder them.
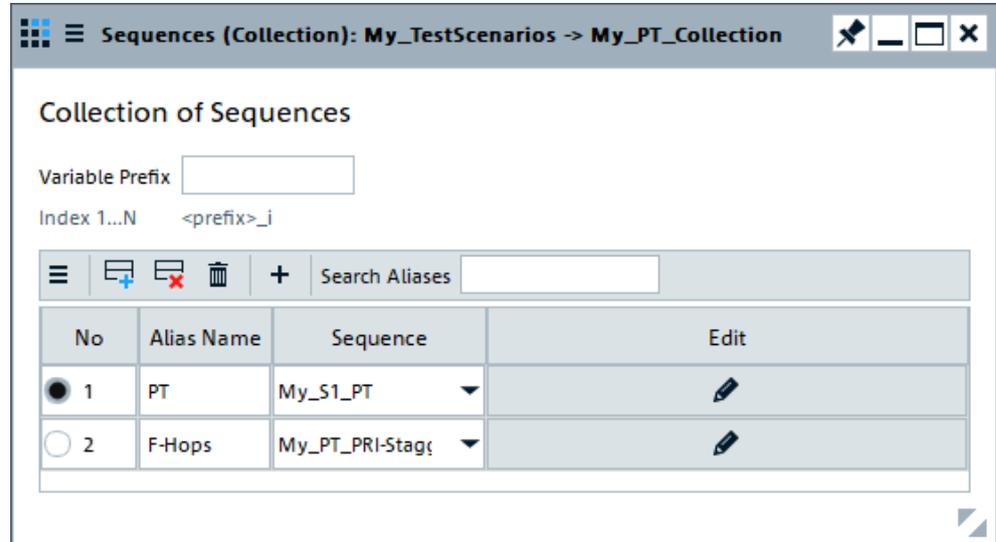
For description of the provided settings, see Chapter 7.2.6, "Lists with multiple sequences", on page 210.

8. In the "Signal Generation" block, select the name of the sequence, e.g. "PT".
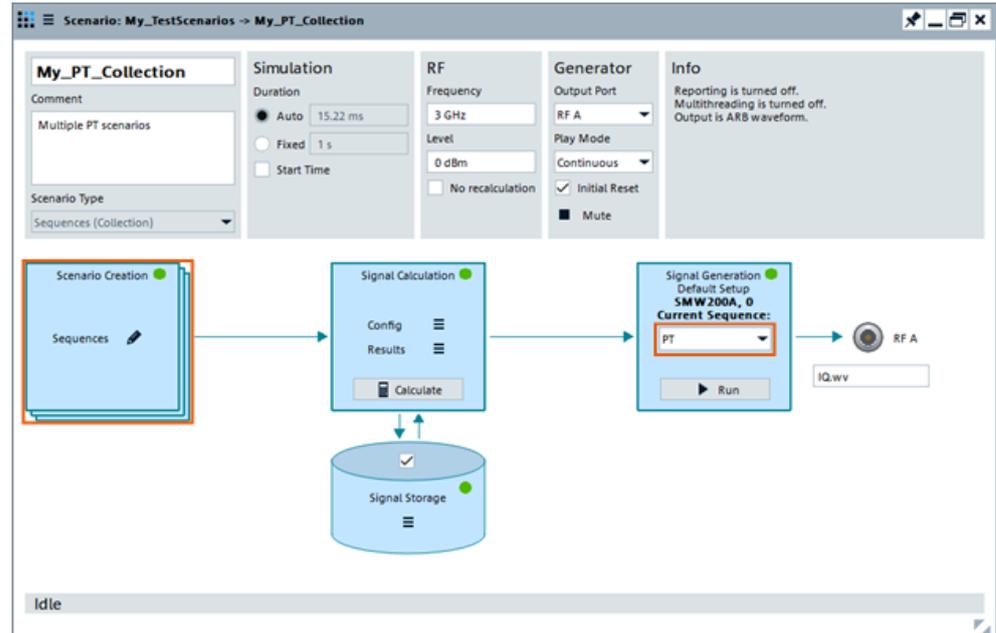
For description of the provided scenario settings, see Chapter 5.2, "Scenario settings", on page 121.

To create a pulse train sequence and scenario automatically with the "Startup Assistant"

Alternatively to the methods described in this section, you can use the "Startup Assistant" to create a pulse train automatically.

1. In the menu bar, select "Help > Wizard".

2. Select "Create a pulse train".
   Click "Next".

3. Follow the instructions, use the default settings and select "Finish".

   For description of the provided settings, see:
   - Chapter 5.2, "Scenario settings", on page 121
   - Chapter 6.2, "Pulse settings", on page 150
   - Chapter 8.1, "IPM profiles settings", on page 227
   - Chapter 21.1, "Marker settings", on page 534

8 Defining and enabling inter-pulse modulation effects

Per default, pulses calculated from the settings of one table row in the sequence description table are identical. To simulate a change of parameters from one pulse to another, like PRI stagger for example, you can define inter-pulse modulation (IPM) effects and assign them to the sequence elements.

Throughout this description, the IPM effects are commonly named parameter variations.

Pulse repetition patterns

The pulse repetition patterns are commonly used to:

- Avoid range ambiguities caused by echoes of far away objects, which fall into the detection period of the next pulse
- Protect the radar signal against jamming
- Deal with clutter (ground, sea, weather)
- Make the radar more difficult to be identified by frequency hopping

IPM effects

In the following, two common IPM techniques, the jittered PRI and staggered PRI, are used to explain the principle:

- *Jittered PRI*
  In general, a jitter is understood as the short-term variations of a significant pulse parameter, deviating from their ideal positions in time in either random or ordered way (see Figure 8-1).
  You can, for example, use the random variation of the rising and falling pulse edge positions to simulate a technically imperfect trigger signal.
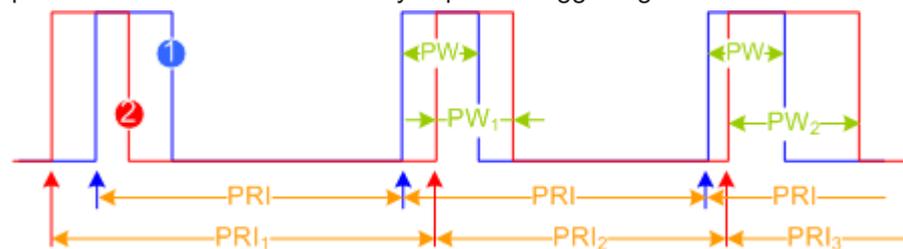
Figure 8-1: Example of Jitter effects

1 = Ideal pulse with constant pulse repetition interval (PRI) and pulse width (PW)
2 = Pulse affected by Jitters: Jitter PRI ($PRI_1 \neq PRI_2$) and varying pulse width ($PW_1 \neq PW_2$)

As illustrated on Figure 8-1, the time between two subsequent pulses varies randomly, on a pulse-to-pulse basis, as does the pulse width.

- *Staggered PRI*
  Staggered pulse trains are commonly used against range ambiguities and blind speeds, see also "Pulse repetition patterns" on page 224.

Staggered PRI is a transmission method based on pulse trains with constant pulse width (PW) and varying PRI. The principle of staggered PRI is illustrated on Figure 8-2
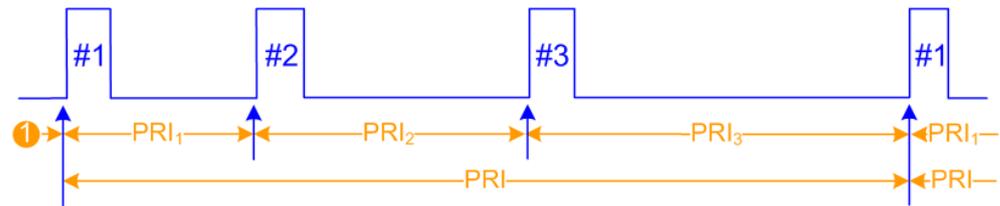
Figure 8-2: Example of an IPM effect: Staggered PRI

1 = Stagger start delay
#1,#2,#3 = Pulse train group composed of three pulses
PRI = Group PRI: Pulse repetition interval of the pulse train group
PRI_1,PRI_2,PRI_3 = Pulses transmitted with different PRI

As illustrated in Figure 8-2, within a pulse train group each successive pulse uses a different PRI; the pulse train group is slewing.

● *Frequency Hopping*
Frequency hopping and frequency agility is a further anti-jamming method. If frequency hopping is used, the radar signal switches rapidly and periodically between different carrier frequencies.

There are no predefined parameter variations in the software but IPM profiles can be assigned to the parameter listed in Table 8-1.

Table 8-1: Parameters to which IPM profiles can be assigned

Parameter group	Pulse parameter	Waveform parameter
Level	Overshoot [%]	-
	Offset [dB] Δ Level	Offset [dB] Δ Level
	Attenuation Top [dB] Attenuation > Top Power	-
	Attenuation base [dB] Attenuation > Base Power	
	Droop [%]	
	Ripple [%] Ripple > Level	
	Ripple Frequency [Hz]	
MOP	AM Modulation Depth [%]	-
	AM Frequency [Hz]	
	FM Deviation [Hz]	
	FM Frequency [Hz]	
	Symbol Rate [Hz]	

Parameter group	Pulse parameter	Waveform parameter
	FSK Deviation [Hz]	
	Chirp Deviation [Hz]	
Timing	Rise Time [s] Pulse > Time > Rising Edge	-
	Pulse Width [s] Pulse > Time > Width	
	Fall Time [s] Pulse > Time > Falling Edge	
	Delay [s]	Delay [s]
	PRI [s]/PRF [Hz]	PRI [s]/PRF [Hz]
Phase	Offset [°]	Offset [°]
Δ Freq.	Offset [Hz]	Offset [Hz]

Source profiles

Parameter variations are described as the combination of a source (profile or variable) and the target element (pulse, sequence or waveform), to that the variation is assigned. If you assign more than one variation to an element, these variations are superimposed (see Chapter 8.1.4, "Inter-pulse modulation (IPM) settings", on page 241).

Source profiles are jitter and IPM profiles. You can use the same profiles for more than one variation; the R&S Pulse Sequencer calculates the profile values independent from each other.

Each jitter and IPM effect affects one of the parameters listed in Table 8-1 and can use the following profiles:

- Steps
- Waveform
- Random List
- Random Steps
- List
- Interpolated Shape
- Random
- Equation
- Plug-in
- Binomial

8.1 IPM profiles settings

Access:

► Select "Repository Tree > IPM > New".

Settings:

8.1.1 Common IPM settings

IPM Profile Name
Enters the name of the profile.

Remote command:
IPM:CREate on page 588
IPM:NAME on page 589
IPM:CATalog? on page 587
IPM:SELect on page 588
IPM:REMove on page 590

IPM Profile Comment
Adds a description.

Remote command:
IPM:COMMent on page 589

Unit of Affected Parameter
Sets the units of the IPM parameters.

Available are: "Time [s]", "Frequency [Hz]","Level [dB]", "Phase [deg]", "Percent [%]".

Remote command:
IPM:UNIT on page 657

Profile
Sets the shape of the profile:
● Steps
● Waveform
● Random List
● List
● Interpolated Shape
● Random
● Equation

Further available parameters depend on the selected shape.

Remote command:
IPM:TYPE on page 657

2D
Opens a preview diagram of the selected profile.

Two views are available:
● "Time Series"
  A visualization of the profile variation over time
● "Histogram"
  A statistical representation of the relative frequency density, calculated for the
  selected number of "Histogram Bins".
  The histogram plots the number of times data points appear in a particular bin.

The displayed characteristics on both views are defined by:

"Values"   Sets the number of values to be displayed.

"Duration"   Sets the displayed duration.

"Histogram Bins"
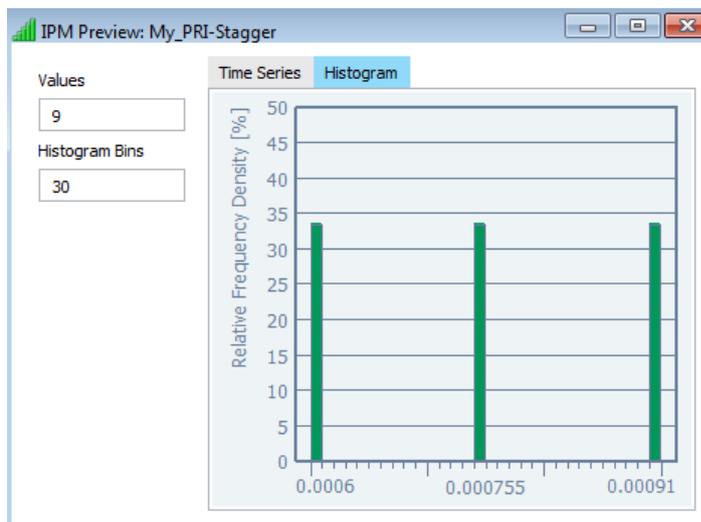
Sets the number of histogram bins, i.e. the histogram sets used to group the obtained data.
Bins of equal size are used. The bin size is calculated from the value range used on the y-axis in the time series display and the number of bins.

Remote command:
DIALog:IPMPlot:VIEW on page 668
DIALog:IPMPlot:SAMPles on page 668

8.1.2 IPM profiles settings

The following IPM shapes can be defined:

Steps

This IPM profile follows a staircase shape and creates a sequence of identical values before it moves on the next one.

The profile shape is defined as a function of "Number of Steps", "Start" and "Increment" values; the stop value is calculated automatically.

The parameters "Burst Length" and "Burst Period" are exclusive. They define how often or how long an increment is repeated, i.e. how many identical values are created.
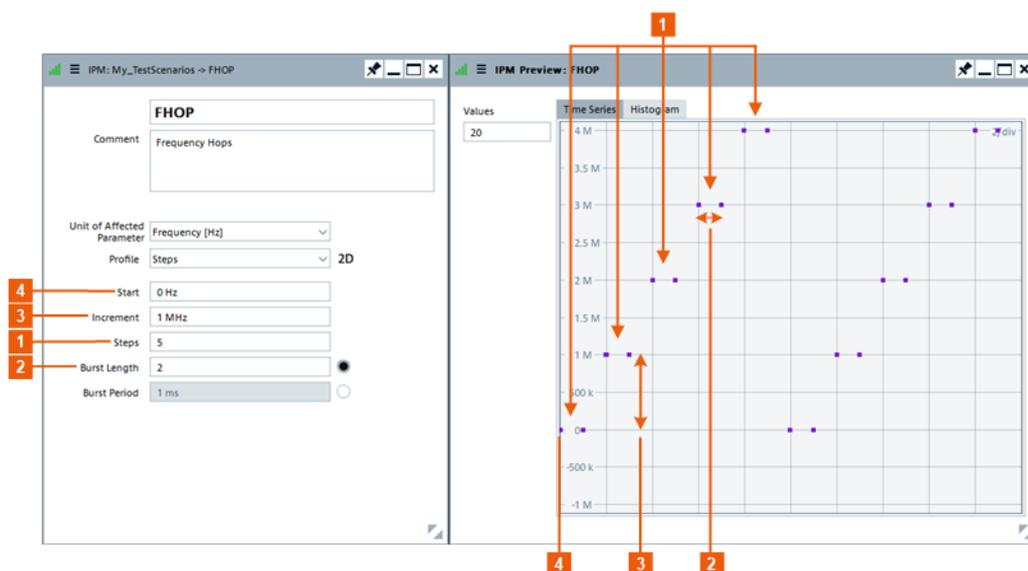
Figure 8-3: IPM with shape "Profile = Steps"

1 = "Number of Steps" = 5
2 = "Burst Length" = 2
3 = "Increment" = 1 MHz
4 = "Start" = 0 Hz

Remote command:
IPM:STEP:STEPs on page 665
IPM:STEP:STARt on page 664
IPM:STEP:INCRement on page 664
IPM:STEP:BASE on page 663
IPM:STEP:BURSt on page 664
IPM:STEP:PERiod on page 664

List

The IPM shape is a sequence of discrete values defined in a list.

The list items are processed sequentially and cyclically. When the list end is reached, the list is processed from the beginning. Each list item is repeated once or several times, or for the defined time duration.
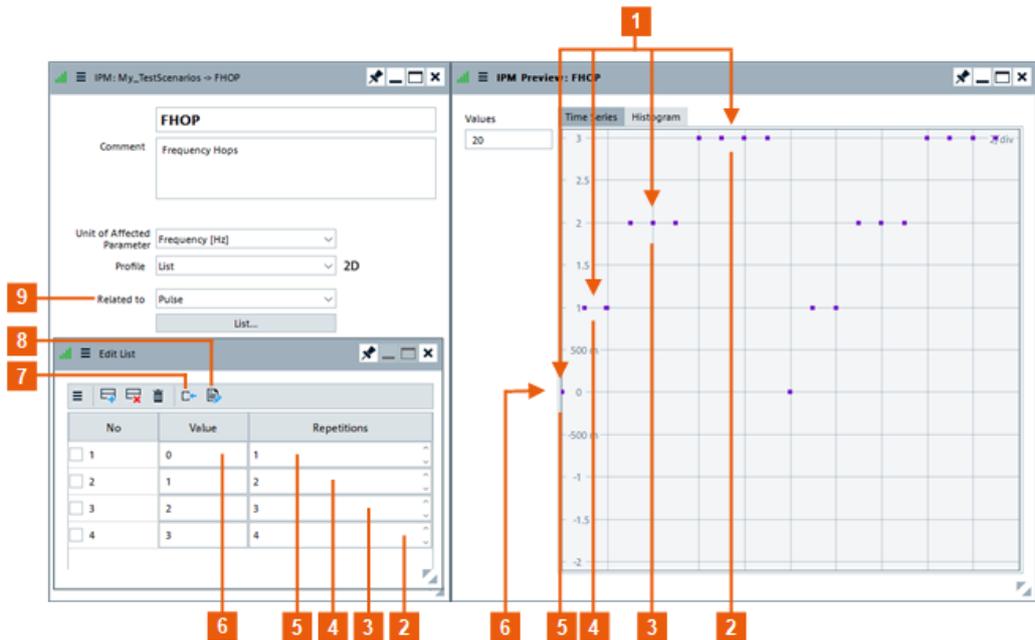
Figure 8-4: IPM with shape "Profile = List"

1	= "List Count" = 4
2, 3, 4, 5	= The increments are repeated different number of times: "Repetitions#1 = 1", "Repetitions#2 = 2", "Repetitions#3 = 3", etc.
6	= "List Start" = 0
7	= "Import Data from File" icon
8	= "Populate list" icon
9	= "Related to = Pulse"

List profiles are defined in the standard list editor in table form, see "Edit List" on page 240.

Use one of the following options to create an IPM list profile:

- To create a list manually, use the standard icons and functions in the context menu (see "Standard function in the context menus" on page 33).
  Enter the list values manually.
- To import a list in ASCII format, select the "Import Data from File" icon.
- To create a list automatically and fill it in with values, select the "Populate list" icon.
  Enter the "Start" and "Increment" values and the number of list elements ("Count") and select "Populate".

"Firing Order"   If "Related to = Pulses", defines the sequencing of the PRIs in time. Without the firing order, PRIs defined in lists are processed in their sequential order, where a list element is repeated the number of times defined as "Repetition" (see Figure 8-4).

By using firing order, PRIs defined once can be reused several times and processed in a different order than their sequence order in the list. The "Repetition" field is ignored and the actual number of times a PRI is repeated each time is defined in the firing order definition field.

Supported are the following expressions:
- M – N
- MxL

Where:
- M and N are list elements numbers
- L is number of repetitions.

Example:

With "Firing Order > On", the expression *1x1,2x2,3x3,4x4* results in the sequence 1,2,2,3,3,3,4,4,4,4, that is the same as the example on Figure 8-4.

You can observe the "Time Series" on the 2D view.

Remote command:
IPM:LIST:FIRing:ENABle on page 659
IPM:LIST:FIRing:SEQuence on page 659

Remote command:
IPM:LIST:ITEM:ADD on page 591
IPM:LIST:ITEM:COUNt? on page 592
IPM:LIST:ITEM:SELect on page 593
IPM:LIST:ITEM:VALue on page 658
IPM:LIST:BASE on page 658
IPM:LIST:ITEM:REPetition on page 658
IPM:LIST:ITEM:TIME on page 658
IPM:LIST:ITEM:DELete on page 594

Waveform

Sets an IPM shape that follows a ramp, sine, or triangular waveform function. The waveform amplitude is defined by the "Peak-to-Peak" value. A constant "Offset" can be applied to the entire waveform. The waveform period is set as "Period Time" or as number of pulses ("Pulse Count") it lasts.
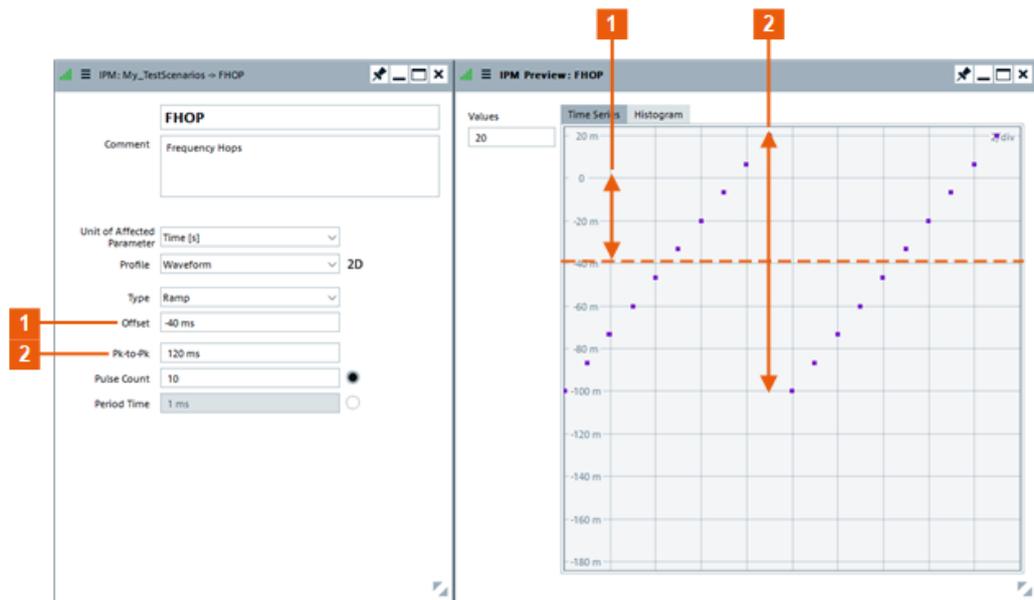
Figure 8-5: IPM with shape "Profile = Waveform" (Type = Ramp)

1 = "Offset" = -40 ms
2 = "Peak-to-Peak" = 120 ms

"Linear Ramp"	The linear ramp is a profile with a sawtooth shape. The generated values monotonically increase from ("Offset"-"Pk-to-Pk"/2) to ("Offset"+"Pk-to-Pk"/2).
"Sine"	The sine profile creates values that follow one period of a sine wave. Enable a "Phase Offset" to change the start phase of the sine wave.
"Triangular"	The triangular profile changes a parameter from a minimum to a maximum value and back to the minimum value, following a triangular shape.

Remote command:
IPM:WAVeform:BASE on page 666
IPM:WAVeform:TYPE on page 666
IPM:WAVeform:PKPK on page 667
IPM:WAVeform:OFFSet on page 666
IPM:WAVeform:PHASe on page 666
IPM:WAVeform:COUNt on page 666
IPM:WAVeform:PERiod on page 667

Interpolated Shape

The IPM shape is a sequence of values defined in a list. The list items are mapped to several repetitions ("Pulse Count") or equally distributed over a selected period of time ("Period Time").

The transition between the increments can also be linearly interpolated.
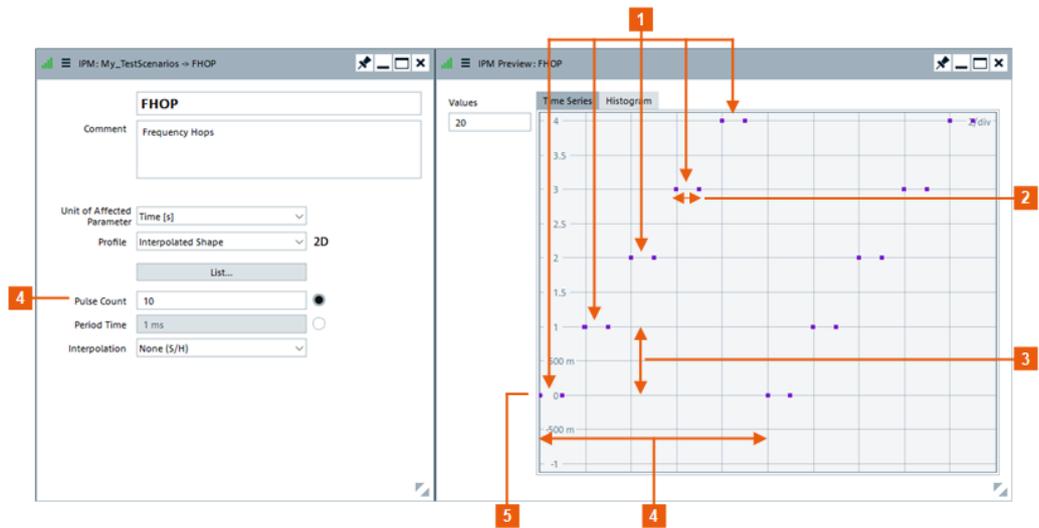
Figure 8-6: IPM with shape "Profile = Interpolated Shape" ("Interpolation = None (S/H)")

1 = "List Count" = 5
2 = Each increment is repeated two times = "Pulse Count"/"List Count"
3 = "List Increment" = 1
4 = "Pulse Count" = 10
5 = "List Start" = 0

"Period Time"   Sets the period of time over that the list items are equally distributed.

"Pulse Count"   Sets the number of pulses for that the data from the list is used.

Remote command:

IPM:SHAPe:BASE on page 663
IPM:SHAPe:PERiod on page 663
IPM:SHAPe:COUNt on page 663
IPM:SHAPe:INTerpol on page 663

Equation

Defines the IPM shape as a function of:

$t = 0 \; .. \; T$ is the absolute time.

$i = 0 \; .. \; N-1$ is a pulse counter, where N is the maximum number of pulses.

Example:

(i<10) ? 5 : (i*0.5)
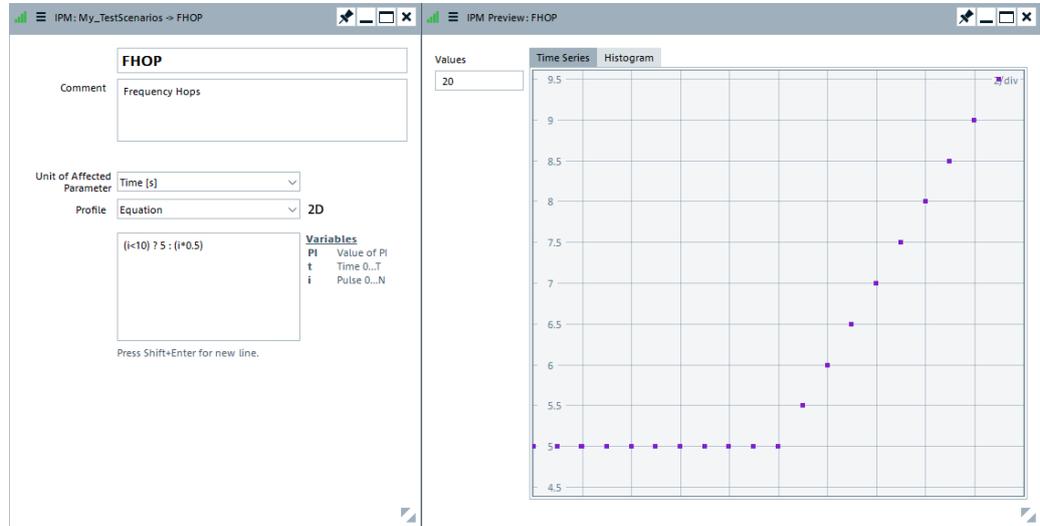
See Chapter B, "Formula syntax", on page 860.

Remote command:
`IPM:EQUation` on page 657

Random List
The IPM shape is a sequence of discrete values that are randomly selected from a list with user defined values.
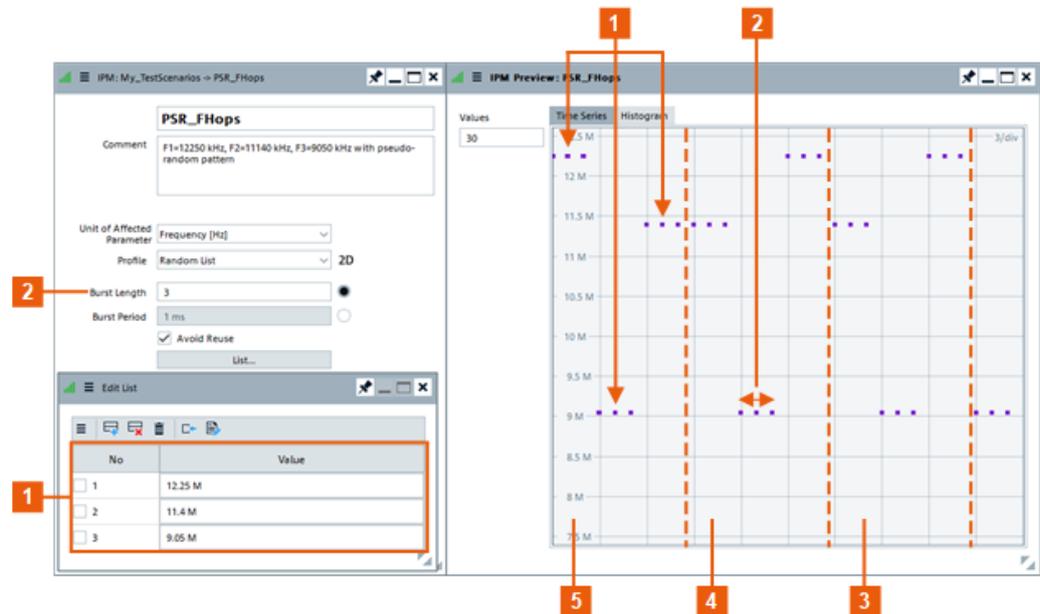
Figure 8-7: IPM with shape "Profile = Random List"

1   = "List Items" = 3
2   = "Burst Length" = 3
3, 4, 5 = Subsequent patterns with values that are selected on a pseudo-random manner from the defined 3 list items

"Burst Length/ Burst Period"	The parameters "Burst Length" and "Burst Period" are exclusive. They define how often or how long an increment is repeated, i.e. how many identical values are created.
"Avoid Reuse"	Ensures that each value is used only once per burst.
"List"	Accesses the standard list editor to define a list in table form, see "Edit List" on page 240

- To create a list manually, use the standard icons and functions in the context menu (see "Standard function in the context menus" on page 33). Enter the list values manually.
- To import a list in ASCII format, select "Import Data from file" icon, navigate to a suitable file, select it and import it.
- To create a list automatically and fill it in with values, select the "Populate List" icon.
  Enter the "Start" and "Increment" values and the number of list elements ("Count") and select "Populate".

Remote command:
`IPM:RLISt:BASE` on page 662
`IPM:RLISt:BURSt` on page 662
`IPM:RLISt:PERiod` on page 662
`IPM:RLISt:REUSe` on page 662

Random Steps

The IPM shape is a sequence of desecrate values and step size that are randomly selected from a user defined value ranges.
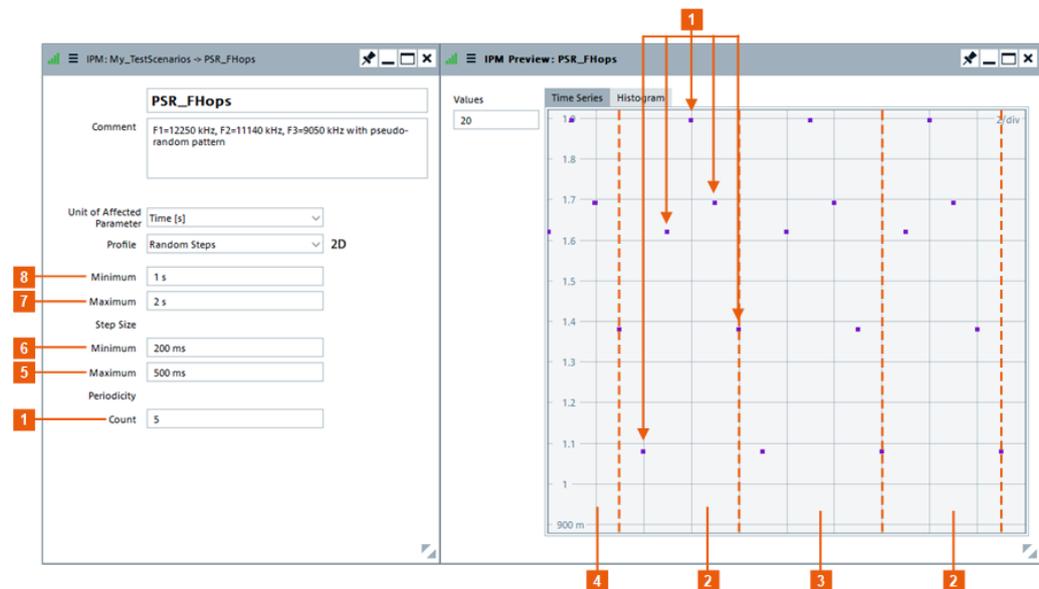
Figure 8-8: IPM with shape "Profile = Random Steps"

1 = "Periodicity > Count = 5", i.e. selected are five random values
2 to 4 = Subsequent patterns
5, 6 = User defined step size
7, 8 = User defined value range

"Periodicity > Count"

Sets the pattern length, i.e. the number of values that are selected on a pseudo-random manner.

Remote command:
IPM:RSTep:MINimum on page 665
IPM:RSTep:MAXimum on page 665
IPM:RSTep:STEP:MINimum on page 665
IPM:RSTep:STEP:MAXimum on page 665
IPM:RSTep:PERiod on page 665

Random

Random jitter is unlimited in its value range and is caused by thermal noise and similar effects. The IPM shape is a sequence of random values according to the select "Distribution" function.
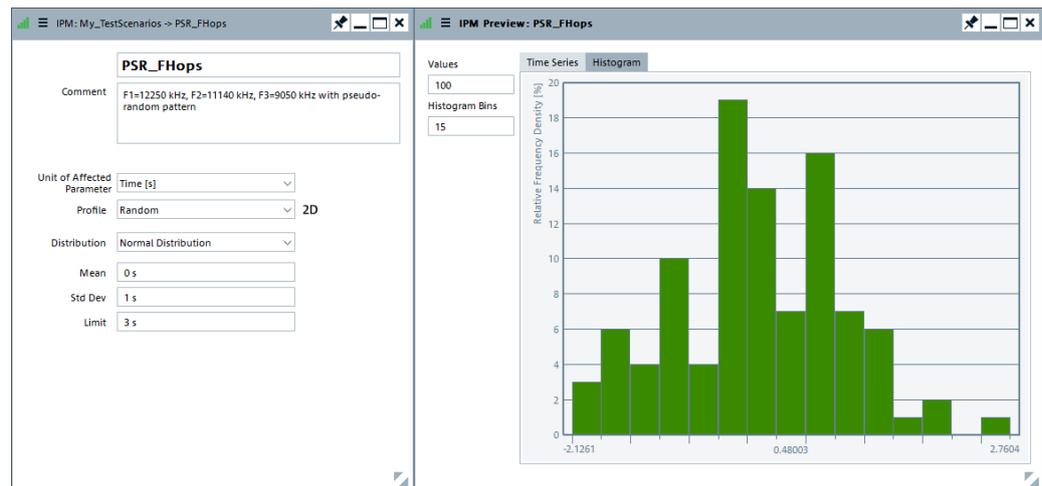
Figure 8-9: IPM with shape "Profile = Random" and "Distribution = Normal Distribution"

"Uniform Distribution"

With this distribution, values occur with the same probability in the range between the minimum and maximum level. The granularity is the value set with the parameter "Step".

"Normal Distribution"

The gauss probability density function or normal distribution is characterized by the static parameters "Mean" (μ), "Standard Deviation" (σ) and "Limit".

The Figure 8-10 illustrates the probability at which values would occur related to the standard deviation if no limit was set.
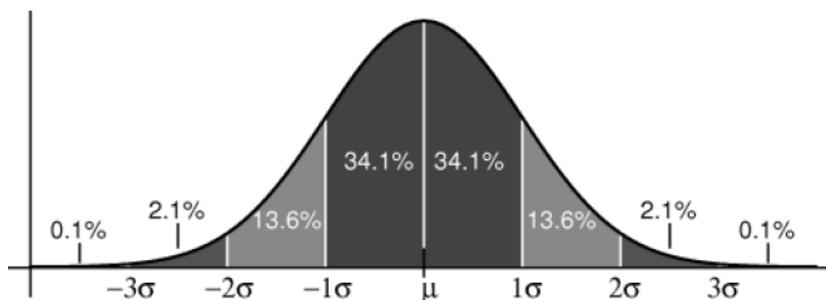
Figure 8-10: Normal distribution

"U Distribution"   The U distribution is characterized by a function with lower limit and upper limit that are calculated from selected "Center" and "Range" values.

Remote command:
IPM:RANDom:DISTribution on page 660
IPM:RANDom:UNIForm:MINimum on page 661
IPM:RANDom:UNIForm:MAXimum on page 661
IPM:RANDom:UNIForm:STEP on page 661
IPM:RANDom:NORMal:MEAN on page 660
IPM:RANDom:NORMal:STD on page 660
IPM:RANDom:NORMal:LIMit on page 660
IPM:RANDom:U:CENTer on page 661
IPM:RANDom:U:RANGe on page 661

Plug-in
Custom IPM profiles are used in the same way as the internal IPM profiles.
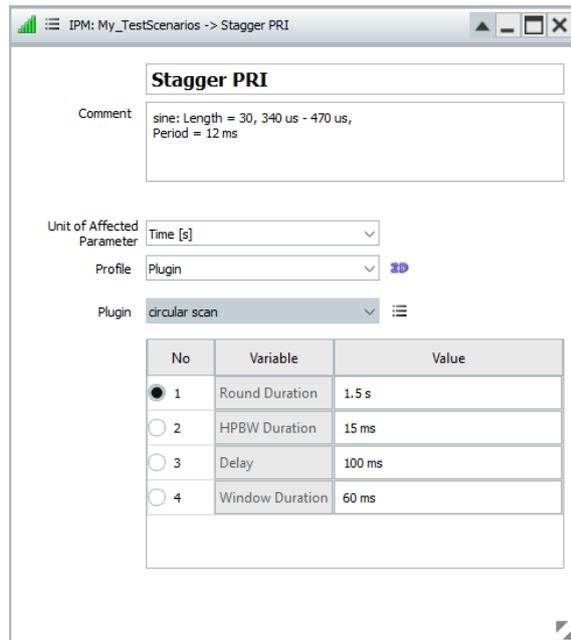
"Plugin Name"   Selects and loads a user-defined IPM profile from a file.
The IPM profile must exist in the "Plug-in" library of the repository.

"Plugin Variables"

Sets the values of the variables defined and used in the plug-in.

Use the standard context menu functions to:
- Reset the plug-in variables
- Access the load "Plug-in" settings, see Chapter 20, "Defining complex modulation schemes and IPM profiles", on page 530

Remote command:

IPM:PLUGin:NAME on page 589

IPM:PLUGin:VARiable:CATalog? on page 689

IPM:PLUGin:VARiable:SELect on page 689

IPM:PLUGin:VARiable:VALue on page 689

Binomial

This IPM profile follows a binomial distribution function.
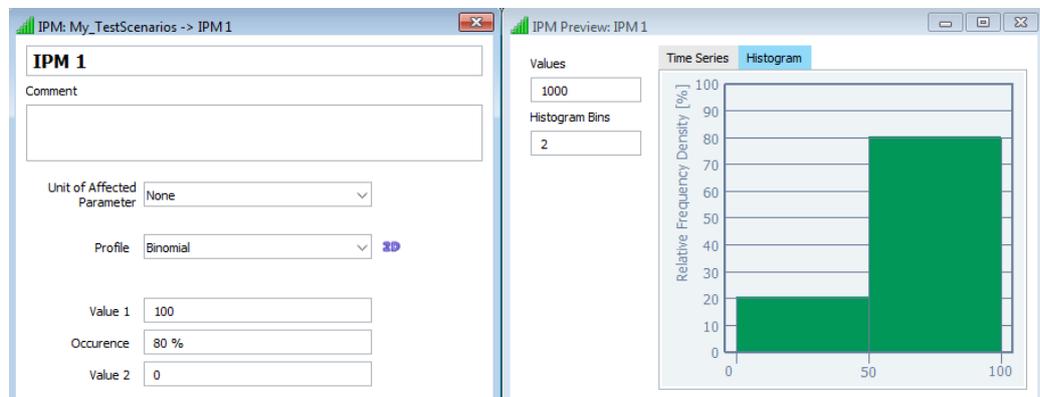
The profile can be used to omit pulses randomly.

"Value 1, Value 2"

Values of the binomial distribution function.

"Occurrence"   Probability of occurrence of value 1 in per cent.
The probability of occurrence of value 2 is calculated as: 100 % - "Occurrence"

Remote command:
IPM:BINomial:VAL1 on page 667
IPM:BINomial:VAL2 on page 667
IPM:BINomial:PVAL1 on page 668

8.1.3 Edit list settings

Edit List
List editor with standard functions.
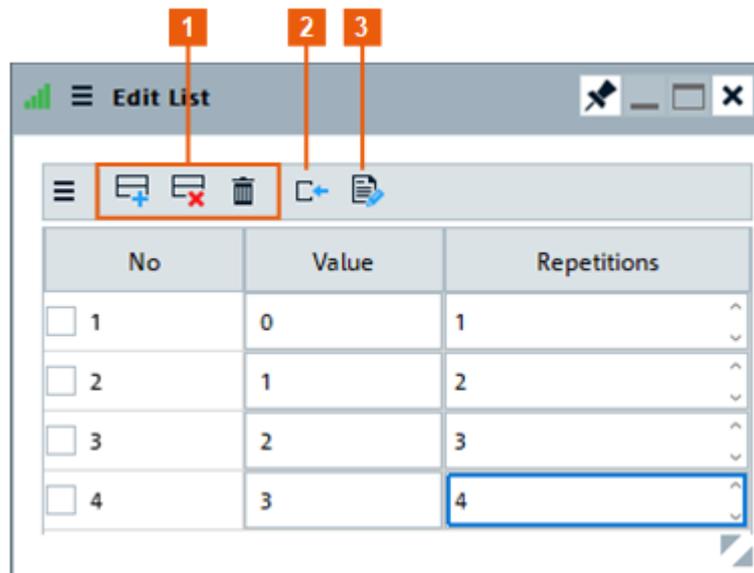
1 = Standard functions for item handling
2 = "Import Data from File" icon
3 = "Populate list" icon

"New/Select"   Accesses standard functions for item handling like "Select/Remove All" or "Insert After".
See Table 2-4.

"Import Data from File"

Opens the Windows Explorer to navigate and select an ASCII file with list data.

"Populate List"   Accesses the "Populate List" dialog.
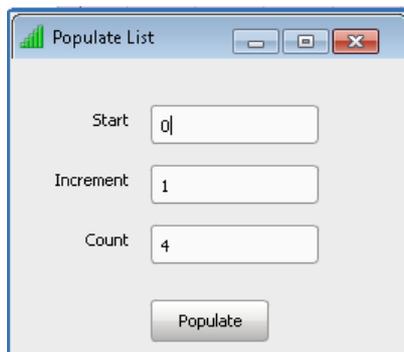
To fill in a list with values automatically, set:
- "Start": start value
- "Increment": step with that the values are incremented
- "Count": number of list items.

"Item"   Selects an item.

"No"   Automatically assigned number to each list item.

"Value"   Enters the value of the list item.

"Repetitions"   Defines how many times a particular list item is repeated.

Remote command:
IPM:LIST:LOAD on page 659
IPM:LIST:SAVE on page 659
IPM:LIST:CLEar on page 595

8.1.4 Inter-pulse modulation (IPM) settings

Access:

► In the "Sequence" dialog, select a pulse item.

Select "IPM > Static".
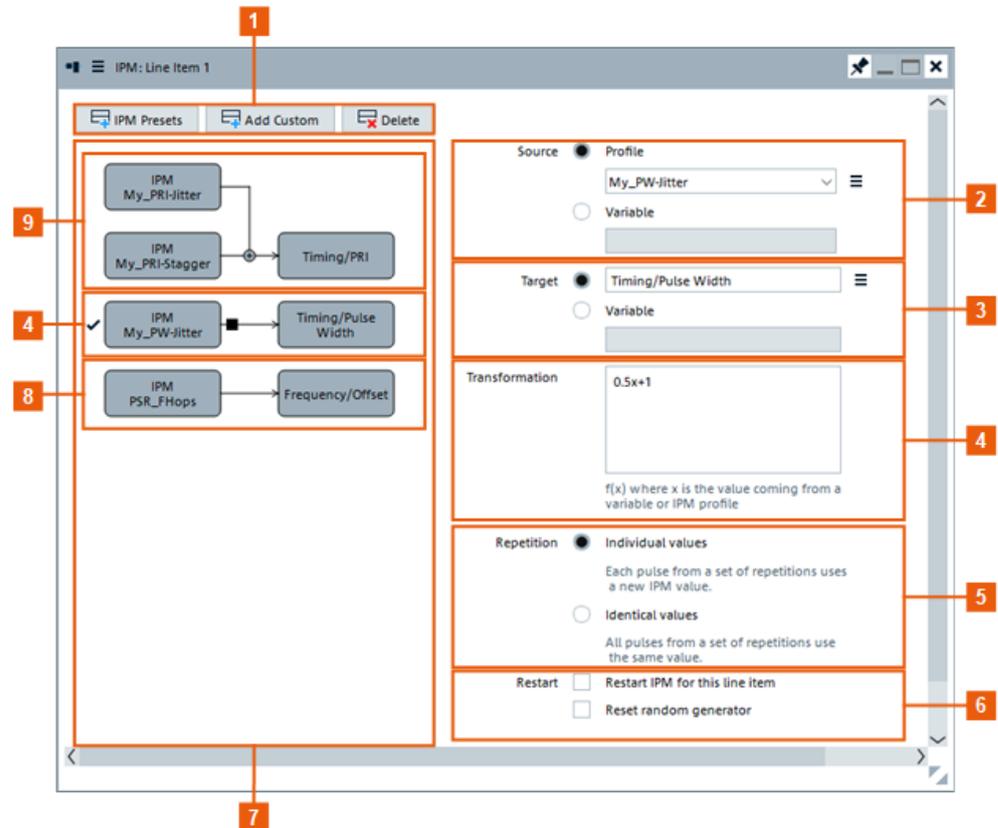
Figure 8-11: IPM: Understanding the displayed information

1 = Standard add/delete functions to create IPM of the selected type or remove the selected IPM from the repository
2 = Variation profile
3 = Defines the pulse parameter or the variable on that the IPM is applied
4 = Enables and defines a transformation
5 = Sets the way repeating pulses are processed
6 = Configures the random generator
7 = Graphical representation of the configured variations and the parameters on that they are applied
8 = Second variation is assigned to a different element ("Target > Frequency > Offset")
9 = Two variations are assigned the same element ("Target > Timing > PRI"); these variations are added

Settings:

Add Custom/Delete

Standard functions to create IPM of the selected type, to disable the selected IPM, or to remove an automatically created one from the repository.

Provided are the following predefined IPM profiles:
- PRI Stagger
- PRI Dwell Switch
- PRI discrete Jitter
- PRI Triangular PFM
- Frequency Agile

Remote command:

SEQuence:ITEM:IPM:ADD on page 591

SEQuence:ITEM:IPM:COUNt? on page 592

SEQuence:ITEM:IPM:SELect on page 593

SEQuence:ITEM:IPM:DELete on page 595

IPM graph

Graphical representation of the configured variations and the elements they are applied on.

The elements are interactive. Right-click to open the context menu.

Source

Parameter variations are defined as profiles or variables.

"Profile"   Lists the available Jitter and IPM profiles in the repository.
To create a profile or to edit an existing one, select the "New/Select" icon.

"Variable"   The variation follows the variable values.
The variable must exist and must be defined. Variable names are case-sensitive.

Variables apply to the following elements:
- **One sequence item** (row in the sequence description table)
- If defined for a **loop**, to all items (pulses, waveforms or nested loops) within the loop.
- In **nested loops**, variables of the parent loop are inherited.

See "To use variables to describe parameter variations" on page 254.

Remote command:

SEQuence:ITEM:IPM:SOURce:TYPE on page 655

SEQuence:ITEM:IPM:SOURce on page 655

SEQuence:ITEM:IPM:SOURce:VARiable on page 655

Target

Profiles can be assigned to an existing pulse parameter or to a variable.

- To select a parameter, use the "New/Select" icon
- To select a variable, enter the variable name (prefix and extension, e.g. "Level_Offset_j"). This variable is available within the current sequence item (row in the sequence description table).
Variable names are case-sensitive.

Parameter variations can be applied on the pulse parameters listed in Table 8-1.

Remote command:

SEQuence:ITEM:IPM:TARGet:TYPE on page 656

SEQuence:ITEM:IPM:TARGet:PARameter on page 656

SEQuence:ITEM:IPM:TARGet:VARiable on page 656

Transformation f(x)

Transformations are an alternative way to describe the output value of the IPM as a mathematical function of the raw value from the IPM profile.

Example:

0.5*x + 1

The example uses the equation $f(x) = a*x+b$, where the multiplier a = 0.5 and the offset b = 1.

See:
- "To use variables to describe parameter variations" on page 254
- Chapter B, "Formula syntax", on page 860

Remote command:

SEQuence:ITEM:IPM:EQUation on page 656

Repetition

Defines the way the variations are applied on repeating pulses:

"Individual value"

A new value is calculated per pulse.

"Identical value"

The same values are used for all repetition of a particular item in the Sequence > Sequence Description Table.
New values are selected each time the same IPM profile is applied to a different item.
Use the "Restart" options to reset an IPM profile.

Remote command:

SEQuence:ITEM:IPM:MODE on page 656

Restart

If the IPM profile uses a random generator, you can:
- Restart the IPM per sequence line item, i.e.:
  - Resets incremented values
  - Starts processing of list from the first item
  - Restarts plug-ins
- Reset the start seed of random generator.

Remote command:

SEQuence:ITEM:IPM:RESTart on page 657

SEQuence:ITEM:IPM:RANDom:RESet on page 657

8.2 How to create IPM profiles and vary pulse parameters

See:

- "To create an IPM with the "Startup Assistant"" on page 245
- "To access the IPM profile dialog" on page 245
- "To create a simple staggered PRI IPM profile" on page 245
- "To assign the staggered PRI profile to the PRI parameter of a pulse" on page 247
- "To visualize the impact of the IPM profiles" on page 249
- "To create and assign a PRI Jitter" on page 252
- "To vary pulse parameters simultaneously" on page 253
- "To create and apply a frequency agile (frequency hopping) pattern" on page 250
- "To use variables to describe parameter variations" on page 254

To create an IPM with the "Startup Assistant"

1. In the menu bar, select "Help > Wizard".

2. Select "Create a new inter-pulse modualtion > On".

3. Select "Next".

4. Select the "Unit" and the "Profile" type.
   See Unit of Affected Parameter and Profile.

To access the IPM profile dialog

Perform one of the following:

1. Select "Repository Tree > IPM > New".

2. Open the "Repository Tree > Sequence > Sequence Description > IPM > IPM/ Static > IPM" dialog and:

   a) On the right of "Source Profile" parameter, open the context menu
   b) Select "New IPM".

   A new empty IPM profile is created and added to the project tree.
   New profiles are named `IPM <n>` where n is a number starting at one.
   You can add information to describe and identify the profile, like a name or a comment.

To create a simple staggered PRI IPM profile

In this example, we create an IPM profile that contains three time values, defined in a list form. We assign this IPM profile to the PRI parameter of a pulse, see "To assign the staggered PRI profile to the PRI parameter of a pulse" on page 247.

Generated is a pulse train containing three pulses, as, for example, the pulse train on Figure 8-2.

1. Open an IPM profile dialog.
   See "To access the IPM profile dialog" on page 245.

2. Select "Unit >Time [s]" and "Profile > List"
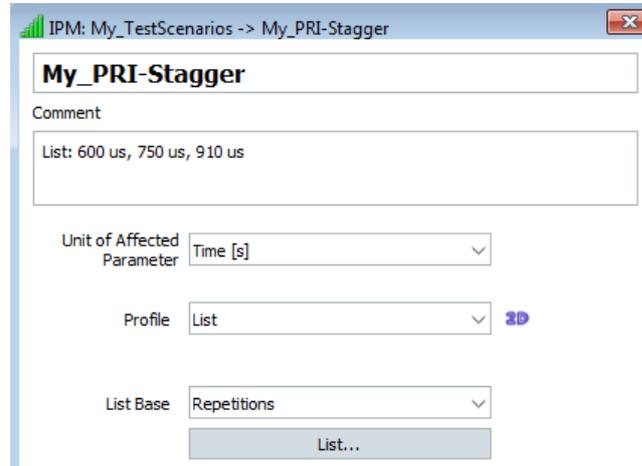
See also Chapter 8.1.2, "IPM profiles settings", on page 229.

3. Select "List...".
   Create a list with three items.

Figure 8-12: Example of a simple staggered PRI IPM profile

1 = PRI_1 = 600 us
2 = PRI_2 = 750 us
3 = PRI_3 = 910 us

The value of each item is set to the required PRI; items are processed once.

4. In the "IPM" dialog, select "2D > Time Series" to visualize the configured IPM profile in the time domain.
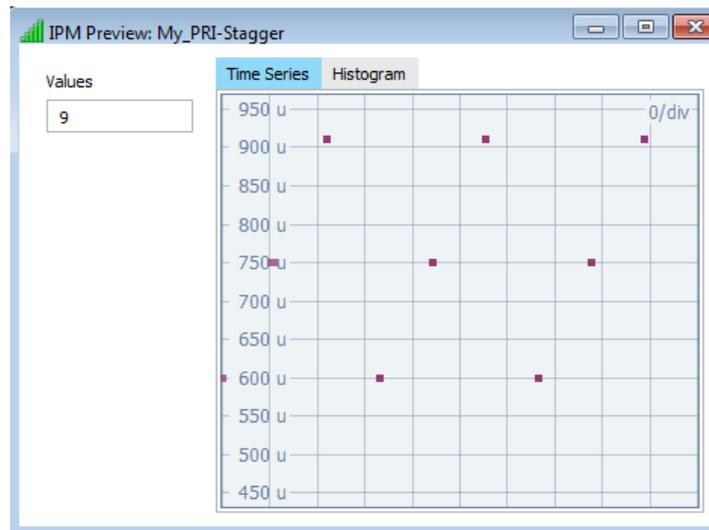
Created is a simple profile with three values only.

5. Select "Histogram" to retrieve statistical information on the IPM profile.
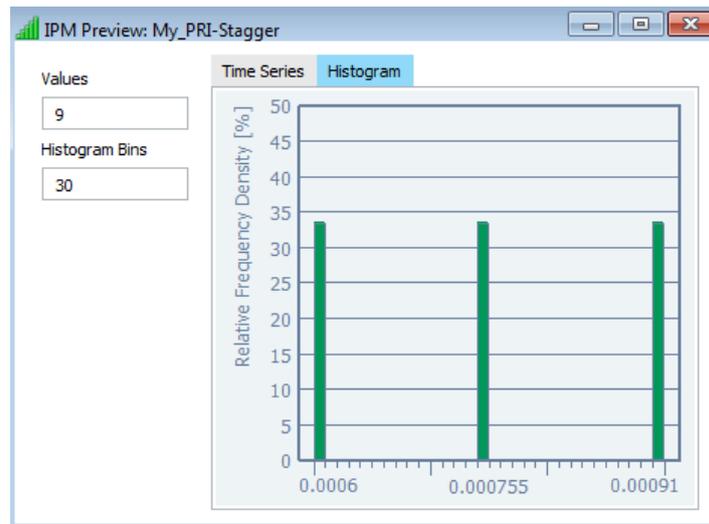
In practice, a staggered PRI profile follows more complex structure and comprises of a larger number of pulses, see for example Figure 8-4.

To assign the staggered PRI profile to the PRI parameter of a pulse

1. Open the pulse sequence to which the created profile is applied, e.g. "My_PT_PRI-Stagger".
   See also "To create a simple sequence" on page 212.

2. In the "Sequence" dialog, select a pulse item.
   Select "IPM > Static".

   Per default, there are no predefined parameter variations in the "IPM" dialog.

3. To assign the created IPM profile:

a) Select "Add Custom".
   New empty profile "IPM None" is created.
b) Select "Source > Profile > My_PRI_Stagger".
c) On the right of "Target" parameter, open the context menu.
   Select "Timing > PRI".
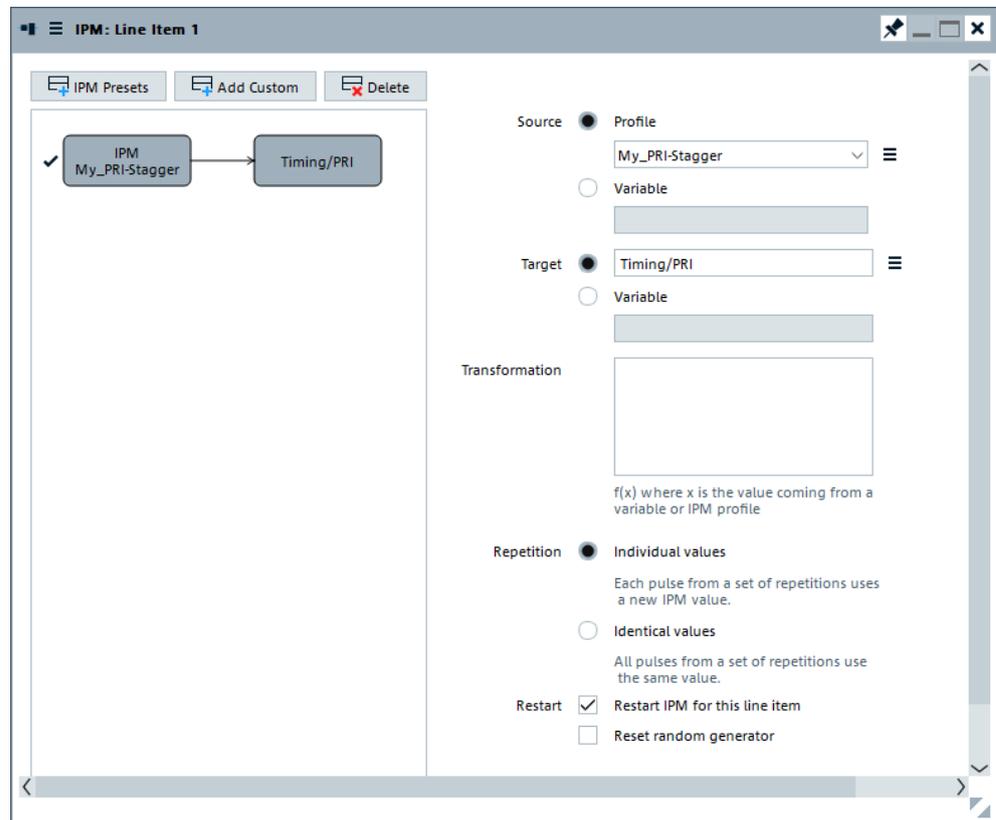
For description of the provided settings, see Chapter 8.1.4, "Inter-pulse modulation (IPM) settings", on page 241.

4. In the "Sequence" dialog:

   a) Set "Repetition Count = 12"
   b) Set "Delay = 230 us" to enable a stagger start delay, see Figure 8-2

The "Sequence" dialog confirms, that a parameter variation varying the PRI is applied ("IPM = IPM" and "PRI = var").

To visualize the impact of the IPM profiles

We assume, that a PRI stagger has been configured and applied to a pulse train, as described in "To assign the staggered PRI profile to the PRI parameter of a pulse" on page 247.

We also assume, that a "Pulse Train" or a "Pulse Train (Collection)" scenario is configured, a signal generator is selected, and the sequence is assigned to this scenario (see Chapter 5.3, "How to select and create a test scenario", on page 145).

To visualize the resulting sequence:

1. In the "Scenario" dialog, select "Signal Calculation > Calculate".

   The software calculates the signal. Green LEDs indicate that the processing is completed.
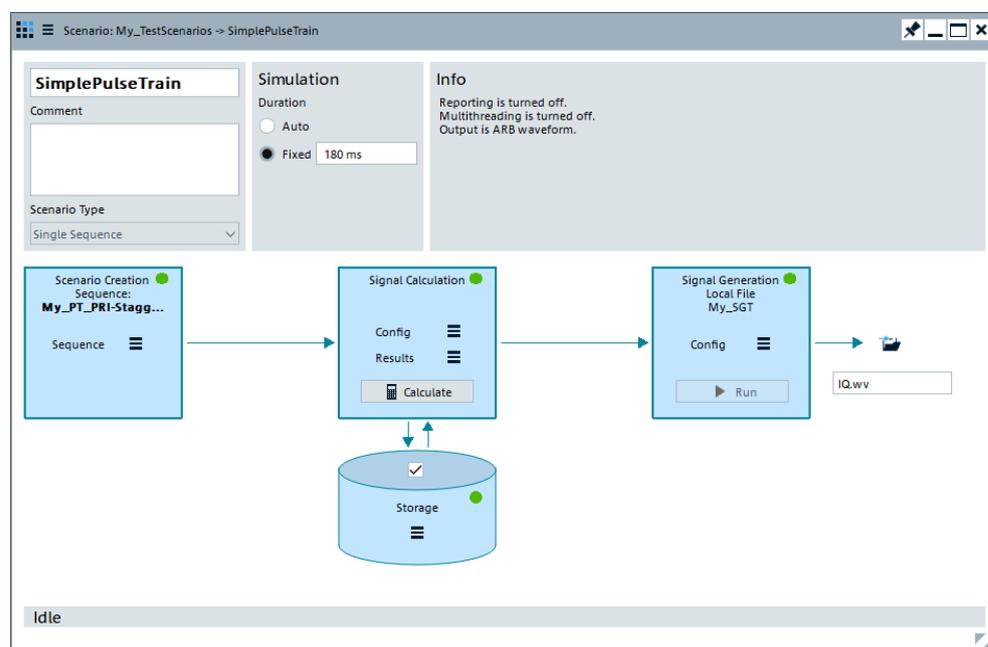
2. Select the sequence in "Signal Calculation > Results > View".

   The "Signal Preview" confirms the configured pulse sequence with enabled PRI stagger.
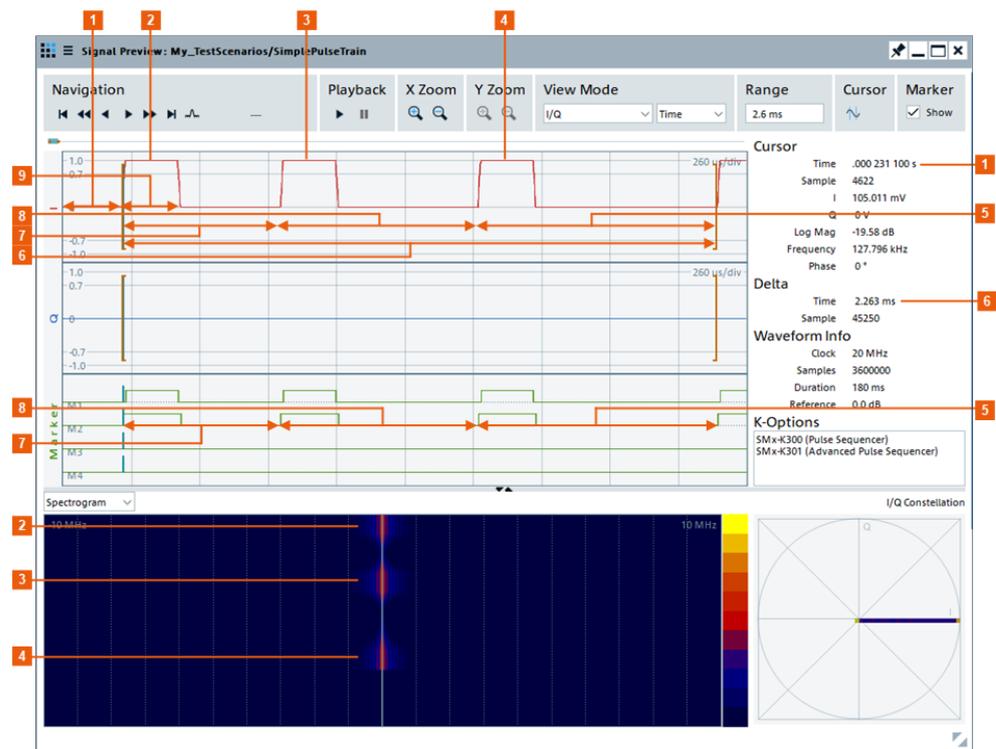
Figure 8-13: Signal preview: Example of a pulse train (PW = 200 us, PRI stagger)

1	= "Delay = 230 ms"
2, 3, 4	= Three unmodulated pulses, also observed on the waterfall diagram
5	= PRI_3 = 910 us
6	= PRI = 2260 us
7	= PRI_1 = 600 us
8	= PRI_2 = 750 us
9	= Pulse width

For detailed description of the provided settings, see Chapter 15.1, "Waveform view and signal preview settings", on page 400.

To create and apply a frequency agile (frequency hopping) pattern

1. In the "Sequence" dialog, select "IPM" to open the "IPM" dialog.

2. Select "Frequency Agile".

   A new IPM profile is created, automatically named and assigned to the "Frequency/ Offset" parameter.

3. Configure the parameters, for example, as shown on the following figure.
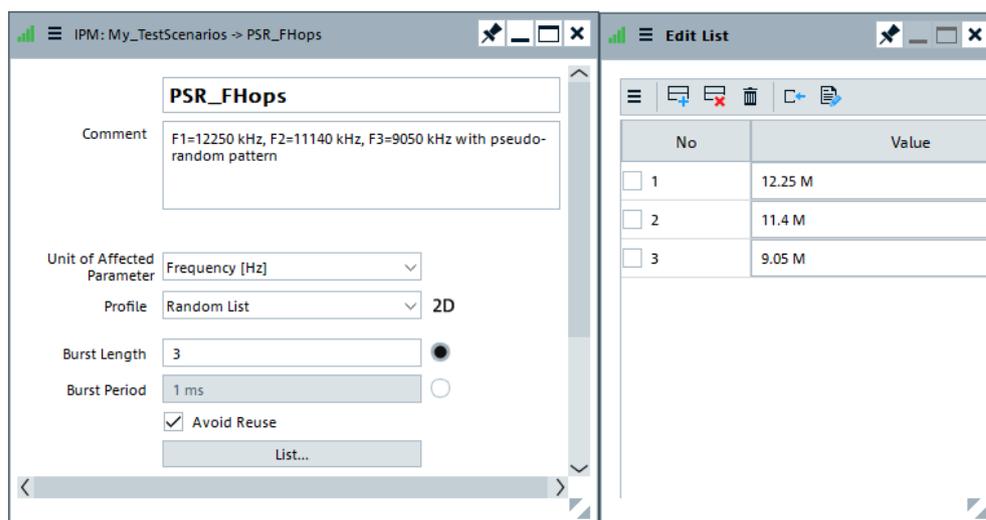
4. Use the 2D view to display the configured pattern.
   See also Figure 8-7.

5. Observe the "Sequence" dialog.

   The sequence configuration settings confirm the configuration.

6. Use the "Signal Preview" dialog to visualize the generated signal.

   See also:
   ● "To visualize the impact of the IPM profiles" on page 249

● Chapter 15.1, "Waveform view and signal preview settings", on page 400
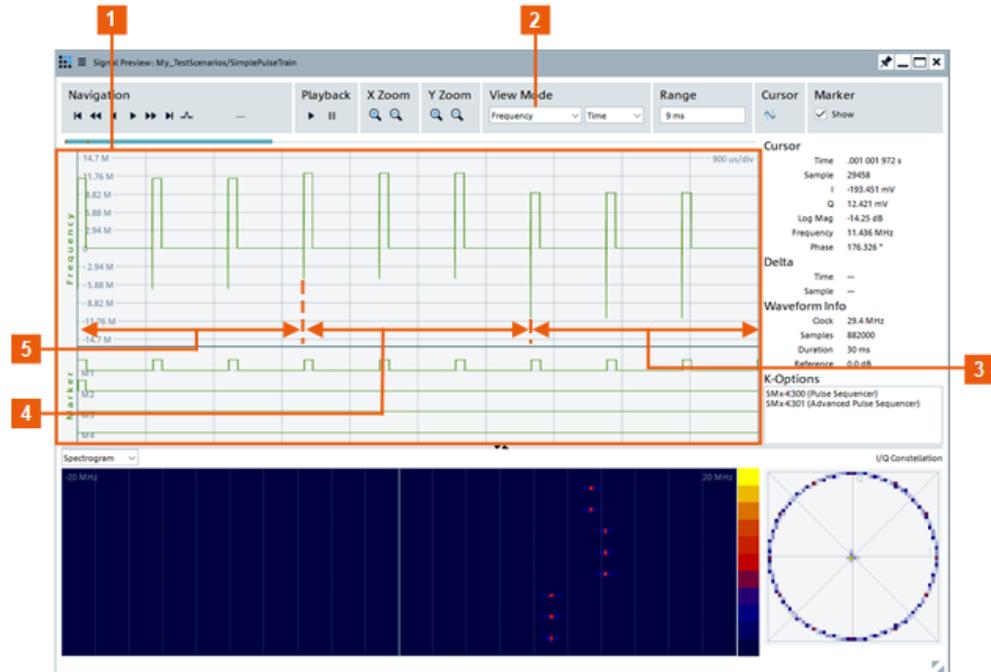
Figure 8-14: Signal preview: Example of a pulse train (PW = 100 us, frequency hopping)

1 = 30 standard pulses; 3 consecutive pulses per hopping frequency
2 = "View Mode > Frequency"
3 = 9050 kHz
4 = 12250 kHz
5 = 11140 kHz

The signal will hop the three frequencies according to a pseudo-random pattern. Each hop frequency is used three times.

To create and assign a PRI Jitter

There is a dedicated function for creating PRI Jitters, the "PRI Discrete Jitter" in the "IPM" dialog (see also "To create and apply a frequency agile (frequency hopping) pattern" on page 250).

In this example, however, we create manually a randomly varying IPM profile with 20 different values. Within 20 ms, the PRI varies in the range 1.1 ms to 1.15 ms.

1. Create an IPM Profile that uses "Profile > Random" and, for example, a "Distribution > Uniform Distribution".
   See also "To access the IPM profile dialog" on page 245.
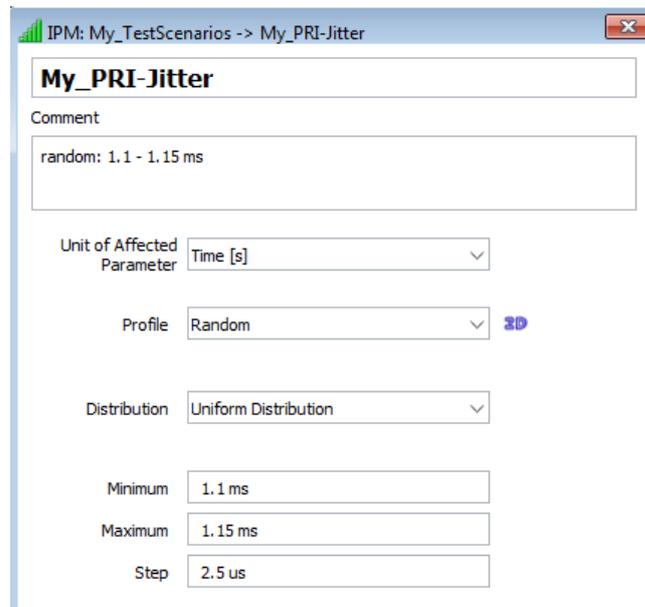
2. Select "2D" to visualize the function.

3. Open the required sequence. Assign the IPM profile to a pulse train.
   See "To create and apply a frequency agile (frequency hopping) pattern"
   on page 250.

To vary pulse parameters simultaneously

We assume, that a PRI jitter has been configured and applied to a pulse train, as
described in "To create and assign a PRI Jitter" on page 252.

To apply a second variation, for example a PW jitter, perform the following:

1. Create an IPM Profile, for example as shown on the following figure.
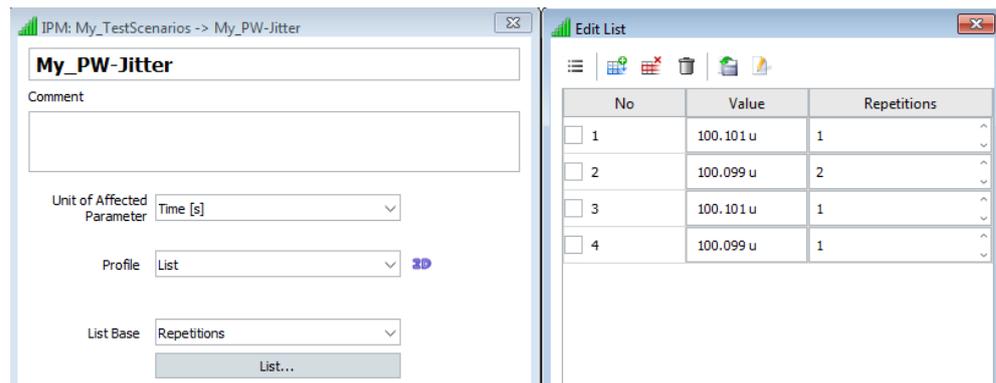
   Configured is a pulse width (PW) pattern with five values.

2. Open the required sequence.
   Assign the IPM profile to the same pulse train as the PRI Jitter.
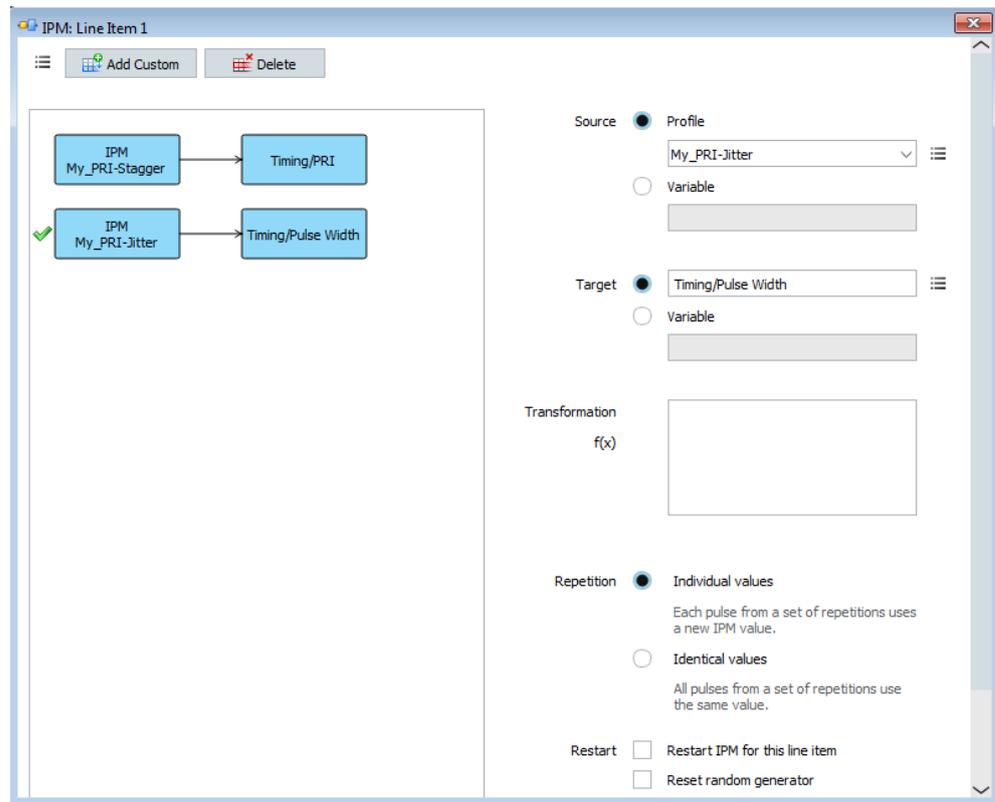
Tip: You can vary one pulse parameter with more than one profile, see Figure 8-11.

Configured is a pulse train, consisting of 20 pulses, each with different PRI value. The PW pattern is applied on five consequent pulses and is repeated four times. In this example, the PRI varies in the range 1.1 ms to 1.15 ms; the PW between 100.101 us and 100.099 us.

To use variables to describe parameter variations

In this example, we create a sequence of two pulses P1 and P2 that are included in a loop. We use an IPM step profile that creates ten values within the range -10 dB to 0 dB.

We assign this IPM profile to the loop and use it to create and vary a variable. This variable in turn is the source for the variations of the two pulses, included in the loop.

1. Create two unmodulated pulses with pulse width of 10 us.

2. Create a sequence with the following settings:

3. Create a step profile with the following settings:

4. Assign the IPM profile to the loop item.

   a) Select "Target > Variable".
   b) Set the variable name.

5. For both pulses:

   a) Assign the created variable as a source of the IPM variations.

b) Set the "Target > Level > Offset".
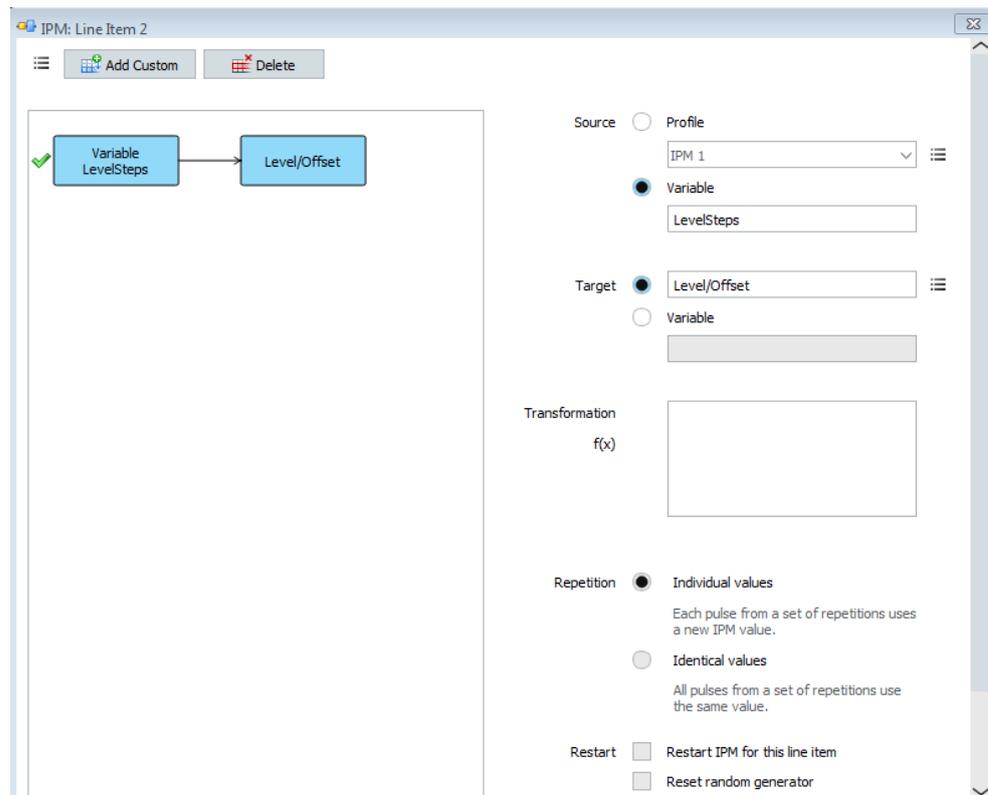
6. Only for "Pulse 2", type into "Transformation f(x)" *x -10*.

7. Assign the sequence to a scenario.
   Calculate the waveform.
   Open the "Signal Preview" dialog.
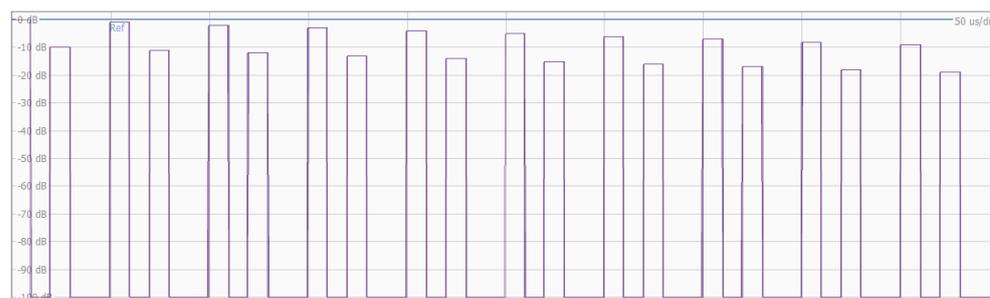
The display confirms that the level of "Pulse 1" decreases from 0 dB with step of 1 dB.
The level of "Pulse 2" follows the same pattern but the level is 10 dB lower than the "Pulse 1" level.

9 Defining antenna patterns and antenna scans

This section informs you about the antenna pattern types and antenna scan types provided by R&S Pulse Sequencer. It also helps you select and configure a subset of antenna characteristics as a basis for the emitter configuration and its different operating modes.

9.1 Overview of the main antenna parameters

Transmitting antennas are elements that radiate energy. Energy emitted in a particular direction is referred to as a *beam* or *lobe*. The direction in which the most energy is radiated is the *main beam* or *main lobe* (see Figure 9-1). Energy radiated energy in other directions (i.e. not in the main lobe) forms the side lobes and the back lobe.

The beamwidth is a key parameter. Beamwidth is usually defined as an angle. The angle is measured at the transmitting antenna (centered on the beam-axis) as shown in Figure 9-1. The value (e.g. 5°) indicates the angle at which the power of the beam is 50% (3 dB) lower than the peak power on the beam-axis. This angle is referred to as the half-power beamwidth (HPBW).
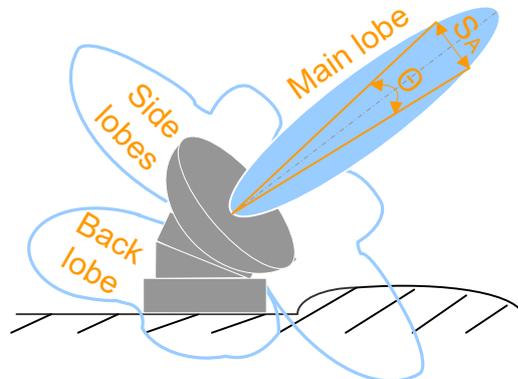
Figure 9-1: Main characteristics of an antenna

Main lobe/bream = The region around the direction with maximum radiation
Side lobes = Energy radiated in undesired direction
S_A = Angular resolution
Θ (theta) = Antenna beamwidth or the -3 dB angle, defined by the half-power (HPBW)

Antenna pattern

Antennas of different type and with different purpose use wider or more directed beams, for example pencil beams have a typical HPBW of about 2 deg.

The R&S Pulse Sequencer provides a wide range of commonly used antenna patterns, such as parabolic, cosecant squared, or planar phased array antennas. Moreover you can:

- Adjust the antenna pattern settings to simulate, for example, a directive antenna that concentrates the radiated energy in a particular direction
- Simulate a back lobe pattern
- Create a custom antenna pattern based on the required antenna characteristics
  (see "To create a custom antenna pattern based on the required antenna characteristics" on page 298)
- Import your custom antenna pattern
  (see "To import a user-defined antenna pattern from file in one of the predefined file formats" on page 299 and "To import an antenna pattern file in custom file format" on page 302)
- Design your phased array antenna
- Visualize the antenna pattern on a 2D and a 3D preview, see Figure 9-12.

Antenna scan

Antenna scans describe the movement of the antenna beam in three-dimensional space. A conical scan, for example, is a movement where the beam points in the direction of the target and is rotated in a circle. The half-power points of the beam draw a circle.

The R&S Pulse Sequencer provides a wide range of commonly used antenna scans, such as a raster scan, helical scan, or conical scan. You can also:

- Adjust the antenna scan settings
- Visualize the configured antenna scan on a 3D preview
- Design your antenna scan
- Observe the signal received by a receiver for a particular antenna pattern and antenna scan configuration
  See:
  - "Received signal amplitude" on page 348
  - "To visualize the signal received by a static receiver" on page 321

Antenna polarization and polarization loss

The polarization of an antenna describes the orientation of the electric wave radiated by the antenna. Polarization can be constant, or it can rotate with each wave cycle.

The Table 9-1 gives an overview and a short description of the supported polarization types.

Table 9-1: Polarization types

Polarization	Type	Short description
Vertical (V)	Linear polarization	The electrical field is vertical to the propagation direction (y-axis).
Horizontal (H)		The electrical field is horizontal to the propagation direction (y-axis).

Polarization	Type	Short description
Circuit right (CR)	Circular polarization	The electrical field rotates clockwise (CW) relative to the propagation direction.
Circuit left (CL)		The electrical field rotates counterclockwise (CCW) relative to the propagation direction.
Slant right (SR)	Linear polarization	The electrical filed is rotated at 45° from the vertical case.
Slant left (SL)		The electrical filed is rotated at -45° from the vertical case

The theoretical maximum received power is achieved if the transmit and the receive antenna are co-polarized and aligned. That is, if the antennas at the emitter and the receiver both use the same polarization and point to each other. If the antennas use different polarizations, the received power decreases. The polarization loss is given by the ratio of the received power to the maximum power. Theoretically, if the transmit and the receive antennas are cross-polarized, the polarization loss is infinite. In practice, however, this value is not reached. In this implementation, the polarization loss value used if the antennas are cross-polarized is a user-definable value.

Table 9-2 lists the polarization loss for any combination of transmit and receive antenna polarizations.

Table 9-2: Polarization loss values depending on the different antenna combinations

Polarization of the transmit antenna	Polarization of the receive antenna	Polarization loss [dB]
Vertical (V)	Vertical (V) (co-polarized antennas)	0
	Horizontal (H) (cross-polarized antennas)	$-\infty$ (As set with the parameter Maximum Attenuation for Antenna Cross Polarization)
	Slant right (SR) or Slant left (SL)	- 3
	Circuit right (CR) or Circuit left (CL)	- 3
Horizontal (H)	Horizontal (H) (co-polarized antennas)	0
	Slant right (SR) or Slant left (SL)	- 3
	Circuit right (CR) or Circuit left (CL)	- 3
Circuit right (CR)	Circuit right (CR) (co-polarized antennas)	0

Polarization of the transmit antenna	Polarization of the receive antenna	Polarization loss [dB]
	Circuit left (CL) (cross-polarized antennas)	–∞ (As set with the parameter Maximum Attenuation for Antenna Cross Polarization)
Circuit right (CR) or Circuit left (CL)	Slant right (SR) or Slant left (SL)	- 3

See:

- "Polarization" on page 263
- "Polarization" on page 265
- "To observe the effect of polarization" on page 302

9.2 Antenna pattern settings

Access:

► Select "Repository Tree > Antenna Pattern > New".

Settings

Antenna patterns are characterized by the following common and specific settings:

Antenna Pattern Name

Enter the name of the antenna pattern.

Remote command:

`ANTenna:NAME` on page 589

`ANTenna:CATalog?` on page 587

`ANTenna:CREate` on page 588

`ANTenna:SELect` on page 588

`ANTenna:REMove` on page 590

Comment

Enter a short description.

Remote command:

`ANTenna:COMMent` on page 589

Type

Selects the antenna pattern. Use the "2D" and "3D" diagrams to visualize the polar antenna gain patterns.

Further provided antenna pattern settings depend on the selected antenna model, see:

● Dipole Antenna Settings
● Parabolic Antenna Settings
● Gaussian Antenna Settings
● Sin(x)/x Antenna Settings
● Pyramidal Horn Antenna Settings

- Cosecant Squared Antenna Settings
- Planar Phased Array Antenna Settings
- Imported antenna settings
- Custom Antenna Settings
- Custom Phased Array Settings
- Cardioid Antenna Settings

Remote command:
ANTenna:MODel:TYPE on page 604

Z-Rotation, X-Rotation

Rotates the antenna pattern around the selected axis.

Table 9-3: Effect of antenna pattern rotation

"X-Rotation = 15°"	"Z-Rotation = 15°"
1) rotation around the x-axis	2) rotation around the z-axis
Positive values rotate the antenna boresight towards the positive z-axis.	Positive values rotate the antenna boresight counterclockwise towards the x-axis.
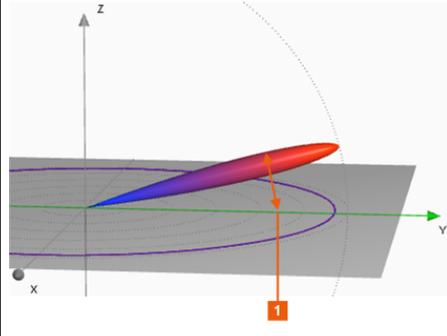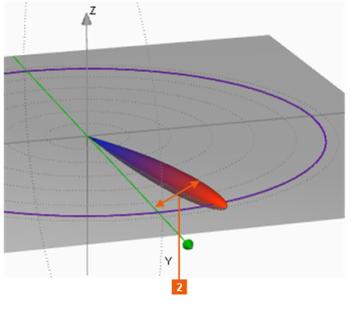
This parameter is useful for example:
- To simulate a radar boresight displacement
- To align a user-defined antenna pattern to the boresight;
  (see "To import a user-defined antenna pattern from file in one of the predefined file formats" on page 299).

Remote command:
ANTenna:MODel:ROTation:X on page 613
ANTenna:MODel:ROTation:Z on page 613

Frequency, Bandwidth

Sets the frequency and bandwidth the antenna pattern is calculated for.

See also:
- Frequency
- Emitter Modes

Remote command:
ANTenna:MODel:FREQuency on page 611
ANTenna:MODel:BANDwidth on page 611

Polarization

Sets the antenna polarization.

The polarization of an antenna describes the orientation of the electric wave radiated by the antenna. Polarization can be constant, or it can rotate with each wave cycle.

The selected polarization is indicated on the 3D diagram.
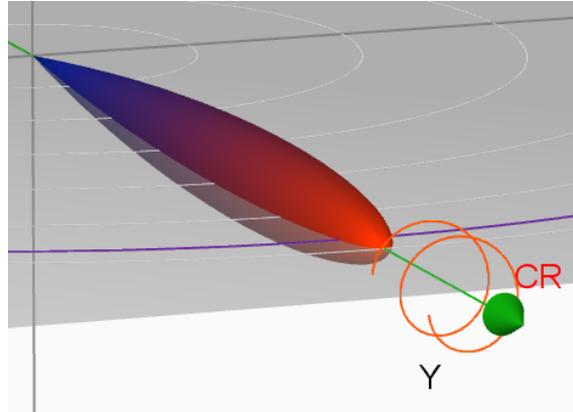
Figure 9-2: Indication of the selected polarization on the 3D diagram

See also "Antenna polarization and polarization loss" on page 258.

"Vertical (V)"   Linear polarization.
                    The electrical field is vertical to the propagation direction (y-axis).

"Horizontal (H)"
                    Linear polarization.
                    The electrical field is horizontal to the propagation direction (y-axis).

"Circuit Right (CR)"
                    Circular polarization.
                    The electrical field rotates clockwise (CW) relative to the propagation direction.

"Circuit Left (CL)"
                    Circular polarization.
                    The electrical field rotates counterclockwise (CCW) relative to the propagation direction.

"Slant Right (SR)"
                    Linear polarization.
                    The electrical filed is rotated at 45° from the vertical case.

"Slant Left (SL)"
                    Linear polarization.
                    The electrical filed is rotated at -45° from the vertical case.

Remote command:
ANTenna:MODel:POLarization on page 604

Simulate Back Lobe, Attenuation, Type
Enables the simulation of a back lobe and sets its power level and shape.

The parameter "Type" defines the shape of the back lobe pattern:
● "Mirror": the back lobe pattern is the mirrored antenna pattern, attenuated with the selected value, see Figure 9-13.

- "Omnidirectional": the back lobe pattern is a hemisphere, attenuated with the selected value.

Remote command:
ANTenna:MODel:BACKlobe:ENABle on page 608
ANTenna:MODel:BACKlobe:TYPE on page 608
ANTenna:MODel:BACKlobe:ATTenuation on page 608

Resolution

This parameter is only available when you use R&S Pulse Sequencer in CPU mode. Either select CPU mode on initial start-up, or change to CPU mode by selecting "Configure > Settings... > Graphics > GPU Support" and deselecting GPU support. Changing modes requires you to restart the application.

Sets a custom resolution for the antenna pattern simulation.

Use this parameter to reduce interpolation errors when simulating antennas with very narrow HPBWs and side lobes.

Remote command:
ANTenna:MODel:SINC:RESolution on page 612

2D and 3D diagrams

You can visualize the antenna characteristics as 2D and 3D interactive diagrams.

2D ← 2D and 3D diagrams

This diagram is a plot of the radiated energy, measured at different angles and at a constant distance from the antenna. The 2D diagram indicates the calculated HPBW and the 3 dB line ("Show 3dB Line > On").

The 2D diagram display the antenna pattern as a XY (azimuth cut) or YZ (elevation cut) graph with polar or with Cartesian coordinates.
- The polar diagram shows the radiation in all possible directions, i.e. the main, side and back lobes, see Figure 9-12.
- The Cartesian diagram is useful to determine the main lobe to back lobe and main lobe to side lobes ratio of highly directive antennas.

Remote command:
PLOT:POLar:TYPE on page 614
PLOT:POLar:CUT on page 614
PLOT:POLar:LOG:MIN on page 615

3D ← 2D and 3D diagrams

This diagram displays the radiated energy as a 3D spatial radiation model.

Minimum Level ← 2D and 3D diagrams

Defines the minimum displayed side lobes level.

Polarization ← 2D and 3D diagrams

Shows the co-polarized and the cross-polarized antenna pattern.

The co-polarized and cross-polarized antennas are selected automatically for the current antenna, see Table 9-2. The cross-polarization is calculated at a polarization loss, as selected with the parameter Maximum Attenuation for Antenna Cross Polarization.

See "To observe the effect of polarization" on page 302.

This function cannot be used for user-defined antenna patterns imported as a file.

Pattern ← 2D and 3D diagrams

A file describing custom antenna pattern can contain more than one antenna patterns. If a custom antenna pattern is loaded, this parameter selects the antenna pattern number that is displayed.
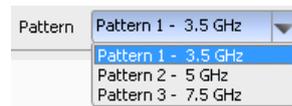
The parameter also indicates the frequency for that the antenna pattern is designed.

Electronic Steering > Z-/X-Rotation ← 2D and 3D diagrams

For "Antenna Model > Custom Phased Array", turns the antenna diagram around the selected axis.

Dipole Antenna Settings

A dipole antenna is a simple antenna with a radiation pattern, shaped like a toroid (doughnut) symmetrical about the axis of the dipole.
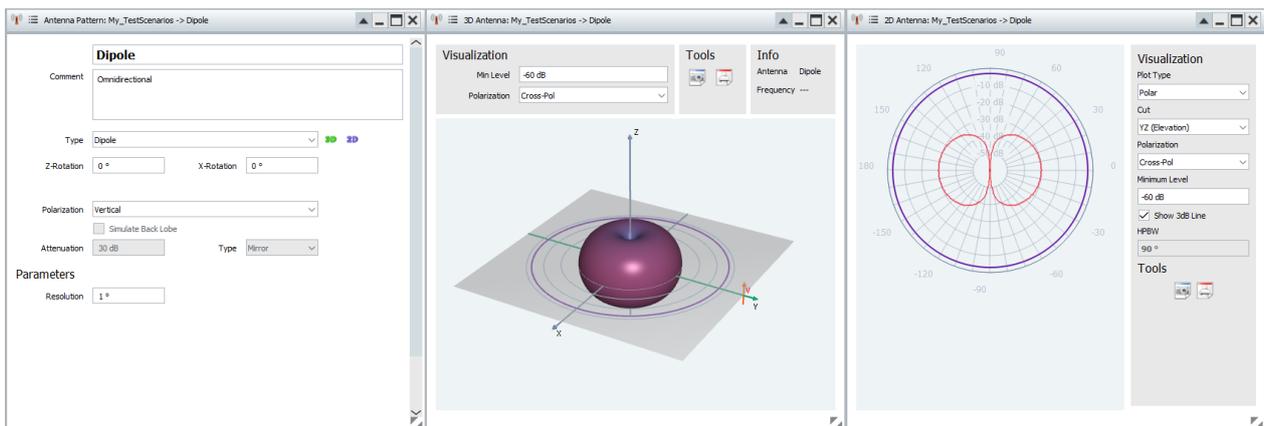
Parabolic Antenna Settings

A parabolic antenna has a curved surface with the cross-sectional shape of a parabola with a user defined "Diameter".

Typically, a parabolic antenna radiates the power in a narrow main lobe along the antenna y-axis.
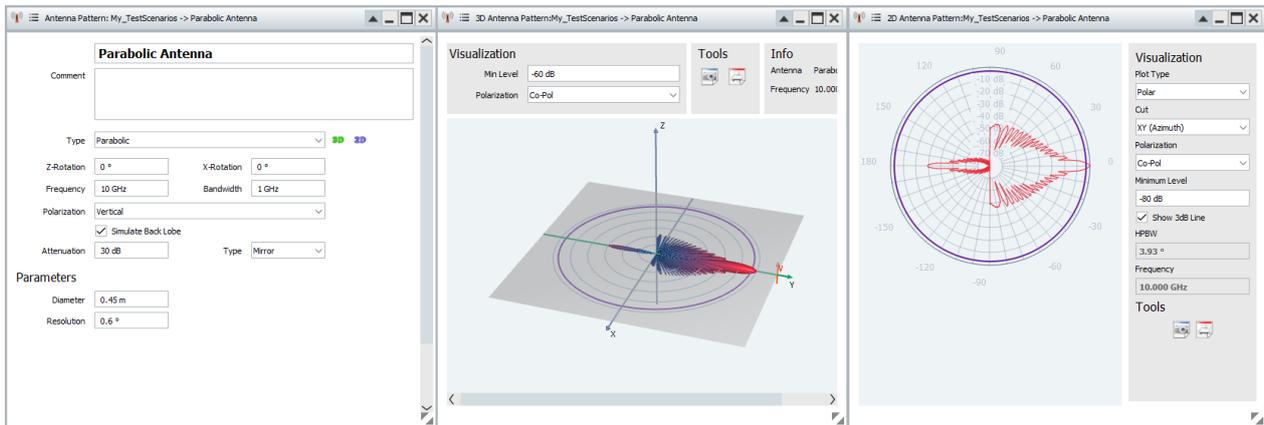
Remote command:

ANTenna:MODel:PARabolic:DIAMeter on page 613

Gaussian Antenna Settings

An antenna with radiation pattern that follows the Gaussian distribution.

The main parameter is the HPBW that specifies the angular width within which the antenna is most sensitive. The HPBW can be defined for azimuth and elevation separately.
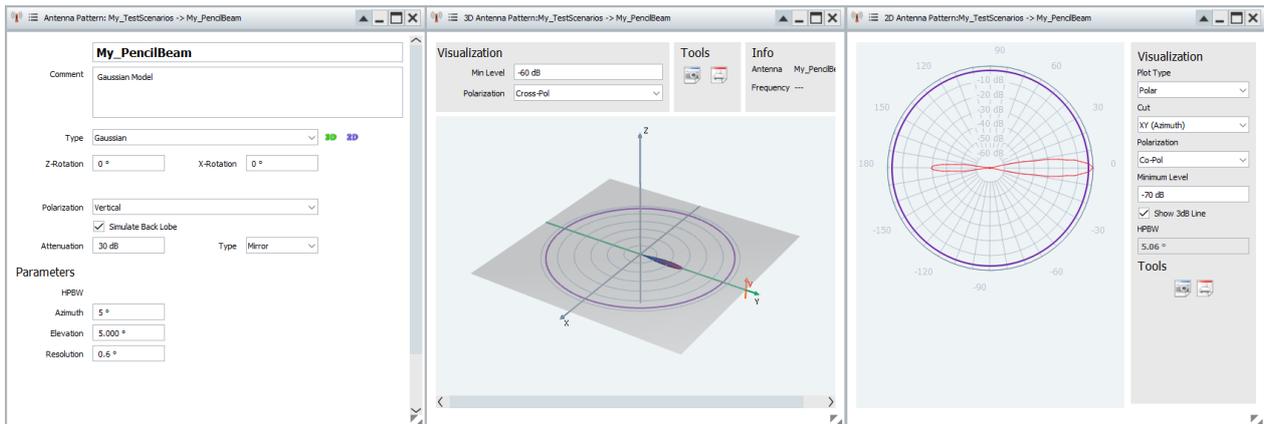
Remote command:

ANTenna:MODel:GAUSsian:HPBW:AZIMuth on page 612

ANTenna:MODel:GAUSsian:HPBW:ELEVation on page 612

Sin(x)/x Antenna Settings

An antenna with radiation pattern that follows the Sin(x)/x distribution.

The antenna is characterized by the HPBW. The HPBW can be defined for azimuth and elevation separately.
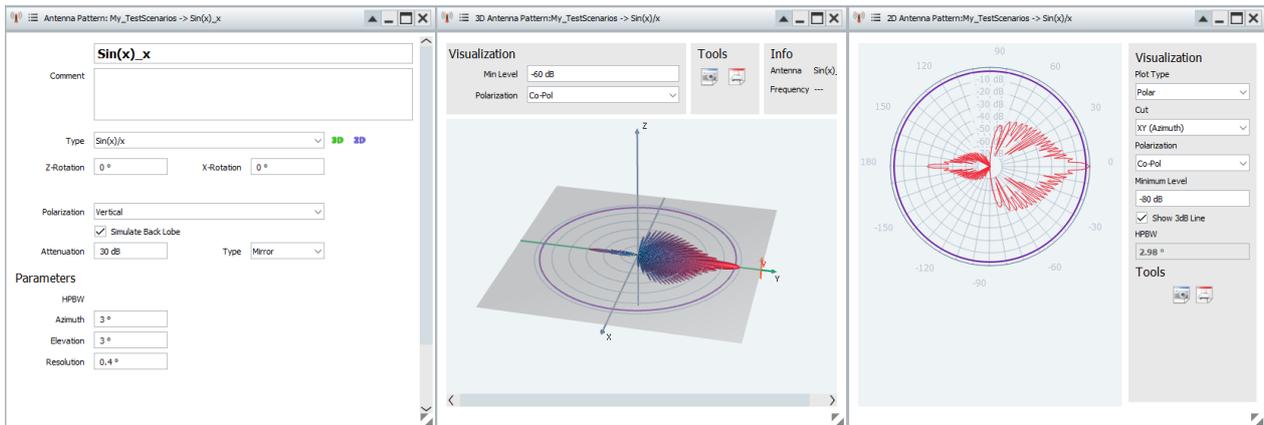
Remote command:

ANTenna:MODel:SINC:HPBW:AZIMuth on page 612

ANTenna:MODel:SINC:HPBW:ELEVation on page 612

Pyramidal Horn Antenna Settings

A pyramidal horn antenna has a horn shape in the form of a four-sided pyramid and a rectangular cross section. The lengths of the rectangular sides "Length X, Length Z" are configurable.
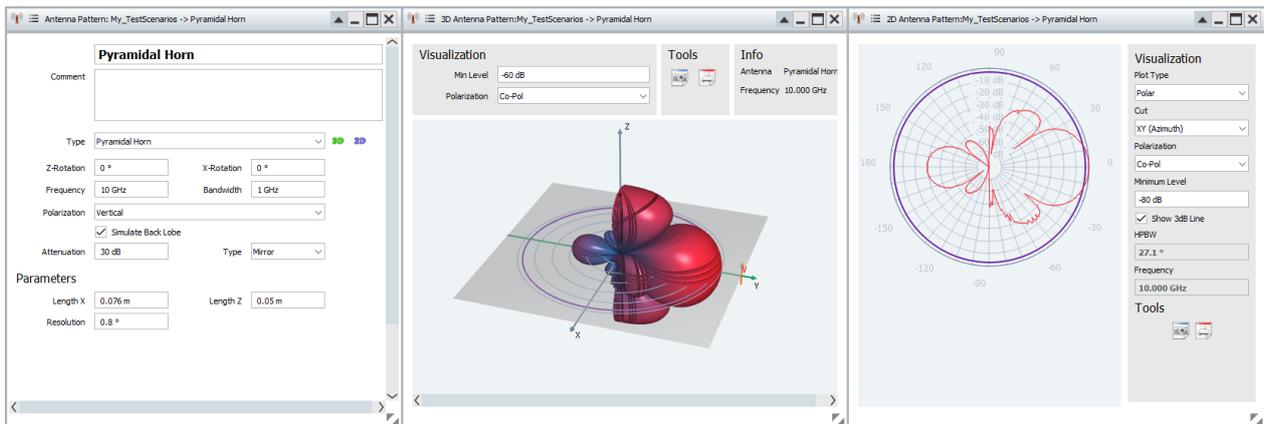
Remote command:

ANTenna:MODel:HORN:LX on page 612

ANTenna:MODel:HORN:LZ on page 612

Cosecant Squared Antenna Settings

Cosecant squared antennas are designed for air-surveillance radar sets.
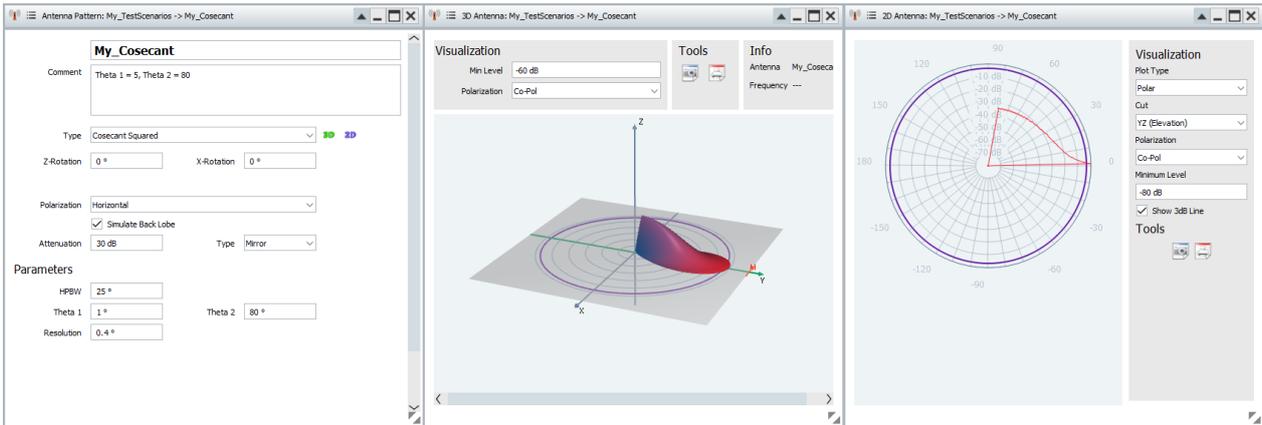
Remote command:
ANTenna:MODel:COSecant:HPBW on page 611
ANTenna:MODel:COSecant:T1 on page 608
ANTenna:MODel:COSecant:T2 on page 608

Planar Phased Array Antenna Settings
A planar phased array antenna consists of several antenna elements.
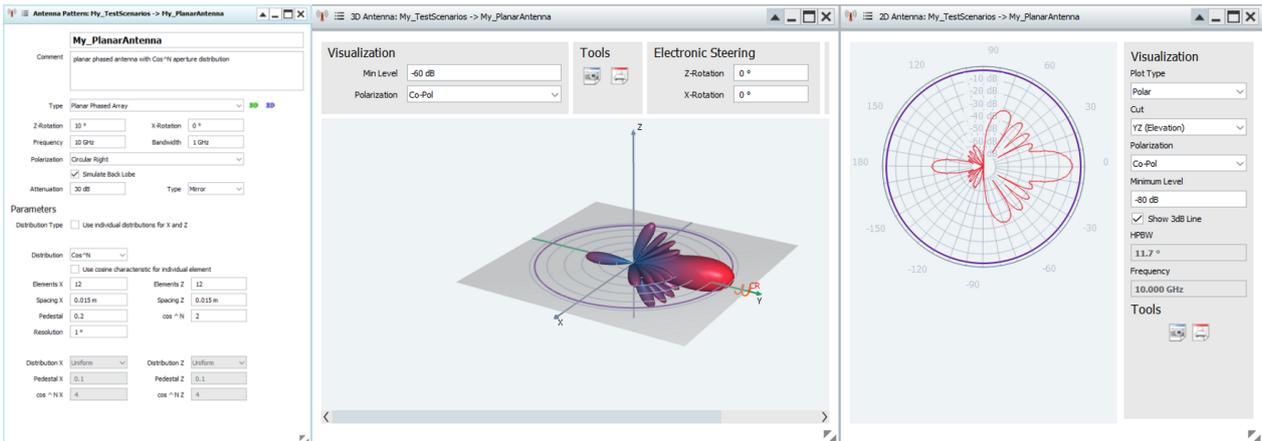
The characteristics of the individual antenna elements influence the transition between the main/side lobes and the back lobes of the phased array antenna, see Table 9-4.

See also "To create an antenna pattern" on page 294.

"Distribution Type"

Enables using the individual distribution function for X and Z direction.

Remote command:
ANTenna:MODel:ARRay:DISTribution:TYPE on page 605

"Distribution" 　　Determines the antenna pattern as a function of the variables x and z, the pedestal level P and the coefficient N:

Where:
- $P = 0$ to 1
  As set with the parameter "Pedestal".
- $N = 2$ to 10
  As set with the parameter "$\cos^N$"
- $x = -1$ to +1
  $z = -1$ to + 1

To simplify the description, only the $f(x)$ functions are listed. The $f(z)$ values are calculated in the same manner.

The distribution functions are calculated as follows:
- "Uniform"
  $f(x) = 1$
- "Parabolic"
  $f(x) = 1 - (1 - P)*x^2$
- "Cosine"
  $f(x) = P + (1 - P)*\cos(0.5*x*\pi)$
- "$\cos^2$"
  $f(x) = P + (1 - P)*0.5*(1+\cos(x*\pi))$
- "$\cos^N$"
  If $N > 2$, $f(x) = P + (1 - P)*(\cos(0.5*x*\pi))^N$
  For $N = 2$, $f(x) = 1$
- "Triangular"
  If $N > 2$, $f(x) = P + (1 - P)*(1 - abs\ (x))$
  (the *abs* function returns the absolute value of the variable)
  For $N = 2$, $f(x) = 1$
- "Hamming"
  $f(x) = P+(1-P)*(0.54-0.46*\cos((1.0+x)*\pi))$
- "Hann"
  $f(x) = P+(1-P)*(0.5-0.5*\cos((1.0+x)*\pi))$

Remote command:
ANTenna:MODel:ARRay:DISTribution on page 605

"Use cosine characteristic for individual antenna element"
　　Per default, the individual antenna elements use omnidirectional characteristic. If the parameter "Use cosine characteristic for individual antenna element" is enabled, the cosine characteristic is used.

Remote command:
ANTenna:MODel:ARRay:ELEMent:COSine on page 606

"Elements, Spacing"
　　Number of radiating elements in X/Z direction and spacing between the elements.

Remote command:
ANTenna:MODel:ARRay:NX on page 606
ANTenna:MODel:ARRay:NZ on page 606
ANTenna:MODel:ARRay:XDIStance on page 608
ANTenna:MODel:ARRay:ZDIStance on page 608

"Pedestal"   Sets the pedestal level P.

Remote command:
`ANTenna:MODel:ARRay:PEDestal` on page 607

"cos^N"   Sets the value of the coefficient N in the $\cos^N$ distribution.

Remote command:
`ANTenna:MODel:ARRay:COSN` on page 604

"Distribution X"/"Distribution Z"
Requires "Distribution Type" enabled.
Sets the individual distribution function for X and Z direction.

Remote command:
`ANTenna:MODel:ARRay:DISTribution:X` on page 606
`ANTenna:MODel:ARRay:DISTribution:Z` on page 606

"Pedestal X"/"Pedestal Z"
Requires "Distribution Type" enabled.
Sets the individual pedestal level for X and Z direction.

Remote command:
`ANTenna:MODel:ARRay:PEDestal:X` on page 607
`ANTenna:MODel:ARRay:PEDestal:Z` on page 607

"cos^N X"/"cos^N Z"
Requires "Distribution Type" enabled.
Sets the individual value of the coefficient N in the $\cos^N$ distribution for X and Z direction.

Remote command:
`ANTenna:MODel:ARRay:COSN:X` on page 604
`ANTenna:MODel:ARRay:COSN:Z` on page 604

Remote command:
`ANTenna:MODel:ARRay:DISTribution` on page 605
`ANTenna:MODel:ARRay:ELEMent:COSine` on page 606
`ANTenna:MODel:ARRay:NX` on page 606
`ANTenna:MODel:ARRay:NZ` on page 606
`ANTenna:MODel:ARRay:XDIStance` on page 608
`ANTenna:MODel:ARRay:ZDIStance` on page 608
`ANTenna:MODel:ARRay:PEDestal` on page 607
`ANTenna:MODel:ARRay:COSN` on page 604

Custom Antenna Settings
You can define an antenna pattern based on the required antenna characteristics.

See Figure 9-15.

"HPBW XY, HPBW YZ"
HPBW specifies the angular width within which the antenna is most sensitive.

"Side lobe level"
Attenuation applied on the first pair of side lobes compared to the power level of main lobe

"Rolloff factor"   Step size, used to calculate the attenuation of the subsequent side lobes.

"Side lobe scale"

Scale factor, used to calculate the HPBW of the side lobes, i.e. how much smaller or wider the side lobes are compared to the main lobe.

Remote command:

ANTenna:MODel:CUSTom:HPBW:XY on page 610

ANTenna:MODel:CUSTom:HPBW:YZ on page 610

ANTenna:MODel:CUSTom:SLSTart on page 610

ANTenna:MODel:CUSTom:SLRolloff on page 610

ANTenna:MODel:CUSTom:SLSCale on page 611

Custom Phased Array Settings

You can define a phased array antenna with user-definable geometry, number of elements and spacing between them.
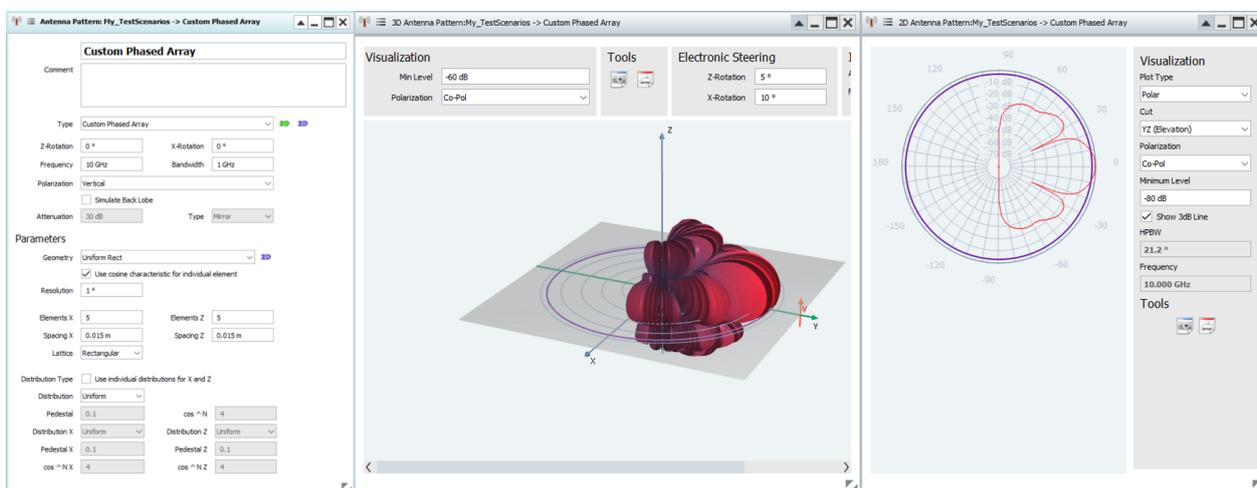
"Geometry"   An antenna array is a set of two or more antennas, called antenna elements.

The geometry describes how the single antenna elements of the array are arranged:
- "Uniform Linear" - in a row
- "Uniform Rectangular" - in a rectangular grid
- "Circular Planar" - in a circle
- "Uniform Hex" - as a hexagon

Remote command:

ANTenna:MODel:CARRay:GEOMetry on page 609

"Use cosine characteristic for individual antenna element"

Per default, the individual antenna elements use omnidirectional characteristic. If the parameter is enabled, the cosine characteristic is used.

The characteristics of the individual antenna elements influence the transition between the main/side lobes and the back lobes of the phased array antenna, see Table 9-4.

Table 9-4: Effect of the parameter "Use cosine characteristic for individual antenna element"

Off (default)	On
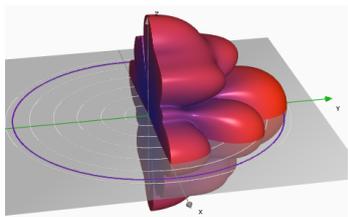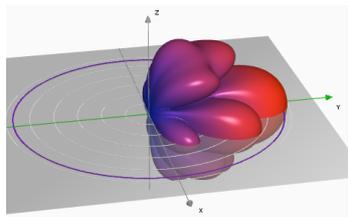
Remote command:

ANTenna:MODel:CARRay:ELEMent:COSine on page 606

"2D"

Illustrates the array geometry.

Click an antenna element to activate or deactivate it.
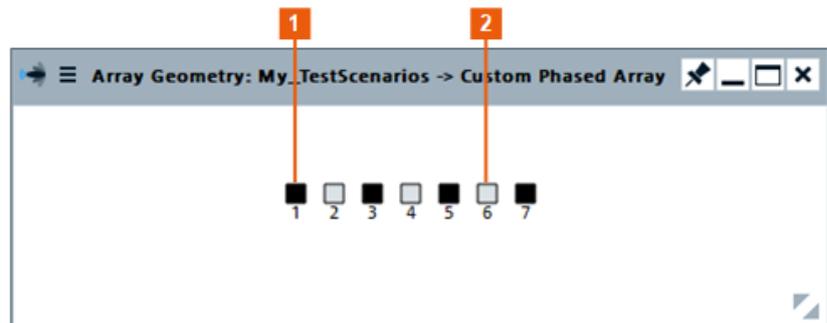
Figure 9-3: Example: Linear array antenna with seven elements

1 = Active element
2 = Deactivated element

Remote command:

ANTenna:MODel:CARRay:ELEMent on page 609

"Uniform rectangular antenna array"

A rectangular array is a planar antenna array with X number of columns and Z number of rows. The antenna elements can be placed on rectangularly or triangularly lattice.

Per default, the number of elements in the rows and in the columns is the same and the elements are uniformly distributed. You can change the spacing and use unequal number of elements.
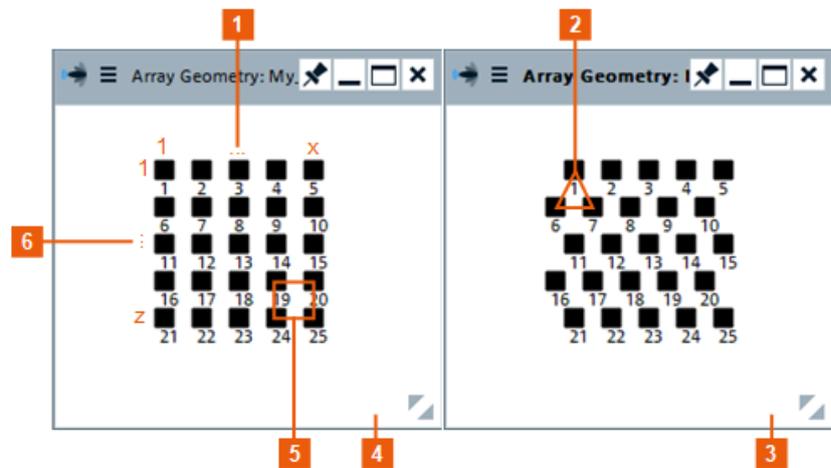
Figure 9-4: Example: Rectangle array antenna with different grids

1, 6 = Number of antenna elements
2, 3 = Triangular lattice
4, 5 = Rectangular lattice

Remote command:

ANTenna:MODel:CARRay:RECTangular:NX on page 606
ANTenna:MODel:CARRay:RECTangular:NZ on page 606
ANTenna:MODel:CARRay:RECTangular:XDIStance
on page 608
ANTenna:MODel:CARRay:RECTangular:ZDIStance
on page 608
ANTenna:MODel:CARRay:RECTangular:LATTice on page 609

"Uniform Linear"

A linear array is a planar array where the antenna elements are placed in a row, see Figure 9-3.

You can change the number of antenna elements and the distance between them. The elements are uniformly distributed.

Remote command:

ANTenna:MODel:CARRay:LINear:N on page 606
ANTenna:MODel:CARRay:LINear:DISTance on page 607

"Uniform hexagonal antenna array"

An array with a hexagonal shape, where each of the six sides has the selected number of antenna elements.
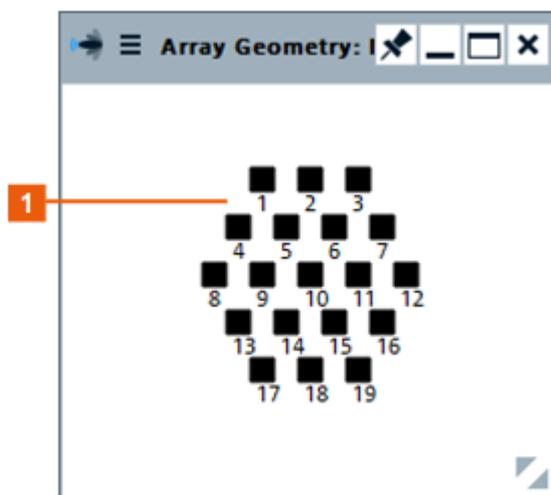
Figure 9-5: Example: Hexagonal array antenna with three elements per side

1 = Elements per side

Remote command:

ANTenna:MODel:CARRay:HEXagonal:N on page 606

ANTenna:MODel:CARRay:HEXagonal:DISTance on page 607

"Planar Circular"

An array with a shape similar to a circle, selectable radius size, and lattice.

Remote command:

ANTenna:MODel:CARRay:CIRCular:RADius on page 609

ANTenna:MODel:CARRay:CIRCular:DISTance on page 607

ANTenna:MODel:CARRay:CIRCular:LATTice on page 609

"Distribution Type"

Enables using the individual distribution function for X and Z direction.

Remote command:

ANTenna:MODel:CARRay:DISTribution:TYPE on page 605

"Distribution"   Determines the antenna pattern as a function of the variables *x* and *z*, the pedestal level *P* and the coefficient *N*:

Where:
- P = 0 to 1
  As set with the parameter "Pedestal".
- N = 2 to 10
  As set with the parameter "$\cos^N$"
- x = -1 to +1
  z = -1 to + 1

To simplify the description, only the *f(x)* functions are listed. The *f(z)* values are calculated in the same manner.

The distribution functions are calculated as follows:
- "Uniform"
  f(x) = 1
- "Parabolic"
  f(x) = 1 - (1 - P)*x²
- "Cosine"
  f(x) = P + (1 - P)*cos(0.5*x*π)
- "$\cos^2$"
  f(x) = P + (1 - P)*0.5*(1+cos(x*π))
- "$\cos^N$"
  If N > 2, *f(x) = P + (1 - P)*(cos(0.5*x*π))^N*
  For N = 2, *f(x) = 1*
- "Triangular"
  If N > 2, *f(x) = P + (1 - P)*(1 - abs (x))*
  (the *abs* function returns the absolute value of the variable)
  For N = 2, *f(x) = 1*
- "Hamming"
  f(x) = P+(1-P)*(0.54-0.46*cos((1.0+x)*π))
- "Hann"
  f(x) = P+(1-P)*(0.5-0.5*cos((1.0+x)*π))

Remote command:
ANTenna:MODel:CARRay:DISTribution on page 605

"Pedestal"   Sets the pedestal level P.

Remote command:
ANTenna:MODel:CARRay:PEDestal on page 607

"cos^N"   Sets the value of the coefficient N in the $\cos^N$ distribution.

Remote command:
ANTenna:MODel:CARRay:COSN on page 604

"Distribution X"/"Distribution Z"
     Requires "Distribution Type" enabled.
     Sets the individual distribution function for X and Z direction.

Remote command:
ANTenna:MODel:CARRay:DISTribution:X on page 606
ANTenna:MODel:CARRay:DISTribution:Z on page 606

"Pedestal X"/"Pedestal Z"

> Requires "Distribution Type" enabled.
> Sets the individual pedestal level for X and Z direction.
>
> Remote command:
> `ANTenna:MODel:CARRay:PEDestal:X` on page 607
> `ANTenna:MODel:CARRay:PEDestal:Z` on page 607

"cos^N X"/"cos^N Z"

> Requires "Distribution Type" enabled.
> Sets the individual value of the coefficient N in the $\cos^N$ distribution for X and Z direction.
>
> Remote command:
> `ANTenna:MODel:CARRay:COSN:X` on page 605
> `ANTenna:MODel:CARRay:COSN:Z` on page 605

Cardioid Antenna Settings

A cardioid antenna is an antenna with a heart-like radiation pattern.

The cardioid equation expressed in polar coordinates is calculated as follows:

$r(\varphi) = [1 + \cos(\varphi)]^{exp}$, where:

- φ is the rotation angle
- *exp* is the exponent.
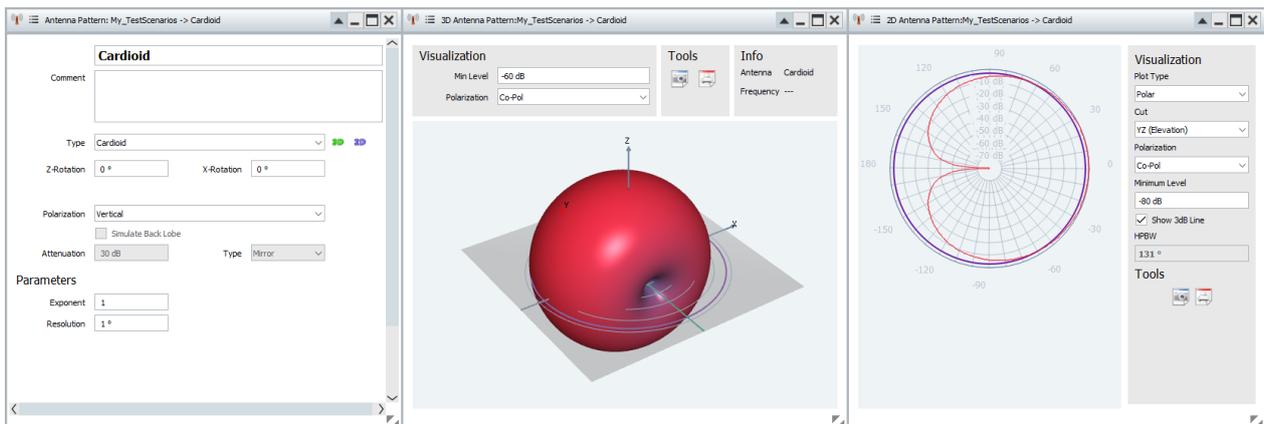
"Exponent"

> Use values greater than 1 to narrow the antenna beam.
>
> Remote command:
> `ANTenna:MODel:CARDoid:EXPonent` on page 613

Imported antenna settings

You can define and load you custom antenna patterns.

See:

- Chapter A.2, "Antenna pattern file formats", on page 843 for description of the file formats.
- "To import a user-defined antenna pattern from file in one of the predefined file formats" on page 299.

Import Filter ← Imported antenna settings

Selects the import filter used to parse the antenna pattern file.

"Internal"　　　　　　Parses antenna patterns in the following file formats:
- FEKO Far Field (`*.ffe`) files
- `*.ant_pat` files in the Rohde & Schwarz proprietary format
- Antenna Magus `*.tsv` files
- ANSYS HFSS `*.ffd` files
- Antenna patterns in `*.csv` file format.
  The Import wizard assists you to import `*.csv` files.

"Plugin Name"　　Any selection other than "Internal" selects a user-defined antenna pattern import filter (plug-in). The import filter must exist in the "Plug-in" library.
See Chapter C.5, "Custom antenna pattern import functions", on page 877.
Antenna patterns in custom file format are files with extension `*.txt`.

Load/Purge ← Imported antenna settings
Standard functions for file management.

Remote command:
ANTenna:MODel:USER:LOAD on page 613
ANTenna:MODel:USER:CLEar on page 595

Number of patterns and points ← Imported antenna settings
Indicates the number of patterns and points as retrieved from the load file.

Import wizard ← Imported antenna settings
When a `*.csv` file is loaded, the "Import Wizard" opens and tries to determine:
- The used column separator
- The decimal delimiter
- The number of the first line with meaningful data, where the numbering start with 1 and empty lines are counted
- The data content of each column.

You can change the values afterwards.

For example, the file on Figure 9-6 contains a line that explains the parameters in each column.

If your file does not contain this information or if the automatically selected column separators do not match the information in the file, change the values. Always check if the parameters in the "Import Wizard" dialog match the particular file format and content.
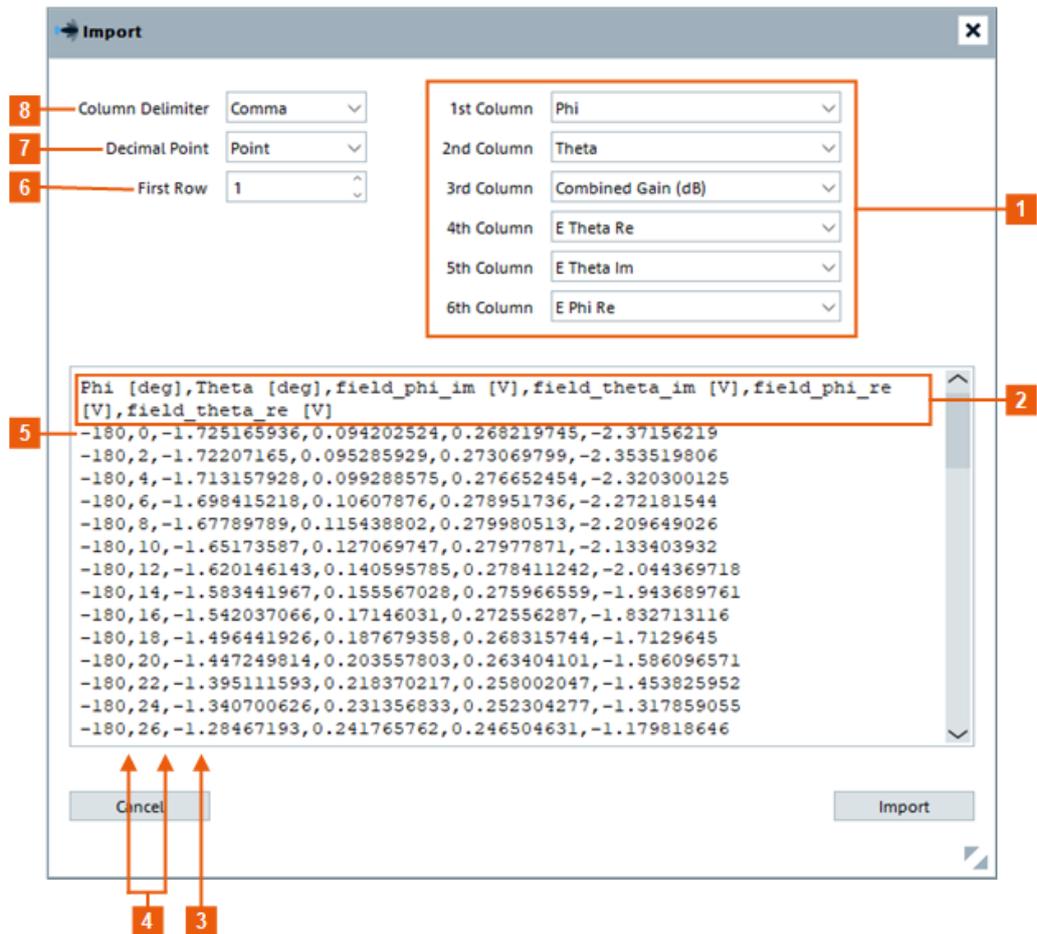
Figure 9-6: Import Wizard: understanding the displayed information

1, 2 = Parameter per column and column header information
3, 7 = Decimal separator: point or comma
4, 8 = Column separator: comma, semicolon, withe space
5, 6 = First row

Remote command:
ANTenna:MODel:USER:CSV:FORMat on page 614

9.3 Antenna scans settings

Access:

► Select "Repository Tree > Antenna Scan > New".

Settings:

Antenna Scan Name

Enter the name of the antenna scan.

Remote command:
SCAN:NAME on page 589
SCAN:CATalog? on page 587
SCAN:SELect on page 588
SCAN:CREate on page 588
SCAN:REMove on page 590

Comment

Enter a short description.

Remote command:
SCAN:COMMent on page 590

Scan Type

Defines the scan type.

Use the "3D Scan View" diagram to visualize the selected scan.

Remote command:
SCAN:TYPE on page 623

3D Scan View

To access the "3D Scan View" open any "Antenna Scan" or "Emitter Type" and click the "Display 3D antenna scan" button.
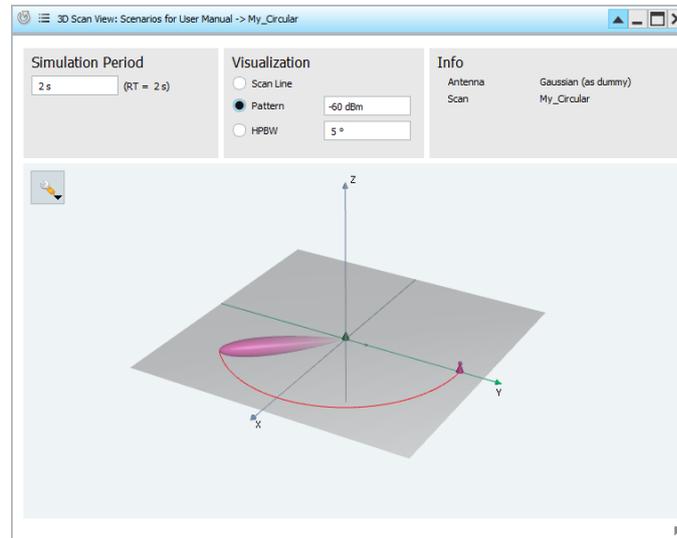
This window contains control and information elements at the top of the window. In the top left corner of the pattern visualization area there is a "Transparency settings" icon that enables you to change the way the scan is displayed. This icon is also available in the "3D Single Emitter Preview" window.

Electronic Scan

Defines how the scan works.

If this setting is activated and an emitter uses this scan together with a phased array antenna, electronic steering is applied.

In a normal scan, the antenna is mounted on a mechanical motor that performs the scan. In electronic scan mode, the antenna is fixed and the scan is performed electronically. The antenna does not move. The pattern is aimed towards the desired location by applying phase offsets to the individual antenna elements.

Remote command:
SCAN:STEering on page 629

3D Scan View

Visualizes the antenna scan in a 3D preview.

See the example on Figure 9-14.

Simulation Period ← 3D Scan View

Sets the time it takes the animation to complete a scan.

The "Real Time" is also displayed.

Visualization ← 3D Scan View

Defines the way the scan is represented:
● "Scan Line = On": the scan is represented by a line; the antenna pattern is disregarded.
● "Pattern Visualization > Pattern": the visualization uses a Gaussian pattern.
  If you have opened this dialog form the "Emitter" dialog or the 2D map, the visualization corresponds to the current selected antenna pattern.
● "Pattern Visualization > HPBW": the antenna is represented by its HPBW.
● "Minimum Displayed Level": the minimum displayed side lobes level.

Info

Summary information.

Circular Scan

A circular antenna turns in a circle and scans 360°.
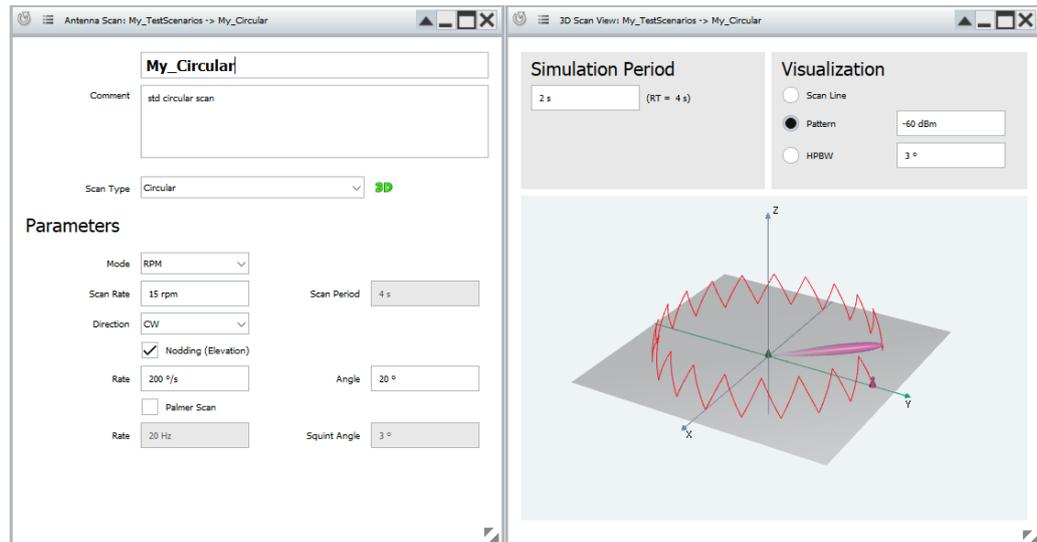
Figure 9-7: Circular scan with enabled nodding

"Mode"	Sets if the scan turning speed is set as a scans rate or as a period. The value "Scan Rate" and "Scan Period" are interdependent and updated automatically whenever one of the two is changed.
"Scan Rate"	Sets speed the antenna is turning with.
"Scan Period"	Sets the time it takes for the antenna to turn once.
"Direction"	Sets the direction, clockwise ("CW") or counterclockwise ("CCW"), the antenna is turning in.
"Nodding (Elevation)"	If enabled, superimposes a horizontal nodding on the circular scan. The height of the nods and their rise time are defined with the parameters "Angle" and "Rate", see Figure 9-7.
"Palmer Scan"	If enabled, superimposes a conical scan on the circular scan, see Figure 9-8.
"Rate"	Enabled if "Palmer Scan > On". Defines how fast the palmer scan is turning.
"Squint Angle"	
	Enabled if "Palmer Scan > On". Sets the radius of the scanned circle.

Remote command:

SCAN:CIRCular:MODE on page 618
SCAN:CIRCular:RPM on page 619
SCAN:CIRCular:PERiod on page 618
SCAN:CIRCular:ROTation on page 618
SCAN:CIRCular:NODDing on page 624
SCAN:CIRCular:NRATe on page 624
SCAN:CIRCular:NELevation on page 624
SCAN:CIRCular:PALMer on page 624
SCAN:CIRCular:PRATe on page 625
SCAN:CIRCular:PSQuint on page 625

Sector Scan

A sector antenna scans only a sector with defined "Sector Width".
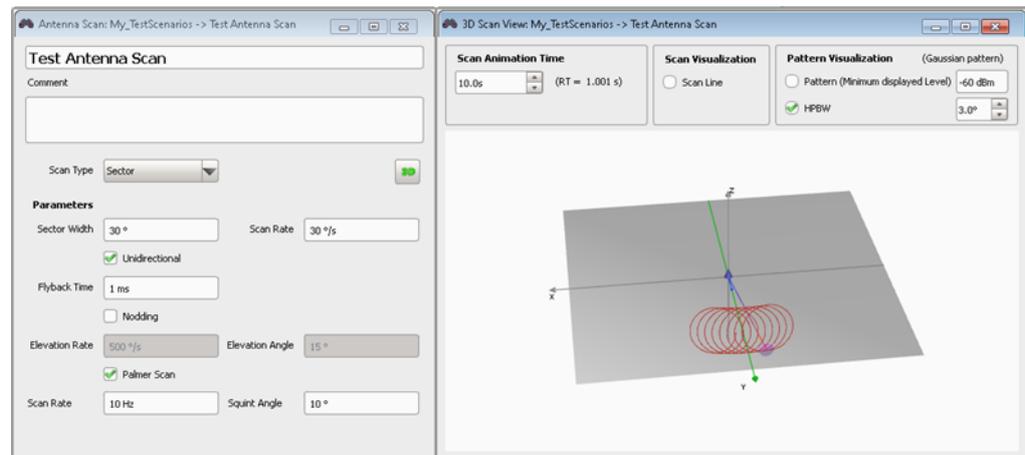
Figure 9-8: Sector scan with enabled Palmer scan

"Sector Width, deg"

        Sets the width of the scanned sector.

"Scan Rate, deg/s"

        Defines how fast the antenna is turning.

"Unidirectional"   The antenna uses a unidirectional scan mode, i.e. the antenna turns in one direction only and "jumps" back within the selected "Flyback Time".

"Nodding"   If enabled, superimposes a horizontal nodding on the circular scan. The height of the nods and their rise time are defined with the parameters "Elevation Angle" and "Elevation Rate", see Figure 9-7.

"Palmer Scan"   If enabled, superimposes a conical scan on the circular scan, see Figure 9-8.

"Rate"   Enabled if "Palmer Scan > On".
                  Defines how fast the palmer scan is turning.

"Squint Angle"

        Enabled if "Palmer Scan > On".
        Sets the radius of the scanned circle.

Remote command:

SCAN:SECTor:WIDTh on page 622

SCAN:SECTor:RATE on page 619

SCAN:SECTor:UNIDirection on page 622

SCAN:SECTor:FLYBack on page 621

SCAN:SECTor:NODDing on page 624

SCAN:SECTor:NELevation on page 624

SCAN:SECTor:NRATe on page 624

SCAN:SECTor:PALMer on page 624

SCAN:SECTor:PRATe on page 625

SCAN:SECTor:PSQuint on page 625

Raster Scan

The raster antenna scans a sector with defined "Sector Width" but performs several scans with different elevations.

See Figure 9-14.

"Raster Width, deg"

>Sets the width of the scanned sector.
>Scanned is an angle of ±"Raster Width"/2 degrees, starting from the Y axis.

"Scan Rate, deg/s"

>Defines how fast the antenna is turning.

"Bar Count, Bar Width"

>Sets the number of bars and the distance between two consecutive scanned bars (sectors).

"Retrace Time"   Defines how fast the antenna returns to its initial orientation after completing the last bar.

"Direction"   Switches between a horizontal and a vertical scanning direction.

"Rewind"   Continuous scan in that upon reaching the end of the last bar, the antenna scans backwards.

"Unidirectional" The antenna uses a unidirectional scan mode. The scan is not a continuous but the antenna turns in one direction only (counterclockwise, CCW) and "jumps" to the next bar within the selected "Flyback Time".

"Palmer Scan"   If enabled, superimposes a conical scan on the circular scan, see Figure 9-8.

"Rate"   Enabled if "Palmer Scan > On".
                  Defines how fast the palmer scan is turning.

"Squint Angle"

>Enabled if "Palmer Scan > On".
>Sets the radius of the scanned circle.

Remote command:

SCAN:RASTer:WIDTh on page 622

SCAN:RASTer:RATE on page 619

SCAN:RASTer:BARWidth on page 621

SCAN:RASTer:BARS on page 620

SCAN:RASTer:RETRace on page 622

SCAN:RASTer:DIRection on page 622

SCAN:RASTer:REWind on page 621

SCAN:RASTer:UNIDirection on page 622

SCAN:RASTer:FLYBack on page 621

SCAN:RASTer:PALMer on page 624

SCAN:RASTer:PRATe on page 625

SCAN:RASTer:PSQuint on page 625

Conical Scan

A conical antenna scans a conus with defined "Squint Angle" (radius); the radar beam is rotated around the y-axis (i.e. boresight).
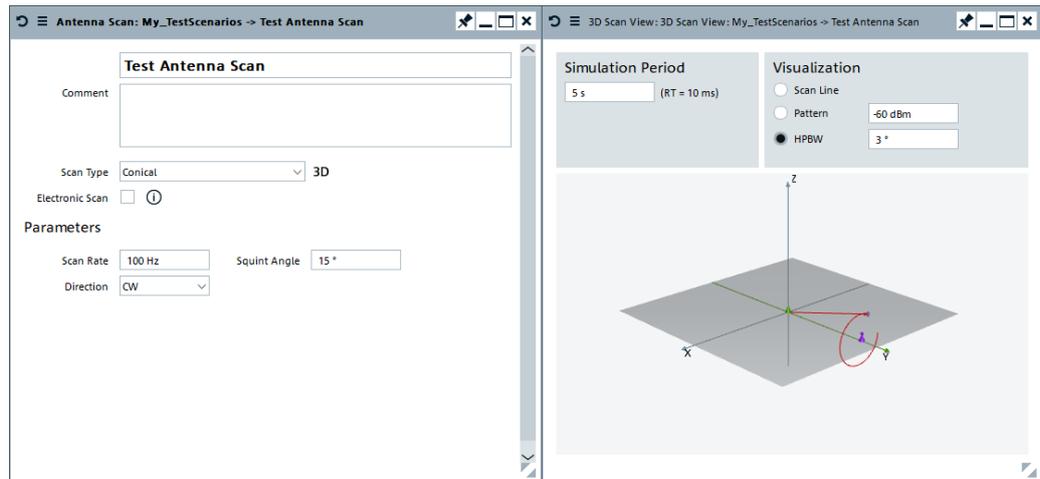
"Scan Rate, Hz"

Defines how fast the antenna is turning.

"Squint Angle, deg"

Sets the radius of the scanned circle.

"Direction"   Sets the direction, clockwise ("CW") or counterclockwise ("CCW"),
the antenna is turning in.

Remote command:

SCAN:CONical:RATE on page 619

SCAN:CONical:ROTation on page 618

SCAN:CONical:SQUint on page 620

Helical Scan

A helical antenna scan turns in a circle (scans 360°) and performs several scans with
different elevations.
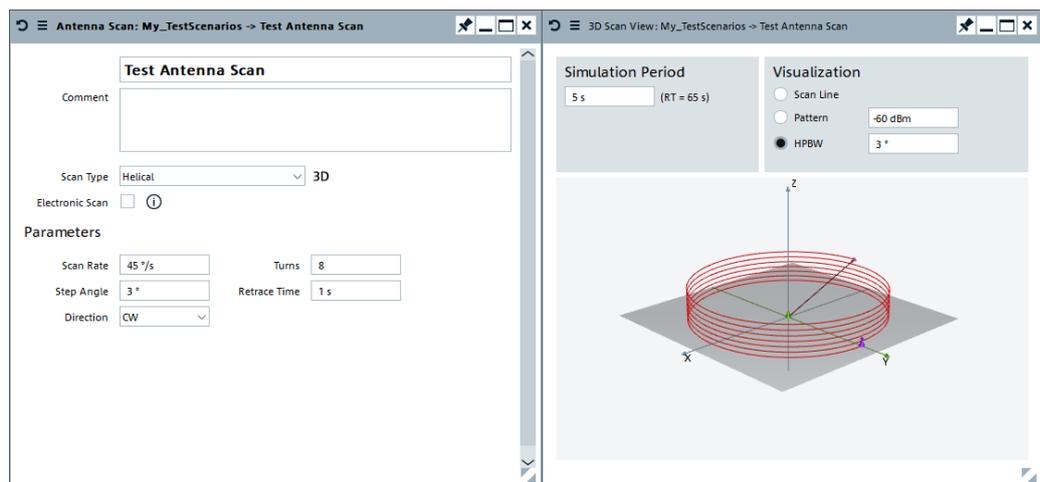
"Scan Rate, deg/s"

Defines how fast the antenna is turning.

"Step Angle, deg"

Sets the step width with that the antenna azimuth changes.

"Turns"   Sets the number of 360° turns the antenna performs.

"Retrace Time"   Defines how fast the antenna returns to its initial orientation after completing the last turn.

"Direction"   Sets the direction, clockwise ("CW") or counterclockwise ("CCW"), the antenna is turning in.

Remote command:
SCAN:HELical:RPM on page 619
SCAN:HELical:ELEVation:STEP on page 619
SCAN:HELical:TURNs on page 619
SCAN:HELical:RETRace on page 622
SCAN:HELical:ROTation on page 618

Spiral Scan

A spiral antenna scan turns in a circle (scans 360°) and performs several scans with different radii.
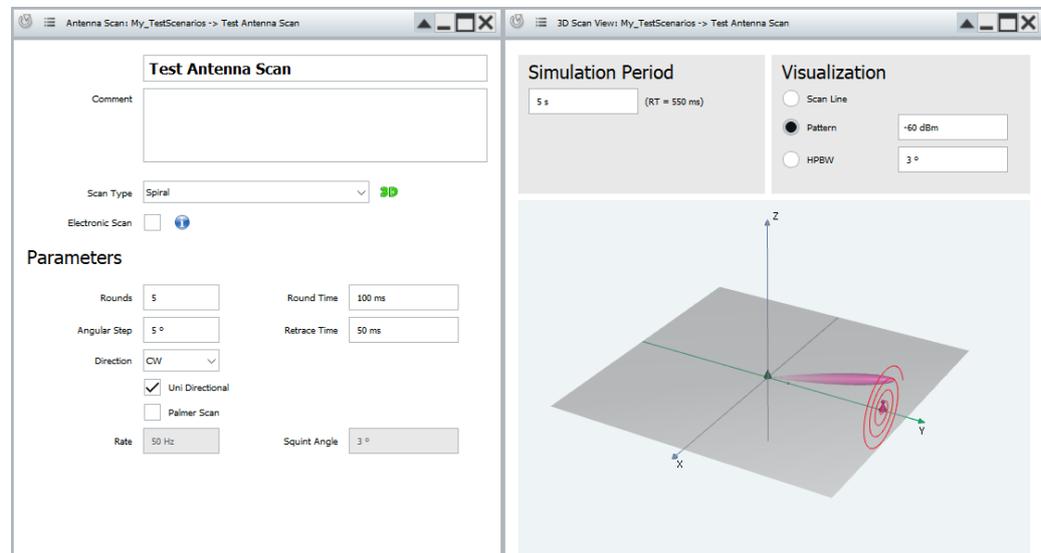
"Rounds"   Sets the number of rounds the antenna performs.

"Angular Step"   Sets the distance in degrees with that the scan radius is increased.

"Round Time"   Defines how fast the antenna is turning.

"Retrace Time"   Define how fast the antenna returns to its initial orientation after completing the last turn.

"Direction"   Sets the direction, clockwise ("CW") or counterclockwise ("CCW"), the antenna is turning in.

"Uni Directional"

        Sets the scan behavior when the scan reaches the last round:
- "Uni Directional = On" - the scan is not continuous.
  It turns for the specified number of rounds and then takes the retrace time to return to the origin of the spiral scan.
- "Uni Directional = Off" - the scan is continuous; it turns forth and backwards.
  When the specified number of rounds is reached, the scan direction turns backwards back to the origin of the spiral scan.

"Palmer Scan"   If enabled, superimposes a conical scan on the circular scan, see Figure 9-8.

"Rate", "Squint Angle"   Enabled if "Palmer Scan > On".
Defines how fast the palmer scan is turning.

"Squint Angle"

        Enabled if "Palmer Scan > On".
Sets the radius of the scanned circle.

Remote command:

`SCAN:SPIRal:ROUNds` on page 623

`SCAN:SPIRal:STEP` on page 623

`SCAN:SPIRal:RTIMe` on page 623

`SCAN:SPIRal:RETRace` on page 622

`SCAN:SPIRal:ROTation` on page 618

`SCAN:SPIRal:UNIDirection` on page 622

`SCAN:SPIRal:PALMer` on page 624

`SCAN:SPIRal:PRATe` on page 625

`SCAN:SPIRal:PSQuint` on page 625

Lobe Switching Scan

A lobe switching antenna uses two or four slightly separated antenna elements.
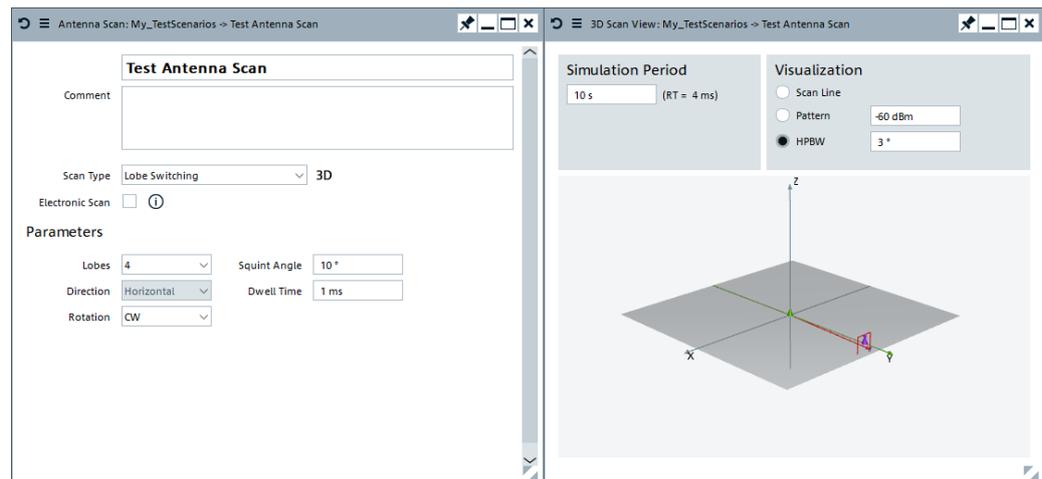
"Lobes"   Set the number of lobes.

"Direction"   Sets the switching direction, i.e. horizontal or vertical.

"Squint Angle"   Sets the angle that the beam axis is offset from the tracking axis (i.e. the y-axis) of the antenna.

"Dwell Time"   Sets the antenna motion, i.e. how fast the switches between the lobes.

"Direction"   Sets the direction, clockwise ("CW") or counterclockwise ("CCW"), the antenna is turning in.

Remote command:
SCAN:LSW:LOBes on page 620
SCAN:LSW:DIRection on page 620
SCAN:LSW:SQUint on page 620
SCAN:LSW:DWELl on page 620
SCAN:LSW:ROTation on page 618

Sine Scan

A sine antenna scan draws a sine wave.
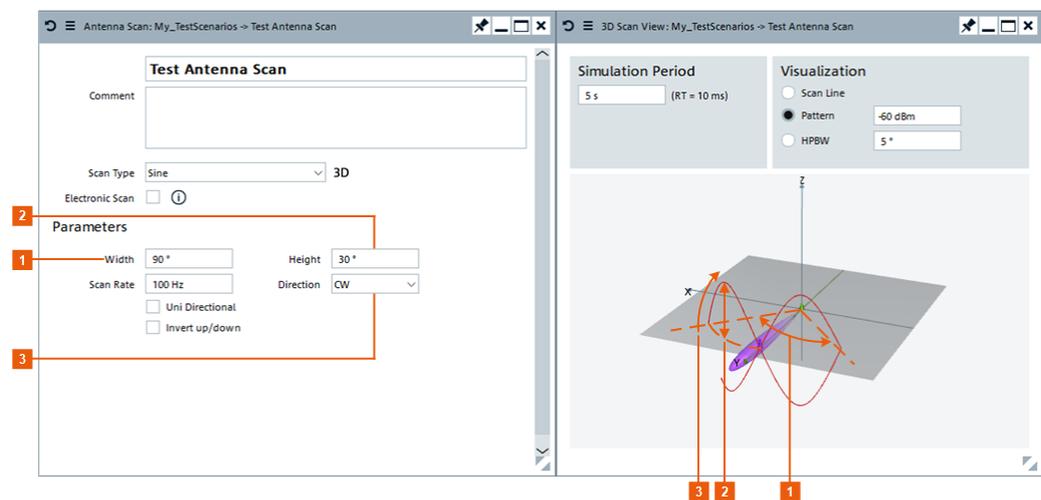
Figure 9-9: Sine scan: understanding the displayed information

1 = Width is the sector width, i.e. the angle on the XY plane between the origin and the end of the scan
2 = Height is the sine amplitude
3 = Seen from the receiver, the scan turns clockwise ("Direction = CW") and draws the upper sine wave first ("Invert up/down scan = Off")

"Width, deg"

Sets the width of the scanned sector. i.e. the angle on the XY plane between the origin and the end of the scan.

"Height, deg"

Sets the amplitude of the sine wave.

"Scan Rate, deg/s"

Defines how fast the antenna is turning.

"Direction"   Sets the antenna turning direction, clockwise ("CW") or counterclockwise ("CCW").

"Uni Directional"

Sets the scan behavior when the scan reaches the last point:
- "Uni Directional = On": the scan is not continuous.
  It draws 1 sine wave and then jumps back to the origin of the scan.
- "Uni Directional = Off": the scan is continuous; it draws two symmetrical sine waves
  When the scan reaches the sine end position, it returns back to the origin drawing a sine wave that is the mirrored version of the sine scan.

"Invert up/down scan"

Sets the sine scan order and defines whether the upper or the down (mirrored) sine scan is used first.

Remote command:

SCAN:SIN:WIDTh on page 628

SCAN:SIN:HEIGht on page 628

SCAN:SIN:RATE on page 619

SCAN:SIN:ROTation on page 618

SCAN:SIN:UNIDirection on page 622

SCAN:SIN:INVersion on page 629

Custom Scan

Describes the antenna scan as a sequence of discrete scan positions.

Each position is defined by an elevation angle and an azimuth angle.

The trajectory between two subsequent positions is interpolated.

Figure 9-10 shows a simple custom scan defined using 4 beam positions.
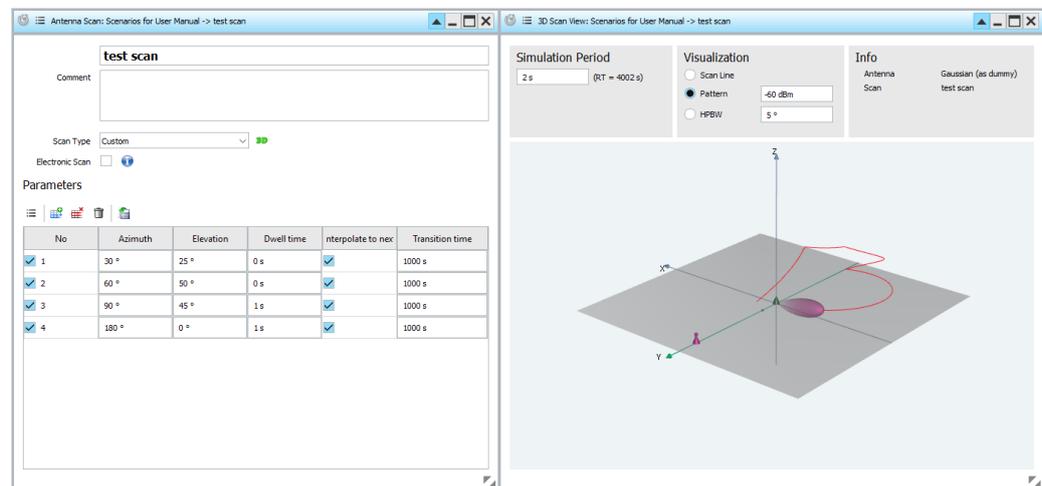
Figure 9-10: Custom scan defined by four positions

"New, Select, Append, Remove, Clear Items"

Standard functions for item handling, see Table 2-4.

"Import data from file"

Opens a dialog that enables you to import a custom scan definition from a CSV file (see Figure 9-10).

The dialog previews the data in the file and enables you to ignore the headers (if present) and map the columns in the file to the columns in the "Parameters" table.

Figure 9-11: Importing custom scan parameters

Positions table

Each row of the table contains parameters that define a position and the behavior of the custom scan. The scan moves sequentially down the table. The scan direction depends on the "Azimuth" (value and sign) and "Elevation" (value and sign) parameters. The scan behavior depends on the other parameters.

Each row consists of the following parameters:
- "Azimuth" (angle offset from the y-axis)
- "Elevation" (angle offset from horizontal)
- "Dwell time" defines how long the beam remains at a particular position.
- "Interpolate to next" defines whether the beam jumps or moves smoothly from the current position to the next position.
  - Selected = smooth movement
  - Deselected = jump
- "Transition time" defines the time over which the beam movement is interpolated.

Note: When the scan repeats, the beam position jumps from the last position to the first position.

If a jump is the correct behavior, do not select "Interpolate to next" for the last position (e.g. like the flyback at the end of a helical scan).

If the correct behavior is a smooth transition (e.g. the custom scan defines a closed shape), select "Interpolate to next" for the last position so that the scan repeats without a jump.

Remote command:

SCAN:CUSTom:ENTRy:ADD on page 591

SCAN:CUSTom:ENTRy:INSert on page 594

SCAN:CUSTom:ENTRy:COUNt? on page 592

SCAN:CUSTom:ENTRy:SELect on page 593

SCAN:CUSTom:ENTRy:DELete on page 594

SCAN:CUSTom:ENTRy:CLEar on page 595

SCAN:CUSTom:IMPort:FILE on page 625

SCAN:CUSTom:IMPort:EXEC on page 625

SCAN:CUSTom:ENTRy:AZIMuth on page 625

SCAN:CUSTom:ENTRy:ELEVation on page 626

SCAN:CUSTom:ENTRy:DWELl on page 626

SCAN:CUSTom:ENTRy:JUMPtype on page 626

SCAN:CUSTom:ENTRy:TRANstime on page 627

Lissajous

The antenna scan follows a Lissajous curve.
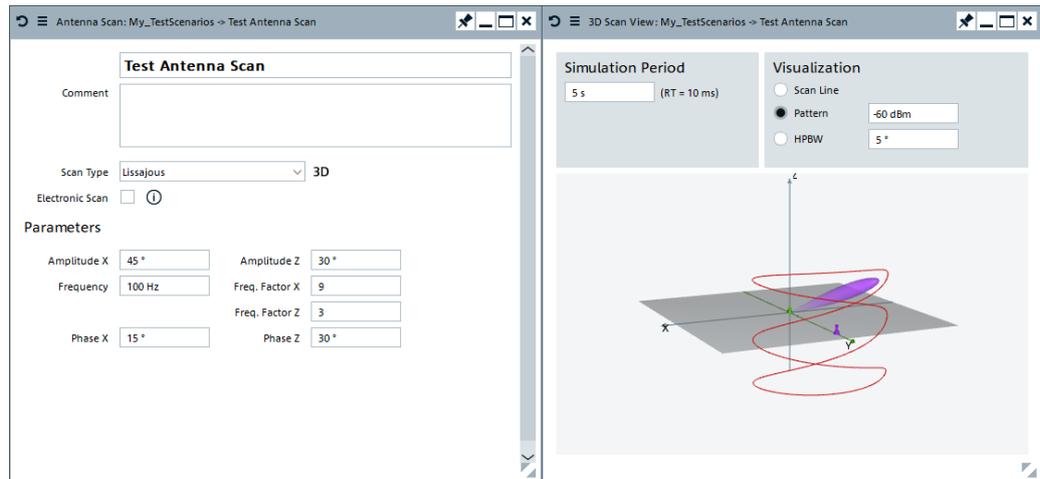
Lissajous curves are mathematically described as follows:

- $x(t) = A*cos(\omega_x*t - \sigma_x)$
- $z(t) = B*cos(\omega_z*t - \sigma_z)$

"Amplitude X, Amplitude Z"

Magnitudes A and B of two harmonic vibrations.

"Frequency"   Base frequency

"Freq X : Freq Z"

The two angular frequencies ω_x and ω_z are defined by their ratios and calculated as follows:

ω_x = "Freq X"*"Frequency"

ω_z = "Freq Z"*"Frequency"

The ratio "Freq X : Freq Z" determines the number of lobes in the curve.

"Phase X, Phase Z"

Phases σ_x and σ_z of the two harmonic vibrations.

Remote command:

SCAN:LISSajous:AMPX on page 627

SCAN:LISSajous:AMPZ on page 627

SCAN:LISSajous:FREQ on page 627

SCAN:LISSajous:XFACtor on page 628

SCAN:LISSajous:ZFACtor on page 628

SCAN:LISSajous:PHIX on page 628

SCAN:LISSajous:PHIZ on page 628

9.4 How to create a library with antenna patterns and scans

See:

- "To create an antenna pattern" on page 294

- "To create an antenna scan" on page 297
- "To create a custom antenna pattern based on the required antenna characteristics" on page 298
- "To import a user-defined antenna pattern from file in one of the predefined file formats" on page 299
- "To import an antenna pattern file in custom file format" on page 302
- "To observe the effect of polarization" on page 302

To create an antenna pattern

1. Select "Repository Tree > Antenna Pattern > New" .

2. Select "Antenna Model", e.g. "Planar Phased Array".

3. Adjust further settings, like "Rotation", "Frequency" or "Aperture Distribution".

4. Select "2D" to display the antenna pattern in a graph in polar or with Cartesian coordinates.
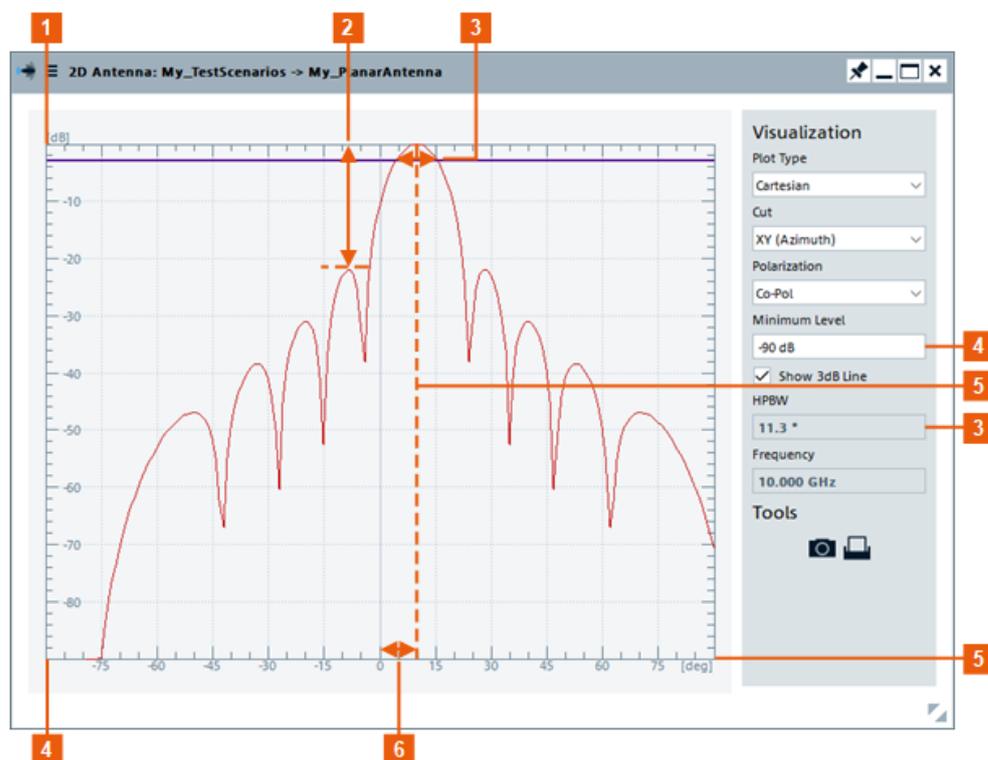
Figure 9-12: 2D antenna pattern: understanding the displayed information

1, 5 = Antenna pattern diagram, as a function of angle (deg) and antenna gain (dB)
2　　= Side lobe level
3　　= HPBW, i.e. the beamwidth at half power (3 dB)
4　　= Antenna gain minimum (-90 dB)
5　　= Beam axis
6　　= "Z-Rotation = 10°"

5. Change the number of used elements ,e.g. "Elements X = 20". Compare the 2D diagram.

   More antenna elements result in a smaller beamwidth and side lobes with lower level.

6. Select "Antenna Pattern > 3D" to display the 3D antenna pattern.
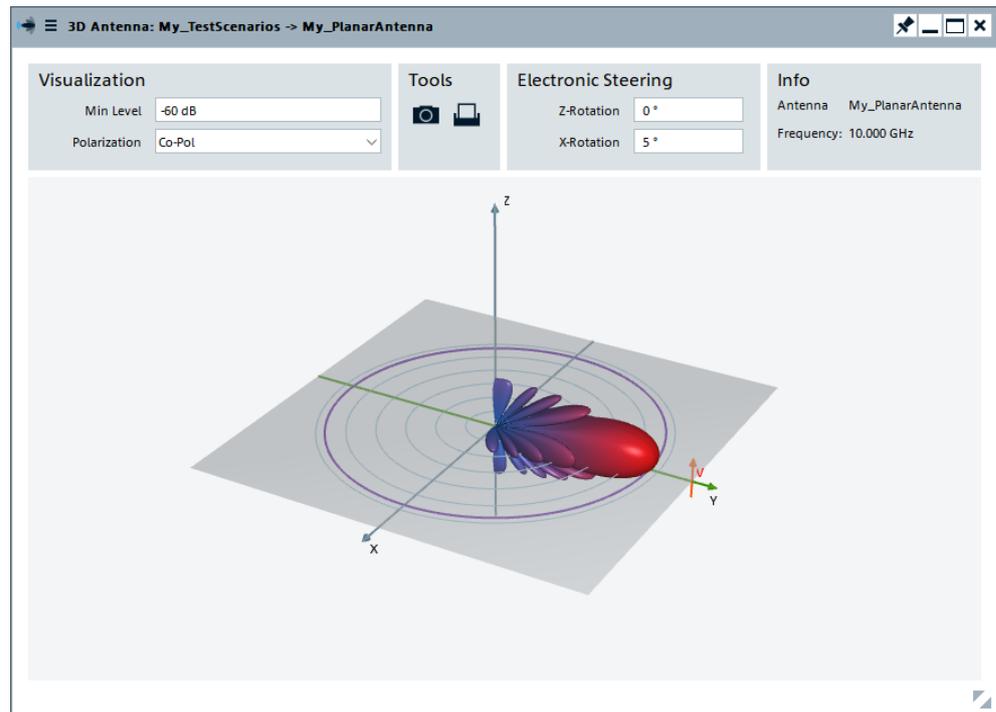
The 3D view is interactive. Use the mouse cursor to turn it around any of the axes and the mouse wheel to zoom in the view.

7. Select "Antenna Pattern > Simulate Back Lobe > On" and "Attenuation = 0 dB".

8. In the 2D view, select "Polar" coordinates.

   The display confirms the antenna pattern with a back lobe, where the back lobe is simulated as the mirrored pattern of the main lobe ("Type > Mirror").
   See Figure 9-13.

9. Select "Antenna Pattern > Back Lobe Attenuation = 40 dB".
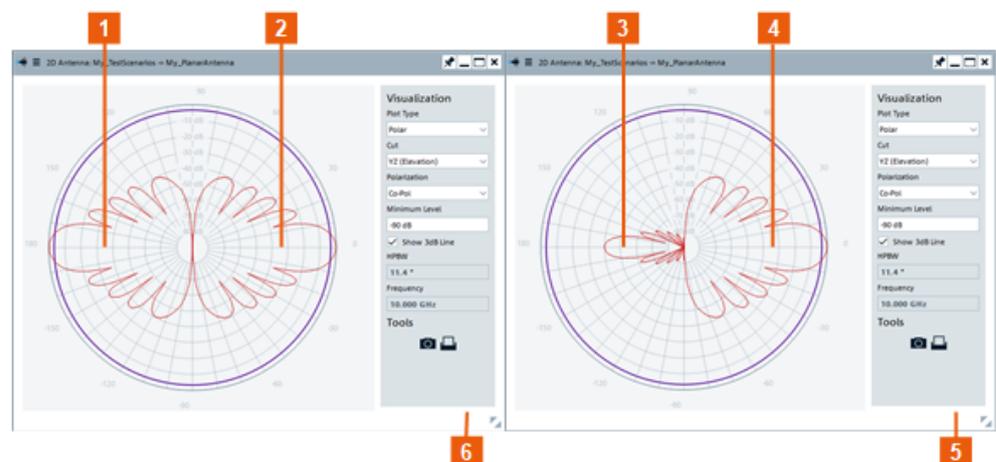
Figure 9-13: Simulate back lobe: understanding the displayed information

1   = Unattenuated back lobe, i.e. the mirrored pattern of the main lobe
2, 4 = Main lobe
3   = Attenuated back lobe
5   = "Attenuation = 40 dB"
6   = "Attenuation = 0 dB"

For description of the provided settings, see Chapter 9.2, "Antenna pattern settings", on page 261.

To create an antenna scan

1. Select "Repository Tree > Antenna Scans > New".

2. Select "Scan Type", e.g. "Raster Scan".
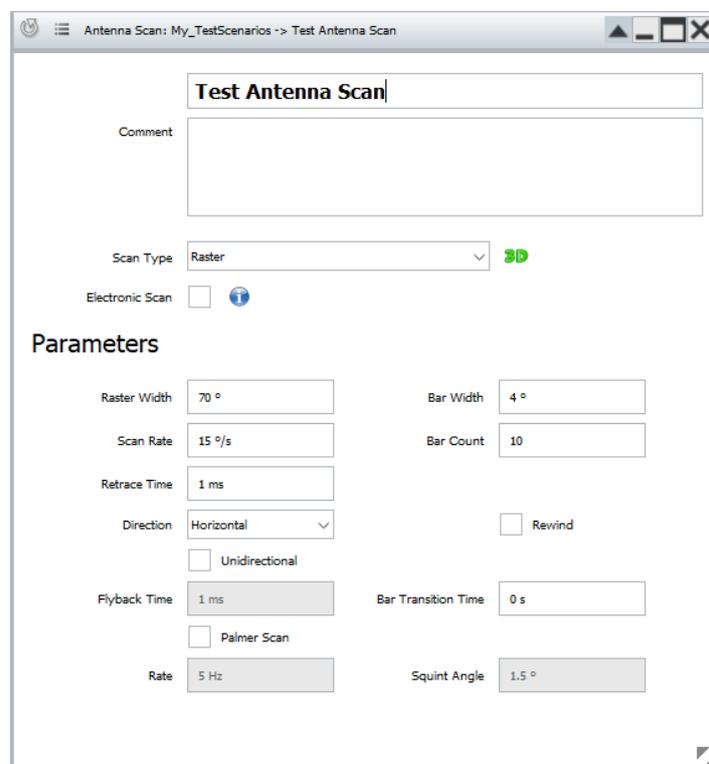
3. Adjust further settings, like "Raster Width" and "Bar Width".

4. Select "3D" to display the 3D antenna pattern.

5. Use the mouse cursor to turn it around any of the axes and the mouse wheel to zoom in the view.

   The antenna scan confirms the settings.
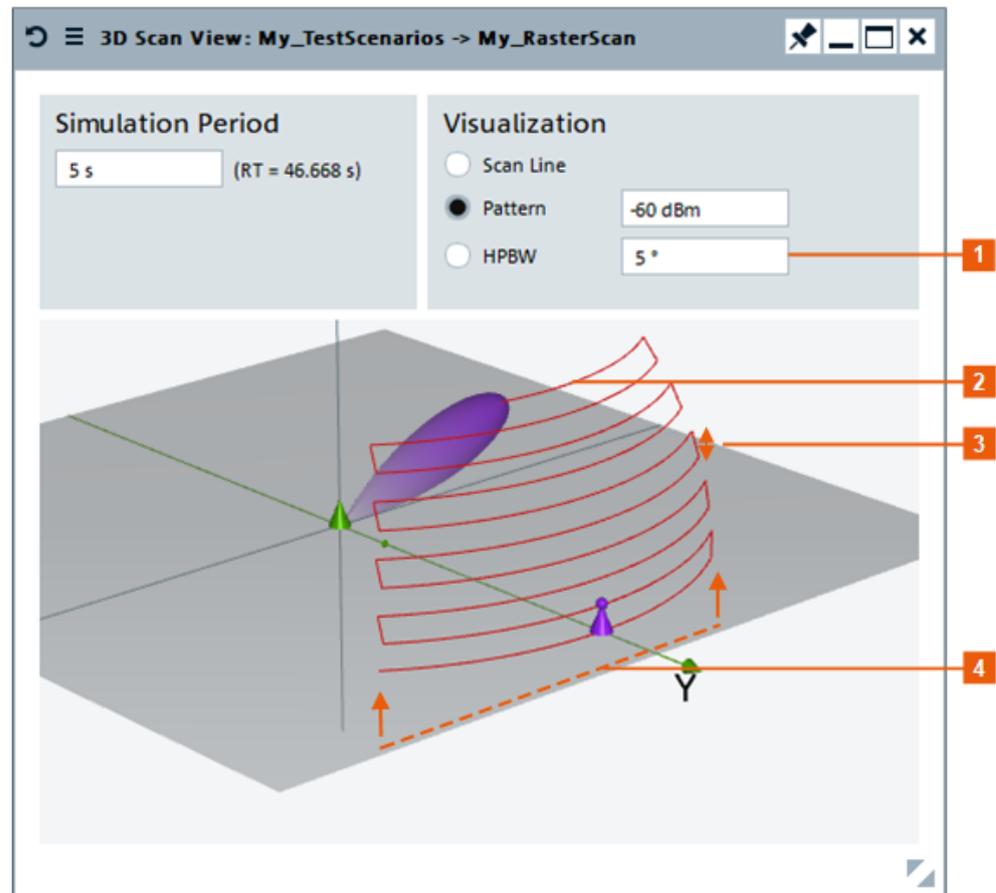
Figure 9-14: Horizontal raster scan: understanding the displayed information

1 = Antenna pattern visualized as "HPBW = 5°"
2 = Tenth scan as defined with "Bar Count"
3 = "Bar Width = 4°"
4 = "Raster Width"

For description of the provided settings, see Chapter 9.3, "Antenna scans settings", on page 280.

To create a custom antenna pattern based on the required antenna characteristics

1. Create an "Antenna Pattern".
   See "To create an antenna pattern" on page 294.

2. Select "Antenna Model > Custom".

3. Adjust the settings, e.g. set the "HPBW XY = 10°", "HPBW YZ = 3°", etc.

4. Observe the antenna pattern on the 2D and 3D views.

Figure 9-15: Custom antenna pattern: understanding the displayed information

1, 6 = Antenna pattern diagram, as a function of angle (deg) and antenna gain (dB)
2 = Roll-off factor or step size, used to calculate the attenuation of the subsequent side lobes
3 = Side lobe level, i.e. the attenuation of the first pair of side lobes
4 = "HPBW XY = 10°", i.e. the main beamwidth at half power (3 dB)
5 = Scale factor to calculate the HPBW of the side lobes, i.e. how much smaller or wider the side
     lobes are, compared to the main lobe

To import a user-defined antenna pattern from file in one of the predefined file formats

Per default, you can import custom antenna pattern in the formats listed in Chapter A.2, "Antenna pattern file formats", on page 843.

1. Create an "Antenna Pattern".

2. Select "Antenna Model > Import from File".

3. Select "Import Filter > internal".

4. Select "Load". Navigate to the folder with your custom antenna patterns. Load a `*.ffe`, a `*.tsv`, a `*.ffd`, a `*.csv`, or a `*.ant_pat` file.

5. Open the 2D view to visualize the antenna pattern.
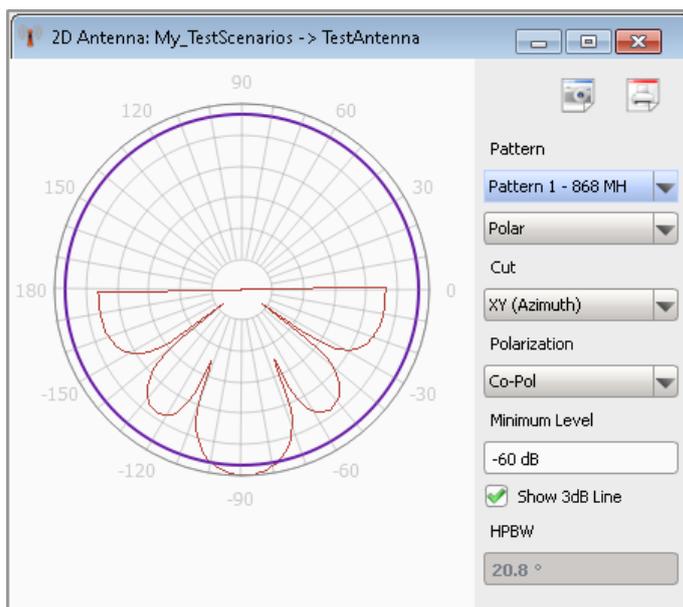
Figure 9-16: 2D view: Example of a user-defined antenna pattern

In this example, the antenna pattern is directed to -90°.

6. Select "Antenna Pattern > Z-Rotation = 90°".
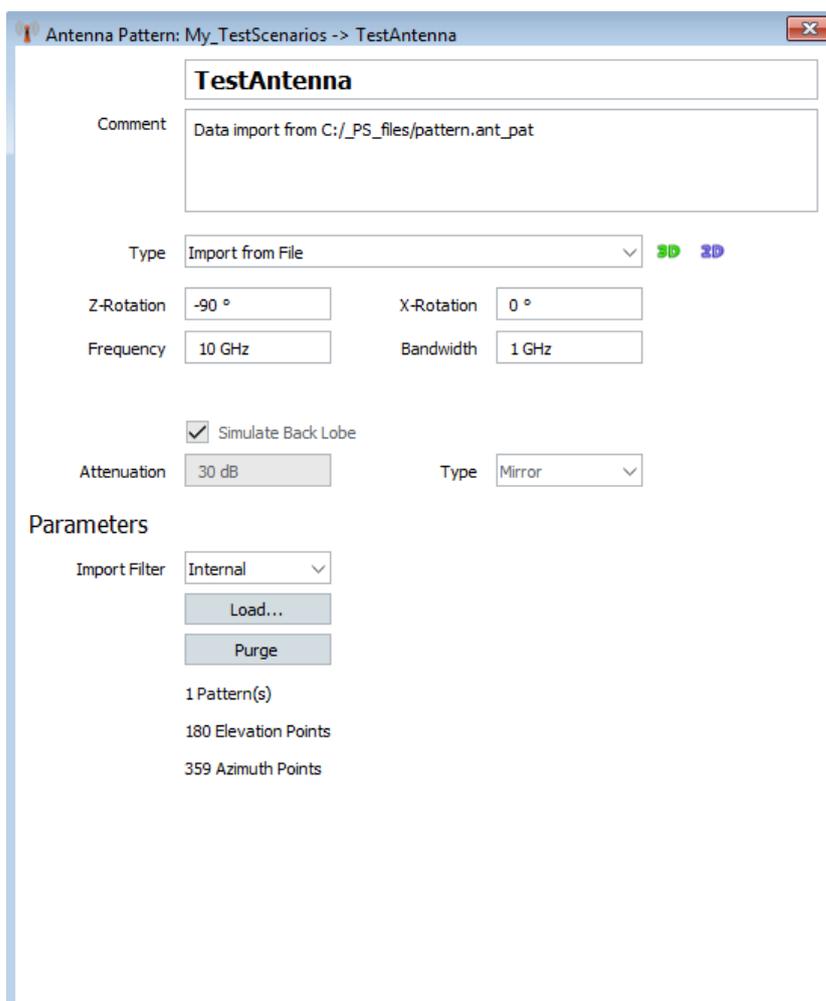
7. On the 2D and 3D views, observe the effect of this parameter.
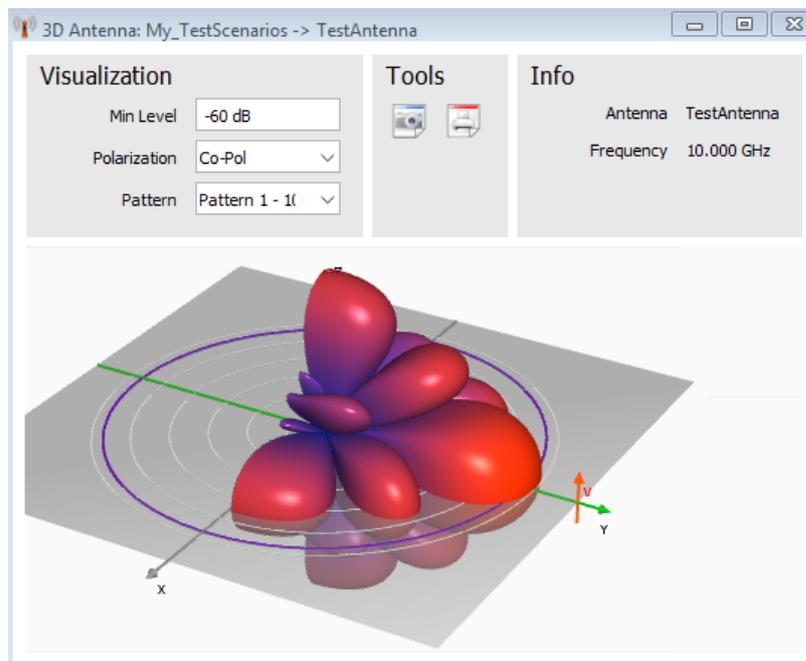
Figure 9-17: 3D view: Example of a user-defined antenna pattern

The antenna pattern is directed on the y-axis.

To import an antenna pattern file in custom file format

Importing antenna patterns in custom file format require that you create a suitable import filter first. That is, that you define and import the file format as a plug-in.

1. Import the plug-in in the repository, for example `My_AntPatternFileFormat`. See "To import a plugin" on page 530.

2. Create an antenna pattern. Select "Model > Import from File".

3. Select "Import Filter > My_AntPatternFileFormat".

4. Select "Load". Navigate to the folder with your custom antenna pattern (`*.txt` file). Select it.

5. Open the 2D and 3D view to visualize the antenna pattern.

   If the file contains more than one antenna patterns, select the pattern to be displayed.

To observe the effect of polarization

1. Create an antenna with a Gaussian beam pattern and a vertical polarization.

2. Open the 3D display. Observe the pattern for "Polarization > Co-Pol" and "Minimum Level = -90 dB".

3. Select "Polarization > Cross-Pol".

The automatically selected cross-polarized antenna is an antenna with *horizontal (H)* polarization.
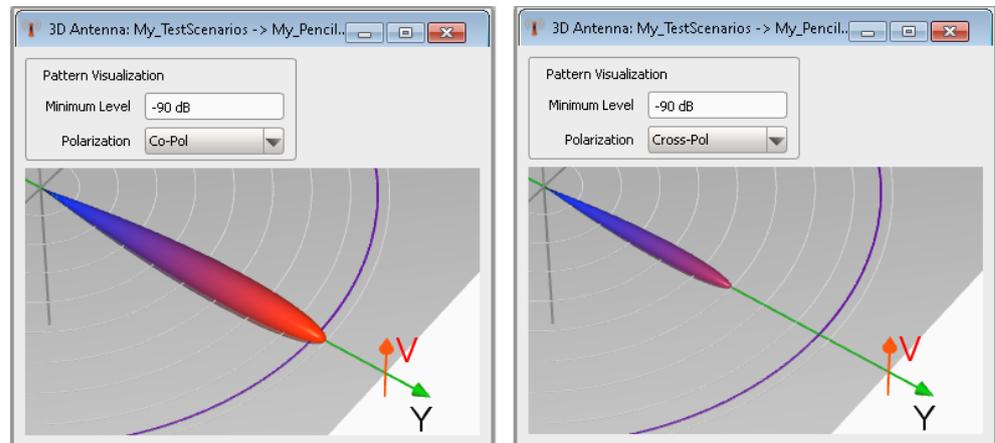
Figure 9-18: Effect of cross-polarization

The left figure shows the Gaussian antenna pattern. The right figure shows the resulting antenna pattern. This pattern has the same shape but the maximum power that is 30 dB lower than the co-polarized case. That is, the applied *polarization loss is -30 dB*.

4. Create a 2D scenario with one emitter and a receiver.

5. Assign the Gaussian beam pattern to the emitter and the receiver.
To simplify the explanation, this example does not use an antenna scan.

   Note: If the receiver uses an isotropic omnidirectional antenna, the antenna polarization is unused.

6. Align the receiver and the emitter: for the emitter, set "Altitude > Point to Receiver > On" and "Roll = 0°".

7. Open the "3D View with Receiver". Observe the maximum normalized power level at the receiver.
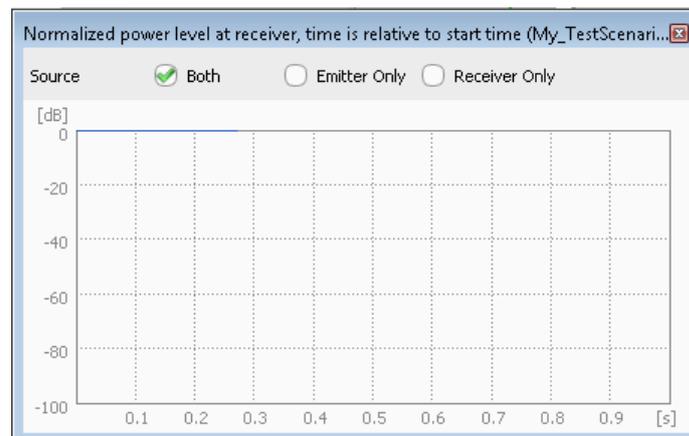
Figure 9-19: Normalized power level at the receiver, if the antennas are co-polarized (Roll = 0°)

The maximum power level is 0 dB.

8. Change the antenna roll angle. Compare the maximum normalized power level at the receiver.

Normalized power level at the receiver for "Roll = 45°"	Normalized power level at the receiver for "Roll = 90°"
The maximum power level is 3 dB less than the co-polarized case.	The antennas are cross-polarized. The maximum power level is 30 dB less than the co-polarized case.
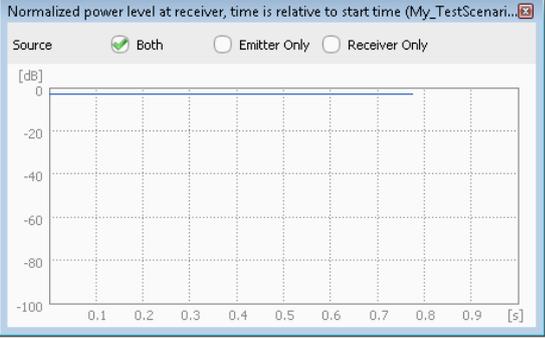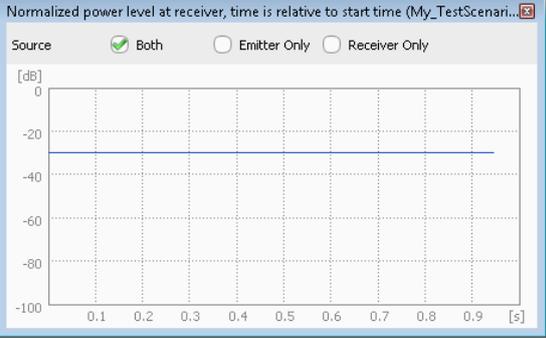
See also:
● Table 9-2
● Chapter 16.1, "How to create scenarios with receiver and TX items", on page 416

10 Emulating emitters

In the context of this software, a simple emitter describes a pulse sequence whereas a complex emitter emulates a radar system. The emitters are characterized by parameters like operating mode, beam, waveform, antenna pattern, and antenna scan.

The available emitter settings depend on the complexity of the simulated system:

- Simple emitters are described only by a sequence of pulses. Antenna beam patterns and antenna scans are not required
- Complex emitters are described by the combination of an operating mode, antenna pattern, antenna scan and a sequence.

Operational mode

As in the real-word where radar systems can run in different operational modes, in the R&S Pulse Sequencer an emitter can also work in more than one operational mode and switch between them. The operational mode is a description of the radar purpose, such as scanning, searching or tracking. For example, an airport surveillance radar can switch between different scan types to observe its airspace.

Each mode can have individual antenna and signal configuration. Multiple beams are possible within a mode. Moreover, the R&S Pulse Sequencer can emulate emitters that change their mode over time.

Related settings:

- "Emitter Modes" on page 308
- "Behaviour" on page 439
- Chapter 10.3.3, "Mode editor settings", on page 317

Emitter attitude

The attitude is a description of the emitter/interferer orientation. Emitters/Interferers can be rotated in three dimensions: yaw, pitch and roll.

The examples in Table 10-1 show an emitter with a cosecant antenna pattern in a single emitter scenario. In direction finding scenarios and scenarios with multiple emitters, the effect is the same.

Table 10-1: Impact of the yaw, pitch and roll angles on the emitter attitude

Attitude	Rotation around axis	Rotation
Yaw	Z-axis	In the x/y plane, to the left/right Yaw = 45° ("Pitch = Roll = 0°")
Pitch	X-axis	In the y/z plane, up/down Pitch = 45° ("Yaw = Roll = 0°")
Roll	Y-axis	In the x/z plane Roll = 45° ("Yaw = Pitch = 0°")
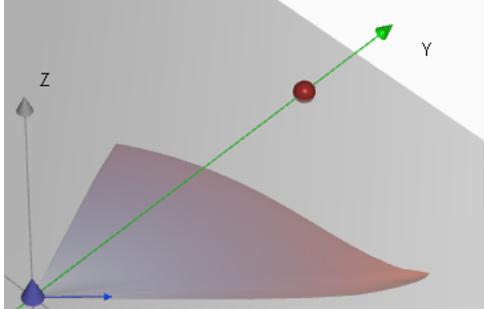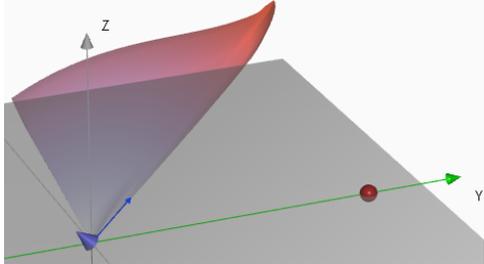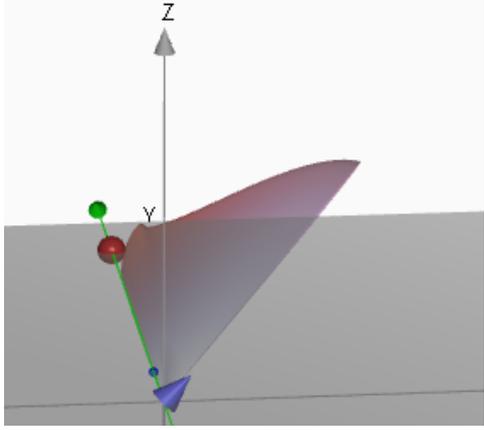
10.1 Emitter settings

Access:

► Select "Repository Tree > Emitter > New".
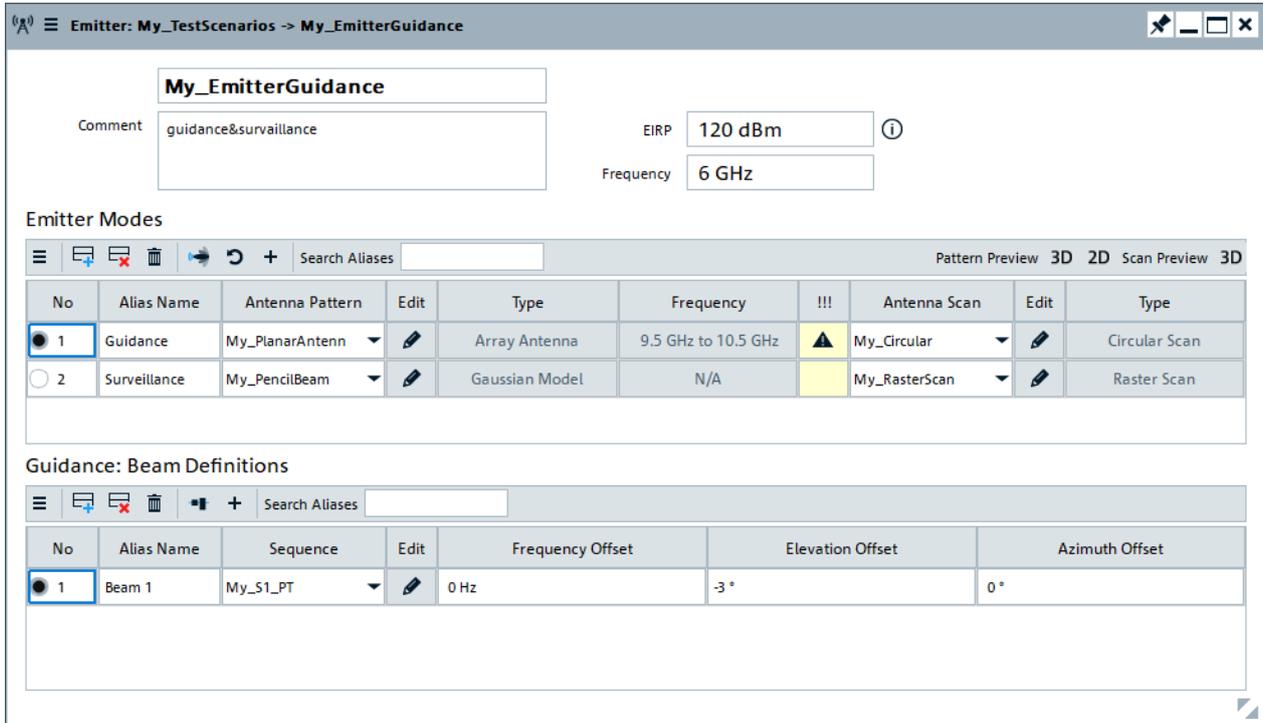
An emitter is described as a combination of an operating mode, an antenna pattern, an antenna scan, and a sequence. Complex emitters support several beams per operating mode.
Enter the settings either in the table or in the dialog fields.

Settings:

Emitter Name
Enter the emitter's name.

Remote command:
EMITter:NAME on page 589
EMITter:CREate on page 588
EMITter:SELect on page 588

EMITter:CATalog? on page 587

EMITter:REMove on page 590

Comment

Enter a short description.

Remote command:

EMITter:COMMent on page 589

EIRP

Sets the equivalent isotopically radiated power (EIRP) of the emitter.

This parameter is used, if the emitter is configured in a "Scenario Type > Localized Emitter".

Example:

For emitter with:
- Transmit power = 1 MW, i.e. 60 dBW
- Antenna gain = 20 dBi

Set "EIRP = 80 dBW".

Remote command:

EMITter:EIRP on page 642

Frequency

Sets the emitter's operating frequency.

If the selected antenna is calculated for a particular frequency, set the emitter's "Frequency" to a value within the suggested frequency range, see "Emitter Modes" on page 308.

Remote command:

EMITter:FREQuency on page 642

Emitter Modes

Defines one or more modes that the emitter works in (see "Operational mode" on page 305).

Use the standard functions in the context menu or in the table to create or rename modes.

Copy a selected line in the table and append it on the existing table via "+".

Search for a defined emitter mode via "Search Alias".

Each emitter mode is defined by the combination of an antenna pattern, antenna scan and at least one beam with assigned pulse sequence.

R&S Pulse Sequencer can emulate radar systems that change their mode over time, see "Behaviour" on page 439.

Use the 3D view to visualize the selected antenna scan and pattern, see:
- "2D and 3D diagrams" on page 265
- "3D Scan View" on page 282

"Antenna Pattern"

Selects an existing antenna pattern or accesses a dialog for creating an antenna pattern.

See "To create an antenna pattern" on page 294.

The antenna pattern and type are displayed.

A warning symbol indicates that the current antennas design frequency does not match the emitter's Frequency.
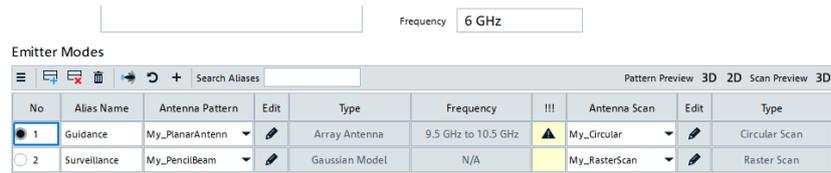
Set the emitter's Frequency to a value within the suggested frequency range.

"Antenna Scan"

Selects an existing antenna scan or accesses a dialog for creating an antenna scan.

See "To create an antenna scan" on page 297

Remote command:

`EMITter:MODE:NAME` on page 589

`EMITter:MODE:ADD` on page 591

`EMITter:MODE:COUNt?` on page 592

`EMITter:MODE:SELect` on page 593

`EMITter:MODE:CLEar` on page 595

`EMITter:MODE:DELete` on page 594

`EMITter:MODE:ANTenna` on page 642

`EMITter:MODE:ANTenna:CLEar` on page 595

`EMITter:MODE:SCAN` on page 643

`EMITter:MODE:SCAN:CLEar` on page 595

Emitter Beams Definition

In each operating mode, the emitter uses one or more beams (e.g. with different pulse sequences), while using the same antenna and scan type. Emitters use multiple beams for exact position detection, especially when the distance to the objects and beamwidth increase.

Use the standard functions in the table to create or rename beams.

Search for a defined beam via "Search Alias".

Assign a pulse sequence to each beam.
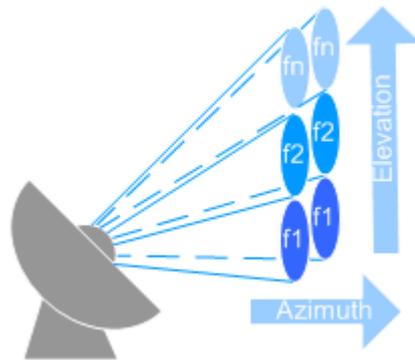
Figure 10-1: Stacked beams (simplified representation)

"Sequence"   Selects an existing sequence or accesses a dialog for creating a sequence.
                         See Chapter 7.3, "How to create sequences and use the control elements", on page 212

"Frequency Offset"
                         Offsets the beam frequency by the selected value, see Figure 10-1.

"Beam Position Offset"
                         Offsets the beam in terms of an "Elevation" and an "Azimuth", see Figure 10-1.

                         To *offset the complete emitter's antenna*, use one of the "Elevation" parameters, depending on your current scenario:
- "Single Emitter" scenario: Elevation
  (See "To configure an emitter in a scenario with static receiver" on page 321)
- "Localized Emitters" scenario: Elevation

Remote command:
EMITter:MODE:BEAM:ADD on page 591
EMITter:MODE:BEAM:COUNt? on page 592
EMITter:MODE:BEAM:NAME on page 589
EMITter:MODE:BEAM:SELect on page 593
EMITter:MODE:BEAM:STATe on page 643
EMITter:MODE:BEAM:SEQuence on page 643
EMITter:MODE:BEAM:OFFSet:AZIMuth on page 642
EMITter:MODE:BEAM:OFFSet:ELEVation on page 642
EMITter:MODE:BEAM:OFFSet:FREQuency on page 643
EMITter:MODE:BEAM:CLEar on page 595
EMITter:MODE:BEAM:DELete on page 594

10.2 3D single emitter preview settings

Access:

1. Open a "Single Emitter" or an "Emitter (Collection)" scenario.

2. In the block diagram, select "3D".

   The dialog displays:
   ● A 3D view of the emitter antenna with its pattern and scan
   ● A live plot of the normalized signal power level at the receiver
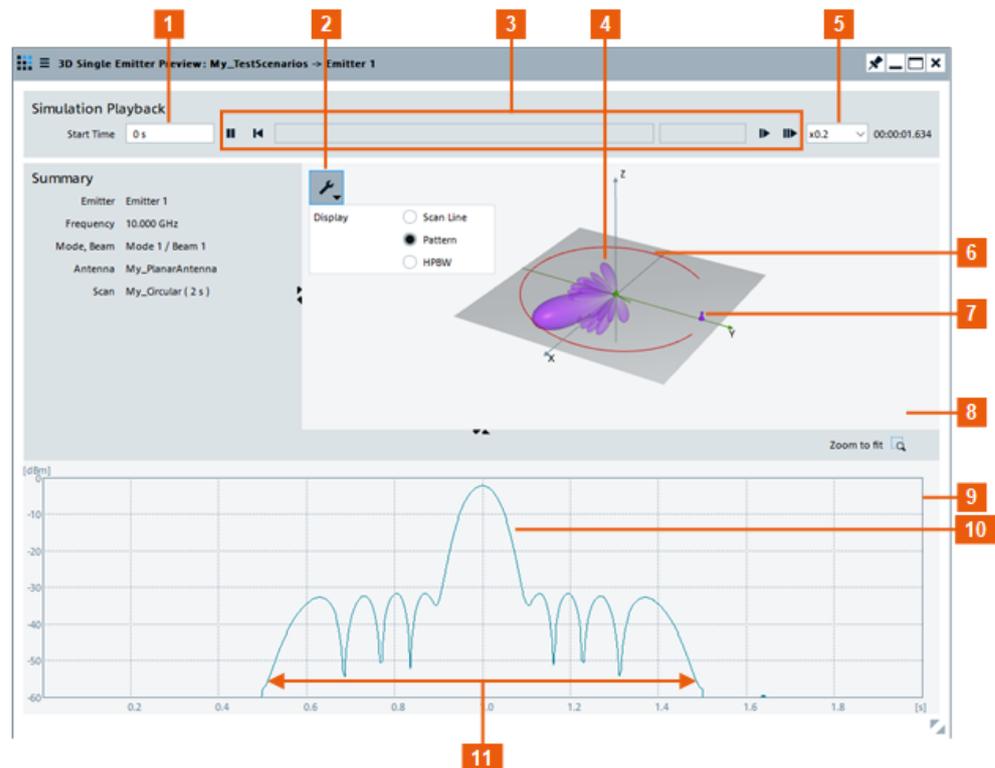
Figure 10-2: 3D Emitter Preview: Understanding the displayed information

1 = Preview time span (selected is one antenna turn)
2 = "Display settings" icon; defines how the scan is visualized
3 = Time bar; use the slider to set the "Start Time"
4 = Antenna pattern diagram of the emitter; antenna pattern without back lobes
5 = Play speed and time counter
6 = Circular antenna scan
7 = Receiver with an isotropic antenna pattern, no antenna scan and at a fixed location
8 = Interactive 3D view
9 = Live plot of the normalized signal power level at the receiver
10 = Antenna pattern without back lobes
11 = Antenna pattern (main and side lobes), compare with the pattern on Figure 9-12

Available settings:

Simulation Playback

Controls the timing of the simulation.

For example, with these settings you can analyze when/where/how often the beams overlap while simulating:

- A static scanning RX and a static scanning TX with different scan cycles.
- A moving RX and static TX
- A moving TX with different modes and a static RX
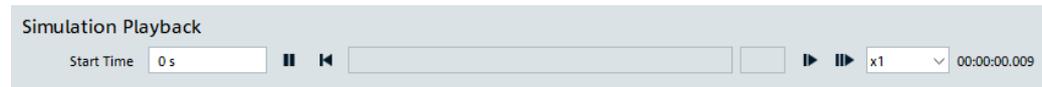
Figure 10-3: Simulation Playback

In addition to setting the overall duration, you can also observe a particular time-segment of the simulation by starting and stopping at particular times.

Note: The duration of a simulation depends on the scenario. An emitter with a long trajectory and low speed causes a long duration. A short trajectory and/or high speed cause a short duration.
The duration provides the maximum value for the "Stop Time" and "Preview Interval" settings.

"Start Time"   Set to *0 s* or *minimum*, to play the simulation from the beginning.
                   Typically, set this value to begin the simulation at a particular point
                   along the trajectory.

Buttons   Control the simulation using the following buttons:

                   - Pause/Play - use this toggle to pause the simulation and resume
                     playing.
                   - Restart - use this button to restart the simulation at the configured
                     "Start Time".
                   - Play slower/Play faster - decrease/increase the "Speed-up factor"

Display settings
Defines the way that the scan is represented:

"Scan Line"   The scan is visualized by a line; the antenna pattern is disregarded.

"Pattern"   The visualization corresponds to the current selected antenna pattern.

10.3 Emitters (Collection) settings

Access:

1. Open an "Emitters (Collection)" scenario.

2. In the block diagram, select "Emitters > Edit".

   The "Emitters (Collection)" scenario comprises several emitters. They display in list form.
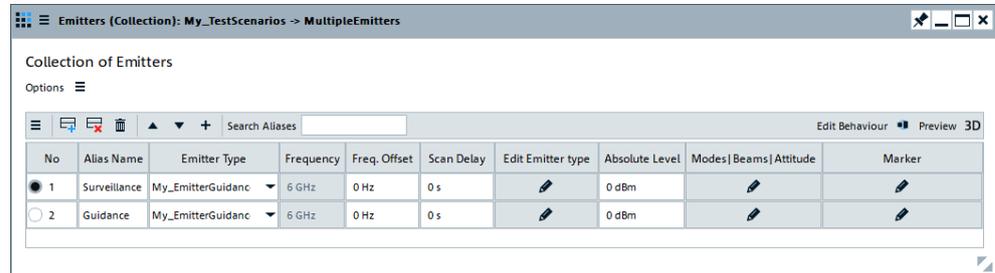
3. To observe and configure the properties of the individual emitters, select "Edit Emitter type". To configure attitude and behavior, select "Modes|Beams|Attitude".
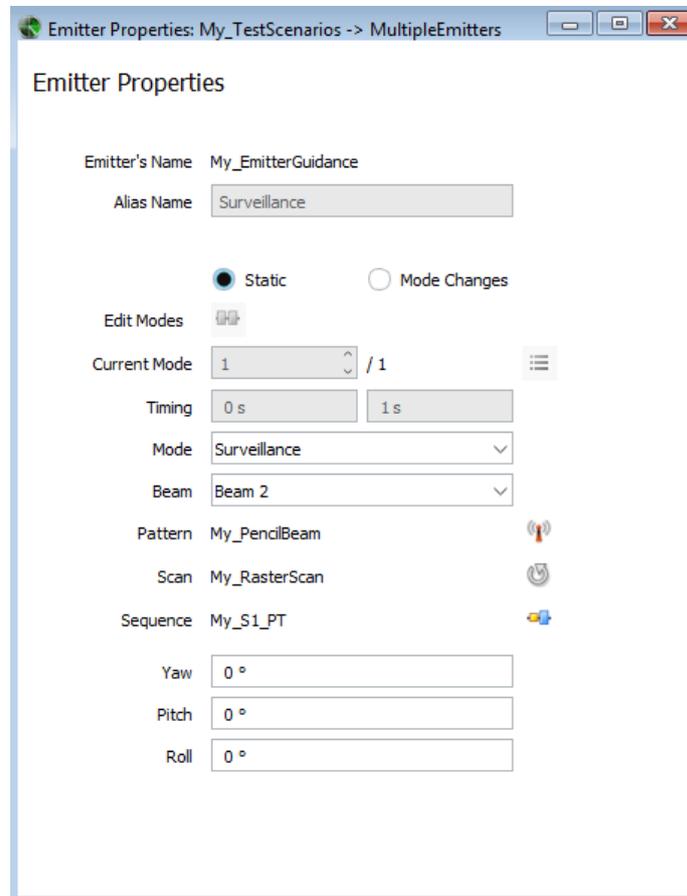
The display resembles the emitters properties as they are configured in the "Emitter" dialogs, see Chapter 10.1, "Emitter settings", on page 307.

10.3.1 Collection of emitters

Settings:

Search Aliases
Allows a fast search of the configured emitters; useful in scenarios with many emitters.

Filters and displays emitters in table rows according to the entered search string.

Options
Joint configuration of these values for all emitters at once.

- "Reset Freq. Offset"
- "Reset Scan Delay"
- "Reset Abs. Level"

Insert, Append, Prepend, Remove, Clear All, Move Up/Down, Copy and append
Standard functions, available as context menu and as icons.

See also "To configure complex scenarios with several emitters" on page 322.

Remote command:
SCENario:CEMit:ADD on page 591
SCENario:CEMit:SELect on page 592
SCENario:CEMit:DELete on page 595
SCENario:CEMit:CLEar on page 596

Preview 3D
Opens the "3D Scan Pair View" for the selected "Emitter".

See Chapter 16.3, "3D scan pair view settings", on page 429.

Edit Modes
Accesses the "Mode Editor" dialog.

This dialog allows graphical configuration for defining the duration that an emitter is working in the different modes.

Works like the Mode Changes settings but rather than defining the start and end time of a mode, you drag the start and end time on the time scale.

See:
● "Behaviour" on page 439
● Chapter 10.3.3, "Mode editor settings", on page 317

List of emitters
Displays an overview of the configured emitters in table form.

No ← List of emitters
Subsequent number.

Selects the emitter for that the "Preview 3D" and "Edit Modes" apply.

Remote command:
SCENario:CEMit:SELect on page 592

Alias Name ← List of emitters
Enters an alias name.

Remote command:
SCENario:CEMit:ALIas on page 746

Emitter Type ← List of emitters
Selects an emitter.

Remote command:
SCENario:CEMit:EMITter on page 748

Frequency ← List of emitters
Indicates the Frequency value of the selected emitter.

Frequency offset ← List of emitters
Sets a frequency offset for the selected emitter.

Use this feature to configure the same emitter transmitting on different frequency.

Remote command:
SCENario:CEMit:FQOFfset on page 754

Scan delay ← List of emitters
Adds a time delay for the selected emitter and thus shifts the scans in time compared to each other.

Use this feature to simulate more realistic situation in that the emitters do not all point to the receiver at the beginning of the simulation.

Remote command:
SCENario:CEMit:SCNDelay on page 754

Edit Emitter Type ← List of emitters
Accesses the Emitter settings dialog.

Absolute Level ← **List of emitters**

Sets the absolute level of the selected emitter.

Remote command:
SCENario:CEMit:LVABs on page 754

Modes|Beams|Attitude ← **List of emitters**

Accesses the Emitter properties dialog that shows an overview information of the properties of the selected emitter.

For emitters that can work in several modes, the dialog allows configuration of the mode changes. Some other settings are read-only.

Markers

Accesses the "Emitter Marker Config" dialog.

See "Emitter Marker" on page 441.

10.3.2 Emitter properties

Each emitter is described with the following parameters:

Emitter's Name, Alias Name

Indicates the emitter name and its alias name, as selected in the Collection of emitters table.

Edit Modes

Accesses the "Mode Editor".

The mode editor allows graphical configuration for defining the duration that an emitter is working in the different modes.

Works like the Mode Changes settings but rather than defining the start and end time of a mode, you drag the start and end time on the time scale.

See:
● "Behaviour" on page 439
● Chapter 10.3.3, "Mode editor settings", on page 317

Static, Mode Changes

Enables emulation of emitters that change their mode over time.

See "Behaviour" on page 439.

| "Mode" | Selects the mode that the emitter is working in. |
| "Beam" | Sets the number of the currently used beam. |

Remote command:
SCENario:CEMit:EMITter:MODE on page 748
SCENario:CEMit:EMITter:MODE:BEAM on page 748

Antenna Pattern, Scan, Sequence
Displays the current selected antenna pattern, antenna scan, and sequence.

To change any of them, select the corresponding icon.

Pitch, Yaw, Roll
Offsets the antenna in terms of a "Pitch", a "Yaw" and a "Roll", see "Emitter attitude" on page 305.

Remote command:
SCENario:CEMit:DIRection:PITCh on page 747
SCENario:CEMit:DIRection:YAW on page 746
SCENario:CEMit:DIRection:ROLL on page 747
SCENario:CEMit:EMITter:MODE:TRACkrec on page 749

10.3.3 Mode editor settings

Access:

1. Open an "Emitter (Collection)" scenario.

2. In the block diagram, select "Emitters > Edit".

3. Select "Edit Modes".

   You access the "Mode Editor" dialog for configuration of the mode changes of *all emitters* available in the scenario.
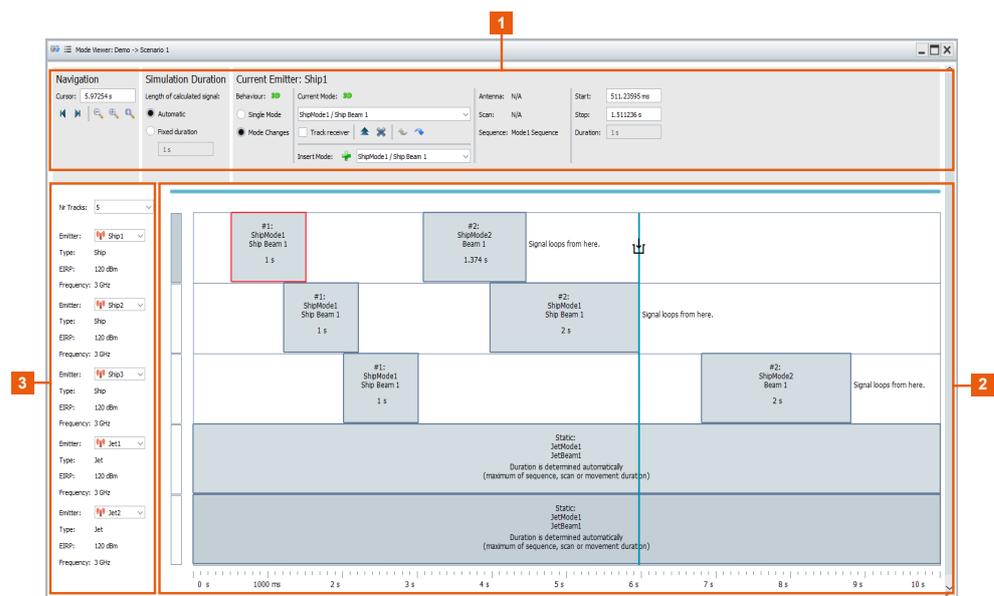
Figure 10-4: Mode Editor in Scenario Type = Localized Emitters

1 = Emitter and visualization settings
2 = Visualization area
3 = Configuration of the visualization area: select the number of displayed tracks ("Nr Tracks") and
　　which emitter is displayed in the respective track

The "Mode Editor" is a graphical editor for joint configuration and defining the duration emitters are working in the different modes.
Works like the Mode Changes settings for each of the emitters. Rather than defining the start and end time of a mode, you drag the start and end time on the time scale.

Settings:

Navigation
Standard functions: Move start/end into view and zooming.

"Cursor" indicates and sets the current cursor position.

Simulation Duration
Resembles the settings in "Clock, Duration" on page 479. Included here for your convenience.

Current Emitter
You can configure mode changes only for emitters with several modes, see "Emitter Modes" on page 308.

Behavior ← Current Emitter
Select the mode for the emitter.

"Single Mode"　　　Constant behavior over time.

"Mode Changes"
　　　　　　　　　Change of behavior over time.

Current Mode ← Current Emitter
Sets the mode of the selected emitter.

Track receiver ← Current Emitter
If activated, modes track and follow the receiver with their antenna scan.

Editing Emitter Sequences ← Current Emitter

Edit emitter sequences directly in the visualization area.

⬆ Extract selected mode

Removes the selected mode and its respective timeslot. Subsequent modes are shifted to close the timeslot of the removed mode.

✖ Lift selected mode

Removes the selected mode. Timeslot of the removed mode is left as gap in the sequence.

↶ ↷ Undo/Redo

Standard functions to reverse or repeat operation.

Insert Mode ← Current Emitter

➕ Inserts an emitter with the selected mode/beam at the cursor position.

Antenna/Scan/Sequence ← Current Emitter

Displays the mode and beam configuration of the selected emitter.

Start/Stop ← Current Emitter

Sets the beginning and the end time of the selected mode.

To define the beginning and end time, use one of the following:

● Enter the time value in the corresponding field
● Drag the beginning and end borders of the bar representing the selected mode on the graph

Duration ← Current Emitter

Indicates the duration of the selected mode. The value is calculated automatically from the selected "Start" and "Stop" values.

Remote command:

SCENario:DF:MCHG:STATe on page 756

SCENario:DF:MCHG:SELect on page 593

SCENario:DF:MCHG:ADD on page 591

SCENario:DF:MCHG:CLEar on page 596

SCENario:DF:MCHG:DELete on page 595

SCENario:DF:MCHG:COUNt? on page 592

SCENario:DF:MCHG:STARt on page 756

SCENario:DF:MCHG:STOP on page 756

Graph

Comprises one or more bars, one per emitter available in the scenario.

To configure a particular emitter, select it. A blue bar next to the name indicates the currently selected one.

To edit a mode, select it. Currently selected modes are highlighted in red. You can set the start and end time of a mode by dragging it on the time axis or by setting the "Start" and "Stop" time.

10.4 How to create and configure emitters

See:

- "To create and configure a new emitter" on page 320
- "To configure an emitter in a scenario with static receiver" on page 321
- "To visualize the signal received by a static receiver" on page 321
- "To configure complex scenarios with several emitters" on page 322

To create and configure a new emitter

You can clone or copy an existing emitter or create an emitter:

1. Select "Repository Tree > Emitter > New".

2. Enter a name and a comment.

3. Select "Mode 1".
   To rename the emitter, double-click the alias name. Enter a name for the emitter mode, e.g. "Guidance".

4. Select "Ant. Pattern" and select an antenna pattern from the antenna library.

5. Select "Antenna Scan" and select an antenna scan from the scan library.

6. To configure the "Beam":

   a) Select the beam via "No".
   b) Select "Sequence" and select a sequence from the library.

7. If necessary, add a second beam.
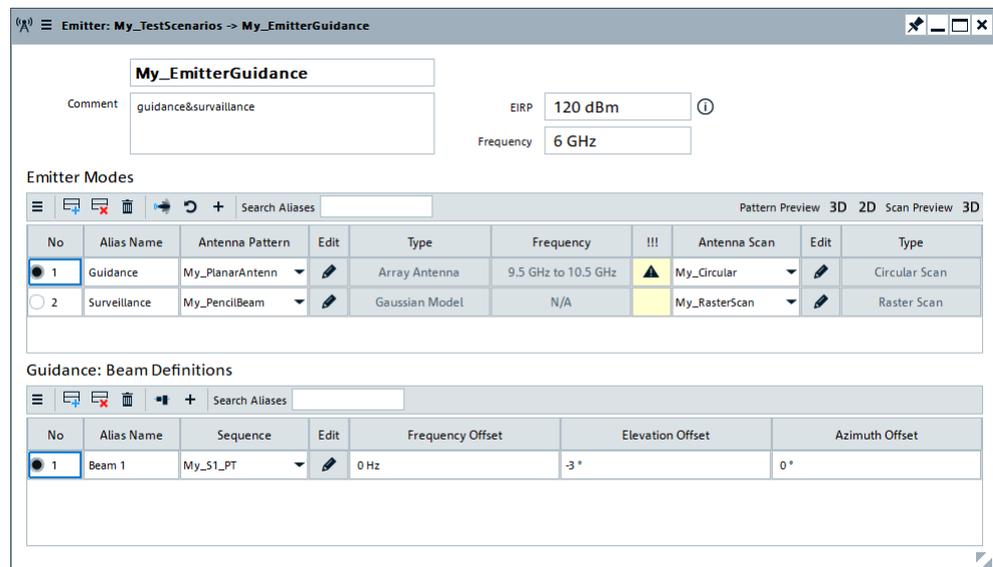
For description of the provided settings, see Chapter 10.1, "Emitter settings", on page 307.

To configure an emitter in a scenario with static receiver

1. Open a "Single Emitter" scenario.
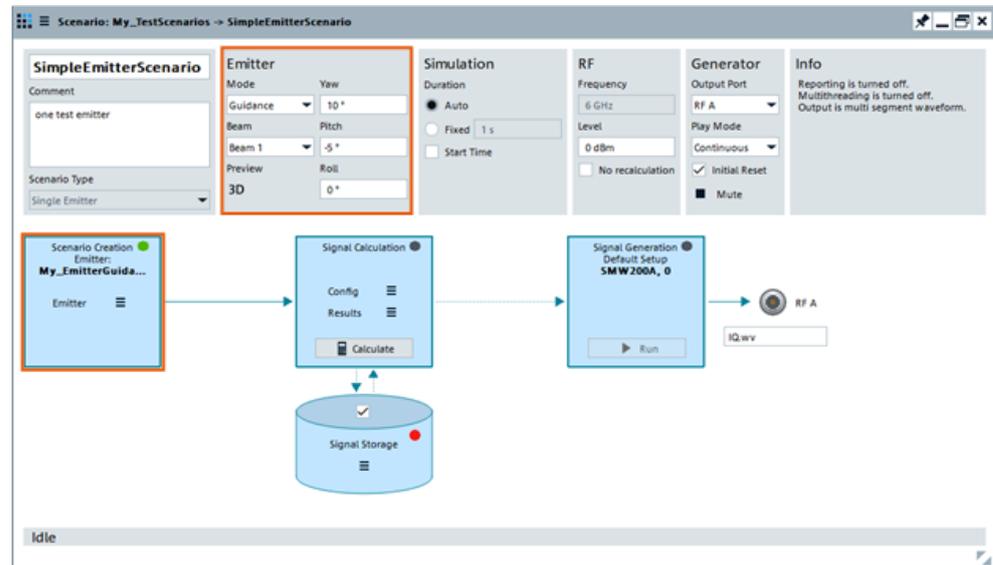
Figure 10-5: Single emitter scenario settings

2. In the block diagram, select "Scenario Creation > Emitter".
   In the context menu, select an emitter from the list of available emitters.

3. Configure the emitter settings, for example, select the current "Emitter Mode" and "Emitter Beam".

   For a description of the possible settings, see Chapter 10.3, "Emitters (Collection) settings", on page 312.

To visualize the signal received by a static receiver

1. Open a scenario with single emitter, e.g. "SimpleEmitterScenario".

2. In the block diagram, select "Emitter" settings > "Preview 3D".

   The dialog displays the signal as "seen" by a static receiver. By default, the receiver is located in such a way that the emitter scan is oriented with 0 deg "Pitch" and "Yaw" at it.

3. To "move" the receiver on the x-axis and on the z-axis, use "Emitter" settings > "Pitch" and "Yaw" (see Figure 10-5). The position on the y-axis (the distance to the emitter) is a fixed value.
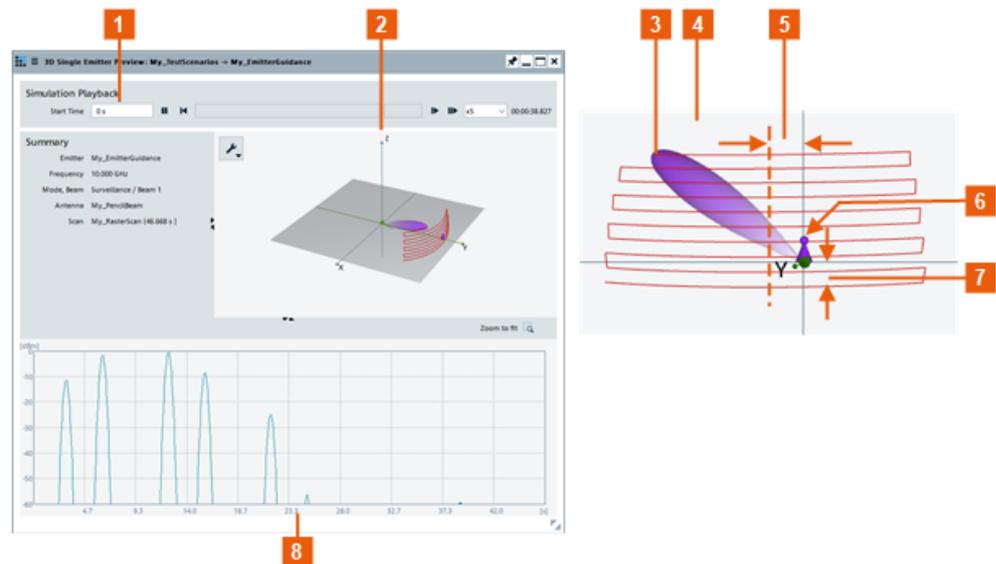
Figure 10-6: Signal at the receiver (XZ view): understanding the displayed information

1 = Selection of preview time span (selected is one antenna turn). See "Simulation Playback" on page 312.

2, 4 = 4 is the XZ representation of the 2 3D view; 2 uses "Scan Line" = "Pattern"; 4 uses "Scan Line" = "On"

3 = Current position of the scanning beam

5 = "Yaw = 10 deg" turns the antenna horizontally, i.e. the receiver is not at the central scan axis but rotated in clockwise (CW) direction on the x-axis

6 = Receiver's position, determined by the selected "Yaw" and "Pitch"; the position on the y-axis (the distance to the emitter) is fixed

7 = "Pitch = -5 deg" turns the antenna vertically, i.e. "moves" the receiver on the z-axis

8 = Amplitude changes in the received signal over time

4. Use the mouse to change the orientation of the interactive 3D diagram and to zoom on it.

For a description of the provided settings, see Chapter 10.2, "3D single emitter preview settings", on page 311.

To configure complex scenarios with several emitters

If your test situation requires testing the receiver's ability to detect the signal from different static emitters, you can use the "Emitter (Collection)" scenario. In this dedicated scenario, you can configure several emitters and switch between them sequentially. You can also configure the receiver and change its position in the scanning beam of the emitters.

The following example creates a complex scenario with several emitters and a receiver to explain the principle.

1. Open an "Emitter (Collection)" scenario.
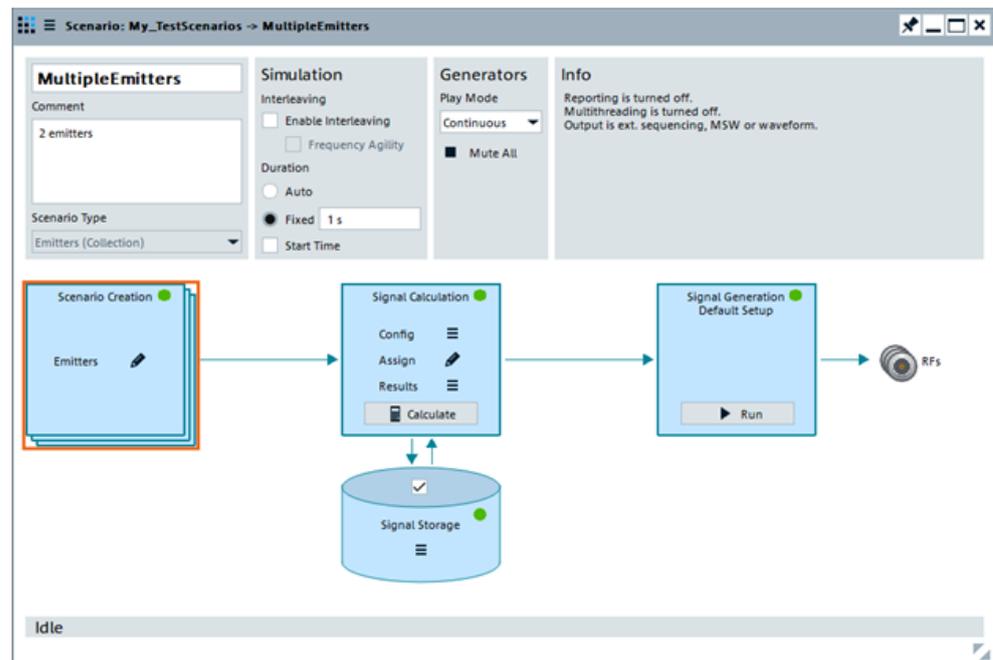
2. In the block diagram, select "Scenario Creation > Emitters".

   The "Emitters (Collection)" dialog opens.

3. Use the standard "Append" function to insert the first emitter in the list.

   You can select from the list of available emitters or create an emitter.

4. Configure the emitter settings, for example, enter an alias name.

5. In the "Emitters (Collection)" dialog, select the emitter that is transmitting, e.g. "Surveillance" in the "No" column.

6. From the controls above the table, select the "Preview 3D" view to visualize the signal received by a static receiver.
   See Figure 10-6.

   For a description of the provided settings, see:
   ● Chapter 10.3, "Emitters (Collection) settings", on page 312
   ● Chapter 10.2, "3D single emitter preview settings", on page 311

To create a single emitter scenario automatically with the "Startup Assistant"

Alternatively to the methods described in this section, you can use the "Startup Assistant" to create a single emitter scenario.

1. In the menu bar, select "Help > Wizard".

2. Select "Create an emitter".
   Click "Next".

3. Follow the instructions, use the default settings and select "Finish".
   For detailed step-by-step description, see Chapter 2.7.5, "Using the wizard to create a complex scenario", on page 40.

   For information on the provided settings, see:
   ● Chapter 10.1, "Emitter settings", on page 307
   ● Chapter 9.2, "Antenna pattern settings", on page 261
   ● Chapter 9.3, "Antenna scans settings", on page 280
   ● Chapter 6.2.5, "Modulation on pulse (MOP) settings", on page 162

11 Combining the signals of multiple emitters

In a real-world scenario, radar receivers have to detect and process several simultaneous signals form different emitters. Such scenarios are referred as multi-emitter scenarios. In multi-emitter scenarios, the signal at the radar receiver is a combination of the signals of several emitters.

In the R&S Pulse Sequencer, you have different possibilities to generate multi emitter signals. Select the one that best fits your test setup.

Signal generator equipped with:	Approach	Method	Suitable for	See
One signal path For example: R&S SMBV100B R&S SMW with one baseband and one RF	Combine the signals of the different emitters Generate time-overlapping pulse signals	Overlay	Scenarios without antenna patterns or antenna scans Example: radar receiver component tests	• "To define and enable overlaying segments" on page 217 • Chapter 7.1, "About the sequencing principles", on page 198
		Interleaving	Test signal comprises the signals of more than two or *many emitters*	"About the interleaving" on page 326
Two signal paths For example: R&S SMW	Generate the signals of *two emitters* and output them at the two RF outputs of the signal generator. Combine the signals with an RF combiner.		Simulation of emitters transmitting in *different frequency bands.* Example: single input radar receiver tests	• Chapter 10, "Emulating emitters", on page 305 • Chapter 16, "Creating complex 2D scenarios with receiver and TX items", on page 415
	Assign the emitter signals to the same RF output of the R&S SMW		Simulation of emitter transmitting at different frequencies but in the *same frequency band*	• Chapter 17, "Playing the generated waveform files", on page 475 • Chapter 3, "Preparing the system for use", on page 55
Two signal paths incl. wideband baseband generators Option pulse-on-pulse simulation **R&S SMW-K315**	Generate the signals of *up to 6 emitters* simultaneously and output then at the two RF outputs the signal generator. Combine the signals with an RF combiner.	Pulse-on-pulse simulation	Test of radar behavior in a *realistic multi-emitter scenario*, where pulse dropping is not allowed	R&S®SMW-K501/-K502/-K315 Extended and Real-Time Sequencing, Pulse-on-Pulse Simulation user manual

In the R&S Pulse Sequencer, you have different possibilities to generate multi emitter signals. Select the one best fitting your test setup.

- Signal generator equipped with **one signal path**, e.g. R&S SMBV100B or R&S SMW with one baseband and one RF
  Obviously, if only one RF output is available, the signals of the different emitters have to be combined. Thus, you have to generate time-overlapping pulse signals.
  - In the R&S Pulse Sequencer, the simplest way to achieve time-overlapping pulse signals is by using the **overlay** functionality.

This method is suitable if no antenna patterns or antenna scans are required, for example for radar receiver component tests.
See "To define and enable overlaying segments" on page 217 and Chapter 7.1, "About the sequencing principles", on page 198.

– Especially if the test signal comprises the signals of more than two emitters, you can use the **interleaving** function, see "About the interleaving" on page 326.
Interleaving signals is the best choice, if scenario comprises of *many emitters*.

● Signal generator equipped with **two signal paths**, e.g. R&S SMW
A two-path signal generator is required, if you need to simulate emitters transmitting in *different frequency bands* and test a **single input radar receiver** for instance.
A suitable approach is to generate the signals of two emitters and output them at the two RF outputs of the signal generator. Then use an RF combiner to combine them into one RF signal.
If, however, the emitter signals are at different frequencies but in the *same frequency band*, you can assign the emitter signals to the same RF output of the R&S SMW.

For more information and examples, see:

– Chapter 10, "Emulating emitters", on page 305

– Chapter 16, "Creating complex 2D scenarios with receiver and TX items", on page 415

– Chapter 17, "Playing the generated waveform files", on page 475

– Chapter 3, "Preparing the system for use", on page 55

● Signal generator equipped with two signal paths, two wideband baseband generators and option pulse-on-pulse simulation **R&S SMW-K315**
If this option is installed, you can generate the signals of *up to 6 single emitters simultaneously* and output then at the two RF outputs. The signals can be then fed to the received individually or combined via external combiner.
This mode of signal generation is known as **pulse-on-pulse simulation** and is available as extension of the extended sequencer functionality in R&S SMW.
Pulse-on-pulse simulation is the best choice, if your task is to test the radar behavior in a *realistic multi-emitter scenario*, where pulse dropping is not allowed.
For details, see R&S®SMW-K501/-K502/-K315 Extended and Real-Time Sequencing, Pulse-on-Pulse Simulation user manual.

About the interleaving

In addition to processing the emitters sequentially, you can also combine them into a single output file.

During the interleaving process, overlapping pulses within the same interleaving group and with lower priority are dropped. The priority of the individual emitters is configurable as is the definition of several groups and the assignment of the emitters to them.
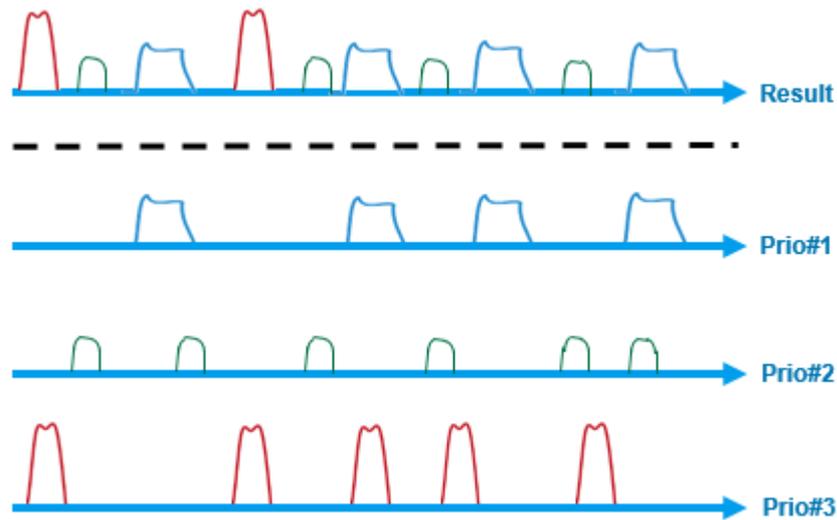
Figure 11-1: Principle of the priority-based interleaving algorithm (applied per interleaving group)

Prio#1 = Pulses with the highest priority (i.e. the priority value is 0 or smaller than the value for the other emitters)
Prio#3 = Pulses with the lowest priority (i.e. the highest priority value within the emitters)

Interleaving signals and combining them into single output signal is function provided in the following scenario types:

- Emitters (collection)
- Localized emitters
- Direction finding
- PDW list (collection)

This section described the settings in the first three scenario types. For description of the settings in the last one, see Chapter 18.6, "PDW list (Collection) interleaving settings", on page 516.

11.1 Emitters (Collection) and localized emitters interleaving settings

Access:

1. In a "Scenario Type = Emitters (Collection)/Localized Emitters/Direction Finding", select "Emitter > Interleaving > On".

2. Select "Interleaving > Config".

   Dialog displays the available emitters as configured in the dialogs:
   - Emitters (Collection)
   - "Localized Emitters" > Properties of TX items (emitters/interferers and plat-forms)
   - "Direction Finding" > Properties of TX items (emitters/interferers and platforms).

   The interleaving related settings, like "Time Offset" and "Priority", are also dis-played.

Alias Name	Emitter Type	Frequency	Freq. Offset	Scan Delay	Marker	Enable	Time Offset	Prio [0=Highest]	Level Offset	Interleaving Group
Surveillance	My_EmitterGuidanc	6 GHz	0 Hz	0 s	✏	✓	0 s	0	0 dB	Default
Guidance	My_EmitterGuidanc	6 GHz	0 Hz	0 s	✏	✓	0 s	1	0 dB	Default

Settings:

Threshold

Enters a level threshold to limit the dynamic range of the signal. Pulses at levels below this threshold are omitted.

Resembles the value set with the parameter "Scenario > Signal Generation > Config > Waveform Generation > Output > Threshold".

Remote command:
SCENario:CEMit:THReshold on page 765
SCENario:DF:THReshold on page 765

`SCENario:LOCalized:THReshold` on page 765

`SCENario:OUTPut:THReshold` on page 765

Search Aliases

Allows fast search of the configured emitters; useful in scenarios with many emitters.

Filters and displays emitters in table rows according to the entered search string.

Interleaving Options

Joint configuration of all emitters at once.

- "Enable All"/"Disable All"
- "Reset Time offset"/"Reset Level offset"
- "Reset Prio"

Edit

Opens the "Interleaving Groups" dialog.

See "Edit Interleaving Groups" on page 330.

Alias Name

Displays the alias name.

Remote command:
`SCENario:CEMit:ALIas` on page 746

Emitter type

Displays the emitter name as selected with Emitter Type.

Remote command:
`SCENario:CEMit:EMITter` on page 748

Frequency

Indicates the Frequency value of the selected emitter, as selected in the Emitter dialog.

Remote command:
`SCENario:CEMit:FREQuency?` on page 753

Freq. Offset

Displays the frequency offset of the emitter as selected with Frequency offset.

Remote command:
`SCENario:CEMit:FQOFfset` on page 754

Scan delay

Displays the scan delay for the emitter as selected with Scan delay.

Remote command:
`SCENario:CEMit:SCNDelay` on page 754

Marker

Opens the "Emitter Marker Config" dialog.

See Emitter Marker.

Enable

Selects the emitter for interleaving.

If disabled, the signal of the emitter is not included in the interleaved output signal.

Remote command:

SCENario:CEMit:ENABle on page 758

SCENario:DF:ENABle on page 758

SCENario:LOCalized:ENABle on page 758

Time Offset

Available if "Enable" = On.

Shifts the signals relative to each other to prevent high drop out rates due to pulse overlap.

Use this function, for example, to decrease the number of dropped pulses.

Remote command:

SCENario:CEMit:LDELay on page 758

SCENario:DF:LDELay on page 758

SCENario:LOCalized:LDELay on page 758

Priority

Available if "Enable" = On.

Sets the priority of the selected emitter. Higher value means lower priority. If pulses are overlapping, the pulses from the emitter with lower priority are discarded.

See Figure 11-1.

Remote command:

SCENario:CEMit:PRIority on page 758

SCENario:DF:PRIority on page 758

SCENario:LOCalized:PRIority on page 758

Level Offset

Available if "Enable" = On.

Adds a relative level offset between the signals.

Remote command:

SCENario:CEMit:LEVel on page 759

SCENario:DF:LEVel on page 759

SCENario:LOCalized:LEVel on page 759

Interleaving Group

Assigns the emitter to one of the available interleaving groups, defined in the Edit Interleaving Groups dialog.

Remote command:

SCENario:CEMit:GROup on page 759

SCENario:DF:GROup on page 759

SCENario:LOCalized:GROup on page 759

Edit Interleaving Groups

Select the icon to access the "Interleaving Groups" dialog.
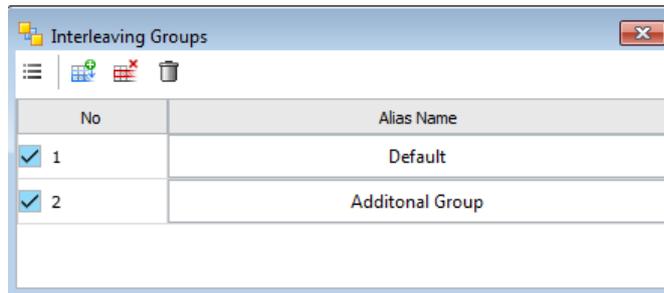
Interleaving groups allow you to combine the emitter signals into different output signals.

Groups can be generated by different signal generators or output at the different outputs of an instrument. See also Chapter 17.2, "Assign signal source or receiver signals to generator", on page 481.

Use the standard functions (icons) to add a new group, rearrange the groups, delete the existing one or all groups.

Select, Insert, Append/Prepend, Remove, Clear ← Edit Interleaving Groups
Standard functions for items handling, see "Standard function in the context menus" on page 33.

Select a row, for example, to delete it or to insert a row before it.

Remote command:
SCENario:CEMit:GROup:CATalog? on page 759
SCENario:CEMit:GROup:SELect on page 592
SCENario:CEMit:GROup:ADD on page 591
SCENario:CEMit:GROup:DELete on page 594
SCENario:CEMit:GROup:CLEar on page 595
SCENario:DF:GROup:CATalog? on page 759
SCENario:DF:GROup:SELect on page 592
SCENario:DF:GROup:ADD on page 591
SCENario:DF:GROup:DELete on page 594
SCENario:DF:GROup:CLEar on page 595
SCENario:LOCalized:GROup:CATalog? on page 759
SCENario:LOCalized:GROup:SELect on page 592
SCENario:LOCalized:GROup:ADD on page 591
SCENario:LOCalized:GROup:DELete on page 594
SCENario:LOCalized:GROup:CLEar on page 595

No. ← Edit Interleaving Groups
Select a row to perform any row-based actions.

Interleaving groups are indicated by consecutive number; also used for indication in remote control.

Remote command:
SCENario:CEMit:GROup:COUNt? on page 592
SCENario:CEMit:GROup:SELect on page 592
SCENario:DF:GROup:COUNt? on page 592

`SCENario:DF:GROup:SELect` on page 592

`SCENario:LOCalized:GROup:COUNt?` on page 592

`SCENario:LOCalized:GROup:SELect` on page 592

Alias Name ← Edit Interleaving Groups

Enter an alias name for the interleaving group.

Remote command:

`SCENario:CEMit:GROup:ALIas` on page 760

`SCENario:DF:GROup:ALIas` on page 760

`SCENario:LOCalized:GROup:ALIas` on page 760

11.2 Emitters interleaving progress statistics

Access:

1. In a "Scenario Type = Emitters (Collection)", select "Emitter > Interleaving > On".

2. Select "Signal Control > Prepare".

3. Select "Signal Control > Interleave".

   The "Interleaving Progress" display opens automatically during the signal calculation, if pulses lists are merged. It indicates the percentage of pulses being processed and dropped for each of the emitters.
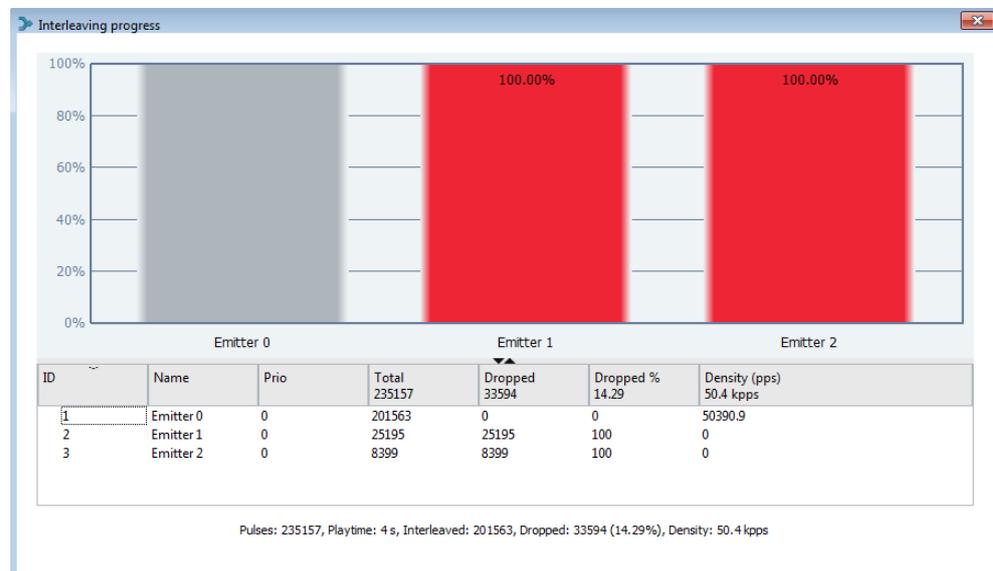
11.3 Signal preview with interleaved and dropping statistics

Access:

1. In a "Scenario Type = Emitters (Collection)", select "Emitter > Interleaving > On".

2. Select "Signal Control > Prepare".

3. Select "Signal Control > Interleave".

4. Select "Interleaving > Results > Interleaved PDW/Dropped PDWs".

The dialog is similar to the "Signal Preview" dialog.

This section describes the settings dedicated to the "Interleaved/Dropped" dialogs. For description of all other settings, see Chapter 15.2, "Signal preview settings", on page 405.

Settings:

Display > Filter
If interleaving is used, the calculated output signal contains pulses originating for multiple PDW lists or emitters. By default, all pulses are displayed. Use the "Filter" parameter to narrow down the displayed information to the pulses of one of the emitters/PDW lists.

The filter does not change the zoom level or the scaling on both axis.

Table

Provides summary information on the number of processes pulses and the content of the output file.

Graphics

Visualize the variation of the parameters over time.

For details, see "Graphics" on page 408.

Graphics > Highlight

Highlights pulses belonging to the selected emitter/PDW list.

Histogram

Resembles the information displayed during the signal processing, see Chapter 18.7, "PDW interleaving progress statistics", on page 519.

Use the filtering option to narrow down to the displayed information according to one of the predefined criteria.

Statistic

The drop out statistic is the table form representation of the same information as the Histogram.

11.4 Analyze interleaved signals

Access:

1. In a "Scenario Type = Emitters (Collection)", select "Emitter > Interleaving > On".

2. Select "Simulation > Interleaving".

3. Select "Signal Calculation > Calculate".

4. Select "Interleaving > Results > View > Analyze".

   The "Interleaving progess" dialog indicates the pulses that are and are not included in the resulting output file. Pulses displayed in **red are dropped out**.
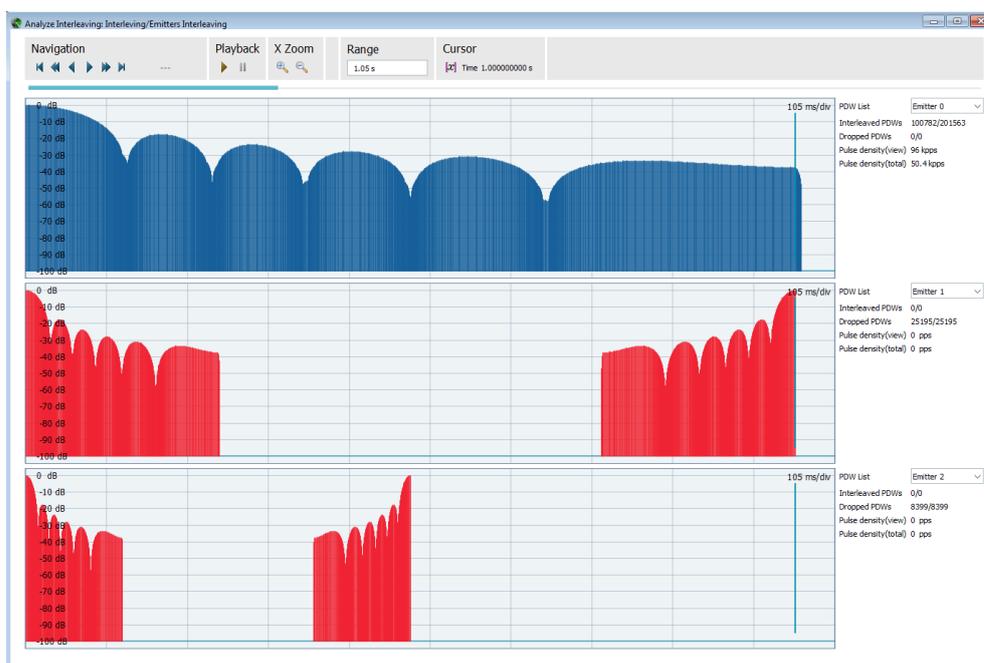
The provided settings are a subset of the settings available in the "Signal Preview" displays and they have the same effect.

For description, see Chapter 15.1, "Waveform view and signal preview settings", on page 400.

5. To increase the number of interleaved pulses, add a *delay to one of the emitters*.

   For example, zoom in the beginning of the signal until you can observe the individual pulses and estimated the required delay.

   Set, for example, "Interleaving > Config > Emitter#1 > Time Offset = 5 us".

   Create the output signal in the same manner. The signal is not recalculated; only the new interleaving settings applied.

   Observe the resulting output signal.

   Depending on the settings, the output file includes more or even all pulses originating from all emitters.

12 Creating platforms

This section describes the use of platforms in the R&S Pulse Sequencer. It covers their characteristics and explains the provided settings and views.

12.1 About platforms

A platform is a group of up to 8 emitters assigned to a single vehicle.

A platform is characterized by its icon, list of emitters, position and trajectory.

The emitters on a platform can be individually configured.

One or more platforms can be included in a scenario. Within a scenario, each platform can be individually enabled or disabled.

12.2 Platform settings

This section describes how platforms can be created and configured.

Access:

► To create a platform, do one of the following:

    a) In the repository tree, select "Platform Types" > context menu > "New".
    b) Select an existing platform > context menu > "Clone".

    A new platform is added to the repository. The "Platform" dialog opens.

    (i) R&S Pulse Sequencer generates new platform names automatically.
    You can edit the platform name.
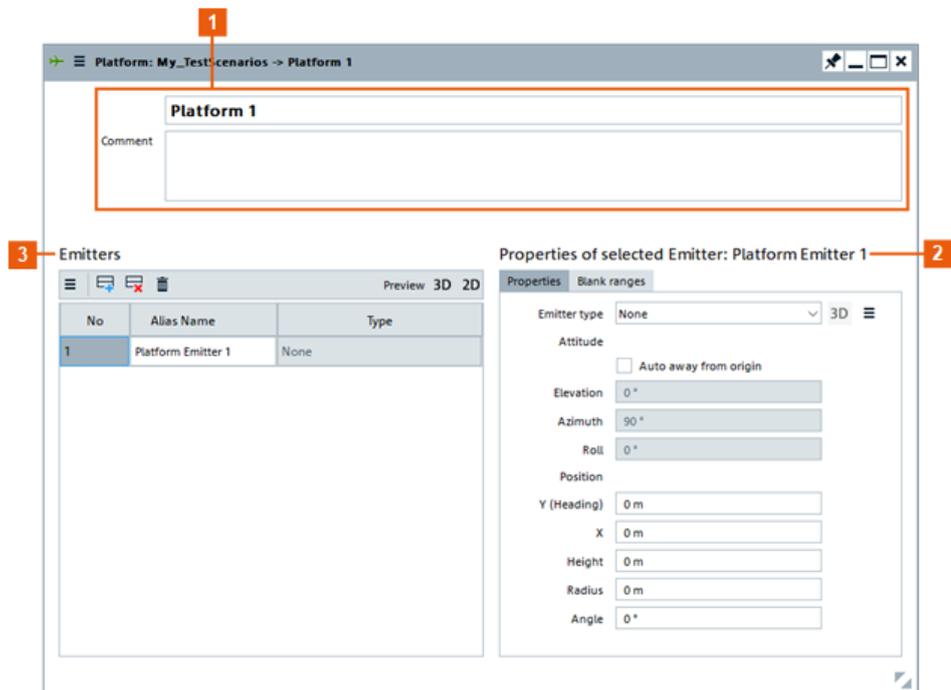
Figure 12-1: New platform in the Platform dialog

1 = Platform information
2 = Properties of selected emitter
3 = Emitter list

Settings

12.2.1 Platform information

Name

The name of the new platform is created automatically.

The format of the new name depends on how it was created.

- **New platform**
  The automatically generated name consists of the word *Platform* and a number -
  e.g. Platform 1. The number is incremented for every new platform.
- **Cloned platform**
  The name of a cloned platform consists of the name of the parent and a numerical
  index. For example:
  - Platform 1
  - Platform 1.1 = first clone of *Platform 1*
  - Platform 1.2 = second clone of *Platform 1*

– Platform 1.2.1 = first clone of *Platform 1.2*

To change the name, select the text and edit it.

Note: R&S Pulse Sequencer does not support duplicate names.
If you enter a duplicate name, a popup error message appears.

Remote command:
PLATform:NAME on page 589
PLATform:CREate on page 588
PLATform:SELect on page 588
PLATform:CATalog? on page 587
PLATform:REMove on page 590

Comment
Use this optional field to provide additional information about the platform.

Remote command:
PLATform:COMMent on page 589

12.2.2 Emitter list

Enables you to add, remove and edit the emitters of the platform.
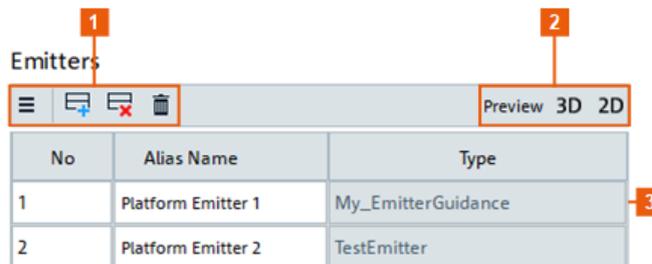
Figure 12-2: Emitter list example

1 = Icons for standard functions
2 = Icons for 2D or 3D previews of the platform
3 = List of emitters: by number ("No") and "Alias Name"

Icons for standard functions
Enable you to access properties, add or delete an emitter (see also Table 2-4).

List of emitters

A platform must have at least one emitter, so one unconfigured emitter is added to a new platform.

You can configure up to 8 emitters on a platform.

Select an emitter to view/edit its properties in Properties of selected emitter.

Remote command:
PLATform:EMITter:SELect on page 588
PLATform:EMITter:ALIas on page 782

Icon for 2D preview

Icon that accesses the 2D preview of the platform.

Figure 12-3 shows an example with two emitters.
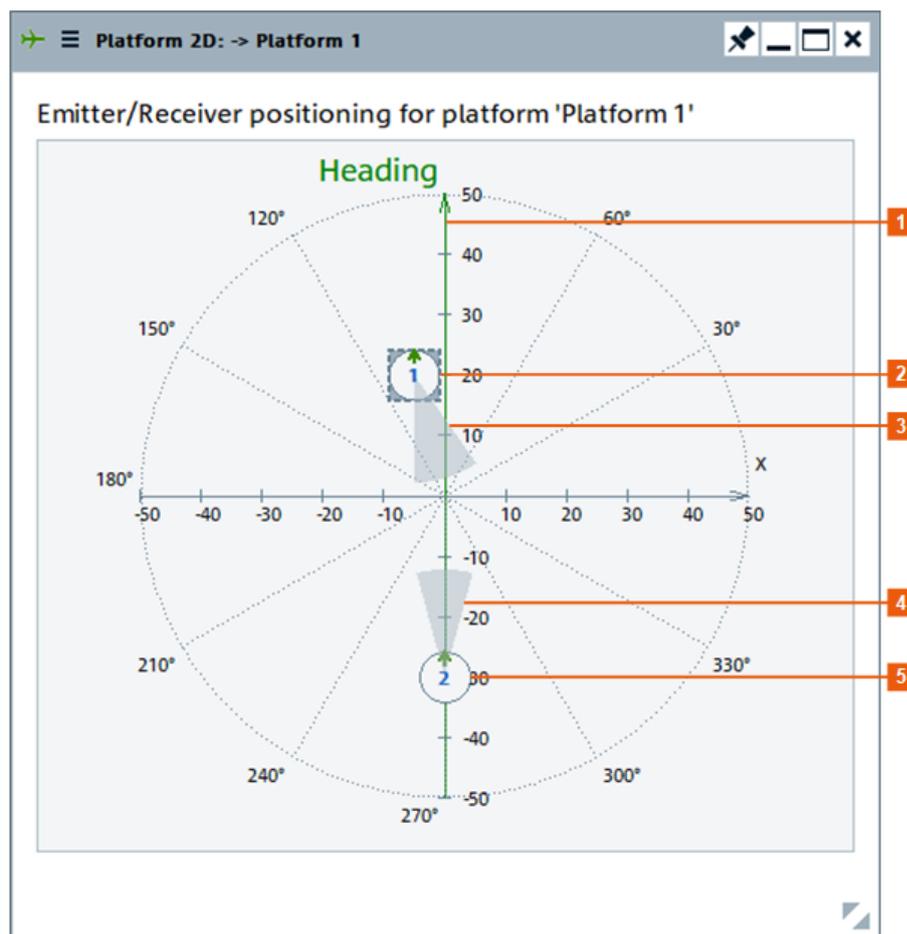
Figure 12-3: Platform 2D view example

1 = Heading of platform
2 = Emitter-1
3 = Blank range of emitter-1
4 = Blank range of emitter-2
5 = Emitter-2

This example shows you:
- A platform that represents two radars aboard a ship.
- "Heading" indicates the platform's direction of travel.
- Emitter-1 is mounted off-axis and has a blanking range towards the rear of the platform.
- Emitter-2 is mounted on-axis and has a blanking range towards the front of the platform.
- The icon for emitter-1 has a frame. This frame indicates that this emitter is selected in the emitter list.

Icon for 3D preview

Icons that access the 2D or 3D preview of the platform.

Figure 12-4 shows an example with the same two emitters as Figure 12-3.

The "2D" view shows the precise horizontal placement of the emitters. The "3D" view shows the placement of the emitters, both horizontally and vertically, relative to the platform's origin.
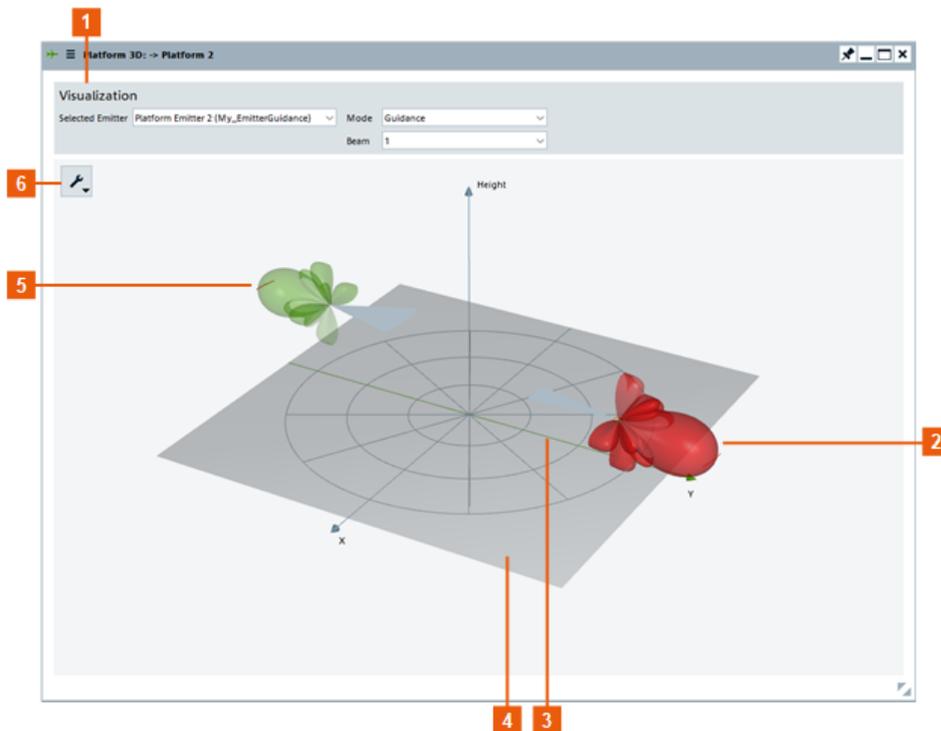
Figure 12-4: Platform 3D view example

1 = Visualization settings
2 = Antenna pattern and blank range of emitter 1
3 = Heading of platform
4 = Ground plain
5 = Antenna pattern and blank range of emitter 2
6 = Tools

Settings and tools:

Selected Emitter ← Icon for 3D preview

Sets the emitter for which the "Mode" and "Beam" settings apply.

The drop-down list contains all emitters of the selected platform.

The selected emitter is colored red in the 3D preview.

Mode ← Selected Emitter ← Icon for 3D preview

Defines which mode of the selected emitter is visualized in the 3D preview.

The drop-down list contains all modes of the selected emitter.

Beam ← Selected Emitter ← Icon for 3D preview

Defines which beam of the selected emitter is visualized in the 3D preview.

The drop-down list contains all beams of the selected emitter.

Opacity ← Icon for 3D preview

Use this function to make the antenna patterns more or less transparent.

12.2.3 Properties of selected emitter

This tab contains the parameters of the emitter selected in the "Emitter" list of the "Platform" dialog.

Figure 12-5: Example of emitter properties configuration

Settings

Emitter

Selects the emitter assigned to the item selected in the "Emitter" list of the "Platform" dialog.

Remote command:
PLATform:EMITter on page 781

Elevation

Sets the elevation of the emitter item's pointing direction, relative to the platform's azimuth plane.

Values can be:

- Positive - pointing above the azimuth plane (e.g. air traffic control radar)
- Negative - pointing below the azimuth plane (e.g. aircraft landing radar)

This setting "Attitude" on page 342 overrides the "Elevation" parameter.

Remote command:
PLATform:EMITter:ELEVation on page 784

Azimuth

Sets the azimuth of the emitter item's pointing direction, relative to the platform's heading.

For static emitters, this parameter sets the azimuth of the beam.

For circular emitters, this parameter sets the beam direction at the start of the scan.

For sector scans, this parameter sets center of the scanned sector.

The setting "Attitude" on page 342 overrides the "Azimuth" parameter.

Remote command:
PLATform:EMITter:AZIMuth on page 782

Roll

Sets the roll of the emitter item's pointing direction, relative to the platform's up direction.

Remote command:
PLATform:EMITter:ROLL on page 785

Attitude

Selecting the "Auto away from origin" checkbox, sets the "Azimuth" and "Elevation" automatically.

- "Elevation" - automatically set to 0°
- "Azimuth" - automatically configured so that the beam axis is radial to the platform origin.

Remote command:
PLATform:EMITter:DIRection:AWAY on page 784

Y (Heading)

The position of an emitter on a platform is defined by three parameters:
- "Y (Heading)"
- "X" (see "X" on page 343)
- "Height" (see "Height" on page 343)

The "Y (Heading)" parameter defines the position of the emitter (in the direction of movement) relative to the platform's origin.

Note: The XY position can also be defined via the "Radius" from the origin and the "Angle" relative to the X-axis.

Remote command:

PLATform:EMITter:Y on page 786

X

The position of an emitter on a platform is defined by three parameters:
- "Y (Heading)" (see "Y (Heading)" on page 343)
- "X"
- "Height" (see "Height" on page 343)

The "X" parameter defines the position of the emitter (at right-angles to the direction of movement) relative to the platform's origin.

Note: The XY position can also be defined via the "Radius" from the origin and the "Angle" relative to the X-axis.

Remote command:

PLATform:EMITter:X on page 785

Height

The position of an emitter on a platform is defined by three parameters:
- "Y (Heading)" (see "Y (Heading)" on page 343)
- "X" (see "X" on page 343)
- "Height"

The "Height" parameter defines the position of the emitter relative to the XY plane.

Note: You can use the "Height" parameter with the *XY* positioning method or the *angle and radius* method.

Remote command:

PLATform:EMITter:HEIGht on page 784

Radius

The position of an emitter on a platform is defined by three parameters:
- "Angle" (see "Angle" on page 344)
- "Radius"
- "Height" (see "Height" on page 343)

The "Radius" parameter defines the position of the emitter on the XY plane, relative to the platform's origin.

Note: The XY position can also be defined via the "X" and "Y (Heading)" parameters.

Remote command:

PLATform:EMITter:RADius on page 785

Angle

The position of an emitter on a platform is defined by three parameters:

- "Angle"
- "Radius" (see "Radius" on page 343)
- "Height" (see "Height" on page 343)

The "Angle" parameter defines the position of the emitter on the XY plane, relative to the platform's X-axis.

Note: The XY position can also be defined via the "X" and "Y (Heading)" parameters.

Remote command:
PLATform:EMITter:ANGLe on page 782

12.2.4 Blank ranges of selected emitter

This tab displays the sectors in which the selected emitter mutes its signal output.

Figure 12-6: Example of blank ranges configuration

1 = Icons that access standard functions
2 = Blank range stop angle
3 = Blank range start angle

On real platforms, blank ranges represent:

- Sectors in which the radar does not transmit.
  For example, to avoid saturating other antennas with the high-power signal from the emitter or to avoid wave-scatter from metal structures such as masts.
- Sectors in which the propagation of the transmitted radar signal is blocked.
  For example, due to obstructions such as masts and superstructure.

Settings

No

Number of the individual blank range entry.

Remote command:
`PLATform:EMITter:BLANkranges:SELect` on page 592

Start
Set the angle for the beginning of the selected blank range.

Remote command:
`PLATform:EMITter:BLANkranges:STARt` on page 783

Stop
Set the angle for the end of the selected blank range.

Remote command:
`PLATform:EMITter:BLANkranges:STOP` on page 783

Set of ranges
You can enter a set of blank ranges for the selected emitter, using a single SCPI command.

The following SCPI example, adds a set of three blank ranges to emitter number 3.

The comma-separated value list (90,120,160,200,350,10) contains three pairs of start/stop values.

● start=90, stop=120
● start=160, stop=200
● start=350, stop=10

This approach is more efficient than using several blank range start/stop commands.

Remote command:
`PLATform:EMITter:BLANkranges` on page 782

12.3 Working with platforms

The platforms you create appear in the repository, as shown in Figure 12-7.
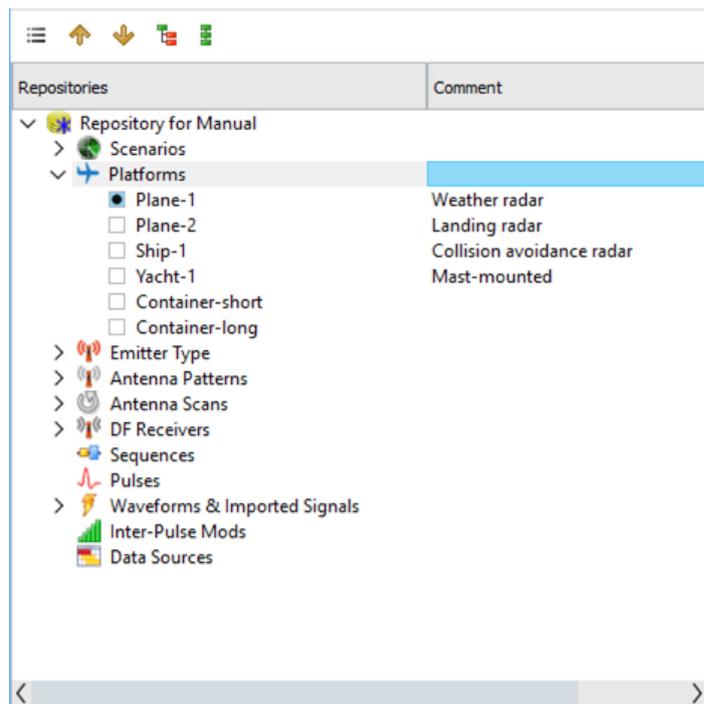
Figure 12-7: Examples of platforms in a repository

To create a platform

1. To create a platform, do one of the following:

   a) In the repository tree, select "Platform Types > New".
   b) Select an existing platform > context menu > "Clone".

   A new platform is added to the repository. The "Platform" dialog opens.

2. Configure the platform as described in Chapter 12.2, "Platform settings", on page 336.

To edit a platform

1. In the repository tree, select a platform.

   The "Platform" dialog opens.

2. Configure the platform as described in Chapter 12.2, "Platform settings", on page 336.

To delete a platform

1. In the repository tree, select a platform.

2. In the context menu, select "Delete".

   A dialog opens to confirm the action.

3. Click "Yes" to delete the platform.

   The selected platform is removed from the repository.

To add a platform to a scenario

1. In the repository tree, select a scenario.

2. In the scenario dialog, click "Scenario Creation > Map".

3. To add a platform to a scenario, do one of the following:

   a) Drag&drop the platform onto the 2D map.
   b) Drag&drop the platform into the "TX Items" list.
   c) In the 2D map: In the context menu, select "Add Platform".

To edit a platform in a scenario

1. To open the platform dialog to edit the platform, do one of the following:

   a) Double-click the platform in the "TX Items" list.
   b) Double-click the platform in the 2D map.

   The "TX Items Properties" dialog opens.

2. Edit the platform parameters.

To remove a platform from a scenario

1. In the repository tree, select a scenario.

2. In the scenario dialog, click "Scenario Creation > Map".

3. To remove a platform from a scenario, do one of the following:

   a) In the "TX Items" list: Select the platform > context menu > "Remove".
   b) In the 2D map: Select the platform > context menu > "Remove".

   The selected entry is removed from the scenario.

13 Emulating receivers

The available receiver settings depend on the complexity of the simulated system:

- Receivers in the "Single Emitter" and "Emitters (Collection)" scenario types.
  These receivers have *an omnidirectional antenna with no antenna scan*.
  These scenarios only simulate the effect of emitter antennas and scans. Attenuation due to distance and free space propagation is not considered.

- Receivers in a 2D multi-emitters receiver scenario have *one receive antenna*.
  They are described by an antenna beam pattern, an antenna scan, and an attitude information. The distance between the emitters and the receiver is configurable.
  This receiver complexity is used in the "Localized Emitters" scenario.

- Direction finding (DF) receivers can have *up to 20 antenna elements*.
  Each antenna element is described by the combination of an antenna pattern, antenna position and attitude. The receiver itself is also described with its position and attitude information.
  This receiver complexity is used in the "Direction Finding" scenario.

Table 13-1: Overview of the receiver types and provided settings

Scenario type	"Single Emitter" "Emitters (Collection)"	"Localized Emitters"	"Direction Finding"
Separate repository elements	-	-	Yes
Related settings	-	Chapter 13.1, "Receiver properties in localized emitters scenario", on page 350	Chapter 13.2, "Receiver settings", on page 352
Scenario	Emitter-receiver	2D multi-emitter-receiver	2D multi-antenna receiver
2D map (gaming area)	-	Yes	
Emitter-receiver distance	Fixed	Configurable (Emitters placed on the 2D map)	
Receiver attitude	-	Configurable (Pitch, yaw, roll, and height)	
Number of antenna (elements)	1	1	≤ 20
Antenna pattern	Isotropic	Configurable	
Antenna scan	None (steady)	Configurable	-
Antenna attitude	-	-	Configurable (Pitch, yaw, and roll per antenna element)

Received signal amplitude

One of the typical receiver characteristics is the minimum detective signal (MDS). The MDS is a measure of the receiver sensitivity and describes the minimum received signal amplitude S_{min} that the receiver is able to detect. A typical receiver achieves an S_{min} within the range of -80 dBm and -110 dBm.

In any type of emitter-receiver scenarios, the R&S Pulse Sequencer calculates the received signal and displays the time variation of the received normalized power.

The principle is illustrated on Figure 13-1.
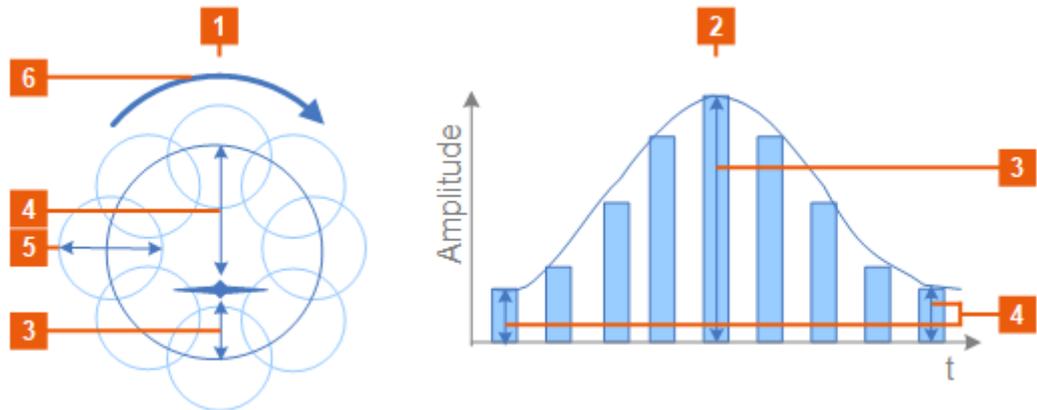
Figure 13-1: Simplified representation of a conical scan and the received signal

1   = Conical scan (boresight perspective)
2   = Amplitude variation of the received signal
3, 4 = Target-to-beam distance determines the amplitude of the received signal
5   = Antenna beam represented as its HPBW
6   = Direction of the rotating scan

The signal received by a static receiver is a sinusoidal waveform. The amplitude of the wave is proportional to the distance between the target and the beam axis. Figure 13-2 shows how this simple case of an emitter-receiver scenario is visualized in the software.
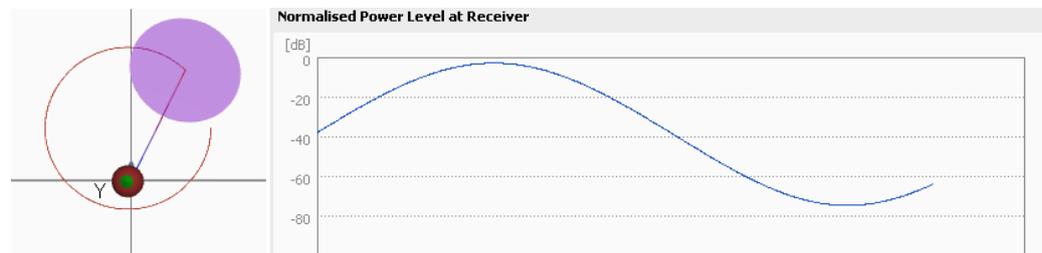
Figure 13-2: Normalized power level at receiver (simple emitter-receiver scenario)

For example, see "To visualize the signal received by a static receiver" on page 321.

For step-by-step instructions, see:

● "How to create a direction finding scenario" on page 417
● Chapter 17.6, "How to play the waveforms of 2D scenarios", on page 492

For description of the provided settings, see:

● Chapter 13.1, "Receiver properties in localized emitters scenario", on page 350
● Chapter 13.2, "Receiver settings", on page 352
● Chapter 13.3, "DF system configuration settings", on page 356

13.1 Receiver properties in localized emitters scenario

Access:

1. In the 2D map of a "Localized Emitters" scenario, select "Receiver Properties > Properties".
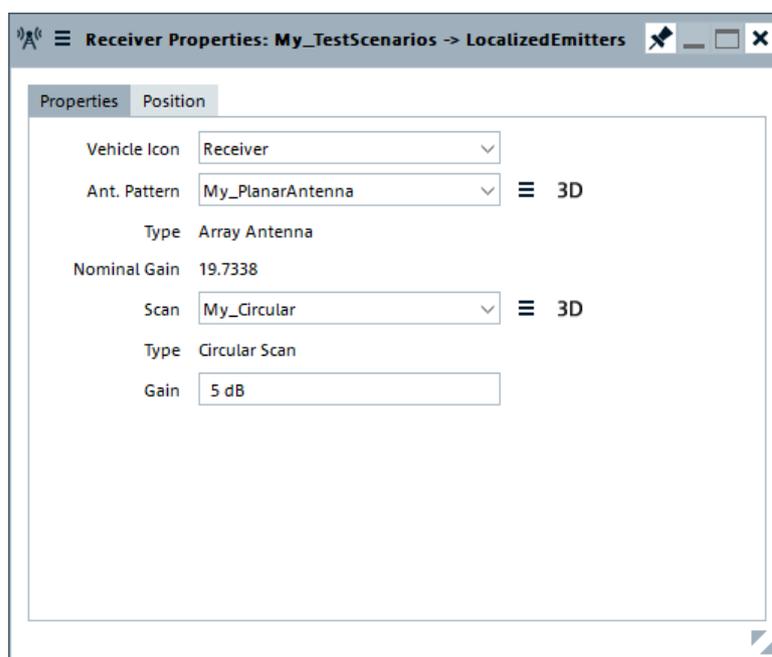
2. Select "Receiver Properties > Position".

Settings:

Vehicle Icon

Selects the icon for the receiver on the 2D map.

The icons are merely indication; the function and the properties of the receiver do not depend on the icon.

Remote command:

SCENario:LOCalized:RECeiver:MOVement:VEHicle on page 777
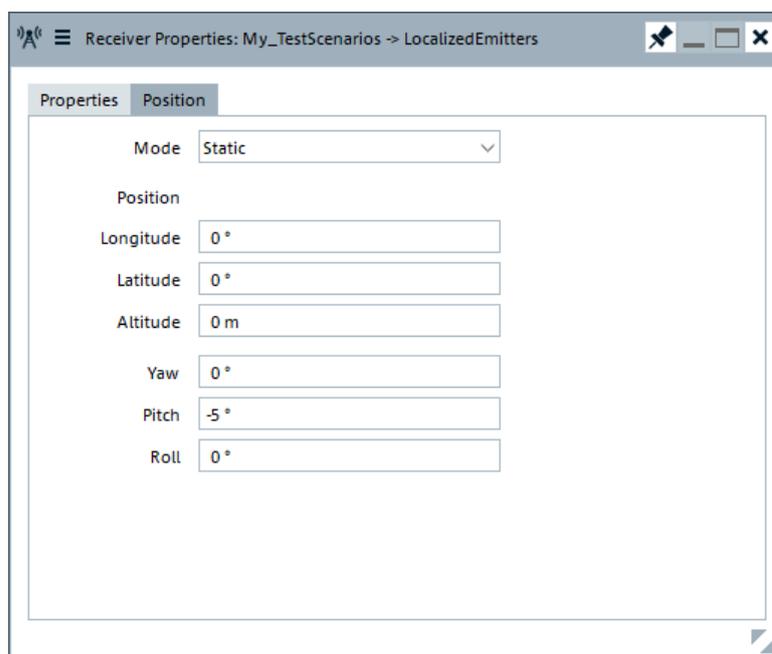
Antenna Pattern/Scan, Type, ▤, 3D

Selects an antenna pattern and antenna scan.

3D opens a plot that visualizes the antenna pattern and scan.

Remote command:

SCENario:LOCalized:RECeiver:ANTenna on page 751

SCENario:LOCalized:RECeiver:SCAN on page 752

Nominal Gain

Displays the nominal gain of the antenna pattern.

Gain

Sets the antenna Gain.

Remote command:

SCENario:LOCalized:RECeiver:GAIN on page 752

Position > Latitude, Longitude, Altitude, Yaw, Pitch, Roll

The receiver's position is defined as latitude/longitude coordinates and attitude parameters. The available settings depend on the selected "Mode".

The position receiver settings in "Localized Emitters" and "Direction Finding" scenarios are identical.

For description, see Chapter 13.4, "Receiver position settings in localized emitters and direction finding scenarios", on page 358.

Remote command:

SCENario:DF:RECeiver:LATitude on page 751

SCENario:DF:RECeiver:LONGitude on page 751

SCENario:DF:RECeiver:HEIGht on page 751

SCENario:LOCalized:RECeiver:LATitude on page 751

SCENario:LOCalized:RECeiver:LONGitude on page 751

SCENario:LOCalized:RECeiver:HEIGht on page 751

13.2 Receiver settings

Access:

► Select "Repository Tree > DF Receiver Types > New".
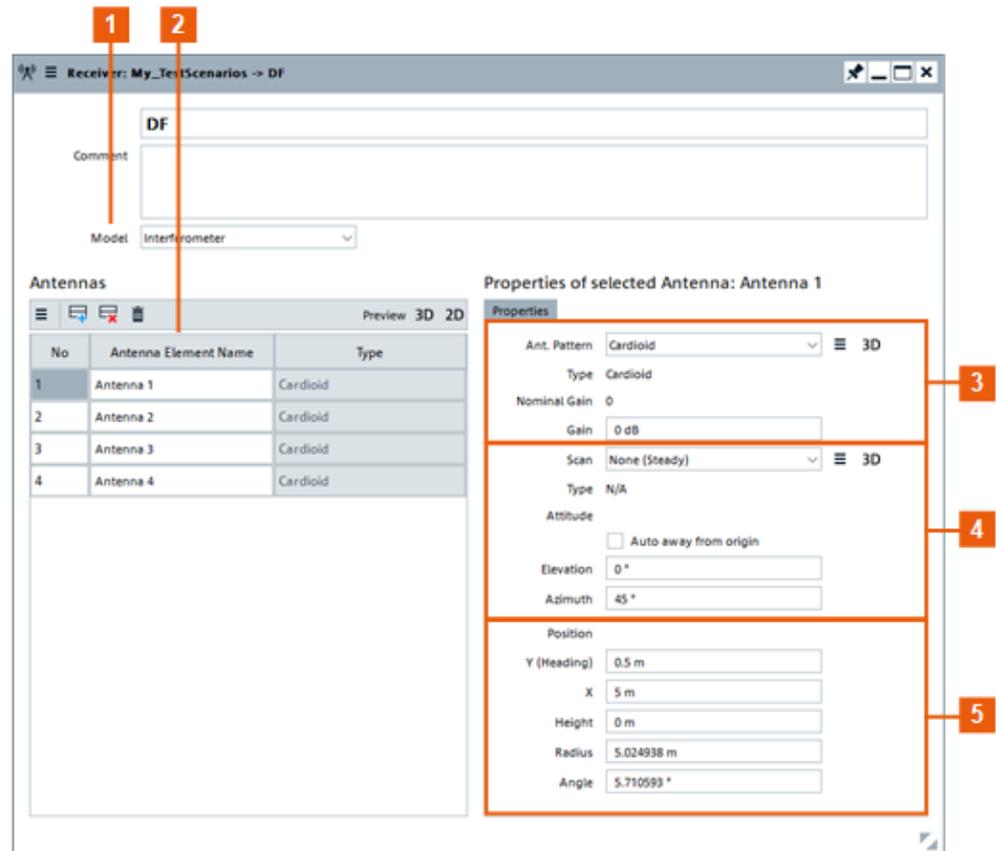
Figure 13-3: Receiver: understanding the displayed information

1 = Receiver model
2 = Up to 20 individual antenna elements
3 = Antenna pattern and gain per antenna element
4 = Altitude and orientation per antenna element
5 = Location and position information per antenna element

Direction finding (DF) receivers can have up to 20 antenna elements, each described with an antenna pattern and position.
See also "How to create a direction finding scenario" on page 417.

Settings:

Receiver Name

Enter the receiver's name.

Remote command:

RECeiver:NAME on page 589

RECeiver:CREate on page 588

RECeiver:SELect on page 588

RECeiver:CATalog? on page 587

RECeiver:REMove on page 590

Comment

Enter a short description.

Remote command:

RECeiver:COMMent on page 589

Model

Defines the model of the DF receiver.

"Interferometer"

Interferometer direction finding receivers work phase coherent and evaluate the **phase difference** between the single antenna ports. They are usually **small**, with a diameter in the range of a wavelength. In this mode, the R&S Pulse Sequencer calculates the relative phase difference between the incoming signal and the antenna positions. The differences in the time of arrival (TOA) of the incoming signals are assumed to be equal for all antenna to prevent small rounding errors in sample granularity.

"TDOA" (Time difference of arrival)

TDOA direction finders use the **absolute time of arrival** of a signal to determine the direction. They do not evaluate phase information, because they are usually **big-sized**. In TDOA receiver, the antenna ports are placed far from one another so that the time delay is large enough and suitable for analysis.
Examples of TDOA receivers are the antennas mounted at the two wings of an airplane or at the outer borders of a ground station. Because of the distance between the antennas, there is a difference in the time the incoming signal arrives at the antennas. In this mode, the R&S Pulse Sequencer calculates the absolute TOA of the incoming signal for each antenna. The phase difference between the single antenna ports is calculated from their absolute distance to the emitter.

"Combined (Interferometer/TDOA)"

The combined model calculates the relative phases between the antenna ports and calculates the individual TOAs for each antenna port.

Remote command:

RECeiver:MODel on page 717

Antenna Configuration

DF receiver can have up to 10 antenna elements.

"Positioning Preview 2D"

Displays the distribution of the antenna elements in a 2-dimensional chart (XY plane).
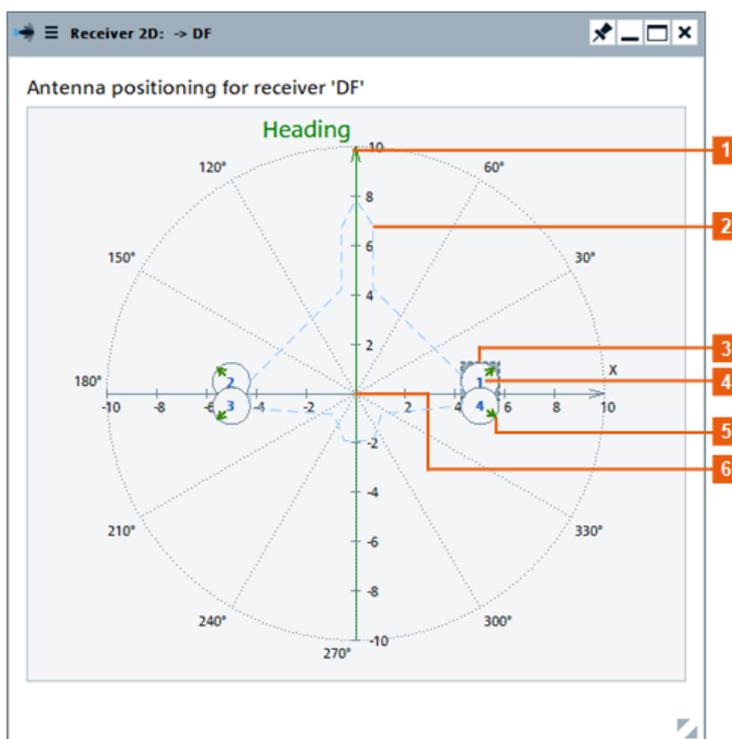
Figure 13-4: Antenna Positioning: understanding the displayed information

1 = Heading (y axis); indicated with a colored line
2 = Airplane graphic is not displayed in the software (shown here for better understanding)
3 = Selected antenna element
4 = Antenna element consequent number, as listed in the "List of antenna elements" table
5 = Antenna element orientation
6 = Receiver origin

See also

"Positioning Preview 3D"

> Displays the distribution of the antenna elements in a 3-dimensional chart (XY plane).
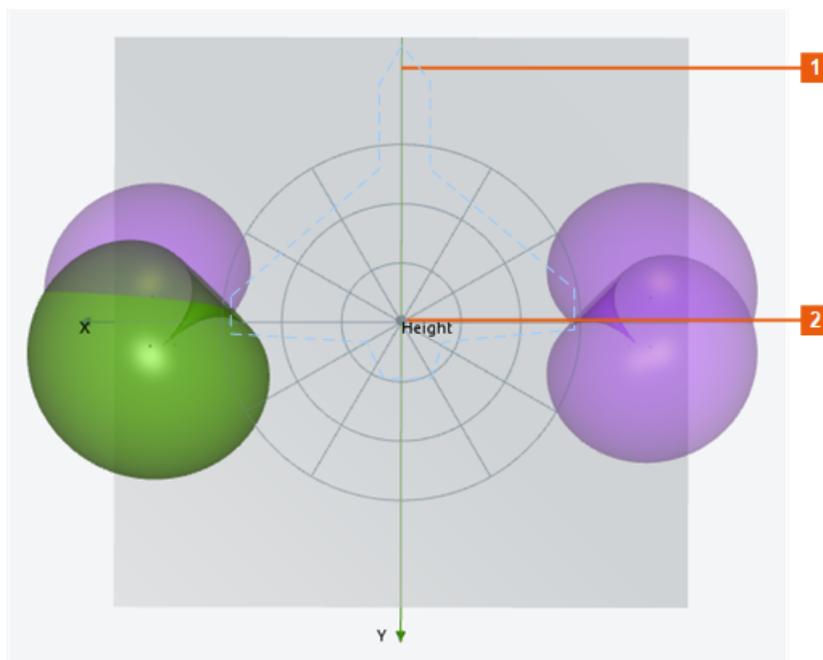
1 = Heading (y axis); indicated with a colored line
2 = Receiver origin

"List of antenna elements"

> Use the standard functions in the context menu and the icons to add, reorder, or remove antenna elements.
> New elements are named automatically but you can change the alias name.

Remote command:

RECeiver:ANTenna:ADD on page 591
RECeiver:ANTenna:SELect on page 593
RECeiver:ANTenna:ALIas on page 717
RECeiver:ANTenna:DELete on page 594
RECeiver:ANTenna:CLEar on page 595

Properties of selected Antenna: Antenna x
Describes the selected antenna element.

Open the "2D/3D Positioning Preview" view to display the configuration, see Figure 13-4.

Properties ← Properties of selected Antenna: Antenna x
Comprises the antenna and receiver orientation-related settings:

"Antenna Pattern, Antenna Scan, Type"

Selects an existing antenna pattern or accesses a dialog for creating a new one.

The antenna pattern, scan and type are displayed.

See also "To create an antenna pattern" on page 294.

Remote command:

`RECeiver:ANTenna:PATTern` on page 718

`RECeiver:ANTenna:SCAN` on page 719

"Nominal Gain"   Displays the nominal gain of the antenna pattern.

"Gain"   Sets the antenna Gain.

Remote command:

`RECeiver:ANTenna:GAIN` on page 719

"Pointing Direction"

Turns the antenna beam axis in the elevation and the azimuth.

"Auto away form origin" sets the azimuth automatically, so that the beam axis is radial to the receiver origin.

Remote command:

`RECeiver:ANTenna:DIRection:AWAY` on page 719

`RECeiver:ANTenna:DIRection:AZIMuth` on page 719

`RECeiver:ANTenna:DIRection:ELEVation` on page 720

Position ← Properties of selected Antenna: Antenna x

Antenna elements are spaced relative to the receiver origin.

Their 3D positions are defined as:

- "X" and "Y (heading)" values relative to the receiver origin.
- "Angle" (azimuth plane) and "Radius" (distance to the receiver origin on the XY plane), where "Angle = 0°" is on the X-axis.
- "Height" offset on the Z-axis relative to the receiver origin.

Example:

"X = 0 m" and "Y = -0.14 m" corresponds to "Radius = 0.14 m" and "Angle = 270 °".

Remote command:

`RECeiver:ANTenna:POSition:X` on page 718

`RECeiver:ANTenna:POSition:Y` on page 718

`RECeiver:ANTenna:POSition:RADius` on page 718

`RECeiver:ANTenna:POSition:ANGLe` on page 718

`RECeiver:ANTenna:POSition:HEIGht` on page 718

13.3 DF system configuration settings

Access:

1. Open a "Direction Finding" scenario.

2. In the block diagram, select "Map > 2D".

3. On the 2D map, select "Receiver > Properties".

4. Select "2D/3D" to display the "Antenna Positioning".
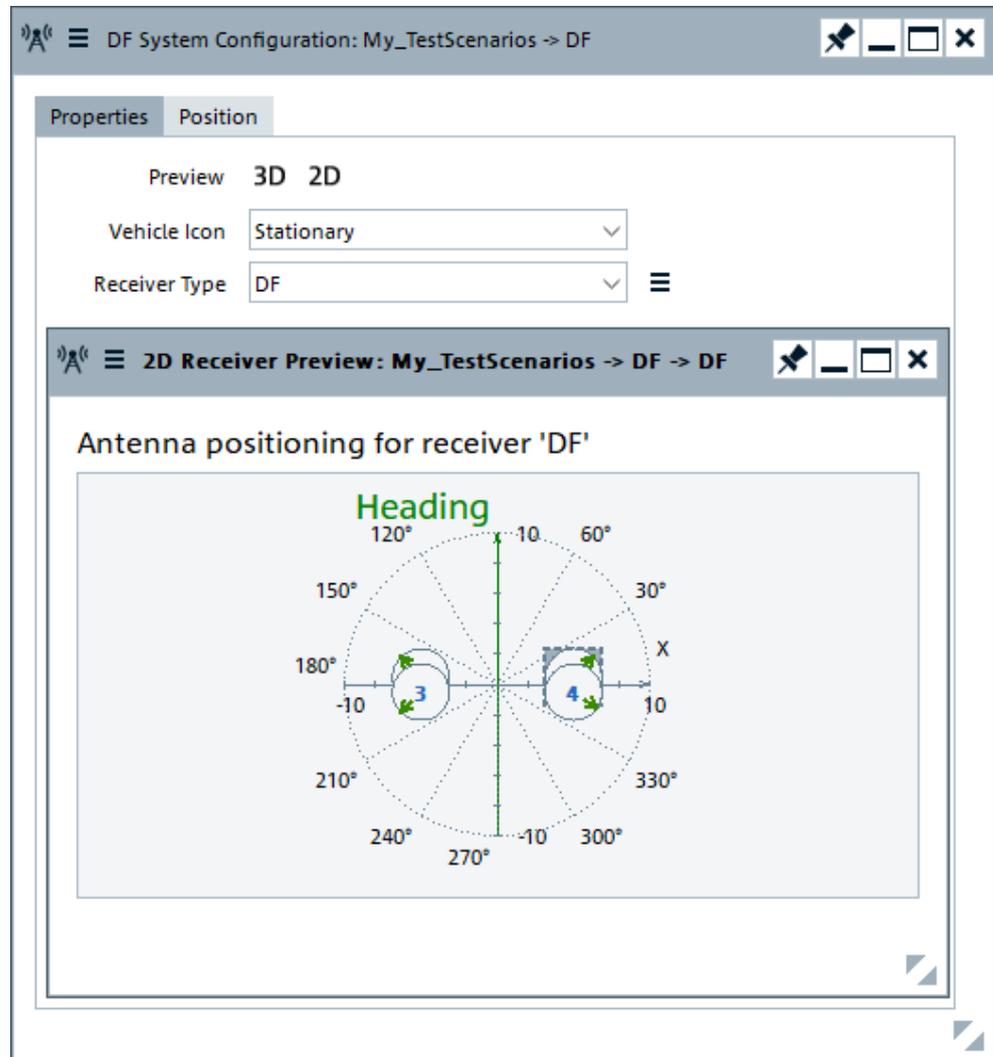
See also "How to create a direction finding scenario" on page 417.

Settings:

Properties

Each receiver is described with the following parameters:

Vehicle Icon ← Properties
Selects the icon for the receiver on the 2D map.

The icons are merely indication; the function and the properties of the receiver do not depend on the icon.

Remote command:
SCENario:DF:RECeiver:MOVement:VEHicle on page 777

Receiver ← Properties
Selects an existing receiver or accesses a dialog for creating a new one.

Remote command:
SCENario:DF:RECeiver on page 755

Position > Latitude, Longitude, Height
The receiver's position is defined as latitude/longitude coordinates and attitude parameters. The available settings depend on the selected "Mode".

The position receiver settings in "Localized Emitters" and "Direction Finding" scenarios are identical.

For description, see Chapter 13.4, "Receiver position settings in localized emitters and direction finding scenarios", on page 358.

Antenna Positioning
Displays the distribution of the antenna elements on a 2D map (XY plane).

The graph resembles the one in the "Receiver" dialog, see Figure 13-4.

Use the split window icon ▣ to detach the diagram and open it in a separate window.

13.4 Receiver position settings in localized emitters and direction finding scenarios

As with the emitters/interferers, you can also define receivers that move along a trajectory with a defined shape. The receiver configuration is similar to the configuration of the moving emitters/interferers.

For introduction to the topic and details on the waypoint files, see:

- Chapter 16.5, "Moving emitters/interferers and platforms", on page 450
- Chapter A.3, "Movement files", on page 850.

Access:

1. In the 2D map of a "Localized Emitters" or "Direction Finding" scenario, select "Receiver Properties > Properties".

2. Select "Position".

   The available settings depend on the selected "Mode".

Properties	Position	States

Movement Type	Moving
Trajectory	Line
Start Position	
East	2902.12 m
North	452.549 m
Height	0 m
Distance	2937.193 m
Azimuth	81.14 °
Elevation	0 °
Speed	100 m/s
Acceleration	0 m/s²
End Position	
East	-2813.61 m
North	950.718 m
Height	0 m
Mode	One Way

For example of possible further configurations, see Chapter 16.5, "Moving emitters/ interferers and platforms", on page 450.

Settings:

Mode
Sets if the receiver is static or moving.

"Static"   The receiver is static; its coordinates are fixed during the whole sce-
                    nario.
                    See Start Position > Latitude, Longitude, Height and Attitude > Pitch,
                    Yaw, Roll.

"Moving"   The receiver is moving from defined start position and follows the
                    selected trajectory.
                    See Trajectory.

Remote command:
SCENario:LOCalized:LOCation:REC:PMODe on page 771

Trajectory
Defines the trajectory shape.

"Line"   The receiver is moving on a straight line, starting from its current
                    position (Start Position > Latitude, Longitude, Height) and ending at
                    the selected "End Position".

"Arc"   The receiver moves on a part of a circle, where the center of the cir-
                    cle is defined with the "Center Position".

"Waypoint"   Receiver with varying coordinates, defined in waypoints files.
                    You find a subset of example files in installation package but you can
                    also create and load your own files.

"Trace"   The receiver follows a user-defined route. You can define the route in
                    the map view using the "Trace" function. R&S Pulse Sequencer gen-
                    erates waypoints from the trace, automatically.

Remote command:
SCENario:LOCalized:RECeiver:MOVement:TYPE on page 772
SCENario:DF:RECeiver:MOVement:TYPE on page 772

Start Position > Latitude, Longitude, Height
Sets the receiver's position.

Remote command:
SCENario:DF:RECeiver:LATitude on page 751
SCENario:DF:RECeiver:LONGitude on page 751
SCENario:LOCalized:RECeiver:HEIGht on page 751

Attitude > Pitch, Yaw, Roll

For static receivers, defines the orientation of the antenna as "Pitch", "Yaw", "Roll".

See Table 10-1.

Remote command:

SCENario:LOCalized:RECeiver:DIRection:YAW on page 747

SCENario:LOCalized:RECeiver:DIRection:PITCh on page 747

SCENario:LOCalized:RECeiver:DIRection:ROLL on page 747

SCENario:DF:RECeiver:DIRection:YAW on page 747

SCENario:DF:RECeiver:DIRection:PITCh on page 747

SCENario:DF:RECeiver:DIRection:ROLL on page 747

Line trajectory definition

A line is defined with:

Speed ← Line trajectory definition

Sets the speed of the moving receiver.

Remote command:

SCENario:LOCalized:RECeiver:MOVement:SPEed on page 774

SCENario:DF:RECeiver:MOVement:SPEed on page 774

Acceleration ← Line trajectory definition

Sets the acceleration of the moving receiver.

Remote command:

SCENario:LOCalized:RECeiver:MOVement:ACCeleration on page 774

SCENario:DF:RECeiver:MOVement:ACCeleration on page 774

End Position ← Line trajectory definition

Sets the "East/North" coordinates and the "Height" of the receiver at the end of the movement.

Remote command:

SCENario:LOCalized:RECeiver:MOVement:EAST on page 775

SCENario:LOCalized:RECeiver:MOVement:NORTh on page 775

SCENario:LOCalized:RECeiver:MOVement:HEIGht on page 775

SCENario:DF:RECeiver:MOVement:EAST on page 775

SCENario:DF:RECeiver:MOVement:NORTh on page 775

SCENario:DF:RECeiver:MOVement:HEIGht on page 775

Mode ← Line trajectory definition

Defines the behavior of the moving object when the end of the trajectory is reached.

"Cyclic"   The trajectory file is repeated cyclically. Once the "End Position" is reached, file movement starts again from the beginning.

Using this mode is recommended in the following cases:
- A circle trajectory
- A trajectory in which the start and the end positions are close to each other.

"Round Trip"   The emitter moves back and forth on the defined trajectory.

"One Way"   The movement is executed once.
                  When the "End Position" is reached, this position is assumed to be a
                  static one.

Remote command:
SCENario:DF:RECeiver:MOVement:RMODe on page 778
SCENario:LOCalized:RECeiver:MOVement:RMODe on page 779

Arc trajectory definition
An arc is defined with:

Speed ← Arc trajectory definition
Sets the speed of the moving receiver.

Remote command:
SCENario:LOCalized:RECeiver:MOVement:SPEed on page 774
SCENario:DF:RECeiver:MOVement:SPEed on page 774

Angle ← Arc trajectory definition
Sets the arc angle and thus defines the arc length.

Remote command:
SCENario:LOCalized:RECeiver:MOVement:ANGLe on page 775
SCENario:DF:RECeiver:MOVement:ANGLe on page 775

Center Position ← Arc trajectory definition
Sets the "East/North" coordinates of the center of the circle on that the receiver is moving.

Remote command:
SCENario:LOCalized:RECeiver:MOVement:EAST on page 775
SCENario:LOCalized:RECeiver:MOVement:NORTh on page 775
SCENario:DF:RECeiver:MOVement:EAST on page 775
SCENario:DF:RECeiver:MOVement:NORTh on page 775

Mode ← Arc trajectory definition
Defines the behavior of the moving object when the end of the trajectory is reached.

"Cyclic"   The trajectory file is repeated cyclically. Once the "End Position" is
                  reached, file movement starts again from the beginning.

                  Using this mode is recommended in the following cases:
                  ● A circle trajectory
                  ● A trajectory in which the start and the end positions are close to
                    each other.

"Round Trip"   The emitter moves back and forth on the defined trajectory.

"One Way"   The movement is executed once.
                  When the "End Position" is reached, this position is assumed to be a
                  static one.

Remote command:
SCENario:DF:RECeiver:MOVement:RMODe on page 778
SCENario:LOCalized:RECeiver:MOVement:RMODe on page 779

Waypoints trajectory definition

A waypoint file is description of a movement with possibly attitude coordinates that can have different forms, like, for example, a sequence of positions or vector arc movement. A waypoint file must have the extension `*.txt`, `*.kml` or `*.xtd`. For description, see Chapter A.3, "Movement files", on page 850.

You find a subset of example files in installation package but you can also create and load custom-specific files. Moreover, waypoint smoothing can be activated if vehicle description files are used.
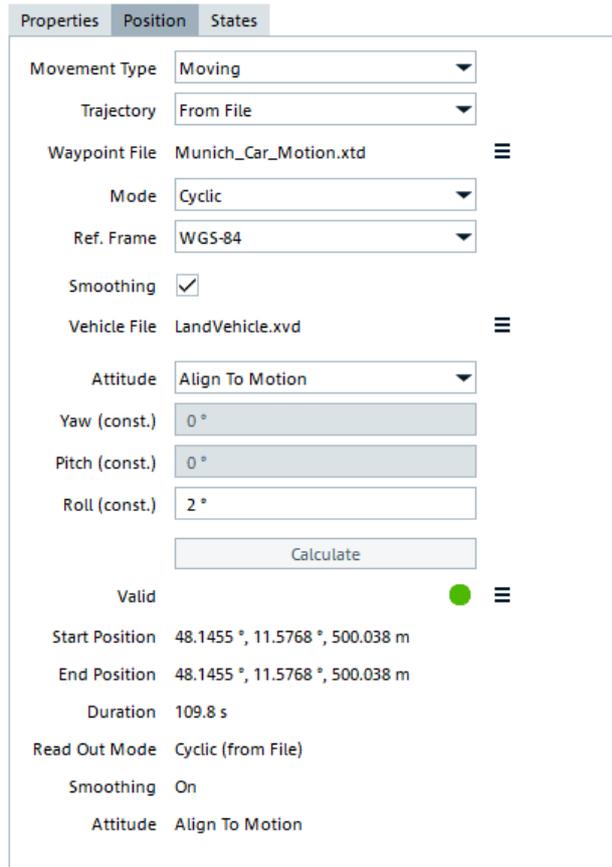
Properties	Position	States
Movement Type	Moving	
Trajectory	From File	
Waypoint File	Munich_Car_Motion.xtd	≡
Mode	Cyclic	
Ref. Frame	WGS-84	
Smoothing	✓	
Vehicle File	LandVehicle.xvd	≡
Attitude	Align To Motion	
Yaw (const.)	0 °	
Pitch (const.)	0 °	
Roll (const.)	2 °	
	Calculate	
Valid	🟢	≡
Start Position	48.1455 °, 11.5768 °, 500.038 m	
End Position	48.1455 °, 11.5768 °, 500.038 m	
Duration	109.8 s	
Read Out Mode	Cyclic (from File)	
Smoothing	On	
Attitude	Align To Motion	

If waypoint files are used, the receiver position and trajectory are retrieved from the file content; the traditional position coordinates parameters are disabled.

Waypoint File ← Waypoints trajectory definition
Indicates the name of the currently used waypoint file.

≡ Load, Delete ← Waypoints trajectory definition
Opens the Windows explorer so that you can navigate to and load waypoint file.

Discards the selected waypoint file.

Remote command:
SCENario:DF:RECeiver:MOVement:WAYPoint on page 776
SCENario:DF:RECeiver:MOVement:WAYPoint:CLEar on page 776
SCENario:LOCalized:RECeiver:MOVement:WAYPoint on page 776
SCENario:LOCalized:RECeiver:MOVement:WAYPoint:CLEar on page 776

Read Out Mode ← Waypoints trajectory definition

Defines the way that the waypoint file is processed.

"Cyclic"　　　　The waypoint file is processed cyclically. Once the last waypoint is reached, file processing starts again from the beginning.

　　　　　　　Using this mode is recommended if the waypoint file describes one of the following:
- A circle trajectory
- A trajectory in which the start and the end positions are close to each other.

"Round Trip"　　By reaching the end of the file, the file is processed backwards.

"One Way"　　　The file is processed once.
　　　　　　　By reaching the end of the file, the last described position is assumed to be a static one.

Remote command:

SCENario:DF:RECeiver:MOVement:RMODe on page 778

SCENario:LOCalized:RECeiver:MOVement:RMODe on page 779

Reference Frame ← Waypoints trajectory definition

Select the reference frame used to define the emitters coordinates.

The transformation between the reference frames is performed automatically.

The following applies:

- $X_{WGS84} = (1 - 0.008*10^{-6})*X_{PZ\ 90} - 0.2041*10^{-7}*Y_{PZ\ 90} + 0.1716*10^{-7}*Z_{PZ\ 90} - 0.013$
- $Y_{WGS84} = (1 - 0.008*10^{-6})*Y_{PZ\ 90} - 0.2041*10^{-7}*X_{PZ\ 90} + 0.1115*10^{-7}*Z_{PZ\ 90} + 0.106$
- $Z_{WGS84} = (1 - 0.008*10^{-6})*Z_{PZ\ 90} - 0.1716*10^{-7}*X_{PZ\ 90} - 0.1115*10^{-7}*Y_{PZ\ 90} + 0.022$

Both reference frames are ECEF frames with a set of associated parameters.

"WGS-84"　　　The world geodetic system WGS-84 is the reference frame used by GPS.

"PZ 90.11 (GLONASS)"

　　　　　　　Parametry Zemli PZ (parameters of the Earth) is a reference frame, used, for example, by GLONASS.

Remote command:

SCENario:DF:RECeiver:MOVement:RFRame on page 778

SCENario:LOCalized:RECeiver:MOVement:RFRame on page 778

Smoothing > State ← Waypoints trajectory definition

The discrete positions (waypoints) defined in the waypoints file can cause abrupt changes in the movement direction.

The R&S Pulse Sequencer provides an internal interpolating algorithm that smooths the movement or the trajectory. This algorithm evaluates the dedicated vehicle description (*.xvd) file, retrieves the velocity vector and the <proximity> value, and inserts waypoints to smooth the trajectory. The resulting movement is more realistic.

To use the algorithm, enable "Smoothing", select "Vehicle File > Load" to load a *.xvd file and select "Calculate". See Chapter A.4, "Vehicle description files (Used for smoothening)", on page 858.

Remote command:
SCENario:DF:RECeiver:MOVement:SMOothening on page 779
SCENario:LOCalized:RECeiver:MOVement:SMOothening on page 779

Vehicle File ← Waypoints trajectory definition
Indicates the name of the currently used vehicle description (*.xvd) file.

Load, Delete ← Waypoints trajectory definition
Opens the Windows explorer so that you can navigate to and load vehicle description file.

Discards the selected vehicle description file.

Remote command:
SCENario:DF:RECeiver:MOVement:VFILe on page 777
SCENario:DF:RECeiver:MOVement:VFILe:CLEar on page 778
SCENario:LOCalized:RECeiver:MOVement:VFILe on page 777
SCENario:LOCalized:RECeiver:MOVement:VFILe:CLEar on page 778

Attitude Behavior ← Waypoints trajectory definition
Defines how the attitude information is defined.

"From Waypoint File"

>The attitude parameters are extracted from the selected waypoint file. Further settings are not required.
>This setting forces the attitude parameters to motion direction even if the waypoint has attitude information, like, for example, in a *.xtd file with <property waypointformat="position_attitude">. For specific emitters like ships or land vehicles, it is realistic to set the yaw and pitch to vehicle's motion direction. This is because the usual body axes angles of a land vehicle are in the direction of the velocity vector.
>For other emitters, however, like landing plane, this parameter is not useful. As an example, the nose of the plane is in an upward direction at the time when the plane is moving downwards.

"Align to Motion"

>Enables a constant rate of change of the roll.

"Constant"   Emitter's attitude is set as the combination of the "Yaw/Heading", "Pitch/Elevation", "Roll/Bank" values.
>The resulting attitude is a constant value.

Remote command:
SCENario:DF:RECeiver:MOVement:ATTitude on page 776
SCENario:LOCalized:RECeiver:MOVement:ATTitude on page 776

Yaw/Heading, Pitch/Elevation, (Start) Roll/Bank ← Waypoints trajectory definition
Sets the angles of rotation in the corresponding direction, i.e. the rotation around the respective yaw, pitch and roll axes. "Yaw/Heading, Pitch/Elevation, Roll/Bank" are defined relative to the local horizon.

Remote command:

SCENario:DF:RECeiver:MOVement:YAW on page 778

SCENario:DF:RECeiver:MOVement:PITCh on page 778

SCENario:DF:RECeiver:MOVement:ROLL on page 778

SCENario:LOCalized:RECeiver:MOVement:YAW on page 778

SCENario:LOCalized:RECeiver:MOVement:PITCh on page 778

SCENario:LOCalized:RECeiver:MOVement:ROLL on page 778

Calculate ← Waypoints trajectory definition

Loads the selected waypoint and vehicle description file.

The "Movement Data" display indicates a summary of the major parameters:
- Start/End position
- Duration of the movement
- Read-out mode as defined in the file
- Smoothing state
- Attitude behavior

The status LED indicates the following states:
- **Red**: the waypoint file is not selected, the file is not imported and the movement not calculated or the file is erroneous.
- **Green**: Movement is calculated.

Remote command:

SCENario:DF:RECeiver:MOVement:IMPort on page 779

SCENario:LOCalized:RECeiver:MOVement:IMPort on page 779

≣ Clear ← Waypoints trajectory definition

Discards the waypoint and vehicle description file.

Remote command:

SCENario:DF:RECeiver:MOVement:CLEar on page 779

SCENario:LOCalized:RECeiver:MOVement:CLEar on page 779

Trace trajectory definition

The "Trace" trajectory definition is similar to the "Waypoint" trajectory definition.

The differences are:

- The "Waypoint" trajectory is defined by a file.
  The waypoint file contains all needed coordinates so no additional settings are required.
- The "Trace" trajectory is defined by a set of user-defined "Trace Points".
  No coordinate information is available, so additional settings are required. As with a "Line" or "Arc" trajectory, the start position must be specified. Also, the position of each trace point must be individually configured.

A trace is defined with:

Start Position > Latitude, Longitude ← Trace trajectory definition

Sets the receiver's starting position.

Remote command:

SCENario:DF:RECeiver:LATitude on page 751

SCENario:DF:RECeiver:LONGitude on page 751

Trace Points ← Trace trajectory definition
Selects a specific trace point.

Remote command:
SCENario:LOCalized:RECeiver:MOVement:PSTep:SELect on page 774
SCENario:DF:RECeiver:MOVement:PSTep:SELect on page 774

Speed ← Trace trajectory definition
Sets the speed of the moving receiver at the selected trace point.

Remote command:
SCENario:LOCalized:RECeiver:MOVement:SPEed on page 774
SCENario:DF:RECeiver:MOVement:SPEed on page 774

East, North, Height ← Trace trajectory definition
Sets the "East/North" coordinates and the "Height" of the receiver at the selected trace point.

Remote command:
SCENario:LOCalized:RECeiver:MOVement:EAST on page 775
SCENario:LOCalized:RECeiver:MOVement:NORTh on page 775
SCENario:LOCalized:RECeiver:MOVement:HEIGht on page 775
SCENario:DF:RECeiver:MOVement:EAST on page 775
SCENario:DF:RECeiver:MOVement:NORTh on page 775
SCENario:DF:RECeiver:MOVement:HEIGht on page 775

Mode ← Trace trajectory definition
Defines the behavior of the moving object when the end of the trajectory is reached.

"Cyclic"   The trajectory file is repeated cyclically. Once the "End Position" is reached, file movement starts again from the beginning.

Using this mode is recommended in the following cases:
● A circle trajectory
● A trajectory in which the start and the end positions are close to each other.

"Round Trip"   The emitter moves back and forth on the defined trajectory.

"One Way"   The movement is executed once.
                When the "End Position" is reached, this position is assumed to be a static one.

Remote command:
SCENario:DF:RECeiver:MOVement:RMODe on page 778
SCENario:LOCalized:RECeiver:MOVement:RMODe on page 779

14 Working with waveforms

In addition to creating pulse sequences, you can also create sequences composed of waveforms.

Waveforms can be created by the R&S Pulse Sequencer or with external software. If the data format is in one of the supported import formats, you can import any waveform into the waveform library. Regardless whether they are created or imported, waveforms can be used as interferers in complex 2D scenarios.

14.1 Waveform types

Waveforms can be created by the R&S Pulse Sequencer or with external software.

Internally created

- Multitone and CW waveforms
- White Gaussian noise signals with configurable bandwidth
- The signal of a group of background emitters

Externally created

- Waveforms in the Rohde & Schwarz proprietary file format (`*.wv`).
  Such files are generated with the signal generation software R&S WinIQSIM2 or with the real-time options of the Rohde & Schwarz signal generators.
- Files with custom I/Q data created with other Rohde & Schwarz instruments or third-party software.
  For example MATLAB (`*.mat`), R&S PR100 (`*.riq`), Rohde & Schwarz signal and spectrum analyzers (`*.iq.tar`), or the Rohde & Schwarz proprietary modified audio file format (`*.wav`).
- Files with custom I/Q data in ASCII format (`*.txt`) or in binary format (`*.bin`).
  ASCII files can describe the I and Q vectors in one file or in separated `*_i.txt` and `*_q.txt` files.
  If file pairs are stored in the same directory, the second file is recognized automatically by the "Import Wizard" on page 378.
- Custom text-based PDW lists in ASCII or coma-separated file format (`*.txt` or `*.csv`), see Chapter 18.2, "PDW import mechanism", on page 507. You can use the imported signals to retrieve information on the reference signal level and to change it.
- AMMOS IF (`*.aif`), AMMOS PDW (`*.pdw` or `*.ppdw`) and AMREC recording (`*.dat`) files
  Rohde & Schwarz proprietary data formats used with Rohde & Schwarz monitoring equipment. These files typically contain multiple data streams.
  The R&S Pulse Sequencer extracts IF (I/Q) or PDW data from these files.

See also Table A-1, on page 837.

14.1.1 Waveforms as interfering signals

Test scenarios often require interfering signals, for example to simulate more realistic conditions or to stress the DUT (receiver).

In the R&S Pulse Sequencer, you can use waveforms as interfering signals. The following is an overview of the different kind of waveforms, their properties, and application fields. You can combine different kind of waveforms in the same scenario:

- Unmodulated (CW) interfering signals
  Waveform types with suitable characteristics are the *multitone and CW waveforms*.
  See Chapter 14.1.2, "About the multitone signals", on page 369.

- Pulse interfering signals
  Waveform types with suitable characteristics are the *background emitters and the imported waveforms containing pulsed signal*.
  A background emitter creates the signal of up to 255 emitters. When included in a "Localized Emitters" scenario, background emitters act as a source of a noise-like pulsed background signal.
  See Chapter 14.1.3, "About the background emitters", on page 370.

- Modulated interfering signal
  Waveforms *containing radiocommunication signals* can be imported in the software.
  These types of waveforms are useful if the test case requires interfering radiocommunication signals or any kind of modulated signals broadcasted at a particular frequency.
  See "Settings for importing externally created Rohde & Schwarz waveforms or files with custom I/Q data" on page 377.

- Noise
  The waveform type *white Gaussian noise* creates this kind of signal.
  See "White noise settings" on page 376.

In the "Localized Emitters" scenarios, the interferers are treated like emitters. They are described with a waveform, an antenna pattern and antenna scan, static location with it X, Y and Z coordinates, and orientation defined with an "Azimuth" and an "Elevation". Per default, interferers broadcast power equally in all directions.

14.1.2 About the multitone signals

Multitone waveforms can consist of up to 1000 sine waves (tones).

The spacing and the number of tones are user-definable. The tones are arranged symmetrically around the RF carrier (see Figure 14-1).

The start phase of each tone (CW) signal is a random value. This approach typically results in signals with lower PAPR compared to multitone signals where all CW start with the same phase.
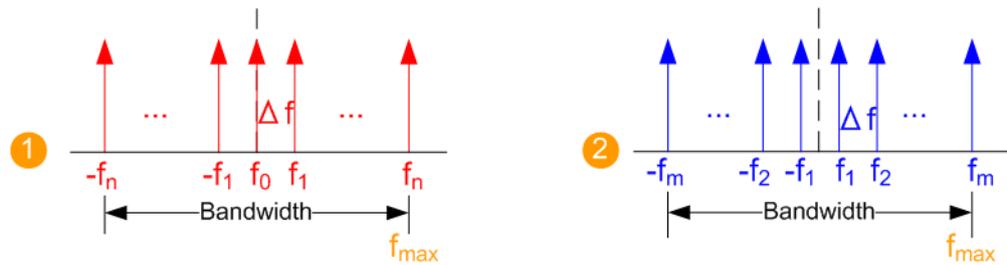
Figure 14-1: Multitone waveforms: arrangement of the tones and calculation of the total bandwidth

1   = Odd number of tones
2   = Even number of tones
Δf = "Spacing"
Bandwidth = ("Tones" - 1) * Δf
f_{max} = Bandwidth/2 = ("Tones" - 1) * Δf/2

The oversampling is set automatically and the clock rate is calculated as follows:

Clock rate = 1.25*Bandwidth = 1.25*2*f_{max}

Example:

The following applies for a multitone waveform with "Tones = 64" and "Spacing = 100 kHz":

- Bandwidth = (64 - 1)*100 KHz = 6.3 MHz
- Clock rate = 7.875 MHz

For a description of the related settings, see "CW and Multitone waveform settings" on page 374.

14.1.3 About the background emitters

A background emitter creates the signal of up to 255 emitters.

The created waveform is the superimposed (overlay) signal of up to 255 pulse sequences, one sequence per emitter. Each pulse sequence is composed of a pulse repeated with random PRIs. The pulse widths are randomly selected values that are constant within a sequence but vary on a per emitter basis.

Each emitter uses a randomly selected frequency within the selected bandwidth.

See Figure 14-2 for illustration of the waveform creation principle.
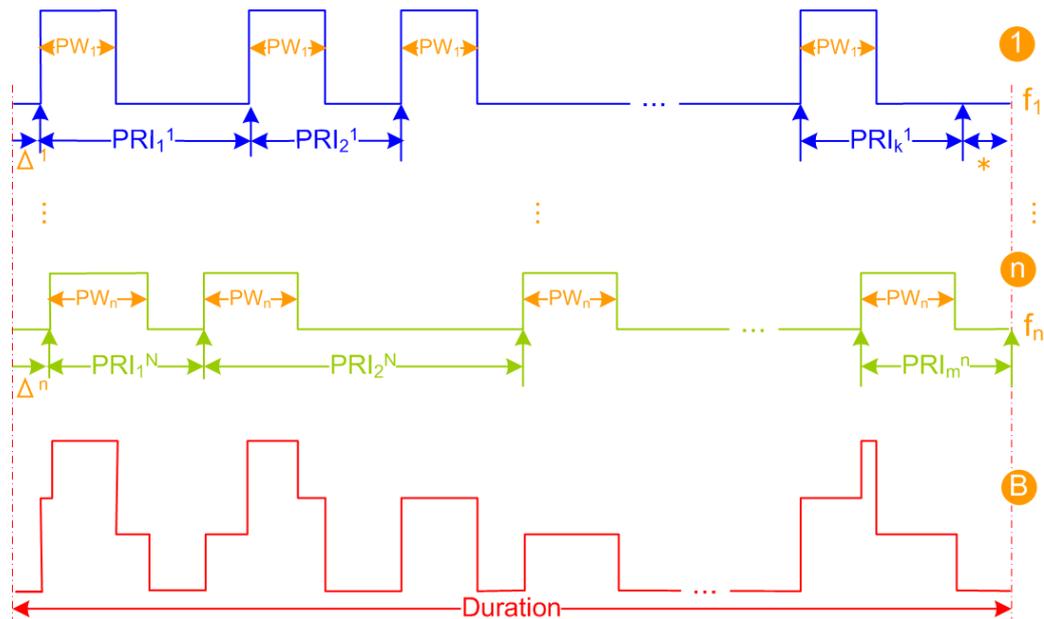
Figure 14-2: Background emitters: simplified representation of waveform structure

n	= Number of emitters ("Count")
1, n	= Pulse sequences of the 1st and the nth emitters
f_1 to f_n	= n random frequency values, within the selected "Bandwidth"
Δ^1, Δ^n	= n random delays
PW_1, PW_n	= n random pulse width values, constant per sequence and different for each emitter
$PRI_1^1 .. PRI_k^n$	= Random PRIs, within the "PRI/PW Ratio" value range
k, m, PRI	= Number of pulses per sequence and PRIs, selected so that the level of the superimposed signal portions does not exceed 12 dB above the "Ref. Level" (see Figure 14-3)
*	= Additional pulse off time at the sequence end to avoid pulse truncation
B	= Resulting signal (Background emitter)

The random process uses the uniform distribution function. The selected values fulfill the user-defined value ranges and the following rules:

● Each pulse uses a "Rise/Fall Time = 1 Sample" and "Pulse Width ≥ 3 Samples"

● Each sequence starts with a randomly selected delay Δ, calculated as follows:
Δ^n = rand () * (PRI_m^n - PW_n), where rand () = 0 .. 1

● Pulses are not truncated.
If the pulse width of the last pulse is longer than the remaining sequence time, the pulse is skipped and the pulse off time is extended until the sequence end.

● The unattenuated pulse top power level of the first emitter is set to the reference level.
All other emitters use reduced pulse top power level, where the maximum level difference between the strongest and the weakest emitter is set with the parameter "Maximum Level Range".
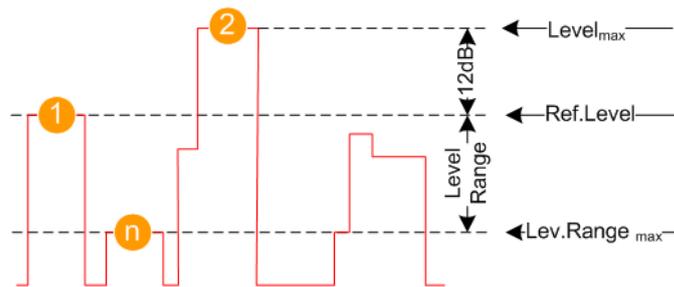
Figure 14-3: Background emitters signal leveling

1	= Pulse top power of the first emitter (i.e. the strongest emitter), set to the reference level (Ref. Level)
n	= Pulse top power of the nth emitter (i.e. the weakest emitter), set to the "Maximum Level Range" (Lev Range$_{max}$) value
2	= Superimposed pulses; the maximum level of the superimposed portion can be 12 dB above the reference level

Level$_{max}$ = Ref. Level + 12 dB

Pulses are superimposed as long as the level of the superimposed signal portions does not exceed 12 dB above the reference level. If this limit is exceeded, a new PRI is selected.

- Pulse top power levels (in dB) are uniformly distributed values

The oversampling is performed automatically, where the used clock rate is calculated as follows:

Clock rate = 1.25*"Bandwidth" and Clock rate ≥ 1 MHz.

The minimum clock rate assures correct sampling of narrow bandwidth signals composed of many fast repeating pulses (small PRIs).

For step-by-step instructions, see:

- "General workflow for creating complex 2D scenarios" on page 416.

For description of the related settings, see:

- "Background emitters settings" on page 375

14.1.4 Waveform scenarios and sequences

Waveforms can be:

- Included in 2D scenarios ("Scenario Type > Localized Emitters") as interferer or as part of a sequence
- In all other scenario types, as part of a sequence

When include in sequences, waveforms are used in similar way as pulses, see Chapter 7.2.1, "Sequence description settings", on page 200.

Dedicated waveform sequences are also described in table form where the rows are processed sequentially. Each row is repeated once or as defined with the parameter "Repetition Count". Complex waveform sequences can contain waveforms and loops. Other control elements are not supported.

For step-by-step instructions, see:

- "To create a waveform scenario" on page 387
- "To create a sequence of waveforms" on page 388.

14.2 Waveform settings

Access:

► Select "Repository Tree > Waveform > New".

Settings:

Waveform Name and Comment

Unambiguous name and an optional description.

Remote command:

`WAVeform:NAME` on page 589

`WAVeform:CREate` on page 588

`WAVeform:SELect` on page 588

`WAVeform:CATalog?` on page 587

`WAVeform:REMove` on page 590

`WAVeform:COMMent` on page 590

Type

Sets the type of the waveform to be created or imported.

See also:

● Chapter 14.1, "Waveform types", on page 368.
● Chapter 18.3, "PDW data settings", on page 509

Remote command:

`WAVeform:TYPE` on page 800

CW and Multitone waveform settings

Unmodulated (CW) waveforms do not require further settings.
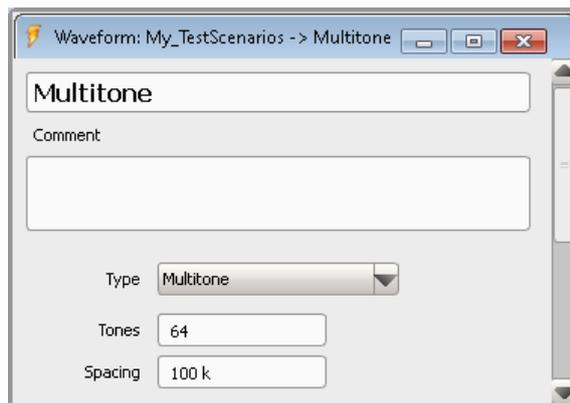
Multitone waveforms can consist of several unmodulated (CW) tones, where the tone spacing and the number of tones are user-definable.

See Chapter 14.1.2, "About the multitone signals", on page 369 for details.

Provided are the following settings:

"Tones"   Sets the number of tones composing the multitone signal.

"Spacing"   Sets the spacing (Δf offset) between the tones.

Remote command:
WAVeform:MT:COUNt on page 801
WAVeform:MT:SPACing on page 801

Background emitters settings

A background emitter is a function that automatically creates the signal of up to 255 emitters.

See Chapter 14.1.3, "About the background emitters", on page 370 for details.

Provided are the following settings:

"Count"   Sets the number of background emitters, that is the number of ran-
                   dom selected frequencies within the defined "Bandwidth".
                   Higher number of emitters leads to higher pulse density.

                   Remote command:
                   WAVeform:BEMitter:COUNt on page 802

"Bandwidth"   Sets the maximum permissible bandwidth of the resulting signal.
                   Emitters are randomly placed within the selected bandwidth.

                   Remote command:
                   WAVeform:BEMitter:BWIDth on page 802

"Duration"   Sets the signal duration (play time) of the waveform.

                   Remote command:
                   WAVeform:BEMitter:DURation on page 802

"Pulse Width Range"

The emitter pulse widths are randomly selected values within the range PW_{min} to PW_{max}.

$PW_{min} \leq PW^N \leq PW_{max}$

Remote command:

WAVeform:BEMitter:PW:MINimum on page 802

WAVeform:BEMitter:PW:MAXimum on page 802

"PRI/PW Ratio"

The PRI values are selected so that the PRI/PW ratio is within the range $(PRI/PW)_{min}$ to $(PRI/PW)_{max}$.

$PRI_m^N = k*PW^N$, where:

- $k = \text{rand}()*[(PRI/PW)_{max} - (PRI/PW)_{min}] + (PRI/PW)_{min}$ and
- $\text{rand}() = 0 .. 1$

Remote command:

WAVeform:BEMitter:PRI:RATio:MINimum on page 802

WAVeform:BEMitter:PRI:RATio:MAXimum on page 802

"Maximum Level Range"

Sets the maximum level difference between the strongest and the weakest emitter.

Remote command:

WAVeform:BEMitter:LEVel:RANGe on page 803

White noise settings

"Bandwidth"   Sets the bandwidth of the generated "White Noise" waveform.

Remote command:

WAVeform:NOISe:BWIDth on page 800

Settings for importing externally created Rohde & Schwarz waveforms or files with custom I/Q data
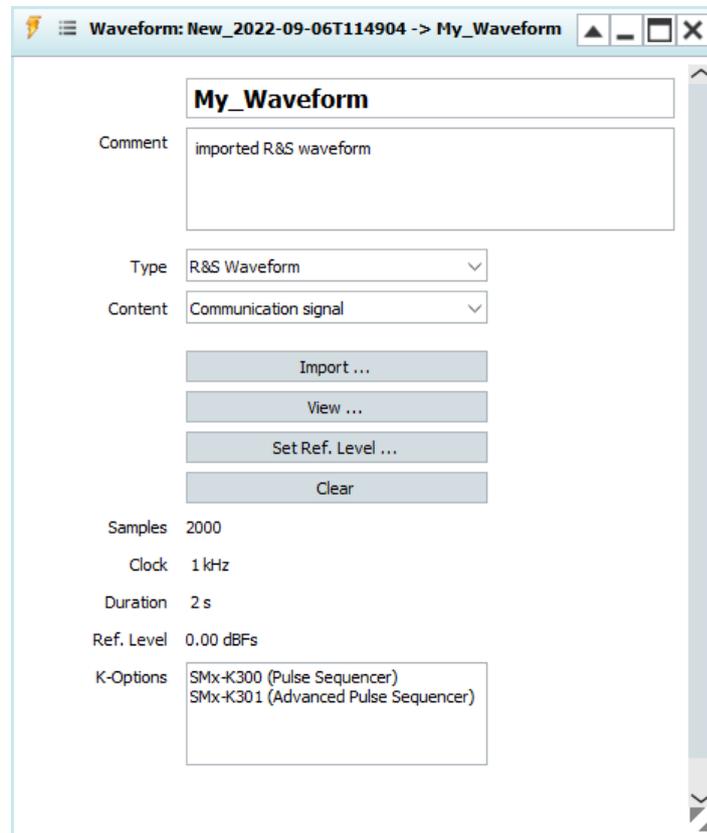
This section describes the available settings.

For step-by-step instructions, see:

- "To import a file with custom I/Q data" on page 385
- "To retrieve more information on an imported waveform" on page 386
- "To change the waveform reference level based on a user-definable signal section" on page 413

"Signal Content"

Sets the waveform signal type and thus determines the filter type used during the resampling process.

The software applies different filters on pulsed signals and on the waveforms containing digital standard signals.

- Waveforms containing digital standard signals are intended for continuous playback. The end of the waveform fits the beginning to avoid spectral regrowth caused by the wrap-around.
- For pulsed signals, however, wrap-around and overshoot effects due to steep filters are undesired. Thus, low-order filters are applied to prevent from overshoots.

Remote command:

WAVeform:SIGCont on page 800

"Import"   Accesses a file import wizard, see Import Wizard.

Remote command:
For loading waveform files
WAVeform:WAVeform:LOAD on page 800

"View, Set Ref. Level"

Access dialogs which visualize the waveform and displays more use-
ful information on it, see:
- Chapter 15.1, "Waveform view and signal preview settings",
  on page 400
- Chapter 15.3, "Waveform reference level settings", on page 409

Remote command:
WAVeform:LEVel:REFerence on page 801

"Clear"   Removes the imported waveform or file with I/Q data.

Remote command:
WAVeform:WAVeform:CLEar on page 801
WAVeform:IQ:CLEar on page 801

"Samples, Clock, Duration, Ref. Level"

After data import, the "Waveform" dialog displays information on the
waveform.

- "Samples": Indicates the number of waveform samples.
  A warning symbol indicates that the waveform contains fewer
  than 512 samples. The waveform length violates the minimum
  required ARB file size.
- "Clock": Indicates the clock rate.
- "Duration": Indicates the waveform duration
- "Ref. Level": Shows the calculated reference level

Remote command:
WAVeform:LEVel:REFerence on page 801

"K-Options"   Displays the required software options.

Import Wizard

The data import wizard converts custom I/Q data into a Rohde & Schwarz waveform
file.

The dialog and the provided settings differ depending on the selected file format but
the differences are self-explanatory.

Input ← Import Wizard

The input settings depend on the custom I/Q data.

The example in Figure 14-4 shows the dialog when importing custom I/Q data in ASCII
format.

ASCII files can describe the I and Q vectors in one file or in separated *_i.txt and
*_q.txt files. If file pairs are stored in the same directory, the second file is recog-
nized automatically by the "Import Wizard".
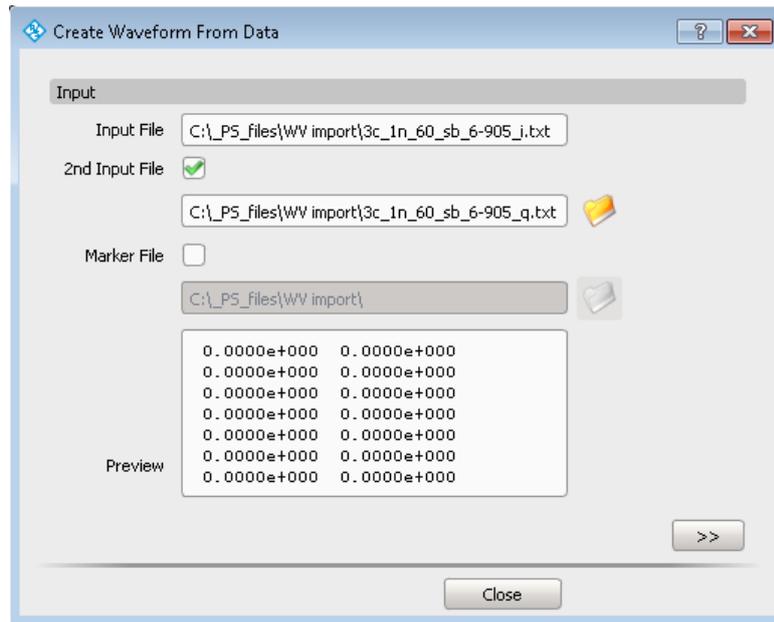
Figure 14-4: Import Wizard > Input (ASCII file)

ASCII and Binary Format ← Import Wizard
The following settings are available for importing of custom I/Q data in ASCII format
(`*.txt`) and binary format (`*.bin`).
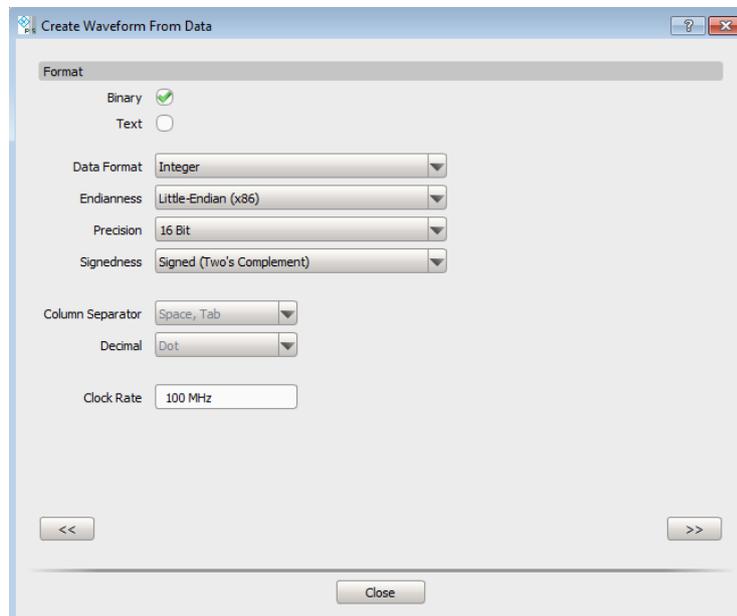
Figure 14-5: Import Wizard > Format (ASCII and binary files)

If the custom I/Q data is in ASCII text format, you can also set the used column and
decimal separators.

MATLAB File Content ← Import Wizard

Indicates the file with custom I/Q data and the data type.
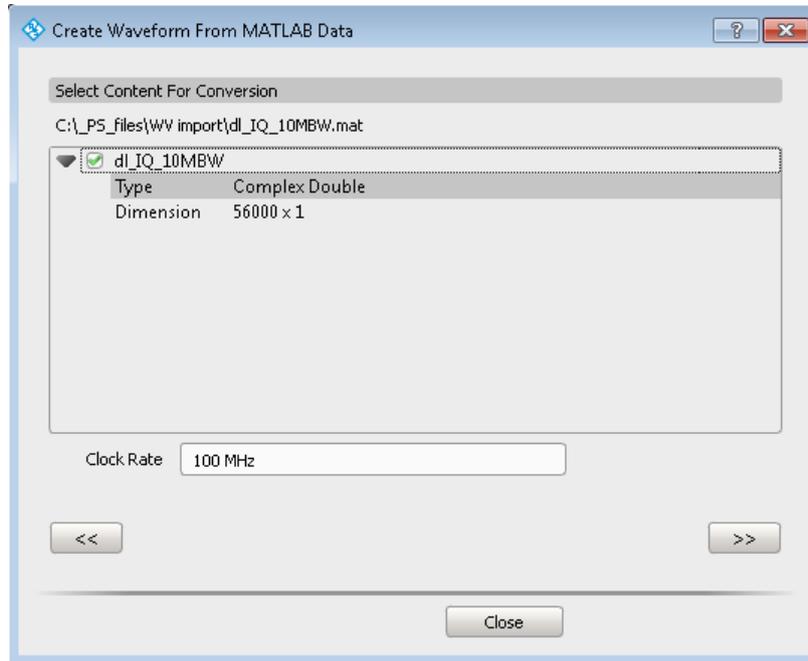
Figure 14-6: Import Wizard > Content (MATLAB file)

RIQ File Content ← Import Wizard

Displays the file content information.
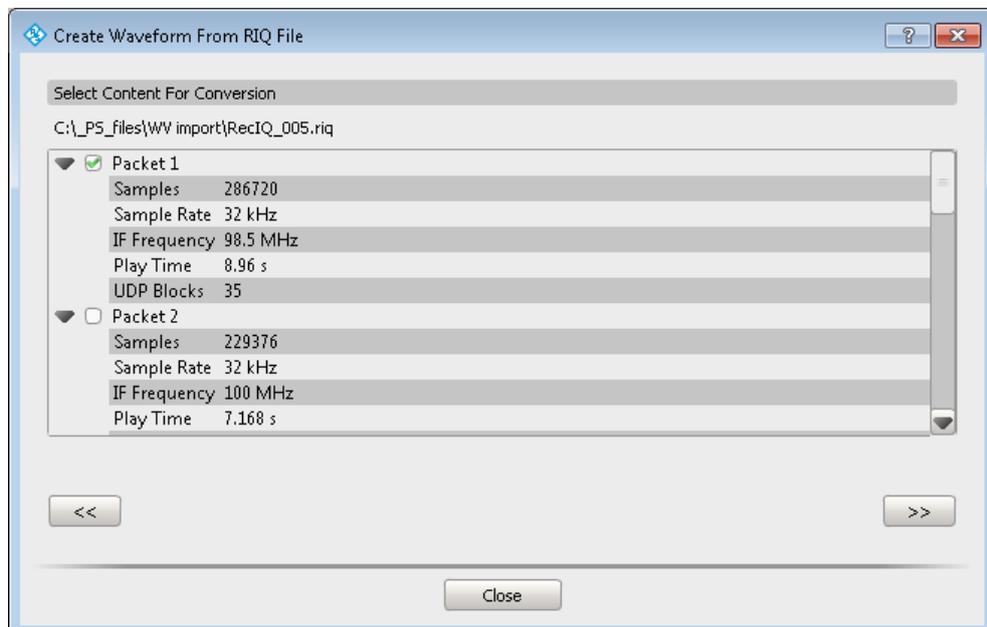
Figure 14-7: Import Wizard > Content (RIQ file)

*.iq.tar and *.wav File Header ← Import Wizard
Displays the file header information retrieved form the file content.
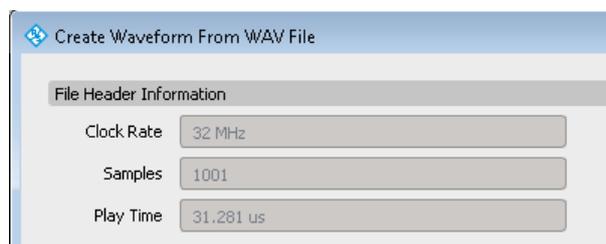
Figure 14-8: Import Wizard > Content (*.iq.tar and *.wav file)

Scaling, IQ vs Time, Constellation, Spectrum ← Import Wizard
The example in Figure 14-9 shows the dialog when importing MATLAB files.

For overview of the supported file types, see Chapter 14.1, "Waveform types", on page 368.
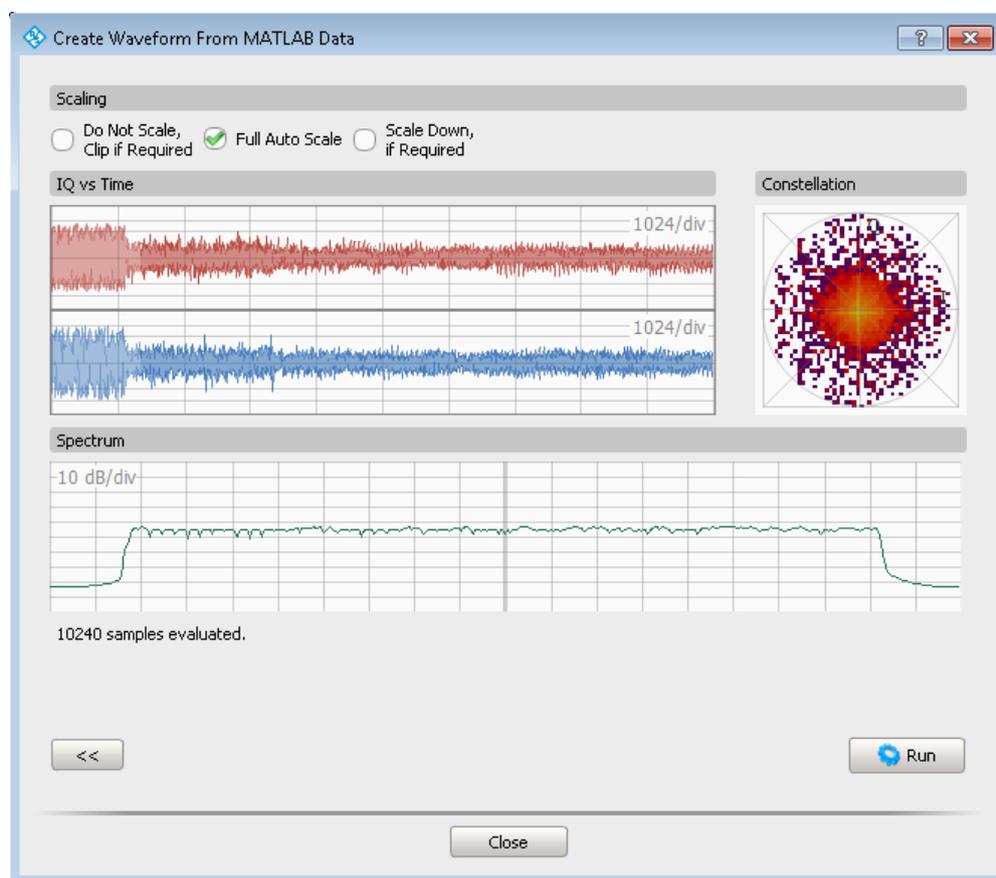
Figure 14-9: Import Wizard > Scaling (MATLAB file)

The dialog shows a preview of the first 10 000 samples in the file.

Per default, the "Full Auto Scale > On" is selected. The waveform is calculated so that it perfectly fits the 16-bit dynamic range of the instrument's ARB. The waveform is not clipped.

The selection "Do Not Scale, Clip if Required > On" leaves the signal amplitude unchanged. This setting could lead to low signal levels or clipping.

AMMOS IF and AMMOS PDW Import

AMMOS IF, AMREC recording files and AMMOS PDW are Rohde & Schwarz proprietary data formats used with Rohde & Schwarz monitoring equipment. These files typically contain multiple data streams.

The R&S Pulse Sequencer extracts IF (I/Q) or PDW data from these files.

The import process and wizard are described in:
- "Import Wizard" on page 378
- Chapter 18.4, "PDW data import wizard settings", on page 511.

The following is a description of the dedicated settings.

Access: "Imported Signals > Type = AMMOS PDW".
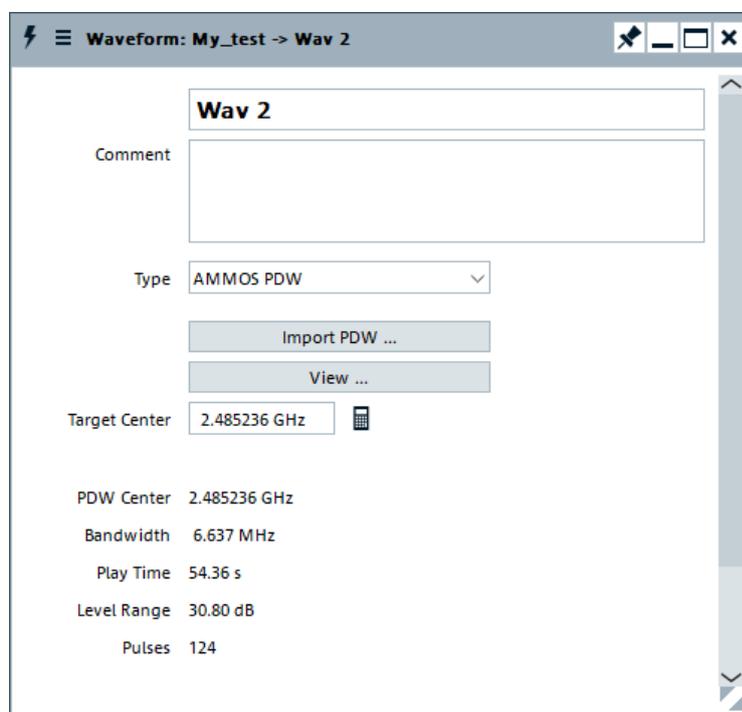
"Import PDW" Accesses a file import wizard, see "Import Wizard" on page 378.

       Remote command:
       WAVeform:WAVeform:LOAD on page 800

"View"

       Access dialogs which visualize the waveform and displays more useful information on it.
       See Level Attenuation.

"Target Center"

 Sets the center frequency of the generator.
When a PDW list file is imported, the value is set to the automatically calculated center frequency of the PDW file. The value resembles the value indicated as "PDW Center".
If the actual frequency of the generator differs from the calculated one, set the parameter "Target Center" to the *center frequency of the generator*.
The frequency offsets of the pulses are recalculated. The bandwidth increases.

 Remote command:
WAVeform:PDW:CENTer on page 803

"Calculate"   The "Calculate" icon resets the "Target Center" field to the value displayed as "PDW Center".

"PDW Center, Bandwidth, Play Time, Level Range, Pulses"

 If the PDW list file is imported in the repository, the dialog displays information on the waveform:
- "PDW Center": *Calculated center frequency*
  The indicated value is the center frequency that the software uses during the import process. The pulses are calculated relatively to this value.
  The "PDW Center" is calculated as the middle frequency between the min and the max frequency values included in the PDW file. Chirp frequency deviations are considered.
- "Bandwidth": Calculated bandwidth
- "Play Time": Duration
- "Level Range": Calculated level range
- "Pulses": Number of pulses in the waveform.

Input File ← AMMOS IF and AMMOS PDW Import

Figure 14-10: Import Wizard for Type = AMMOS PDW

"R&S PDW File"

 Selects the source file.

"ACH Filter"   Sets the analysis channel filter that defines which of the four analysis channels within the bandwidth is used.
                   If ACH filter is not used ("ACH Filter = Off"), the import mechanism discards all pulses that overlap with the current one. This behavior cannot be changed. To ensure processing of one pulse per time, apply an ACH filter.

"MOP Filter"   Filters out pulses based on the used modulation.

"CW Support"   Enables import and replay of CW signals.

Level Attenuation ← AMMOS IF and AMMOS PDW Import

Graphical representation of the retrieved information, together with statistical information on the processed data.
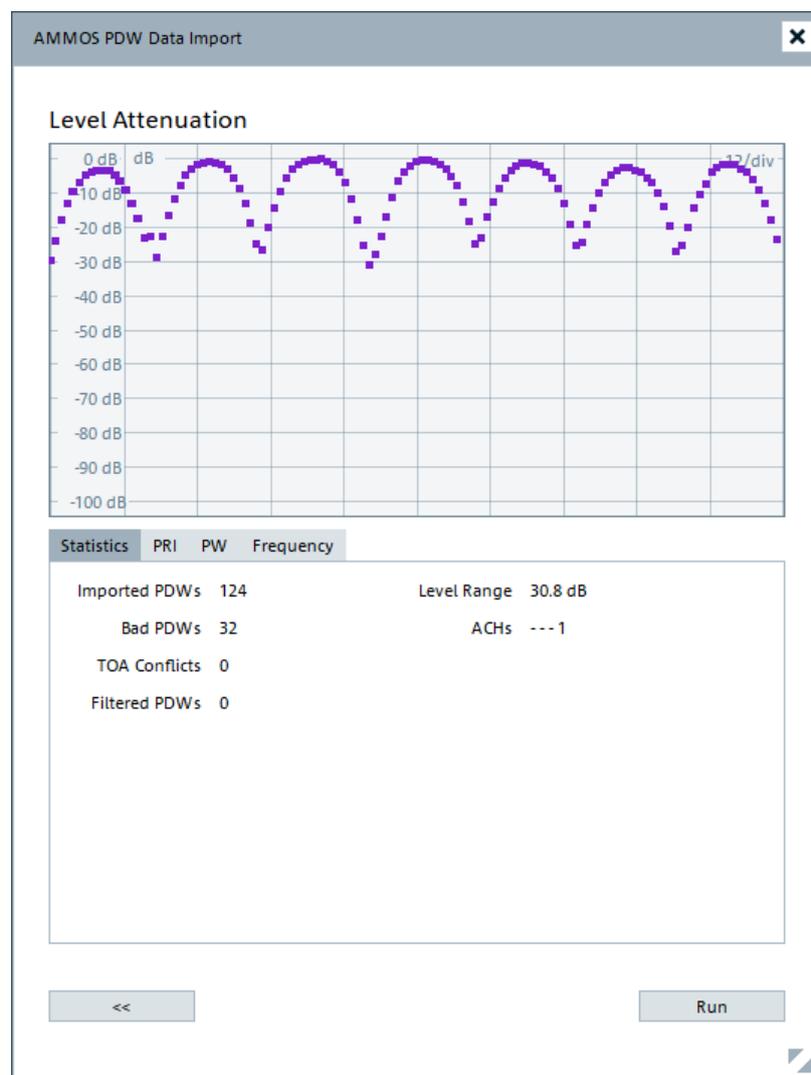
Displayed is the time variation of the following parameters:
- Level Attenuation
- PRI
- PW

● Frequency

See also Chapter 15.2, "Signal preview settings", on page 405.

14.3 How to create a waveform scenario and work with waveforms

See:
● "To create a waveform scenario" on page 387
● "To create a sequence of waveforms" on page 388
● For how to build a waveform library, see:
    – "To create a waveform" on page 385
    – "To import a file with custom I/Q data" on page 385
    – "To retrieve more information on an imported waveform" on page 386
    – "To import PDW list data" on page 501

To create a waveform

1. Select "Repository Tree > Waveform > New".

    A new waveform with default settings is created and added to the project tree. New waveforms are named `Wav_<n>` where `n` is a number starting at one.

2. Add information to describe and identify the waveform, like a name and a comment.

3. Select "Type > White Noise".
    Set the "Bandwidth".
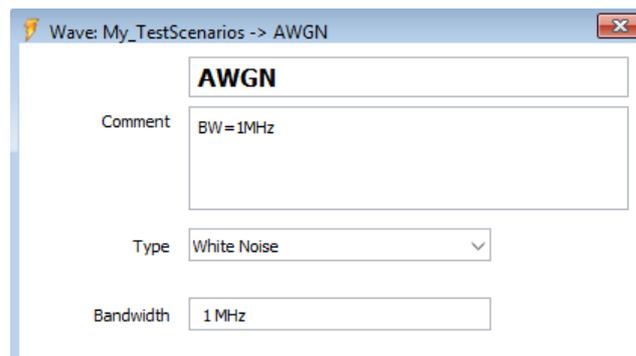
To import a file with custom I/Q data

1. Select "Repository Tree > Waveform > New".

2. Enter a name and a comment.

3. Select "Type > Custom I/Q Data".
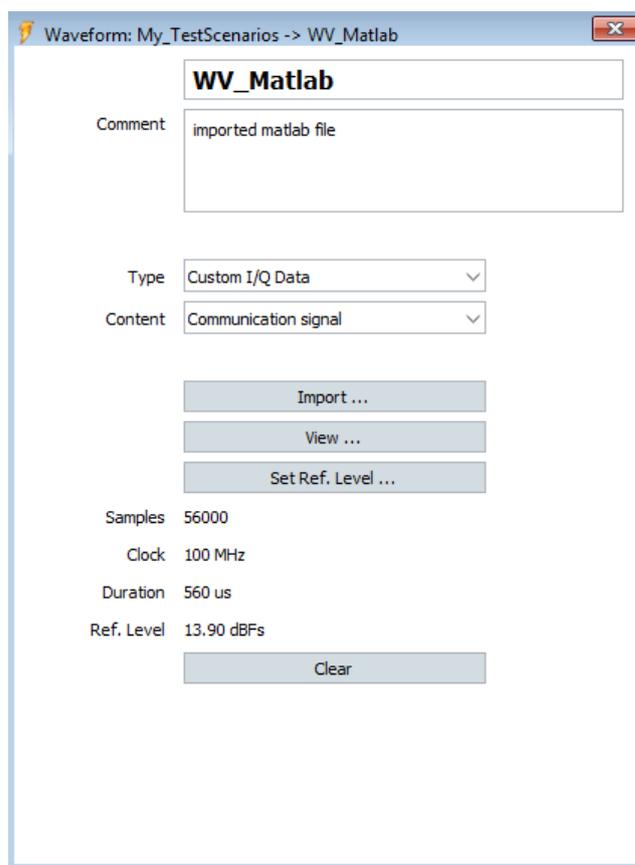
4. Select "Import".

   Select the `*.mat` file. Follow the instructions.

   An "Import Wizard" opens. See the example in "Import Wizard" on page 378. The wizard differs depending on the selected file format but the differences are self-explanatory.

   After the import has been completed, the "Waveform" dialog displays information on the waveform.

   See also "To retrieve more information on an imported waveform" on page 386.

To retrieve more information on an imported waveform

1. Select "Repository Tree > Waveform", e.g. "WV_Matlab"

2. To display the I/Q diagram or the spectrum of the signal, in the "Waveform" dialog, select "View".
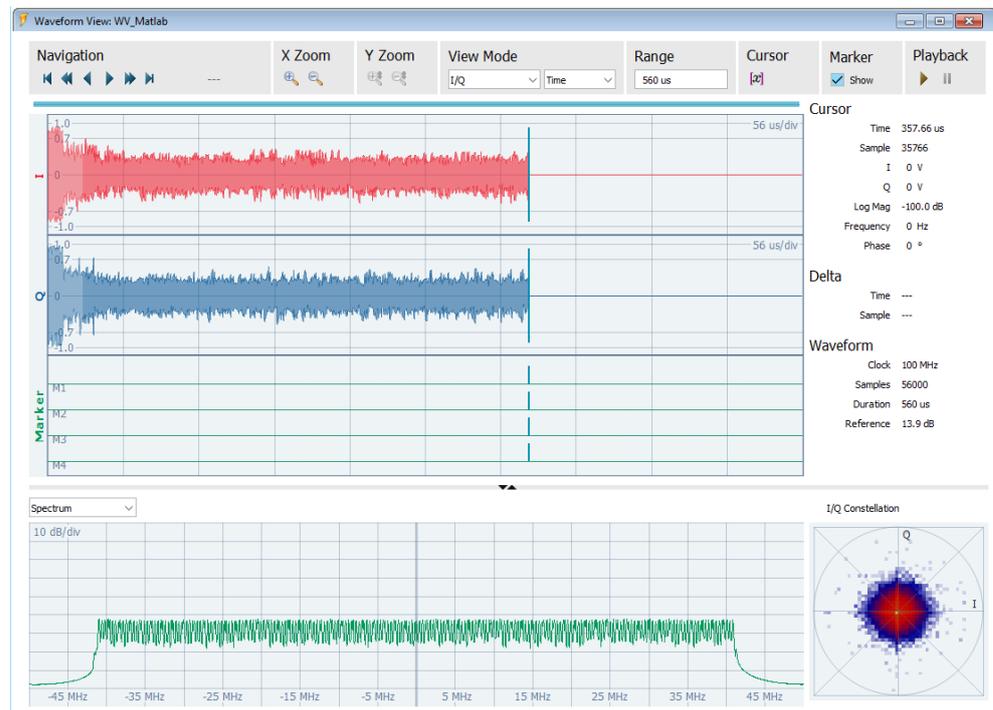
For description of the provided settings, see:
- Chapter 15, "Visualizing and analyzing signals", on page 400
- Chapter 15.1, "Waveform view and signal preview settings", on page 400

To change the waveform reference level

► Follow the instructions in "To change the waveform reference level based on a user-definable signal section" on page 413.

For description of the provided settings, see:
- Chapter 15.3, "Waveform reference level settings", on page 409

To emulate an interferer by using waveforms

Waveforms can be used in sequences or, in "Localized Emitters" and "Direction Finding" scenarios, used to emulate interferes.

► To include interferers in a scenario, drag&drop waveforms to the 2D map.
See Chapter 16.1, "How to create scenarios with receiver and TX items", on page 416.

To create a waveform scenario

The "Waveform Sequence" scenario is a dedicated scenario type for combining waveforms.

► Create a scenario with "Scenario Type > Waveform Sequence".
See also "To create a single pulse train scenario" on page 146.
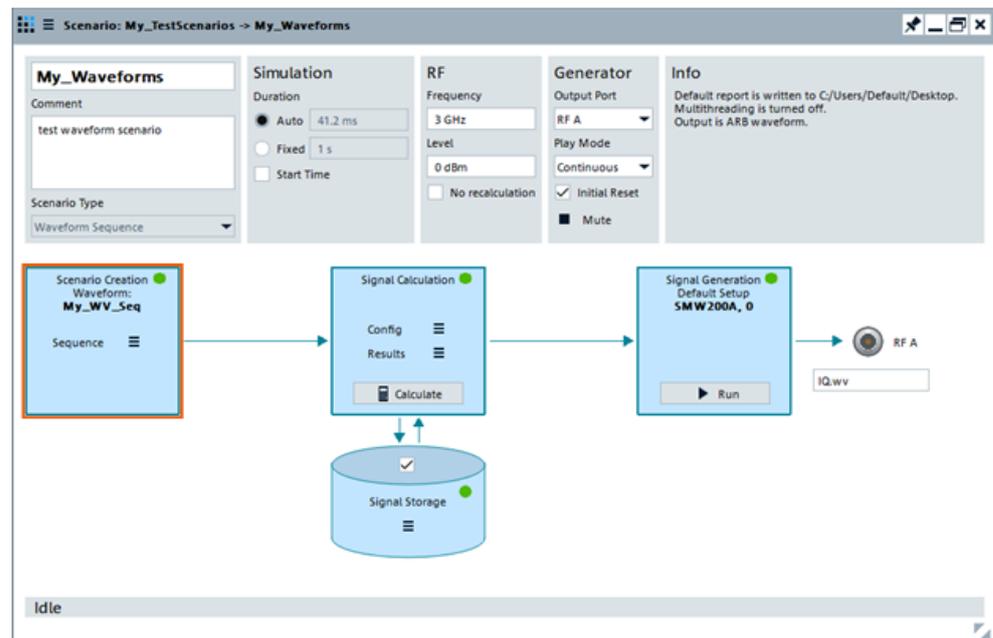
Figure 14-11: Waveform scenario dialog

To create a sequence of waveforms

1. Perform one of the following:

   a) In the "Waveform Scenario" dialog, select "Scenario Creation > Sequence > New" (see Figure 14-11).
      The "Sequence" dialog opens.
   b) Select "Repository Tree > Sequence > New".
      Select "Sequence Type > Waveform Sequence".

   The "Waveform Scenario" dialog opens, then perform step a.

2. To add waveforms to the sequence, in the "Sequence" dialog perform one of the following:
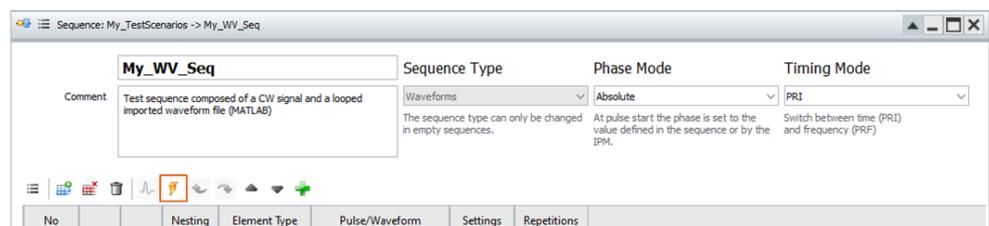
- Select the "Create new waveform and append to sequence" icon.
  See "To create a waveform" on page 385.
- If there are waveforms in your repository, select the "Append item to list" icon.
  Select "Element Type > Wave".

- To add a waveform from the list once more, select the waveform in the table and select the "Copy and append" icon.

A waveform sequence contains one or more waveforms. Complex sequences can also contain loops.

See also

14.4 Waveform composer

The waveform composer provides tools for waveform editing and composing function-ality. You can arrange various waveforms in time or in frequency by placing waveform segments in up to 16 parallel tracks. The waveform composer builds the final wave-form by adding all waveform data from all tracks on one common time scale.

Access:

► Open the waveform composer in one of the following ways:

- In the menu bar, select "ARB Tools > Composer".
- In the toolbar, click "Composer".
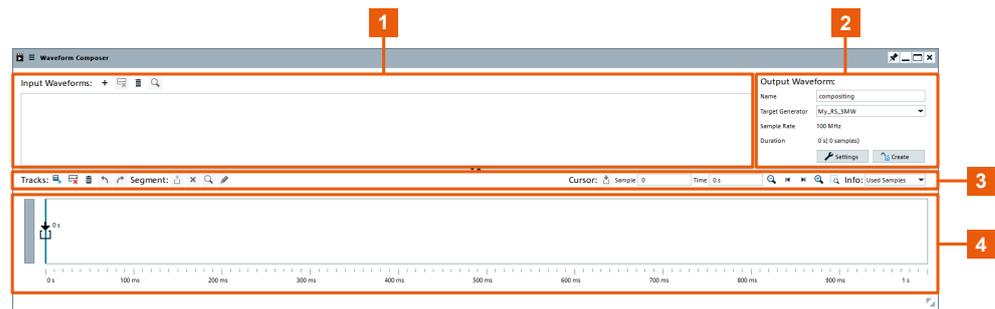
Figure 14-12: Waveform composer

1 = "Input Waveforms": Tools for standard functions and display of the library
2 = "Output Waveforms": Tools to define the settings of the output waveform
3 = Tools for working in the visualization area
4 = Visualization area

14.4.1 Input Waveforms

This section provides the tools to work with input waveforms.
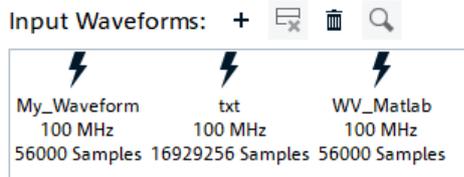
Add

Opens a dialog to select a waveform from the repository to add to the library.

Import waveforms into the repository beforehand. The waveform composer displays added waveforms as tile in the library.

Delete

Removes the selected waveform from the library.

Clear

Removes all waveforms from the library.

View

Opens the waveform viewer and displays the selected waveform.

See Chapter 15.1, "Waveform view and signal preview settings", on page 400.

14.4.2 Output Waveforms

In this section, define the settings for the output waveform.

Name
Enter the name of the created waveform file.

Target Generator
Select a signal generator where you want to play back the created waveform.

The creation process takes the available memory of the signal generator and the maximum bandwidth into account.

Sample Rate
Displays the sample rate of the created waveform.

Duration
Displays the duration of the created waveform.

Settings
Define the parameters related to the waveform created by the composer engine.
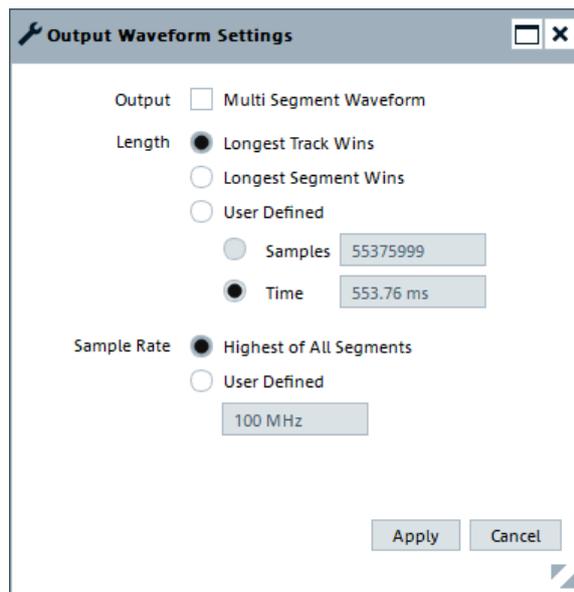
Output Multi Segment Waveform ← Settings
Creates a multi-segment waveform file instead of a regular waveform if one single track is used. Multi segment waveform files contain individual waveform segments that you can select for single or repeated playback.

Length ← Settings
The length of the target waveform is determined automatically or set by the user. All waveform data beyond the end of the waveform gets truncated or is wrapped around to the beginning.

"Longest Track Wins"

> The length of the longest track determines the overall waveform length. This is the default setting, and it is useful in most applications. The length is updated with each editing operation.

"Longest Segment Wins"

> The longest segment in any of the tracks defines the overall waveform length. This setting may be useful when multiple segments are added at the same point in time. The length is updated with each editing operation.

"User Defined"   In this mode, the length is fixed and set in samples or time. The length does not change with editing operations.

Sample Rate ← Settings

The target clock rate is determined automatically or set to a fixed value.

Segments that differ in clock rate require resampling during the building process. These segments are indicated with an "R" symbol in the track editor.

"Highest of All Segments"

> This mode is the default setting and determines the clock rate automatically. This mode selects the highest of all segment clock rates. This, however, may not be sufficient when frequency offsets are used. The target clock rate is automatically adjusted during edit operations.

"User Defined"   Sets the target clock rate to a fixed value.

Apply ← Settings

Confirms the settings for the created waveform and closes the dialog.

Cancel ← Settings

Aborts the changes and closes the dialog without saving the configurations.

Create

The composer engine creates the defined waveform.

14.4.3 Visualization area

In the visualization area you can put together your waveform. This area provides tools to work with the tracks and segments and offers drag and drop functionality.
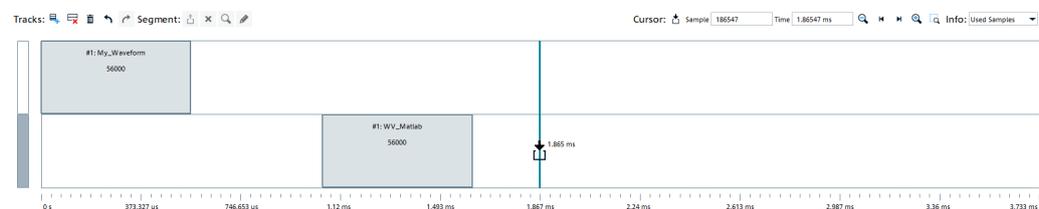
Tracks

Provides standard functions to work with the tracks.

"Add"   Adds a new track in the visualization area.

"Delete"	Deletes the selected track from the visualization area.
"Clear"	Removes all tracks from the visualization area.
"Undo"/"Redo"	Standard functions to reverse or repeat operation.

Segment

Provides fuctions to work with a selected waveform segment.

"Extract"	Removes the selected waveform segment and its respective timeslot. Subsequent segments are shifted to close the timeslot of the removed segment. This operation shifts track content individually depending on the size of the removed segments. Therefore, the timing relationship of the tracks may become unsynchronized after the operation.
"Lift"	Removes the selected waveform segment. The timeslot of the removed segment is left as gap in the sequence. This operation does not shift content and therefore leaves the tracks synchronized.
"View"	Opens the waveform viewer and displays the selected waveform segment. See Chapter 15.1, "Waveform view and signal preview settings", on page 400.
"Edit"	Opens the dialog to edit the selected waveform segment. See Chapter 14.4.4, "Waveform segment editor", on page 395.

Cursor

Provides functions to work with the cursor.

"Insert"	Inserts the selected waveform at the cursor position in all marked tracks. This operation shifts all content following the insert position by exactly the space that the inserted waveform requires. If the insert mark is placed within a segment, the segment is split and the remaining part is shifted beyond the inserted segment. This operation shifts track content individually depending on the size of the inserted segment. Therefore, the timing relationship of the tracks may become unsynchronized after the operation.
"Sample"	Displays the cursor sample. You can also enter a value to place the cursor at a certain sample.
"Time"	Displays the cursor time. You can also enter a value to place the cursor at a certain time.

Navigation and zoom

Tools for navigating and zooming in the visualization area. You can move start or end into view, zoom in/out or fit the zoom that the visualization area displays all segments.

Info

Provides information about the segment.

Center information ← Info

Selects the information displayed in the center of the waveform segment.
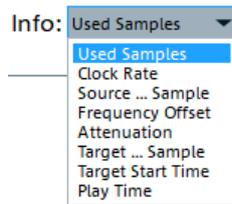
"Used Samples"

> This is the number of samples that are taken from the source waveform. The waveform is truncated at the beginning or the end if the number of used samples is less than the total number of waveform samples.

"Clock Rate"

> This is the source clock rate of the waveform used in the segment. The source clock rate may differ from the output waveform's clock rate. In this case, resampling is required, and the segment is marked with a "Δclk" symbol.

"Source Start Sample"

> This is the number of the first sample that is taken from the source waveform. The waveform is truncated at the beginning if the first sample is greater than zero.

"Frequency Offset"

> This is the frequency offset that is set for the segment. Setting a frequency offset shifts the signal away from the carrier frequency. The frequency offset can be positive or negative. If a frequency offset is set, the segment gets marked with the "ΔFreq" symbol.

"Attenuation"

> This is the level attenuation set for the segment. Setting a level offset reduces the signal level of this segment in the output waveform. Positive values reduce the signal level. If a level offset is set, the segment gets marked with the "ΔLev" symbol.

"Target Start Sample"

> This is the start sample of the segment in the output waveform. The start sample is related to the output waveform's clock rate.

"Target Start Time"

> This is the start time of the segment in the output waveform.

"Play Time"

> This is the segment's play time based on the source clock rate and the number of used samples.

Segment states ← Info

The third row in the waveform segment contains icons that indicate certain segment states.

ΔClk	The source clock rate differs from the output clock rate. Resampling is required for the segment.
ΔLev	The segment is attenuated.
ΔFreq	The segment is frequency-shifted.

14.4.4 Waveform segment editor

Access:

1. In the visualization area, select the waveform segment you want to edit.

2. In the toolbar of the visualization area, "click Segment > Edit".

   The "Edit Waveform" dialog opens.
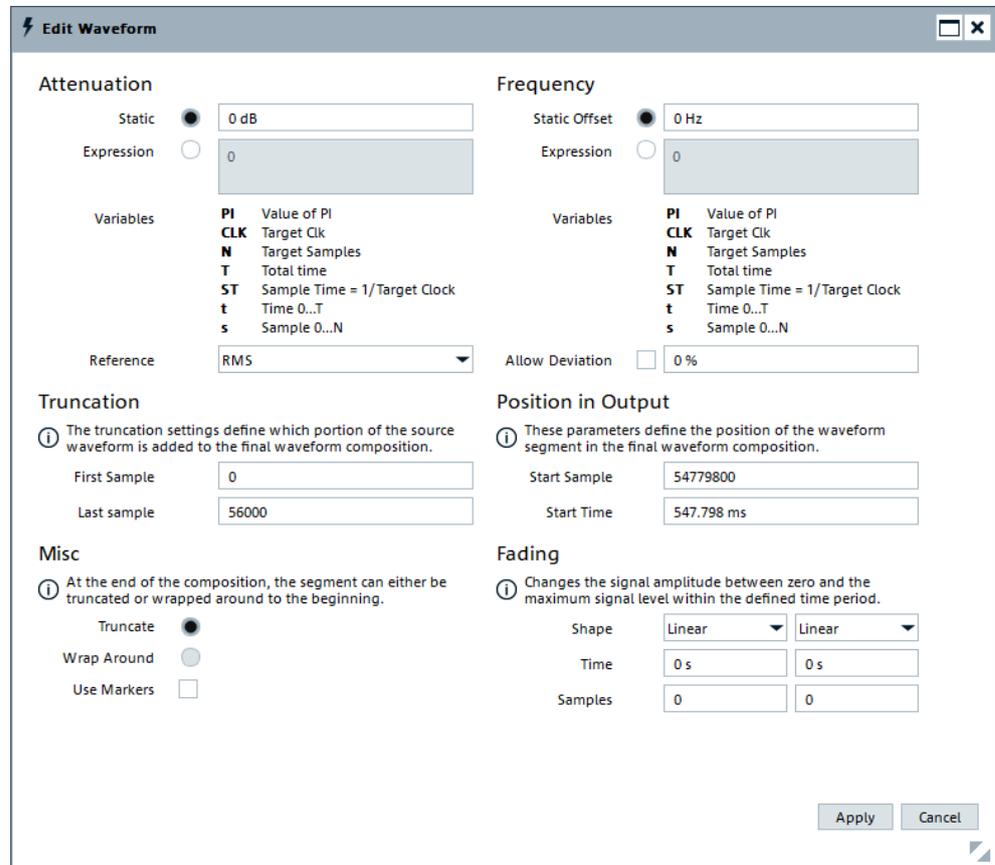
Attenuation

Adds attenuation to the signal.

"Static"
Sets a fixed attenuation value.
The attenuation reduces the signal level of the segment in the output waveform. A positive value of 6 dB, for example, reduces the RF signal level by 6 dB and the I/Q baseband level to 50 %.

"Expression"
Sets the attenuation according to a mathematical expression which is used to define the attenuation as a function of time.

"Variables"
Displays the available set of variables which you can use in the expression.
- **PI**: The number pi.
- **CLK**: The target clock rate in Hz.
- **N**: The number of samples of the output signal.
- **T**: The total signal time of the output signal.
- **ST**: The sample time (ST = 1/Target clock).
- **t**: The current time (0 ... T).
- **s**: The sample index (0 ... N).

"Reference"
Sets the reference for leveling a waveform segment.
You can select "RMS" (root mean square) or "Peak" (peak level). For example, modulated telecommunication signals use the RMS value whereas pulsed signals often use the peak value.

Frequency

Adds a frequency offset to the signal.

"Static Offset"
Sets a fixed value for the frequency offset.
The frequency offset shifts the signal in frequency. You can add the same waveform at the same position to multiple tracks and then apply individual frequency offsets. The result is a multicarrier scenario.
Make sure that the target clock rate is large enough for the desired frequency shift and signal bandwidth. We recommend a minimum target clock rate of twice the signal bandwidth plus the frequency offset.

"Expression"
Sets the frequency offset according to a mathematical expression which is used to define the frequency offset as a function of time.

"Variables"
Displays the available set of variables which you can use in the expression.
- **PI**: The number pi.
- **CLK**: The target clock rate in Hz.
- **N**: The number of samples of the output signal.
- **T**: The total signal time of the output signal.
- **ST**: The sample time (ST = 1/Target clock).
- **t**: The current time (0 ... T).
- **s**: The sample index (0 ... N).

"Allow Deviation"

Activate to allow frequency offset adjustments to remove phase shifts between the first and the last sample of a continuous signal in wrap around mode. Enter a maximum value for allowed deviation in percent.

The frequency offset entered by the user causes a constant phase shift between adjacent samples. The overall phase change between the first and the last sample depends on the target clock rate, frequency offset and the number of samples. If this total phase change is not a full multiple of 2*π, the phase at the waveform start does not match the phase at the waveform end, which may cause spectral regrowth.

Example:
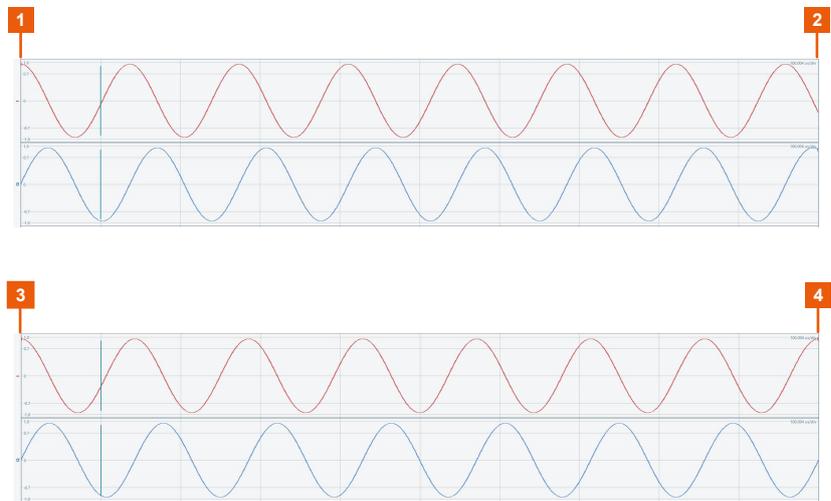
Figure 14-13: Waveform segment with frequency offset causing a phase shift and level difference

1 = Start of waveform segment with frequency offset
2 = End of waveform segment with phase shift. The phase shift and level difference at the start of the next segment may cause spectral regrowth.
3 = Start of waveform segment with frequency offset
4 = End of waveform segment with allowed deviation. The phase shift and level difference between start and end of the waveform segment is corrected.

Truncation

Reduces the size of the waveform segment to use a part of the segment in the created waveform.

"First Sample"   Defines the first sample of the truncated segment. Enter a value higher than zero.

"Last Sample"   Defines the last sample of the truncated segment. Enter a number lower than the maximum value.

Position in Output

Sets the position of the waveform segment in the created waveform.

"Start Sample"   Sets the number of the first sample for the current segment in the created waveform. The start sample number is related to the target sample rate.

"Start Time"   Sets the time where the waveform segment in the created waveform starts.

Misc

Sets flags how to handle the waveform segment in the created waveform.

"Truncate"   Truncates all segment content beyond the end of the created waveform.

"Wrap Around"   This function is not yet implemented.

"Use Markers"   Enables adding marker data from the waveform segment to the created waveform.

Fading

Changes the signal amplitude at the beginning or end of the segment from zero to the full level.

For example, use fading to reduce spectral regrowth at the joint between two segments. Fading is indicated by a small arrow at the beginning or at the end of the segment. The arrow symbolizes the direction of the level change.

"Shape"   The shape defines the profile that is used for changing the amplitude between zero and the full level.
As profile, you can select "Linear", "Raised Cosine", "Square Root" or "Blank".

"Time"   The period of time used for the level change between zero and full level. Leaving the time at zero disables fading.

"Samples"   Samples used for the level change between zero and full level. Leaving the samples at zero disables fading.

14.4.5 Creating a waveform with the waveform composer

To create a waveform with the waveform composer, do the following:

To add waveforms

1. Import input waveforms to the library.

   a) Click "Input Waveforms > Add".
   b) In the dialog, select the waveform.
   c) Click "Add".
   d) Repeat steps a) to c) for all waveforms you want to add to your library.

2. Select a input waveform from the library.

3. Insert the input waveform at the desired positon in the track.

   a) If necessary, add more tracks.
   b) Insert the input waveform at the desired position in the respective track.

To edit waveform segments

1. If you want to edit waveform segments, select the respective segment.

2. Open the waveform editor in one of the following ways:
    ● Click "Edit".
    ● Double-click the waveform segment.

3. Edit the waveform.
    See Chapter 14.4.4, "Waveform segment editor", on page 395.

4. To confirm the settings, click "Apply".

To create a waveform

Define the settings in the "Output Waveform" section.

1. Enter the "Name" of the waveform.

2. Select the "Target Generator".

3. Click "Settings".

    The "Output Waveform Settings" dialog opens.

4. Define the settings.
    See Chapter 14.4.2, "Output Waveforms", on page 391.

5. Click "Apply".

6. Click "Create".

    The waveform composer creates the waveform and and adds it automatically to the pulse sequencer repository. Now you can use the created waveform in your scenarios.

15 Visualizing and analyzing signals

Signals generated with the software or imported into the repository can be visualized with the built-in "Signal Preview" function. The R&S Pulse Sequencer reads a generated signal, evaluates it and displays the I and Q data, the spectrum, and the constellation diagram of a signal section or of the entire signal. The "Signal Preview" is only available if **a signal was calculated (created) successfully**.

Moreover, with R&S Pulse Sequencer you can visualize the content of signals in Rohde & Schwarz format created by external software. Double-clicking the signal name in your file explorer automatically opens the "Signal Preview" dialog. Furthermore, you can select the R&S Pulse Sequencer as the default program for opening such files.

15.1 Waveform view and signal preview settings

The waveform viewer reads the entire waveform data, evaluates it and visualizes the signal in three sections: The time domain view, the spectrum/spectrogram, and the I/Q constellation plot.

Access:

► In the "Scenario" dialog, select "Volatile > View".
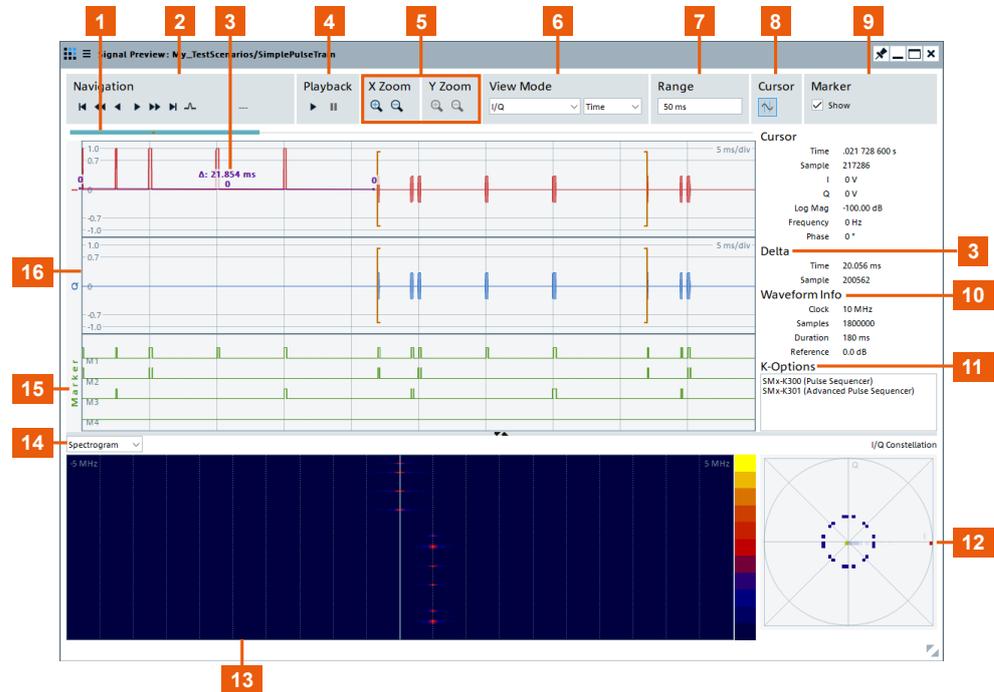
Figure 15-1: Signal Preview/Waveform View: Understanding the displayed information

1 = Timeline
2 = Timeline navigation
3 = Retrieved delta information
4 = Play/Stop
5 = Zoom in/out, see Range
6 = View mode, e.g. I/Q data, frequency, phase, magnitude
7 = Signal part as number of samples or a time duration
8 = Delta cursors, see Playback
9 = Enabled marker traces (M1 = all, M2 = first, M3 = last), see Marker
10 = File information: clock rate, number of samples and duration
11 = Information on the required software options
12 = I/Q constellation plot
13 = Spectrogram in the default rainbow colors, see Display Mode
14 = Display Mode selection
15 = Enabled marker traces (M1 = all, M2 = first, M3 = last), see Marker
16 = Time domain view, "View mode = I/Q" (screen points represent multiple signal samples)

Settings:

Navigation

Steps forward or backward to the beginning or end of the waveform.

Searches for and jumps to the subsequent pulse with level above -80 dB outside the displayed waveform part ("Range").

This function is useful in the following situations:
- To switch between the zoomed in displays of long waveforms.
- To display the first pulse in the waveforms of scenarios with antenna scans, if:
  - At the simulations begin, the emitter antenna does not point to the receivers antenna.
  - It takes time until the first pulse is detected, for example if raster scan is used.

X/Y zoom

Zooms in and out, i.e. changes the resolution on the x and y axis.

Remote command:
SCENario:VOLatile:VIEW:ZOOM:POINt on page 804
SCENario:VOLatile:VIEW:ZOOM:RANGe on page 804

View mode

Switches between the trace types.

The dialog size and the screen resolution limit the number of screen points in the time domain display. They also limit the resolution at which level and time information is represented. Particularly when a large signal section is analyzed, one single screen point can contain the information from many signal samples.

Depending on the zoom level, each screen point represents multiple signal samples. If so, the area between the minimum and maximum value is filled with a semi-transparent trace color.

"I/Q"   I and Q trace on a linear scale in the range between –1.0 and +1.0

"Magnitude dB/Watt/Volt"

Signal amplitude on a 100 dB/1000 mW/100 mV scale

"Phase"   Phase trace on a linear scale in the range between –pi and +pi

"Frequency"   Frequency trace on a linear scale with adjustable scale

"Average Power"

Average value when multiple samples are represented by one screen point.
See also Chapter 15.3, "Waveform reference level settings", on page 409.

Remote command:
SCENario:VOLatile:VIEW:YMODe on page 804

Units

Sets the units (time or samples) used on the x axis.

Remote command:

SCENario:VOLatile:VIEW:XMODe on page 804

Range

Sets the number of displayed signal samples, e.g. to display only a part of the entire waveform.

The "Signal Preview" displays a maximum of 100000 samples. To visualize a longer part of a waveform, reduce the clock rate used by the calculation, see Clock, Duration.

Current clock rate and further information on the waveform is displayed in the "Waveform" section of the "Signal Preview" view.

Cursor [x]

To retrieve delta information, use one of the following:

"Temporary delta information"

Drag the mouse cursor on the diagram.

The delta information is displayed temporary above the line.
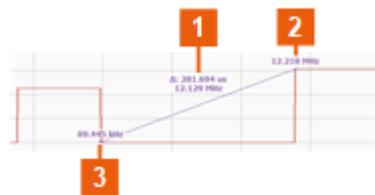
1 = Delta information: upper value is the Δx [in the selected "Units"], lower value is the Δy [units depending on the selected "View mode"]

2, 3 = Start and end y values [units depending on the selected "View mode"]

"[x] Delta cursors"

Select the "[x]" icon and Drag&Drop the cursors.

The delta cursors are indicated by brackets. The delta information is displayed in the "Delta" section of the dialog.

Marker

In each of the view modes, enable "Marker > Show > On" to display all four markers traces M1 to M4.

See also Chapter 21, "Defining and enabling marker signals", on page 534.

Playback

Starts and stops the graphical waveform playback. The waveform section is shifted in time by one half of a display section every 100 ms.

Timeline

1 = Searches pulse outside current "Range" abd jump to it (see "Navigation" on page 402)
2 = Time at mouse position
3 = Timeline
4 = Slider
5 = Delta cursor and mouse position (red and green dots)

- The length of the timeline symbolizes the play time of the entire waveform.
- The blue slider indicates the area of the waveform that is visible in the viewport.
- The red dots indicate the time that corresponds to the cursors and mouse positions.
- The numerical value above the timeline indicates the time at the mouse position.
- A left mouse click moves the viewport center position to the selected time.
- Sliding the blue slider shifts the viewport, i.e. shifts the visible waveform area.

Display Mode

Sets the display mode:

"Spectrum"   The spectrum shows the FFT of the signal portion that is visible in the viewport. The FFT algorithm processes the waveform data in sections of 512 points.
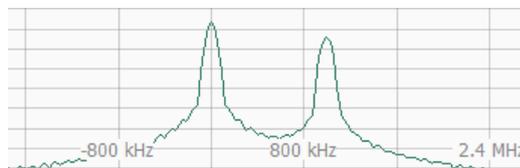
"Spectrogram"

The spectrogram (waterfall diagram) shows the spectrum of a signal portion as a function of time. Each line in the spectrogram represents one FFT with 512 points.
The spectrogram uses a color scale to represent the dynamic range. You can change the default rainbow color scheme in the context menu of the diagram.

I/Q constellation plot

Depending on the number of samples visible in the viewport, displays the individual I/Q samples or turns to a density plot display.

15.2 Signal preview settings

Per default, the R&S Pulse Sequencer evaluates the settings and creates waveforms out of them.

However, if extending sequencing functionality is used, the output is in Rohde & Schwarz proprietary file format.

Files generated in this way cannot be evaluated as I and Q data, as this evaluation is done with the waveforms. Instead, a dedicated "Signal Preview" dialog displays the main pulse characteristics in a table form.

Access:

1. In the "Scenario" dialog, select "Signal Generation > Config > Target > Signal Generators" and select an instrument with extended sequencing option.

2. Select "Signal Calculation > Calculate".

3. Select "Signal Calculation > Results > View" and select the required file.

4. In the "Signal Preview" dialog, select "Table".

5. Select, for example, "Graphics > Frequency".

6. Set "Unit > PDW Count". Press the "Zoom to Fit".

    The "Level", "PW", "Frequency", "Phase" and "PRI" graphs visualize the variation of the corresponding parameter over time.
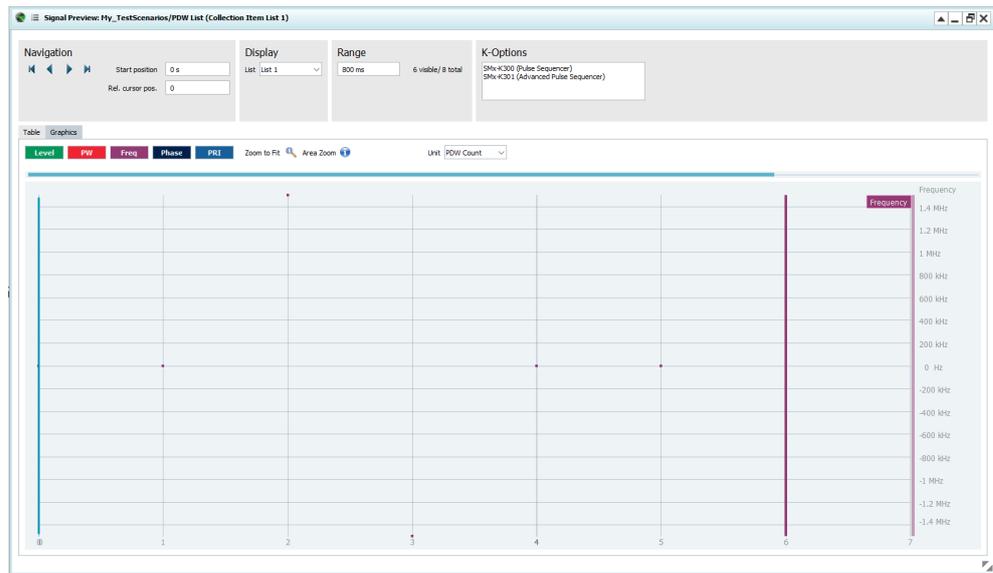
7. You can observe the variation of one or several PDW parameters simultaneously. Select a parameter to include it on the display.

    The display is color-coded.

8. Click the parameter name to change the units on the y-axis.

    The color bar on the right of the display indicates the current PDW parameter.

Settings:

Navigation

Steps forward or backward to the first or last page.

Start Position

Sets the start time of the current page.

Cursor Position/Rel.Cursor Position

Sets the positon of the cursor, expressed in seconds.

Sets the position of the cursor, expressed in:
*   Time units, if "Unit = Time".

- Number of PDWs, if "Unit = PDW Count".

Display

Selects a list to display if more than one list is used in the calculation.

Range

Selects the time range of the current page.

Selecting a lower value reduces the number of pulses visible. The "visible/total" display next to the field updates accordingly.

You can also change this value by holding the cursor over the graph and using the mouse scroll wheel.

K-Options

Displays a list of the required software options.

Table

Lists the file content in table form.

Each row represents one pulse. The sequence in this example consists of 9999 pulses, each described with:

"TOA"	Time of arrival The value reflects the selected PRI/PRF, incl. Delay. The last entry "EOF" indicates the file duration.
"Mode"	- RT = real time For unmodulated rectangular CW pulses and pulses with linear frequency modulation or Barker codes. - WV = waveform For all other pulse shapes
"Level"	Pulse level, as configured with the parameterLevel, incl. IPM variations.
"PW"	Pulse width, as configured with the parameter Width, incl. IPM variations.
"Att"	Attenuation, as configured with the parameters Top Power and "Δ Level" on page 205, incl. IPM variations.
"Freq."	Frequency as set with the parameter Δ Freq., incl. IPM variations. In this example, a frequency hopping IPM profile varies the frequency in the range 0 Hz to 5 MHz.
"Phase"	Value as configured with the parameter Phase, incl. IPM variations.
"MOP"	Indicates the modulation schemes, as configured with the parameter "MOP Type" (see Chapter 6.2.5, "Modulation on pulse (MOP) settings", on page 162) In this example, the modulation is a linear chirp (LFM).

"Param."   The indicated information depends on pulse "Mode".
- "Mode = RT": indicates the major modulation parameter.
  In this example, the parameter is the chirp deviation. The value
  +500.25 Hz/us correspond to a deviation of 5 MHz.
  The positive sign indicates the chirp "Type = Up".
- "Mode = WV": indicates the waveform segment index.

"Marker [4321]"

Indicates if the corresponding marker is active.

Graphics

Visualize the variation of the corresponding parameter over time.

Available parameters ← Graphics

Each PDW is described by the parameters level, pulse width (PW), frequency, pulse repetition interval (PRI), azimuth (Azi) and elevation (Elev).

Each PDW is described by the parameters level, frequency, pulse width (PW), phase and time of arrival (TOA).

The difference in TOA between two consecutive pulses is used to calculate the pulse repetition interval (PRI).

You can observe the variation of one or more PDW parameters simultaneously. The display is color-coded.

The color bar on the right of the display indicates the current PDW parameter. Click the PDW parameter name on the right corner of the graph to change the units on the y-axis.
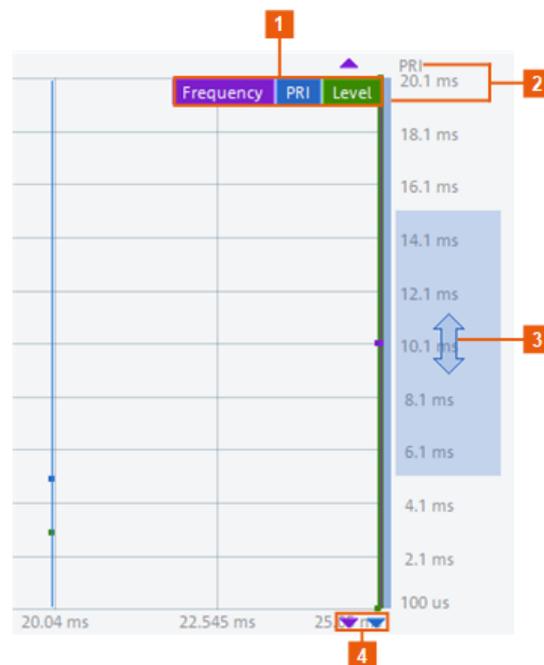
Figure 15-2: Signal Preview: understanding the displayed information

1 = PDW parameters displayed on the graph

2 = Current PDW parameter, indicated by the label on the y-axis and the color bar. To change it, click the colored label of the PDW parameter of interest

3 = Zoom/shift slider scales and repositions the PDW parameters on the display

4 = Auto-scale arrows indicated that the PDW parameter is zoomed in, that is not all values are visible. Click the arrow to zoom out and display all values

The value ranges for both axis are selected automatically. To change the zoom:

- On the x-axis, select the Zoom to Fit or turn the mouse wheel.
- On the y-axis, click the labels and move the "Zoom/shift slider".
  To zoom out, click the auto-scale arrow (see Signal Preview: understanding the displayed information).

Unit ← Graphics

Sets if the x-axis represents the PDW indexes or the time.

"PDW Count"	The PDWs are plotted sequentially with no regard to time; hence all pulses are visible without zooming. This representation is suitable for time-independent measurements.
"Time"	The PDWs are displayed along a time axis and are distributed according to their individual timestamp that is their TOA (time of arrival). This mode shows the pulses in their relation to each other. You can retrieve time-depended information.

Zoom to Fit ← Graphics

Scales the graphs automatically to fit into the current time range.

Turn the mouse wheel to zoom in/out in the display.

Zoom Area ← Graphics

Provides information on defining a zoom area.

Hold the cursor over the blue icon to see the Tooltip.

On the graph, hold [CTRL] and the left mouse button, and drag the mouse to define the zoom area.

15.3 Waveform reference level settings

Access:

1. Select "Repository Tree > Waveform".

2. Select "Waveform > Set Ref. Level".
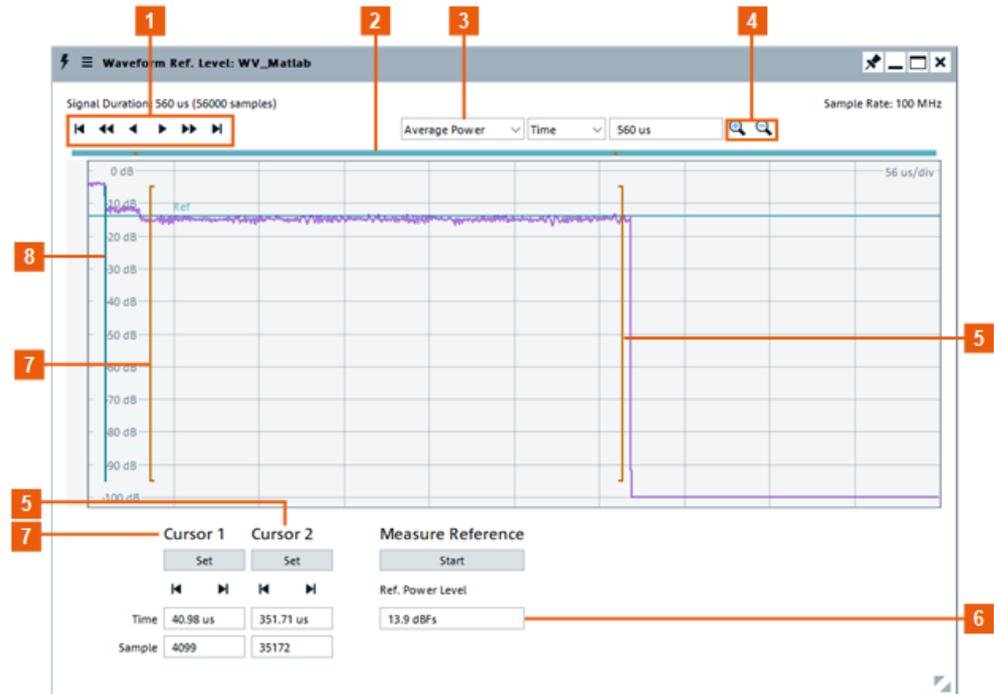
Figure 15-3: Waveform Reference Level: Understanding the displayed information

1 = Navigation
2 = Timeline
3 = View mode, e.g. I/Q data, power, etc.
4 = Zoom in/out and signal part as number of samples or a time duration, see Range
5, 7 = Cursors
6 = Calculated reference level
8 = Primary cursor

The "Waveform Reference Level" reads the entire waveform data and provides a view of the I/Q data versus time. The provided navigation controls, zooming functions, and view modes are similar to the functions provided in the "Waveform View" dialog, see Figure 15-1.

The additional controls define the signal portion that is used for calculating of the reference signal level, see "To change the waveform reference level based on a user-definable signal section" on page 413.

Settings:

Cursors

Define the signal portion used for calculating of the reference signal level.

- The primary cursor is indicated by a single line. It marks the center point for all zoom operations.
- The other two cursors are indicated by brackets. They define the beginning and end point for the reference level evaluation period.

The "Set" buttons set the brackets to the current cursor position.
Both brackets can also be set numerically by using the controls below the cursor
navigation buttons.

Measure Ref. Level

If the imported file contains information on the reference level, this information is displayed as "Ref. Level". In files in the Rohde & Schwarz waveform file format (* .wv files), the reference level is present and correct, i.e. the corresponding tag is set.

If the tag is missing, the software evaluates all samples in the file to calculate the reference level. For same signals, such as pulses, the reference level calculated in this way is of little interest. In this case, you can select the waveform section to be evaluated, e.g. the pulse top power.

The reference level is recalculated for the particular waveform section (see the example in section "To change the waveform reference level based on a user-definable signal section" on page 413).

If a signal generator plays the waveform, the signal generator uses the reference level for leveling.

Remote command:
WAVeform:LEVel:REFerence on page 801

15.4   Analyzing the calculated results

See:

- "To access the "Signal Preview" dialog" on page 411
- "To adjust the display and retrieve more information on the waveform" on page 412
- "To change the waveform reference level based on a user-definable signal section" on page 413

To access the "Signal Preview" dialog

Perform one of the following:

1. In your file explorer, navigate to the waveform of interest.
    Double-click the file name.

    The "Waveform Viewer" dialog opens.
    If R&S Pulse Sequencer was not started, it is launched.

2. Select "Repository Tree > Waveform" and:

    a) Select an imported waveform, e.g. "WV_Matlab".
        See "To import a file with custom I/Q data" on page 385.
    b) Select "Waveform > View".

3. Select "Repository Tree > Scenario".
    Select, for example, "SimplePulseTrain".
    We assume, that you have configured the scenario, selected a signal generator and assigned a sequence to this scenario.

See:
- Chapter 5.3, "How to select and create a test scenario", on page 145.
- Chapter 7.3, "How to create sequences and use the control elements", on page 212.

a) In the "Scenario" dialog, select "Prepare".
   The software calculates the waveform. Green LEDs indicate that the processing is complete.
b) Select "Signal Generation > Result > View" and select the name of the calculated waveform file.

See Figure 15-1.
The "Waveform View/Signal Preview" displays the I and Q data, the spectrum, and the constellation diagram of the signal. It also displays the evaluated information from the waveform file, such as the clock rate, number of samples or the duration.

To adjust the display and retrieve more information on the waveform

1. Select the signal characteristic to be visualized, e.g. "View Mode > I/Q".

2. Set the cursor on a particular point to retrieve detailed information.

   A blue line indicates a single cursor. It sets the center point for all zoom operations.

3. Use the mouse wheel or the "Zoom In/Out" functions to display a part of the waveform or the whole generate waveform.

4. To retrieve delta information:

   a) Select the "[x]" icon and drag and drop the cursors.
      The delta cursors are indicated in brackets. The delta information is displayed in the "Delta" section of the dialog.
   b) Drag the mouse cursor on the diagram.
      The delta information displays temporary above the line.
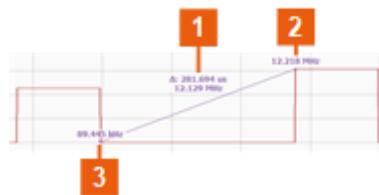
1   = Delta information: upper value is the Δx [in the selected "Units"], lower value is the Δ y [units
        depending on the selected "View mode"]
2, 3 = Start and end y values [units depending on the selected "View mode"]

5. Select "Spectrogram" or "Spectrum".

6. To change the color scheme of the spectrogram, in the context menu, select e.g. "Green".

7. If you observe a part of the waveform, select the play icon to play back the waveform. Observe the signal changes over time.

To change the waveform reference level based on a user-definable signal section

1. Select "Repository Tree > Waveform".

2. Select an imported waveform, e.g. "WV_Matlab".

   The "Waveform" dialog displays information on the waveform. In this example, "Ref. Level = 14.47 dBFs"

Samples	56 k
Clock	100 MHz
Duration	560 us
Ref. Level	14.47 dBFs

3. Select "Waveform > Set Ref. Level".

   The "Waveform Reference Level" dialog opens, see the example on Figure 15-3. It displays the reference signal level of the complete signal. In this example, "Ref. Power Level = 14.47 dBFs".

4. Set the cursor on a particular point.

   A blue line indicates a single cursor. It sets the center point for all zoom operations.

5. Use the mouse wheel or the "Zoom In/Out" functions to display a part of the waveform or the whole waveform.

6. To measure the reference signal of a particular signal part:

   a) ● Set the cursor on the start point of the waveform interval you want to measure.
      Or
      ● Select "Cursor 1 > Time" and enter the beginning of the evaluation period.
   b) Select "Cursor 1 > Set".
   c) Perform the same for the second cursor.

   The two brackets indicate the two cursors. They indicate and define the beginning and end point for the reference level evaluation period.

7. Select "Measure Reference > Start".

   Evaluated are all samples *in the defined period*, see the example on Figure 15-3.

   The calculated reference signal level is displayed. The software stores this value and uses it as a reference level.

   In this example, the calculated value is "Ref. Power Level = 13.9 dBFs".

   Tip: You can query the reference level with the command `WAVeform:LEVel:REFerence`.

8. Close the "Waveform Reference Level" dialog.
   Observe the information in the "Waveform" dialog.

Samples	56 k
Clock	100 MHz
Duration	560 us
Ref. Level	13.90 dBFs

The dialog confirms the calculated value ("Ref. Level = 13.9 dBFs").

16 Creating complex 2D scenarios with receiver and TX items

The dedicated scenario types for complex 2D emitter-receiver scenarios are:

- "Localized Emitters"
- "Direction Finding"

Complex 2D emitter-receiver scenarios can include the following elements:

- **"Receiver"**
  Simple or complex, depending on the scenario.
  There is exactly one receiver per gaming area (2D map).
  The receiver is described by one ore more antenna elements, static position and attitude information.
- **"Emitters"**
  Simple or complex, depending on their definition.
  Emitters are described by the combination of an operating mode, antenna pattern, antenna scan and a sequence, static position and attitude information.
- **"Interferers"**
  Interferers are simulated by waveforms. If placed on a 2D map, interferers are treated like emitters.
  They are described by a waveform, an antenna pattern, an antenna scan, static position and attitude information.
- **"Background emitters"**
  Background emitters are simulated by waveforms. They are a source of a noise-like pulsed background signal, broadcasted with equal power in all directions.
- **"Platforms"**
  A platform is a group of up to 8 emitters associated with a single icon.
  Platforms are differentiated from other emitters by a configurable colored background.
  Each emitter on the platform can be individually configured and controlled.

Complex 2D emitter-receiver scenarios support the following georeferenced map formats:

- GeoTIF

For step-by-step instructions, see:

For a description of the related settings, see:

16.1　How to create scenarios with receiver and TX items

Depending on the test situation, select one of the following scenario types:

- If your test situation requires testing the receiver's ability to detect the sum signal from different static emitters and interferers, you can use a **"Localized Emitter"** scenario.
  In this dedicated scenario, you configure the signal of one or more emitters that are received by a receiver with defined characteristics. You can also configure the receiver and change its position in the scanning beam of the emitters.

- If your test situation requires testing a multichannel receiver and its ability to detect the origin of a signal, you can use a **"Direction Finding"** scenario.
  In this dedicated scenario, you configure the signal of one or more emitters that is received by a receiver with up to 10 antenna elements. The receiver evaluates the receiver signal and estimates the origin of the emitter, based on received power level and time delays.

See:

- "General workflow for creating complex 2D scenarios" on page 416
- "How to create a direction finding scenario" on page 417
- "How to create the scenario and configure the receiver" on page 418
- "How to configure moving emitter and receiver" on page 420
- Chapter 16.8, "Platforms with multiple emitters", on page 464
- Chapter 16.10, "Using georeferenced maps", on page 469

General workflow for creating complex 2D scenarios

To explain the principle, the provided example creates a complex "Direction Finding" scenario. One emitter is placed on the 2D map at a given distance to a multichannel receiver. The receiver and the emitter characteristics are configured.

To create "Localized Emitter" scenarios, follow the same workflow.

For example, drag and drop further elements, such as emitters/interferers, platforms and background signals to the 2D map and configure their settings.

Follow the following general steps:

1. Open a suitable scenario.

2. In the block diagram, select "Map".

   The "2D" dialog displays a 2D view of the receiver and currently configured emitters and interferers, together with their main characteristics.

3. Add a georeferenced background map to the "2D" dialog, if necessary.

4. Drag and drop items from the repository onto the 2D map (or relevant list).

   - Emitters (drop on map or "TX Items" list)
   - Platforms (drop on map or "TX Items" list)
   - Waveforms (drop on map or "Background" list)

5. Use the mouse to reposition (drag and drop) items on the map.

6. Use the context menu of any of the elements on the 2D map to:

    a) Access and configure the "Properties".
        For example, define the elements position more precisely. Or change the atti-tude (orientation) of the antenna.
    b) Display the 3D antenna patterns and scans.
        For example, select "Normalized power level at Receiver > Source > Emitter Only > On" to display only the signal at the receiver.
        The calculation assumes a receiver with an omnidirectional antenna pattern.
    c) To add/remove an emitter/interferer or platform.
    d) To select an icon.
    e) To configure a trajectory (movement path).

7. Use the mouse wheel to change the scale of the 2D diagram, i.e. to change the distance between the receiver and the emitters/interferers.

    For a description of the provided settings, see:
    ● Chapter 16.2, "2D map settings", on page 424
    ● Chapter 16.3, "3D scan pair view settings", on page 429
    ● Chapter 16.4.1, "Available TX items", on page 435
    ● Chapter 16.7, "Background emitters properties", on page 462

How to create a direction finding scenario

This example shows you how to create and configure a scenario for testing of a four-channel receiver. A four-channel receiver can be located for example on the wings of an airplane, see Figure 16-1.

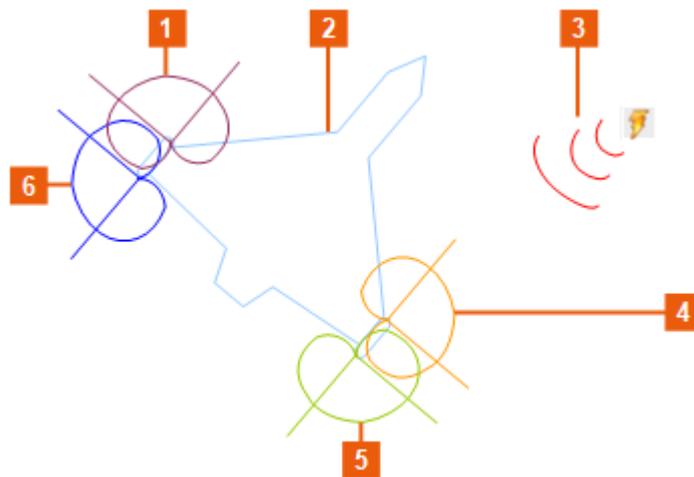*Figure 16-1: Direction finding: four channel receiver*

1, 4, 5, 6 = Four sectors (i.e. the four antennas of a multichannel receiver)
2   = Airplane with a direction finding receiver
3   = Emitter

Each of the four antennas receives a signal with a different power level. In this exam-ple, the antenna at the first sector (at the front on the right side) receives the strongest

signal. The signals are also received with different time delays. The signal at the third sector, for example, has the longest time delay. Multichannel direction finding receivers measure the received power levels and time delays and estimates the origin of the emitter.

The illustrations on Figure 16-2 show the principle of amplitude and time delay-based direction finding.
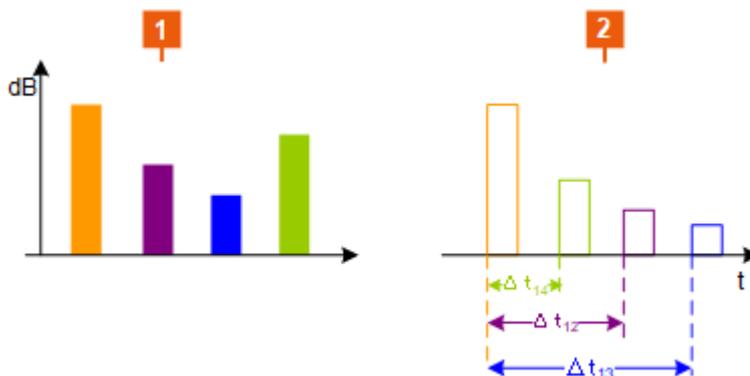
Figure 16-2: Principle of the amplitude and time delay based direction finding

1 = Power levels received by each of the four antennas at the receiver
2 = Relative time delays between the received

How to create the scenario and configure the receiver

1. Create a direction finding scenario.

2. Create a receiver.

   For an example of suitable receiver settings, see Chapter 13.2, "Receiver settings", on page 352 and:
   - Figure 13-3
   - Figure 13-4.

3. In the scenario dialog, select "Map > 2D".

4. Create an emitter.
   For example, use a raster scan and an antenna with a narrow (pencil) antenna beam.
   Enable a "Pitch = 10°" because the receiver is 10000 higher than the emitter.
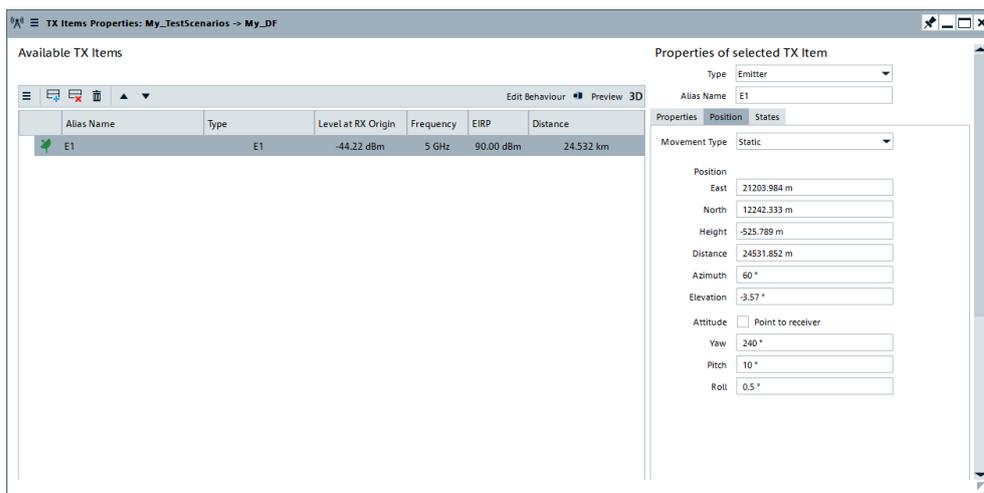
5. Drag and drop the emitter onto the 2D map.
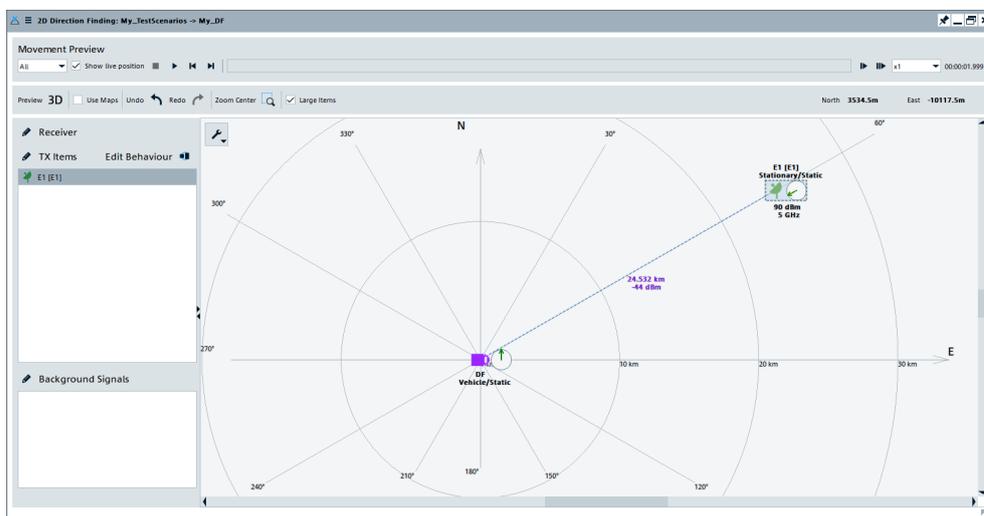
6. Select "Receiver > context menu > Properties".
   Set "Height = 10000 m".

7. Select "Available TX Items > Preview 3D".
   Observe the display. Zoom in; turn on the 3D display for a better overview.

Change the antenna selected with the parameter "Receiver Antenna".
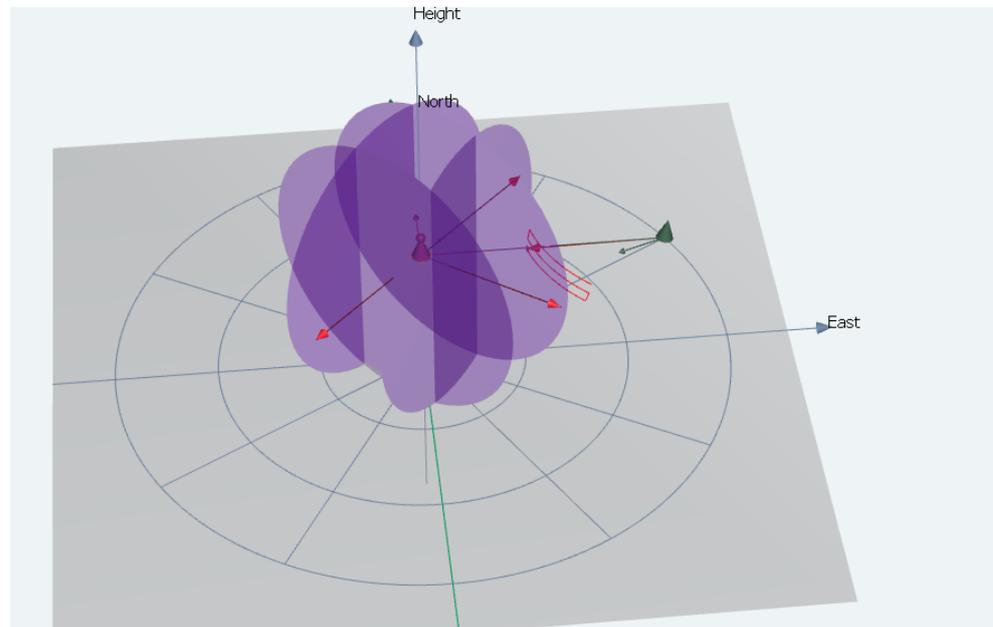
The 3D view displays the four antennas of the multichannel receiver. The receiver and the emitter are at different heights.

The power level of the received signal changes depending on the current "Receiver Antenna".

In this example, the signal from the antenna placed on the right side at the front of the airplane ("Antenna = front right") receives the strongest signal (see Figure 16-2).

See also:
● Chapter 17.6, "How to play the waveforms of 2D scenarios", on page 492

How to configure moving emitter and receiver

1. On the 2D map, select "Properties > TX Items".

2. In the list of "Available TX Items", select an emitter.

3. Select "Properties of the Selected TX item > Position".

4. Select "Mode > Moving".
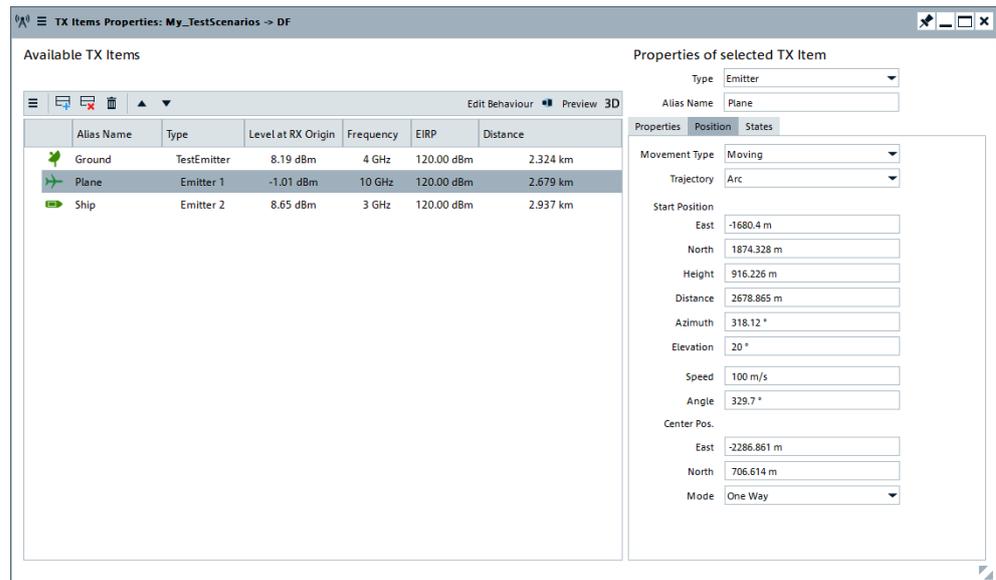
5. Configure the "Trajectory" and further settings.
   See Chapter 16.9, "Creating trajectories on a 2D map", on page 468.

6. To observe the movement on the 2D map, select "Movement Preview > Play".
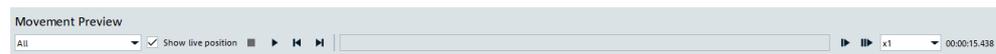
Figure 16-3: 2D map > Movement Preview

If "Show live position" is activated, an overview dialog opens and displays a live preview of important position parameters per TX item and RX item. You can close the "2D Live Preview" at any time. The movement is not interrupted.

The "Movement Preview" dialog displays the duration of the moving scenario. With "Scenario > Signal Calculation > Config > Config > Signal Calculation Settings > Duration > Simulation Duration = Auto", the duration is calculated automatically so that the movement of all objects is completed.
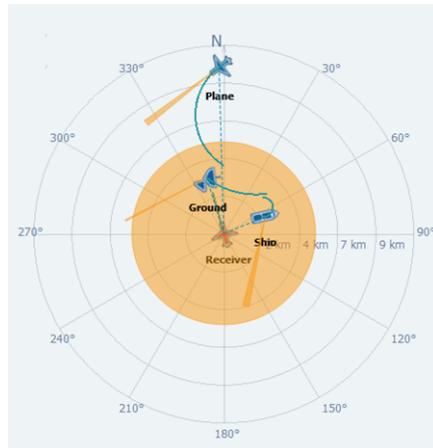
Figure 16-4: 2D map showing antenna types

Beam = Indicates a directional antenna of any kind. The sector direction corresponds to the actual scan direction.
Circle = Represents an omni-directional (isotropic) antenna.

7. To observe the signal variation at the receiver, select an "Emitter > context menu > 3D View with Receiver".
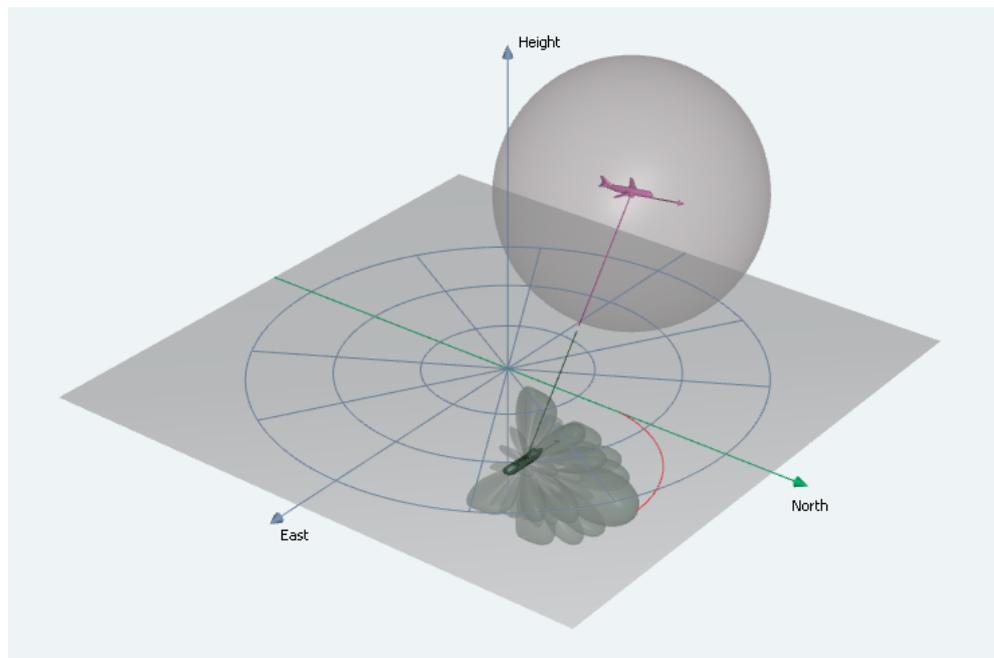
The power level at the receiver varies due to the receiver's antenna pattern and scan. The free-space attenuation over the dynamically changing distance between the emitter and the receiver also affects the level.

When interpreting the results, consider the type of receive antenna and the emitter's trajectory and altitude. A low received power level does not always correspond to a long distance between emitter and receiver. A receive antenna with high directivity and gain only provides a high signal power if the emitter is in the main beam. If the emitter overflies the receiver, you can expect large sudden changes in the signal level. If the receive antenna is omni-directional, you can expect a smooth rise and fall in level as the emitter passes by.

8. Calculate the signal. Observe the generated signals:

   a) In the scenario dialog, select "Signal Calculation > Config > Config > Signal Calculations Settings > Output > Use extended sequencing option".
   b) Select "Signal Generation > Target > File".
   c) Select "Signal Calculation > Calculate".
   d) Select, for example "Signal Generation > Results > View > Ground".
   e) In the "Signal Preview" dialog, select "Frequency".

   In this example, the receiver and the emitter (Ground) are moving objects, where the receiver approaches the emitter and later departs from it.
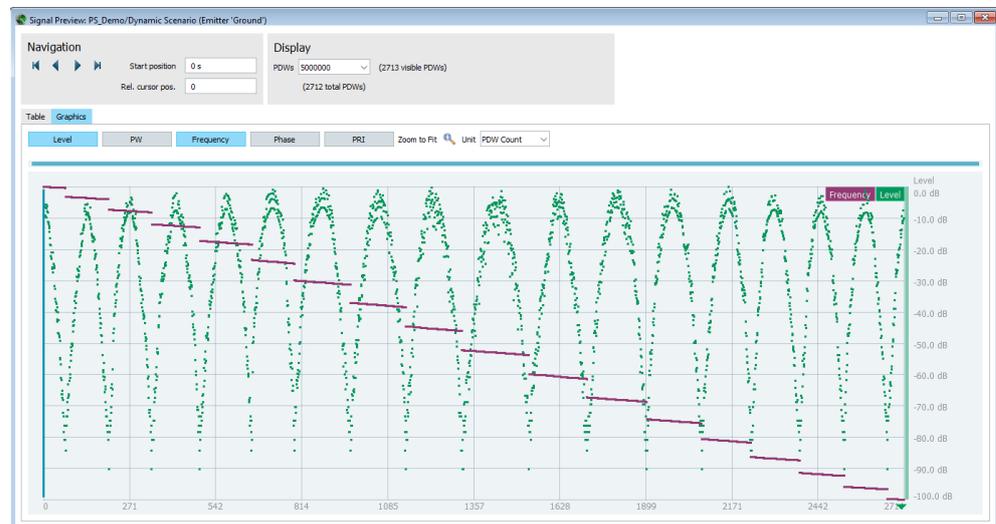
The "Frequency" and "Level" displays confirm the expected frequency variation caused by the Doppler effect. The Doppler is a positive value during the approaching phase and a negative one during the time the receiver is flying away from the emitter.

16.2   2D map settings

Access:

► Perform one of the following:
- In a "Scenario Type = Localized Emitters", select "Map".
- In a "Scenario Type = Direction Finding", select "Map".

The name of the dialog changes depending on the scenario but the settings and the workflow are similar.
The 2D diagram displays the configuration on a map with North/East coordinates.
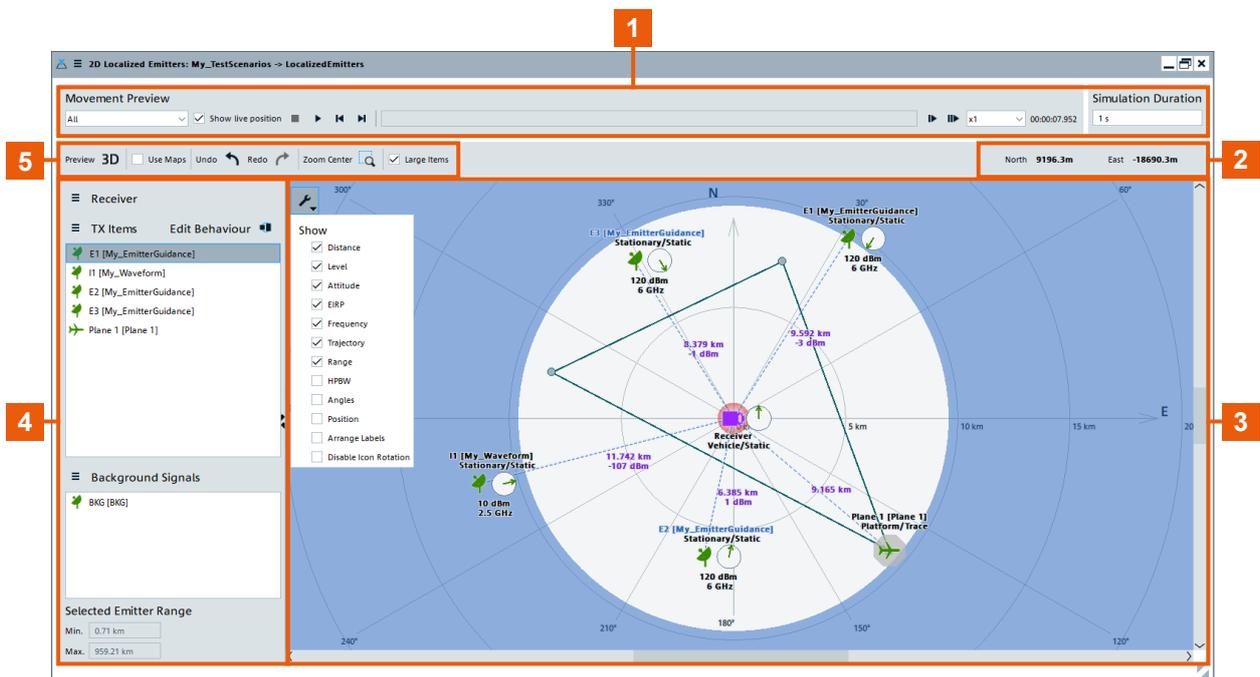
Figure 16-5: 2D view: understanding the displayed information

1 = Simulation tools and information
2 = Cursor coordinates
3 = Visualization area
4 = Receiver/TX Items
5 = Display controls

See also Chapter 16.5, "Moving emitters/interferers and platforms", on page 450.

Settings:

Movement Preview

Use the drop-down menu to select the trajectory of a specific platform. Selecting "All" displays the trajectories of all platforms.

"Show live position"

> Activates the dialog "2D Live Preview Stats" on page 425.

"Start/Stop, Pause, Move to Start/End"

> Standard play functions.

"Time Line"

> Indicates the current time of the simulation.

"Play slower/faster"

> Switches to the next/previous value of the time scale coefficient.

"Time Scale Coefficient"

> Applies a coefficient to speed up or slow down the display.

Position Step

Available if an emitter with "Position > Mode = Steps" is present. You can select a specific step for the emitter on the 2D map.

Simulation Duration

Available if a fixed simulation time is defined for the selected scenario, see Clock, Duration.

2D Live Preview Stats

If "Show live position" is activated and a preview is started ("Play"), displays a live track of the positions of the TX items and RX items.

Closing the dialog does not interrupt the movement preview.

Remote command:
PREView:POSition? on page 780

Undo/Redo

Reverses the drag&drop operation on the 2D map.

Pan, Zoom
- Pan: Use the mouse to move the displayed map area.
- Zoom in/out: Turn the mouse wheel to zoom in or out.
- Zoom center: Press the zoom icon.

Large Items

Activates the larger size for items displayed in the visualization area.

Visualization settings > Distance, Level, Attitude, EIRP, Frequency, Trajectory, Range, HPBW, Angles, Position, Arrange Labels

To change the displayed information, enable or disable the corresponding parameter.

"Range"　　　　If enabled, indicates the dynamic range for the simulation, numerically and as a color-coded indication.
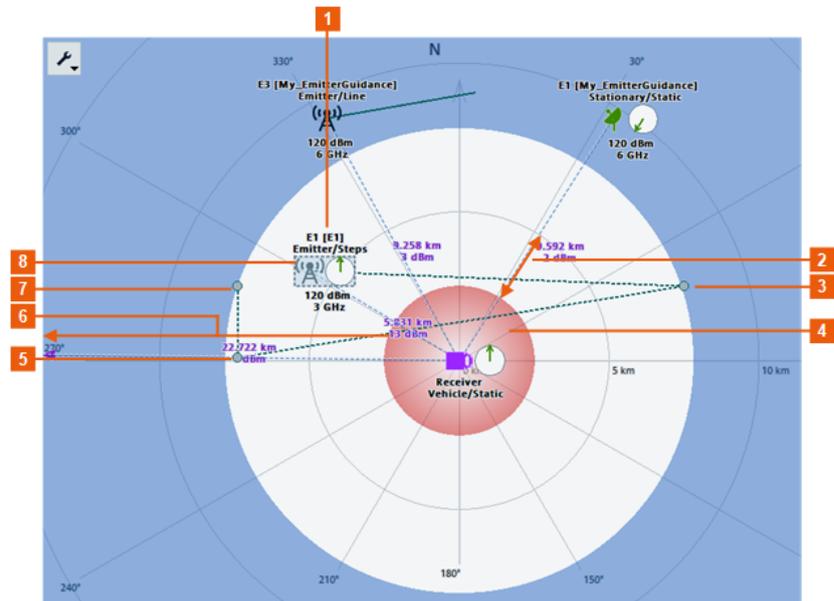
Figure 16-6: 2D map > Visualization > Range: Understanding the displayed information

1　　　　= Selected emitter (Emitter 1, movement = position steps)
2　　　　= Unused area, where the radius (outer border) is the minimum distance between the receiver and the selected emitter; in this example, the position of step 1
3, 5, 7,　= Four position steps, emitter currently displayed at position step 3
8
4　　　　= Minimum distance from the receiver that can be simulated for the selected emitter
6　　　　= Dynamic range of the signal generator (outer border outside of the displayed area)

- Red background
  Range where the maximum output power of the simulated signal generator is exceeded.
- Blue background
  Range where you can place an emitter or emitters can move and the signal generator generates the signal in its dynamic range.
  – Outer border = lowest output power that can be generated for the selected emitter and position mode.
  – Inner border = minimum distance between the receiver and the selected emitter.
    "Position > Mode = Static", the position of the emitter
    "Position > Mode = Steps/Moving", among all steps/trajectory points, the position of the closest to receiver one

If an emitter moves beyond the outer-border of the blue area, the signal becomes too weak to be generated.
If emitters are too close to the receiver and enters the red area, the output power of the signal generator can be insufficient to

generate such high power. In such cases, the power levels clipped to the value specified in the signal generation settings (see "Clipping Level" on page 478).

● White background
Unused area on the map, showing the cross section between the red and blue area.

"Angels"   If enabled, displays the azimuth and elevation angles in a label near the emitter.

"Position"   If enabled, displays the positions in terms of north/east coordinates and height.

"Arrange Labels"

If enabled, moves the labels to avoid any overlapping.

Platform, Emitter/Interferer, Properties
Double-click the "Platform/Emitter/Interferer" name to access the "Emitter/Interferer Properties" dialog, see Chapter 16.4.1, "Available TX items", on page 435.

Signal Assigment > Assignment Editor
Accesses the "Assign Signal Source to Generator Config" dialog.

See Chapter 17.2, "Assign signal source or receiver signals to generator", on page 481.

Background Signals
Displays the sequences or the background emitters that compose the background signal.

To add a background emitter, drag and drop a waveform in the block.

Remote command:
See `SCENario:LOCalized:SEQuence` on page 752.
See `SCENario:DF:SEQuence` on page 752.

Selected Emitter Range
Displays the minimum and maximum distance of the selected emitter to the receiver.

Strength of the received signal
The displayed value is the reference value used to calculate the power at the receiver antenna output. It is the virtual power level at the receiver output for two *perfectly aligned isotropic antennas* of emitter and receiver. Hence, this value is the maximum signal level that can be expected at the receiver antenna output.

In practice, the received signal is weaker than the displayed signal strength. The following parameters influence the signal strength:
● Distance between the emitter and the receiver, and the resulting free-space attenuation.
● Alignment of the antennas and used antenna patterns
● Antenna gain

Platform/Emitter/Interferer position on the 2D map and distance to the receiver
The 2D map indicates the current distance, calculated from the current platform/emitter/interferer position on the 2D map.

To change the current position of the emitter/interferer, use one of the following possibilities:
- Drag the platform/emitter/interferer to the new position on the 2D map.
- Open its context menu, select "Properties > Position" and change the coordinates, the "Attitude" and the "Distance".
  If several position steps are defined, the changes apply to the current step ("Current Step"), see Discrete position steps.

16.3 3D scan pair view settings

Access:

1. Perform one of the following:
   - In a scenario type "Localized Emitters", select "Map" > " 3D"
   - In a scenario type "DF" (Direction Finding), select "Map" > " 3D"

2. In the 2D map, perform one of the following:
   - In one of the lists, select an emitter/interferer or platform > "Properties" > "3D"
   - In the map, right-click an emitter/interferer or platform > context menu > "3D View with Receiver".

The "3D Scan Pair View" appears.

If you loaded a georeferenced map in the 2D view, it also appears in the 3D view.

The available settings depend on the scenario:
- A simple scenario containing an emitter and a receiver (each with only one antenna) offers fewer settings.
- A complex scenario containing moving platforms (each with multiple emitters) and a scanning DF receiver (with multiple antennas) offers more settings.

However, if you understand how your emitters are configured, the differences are self-explanatory.
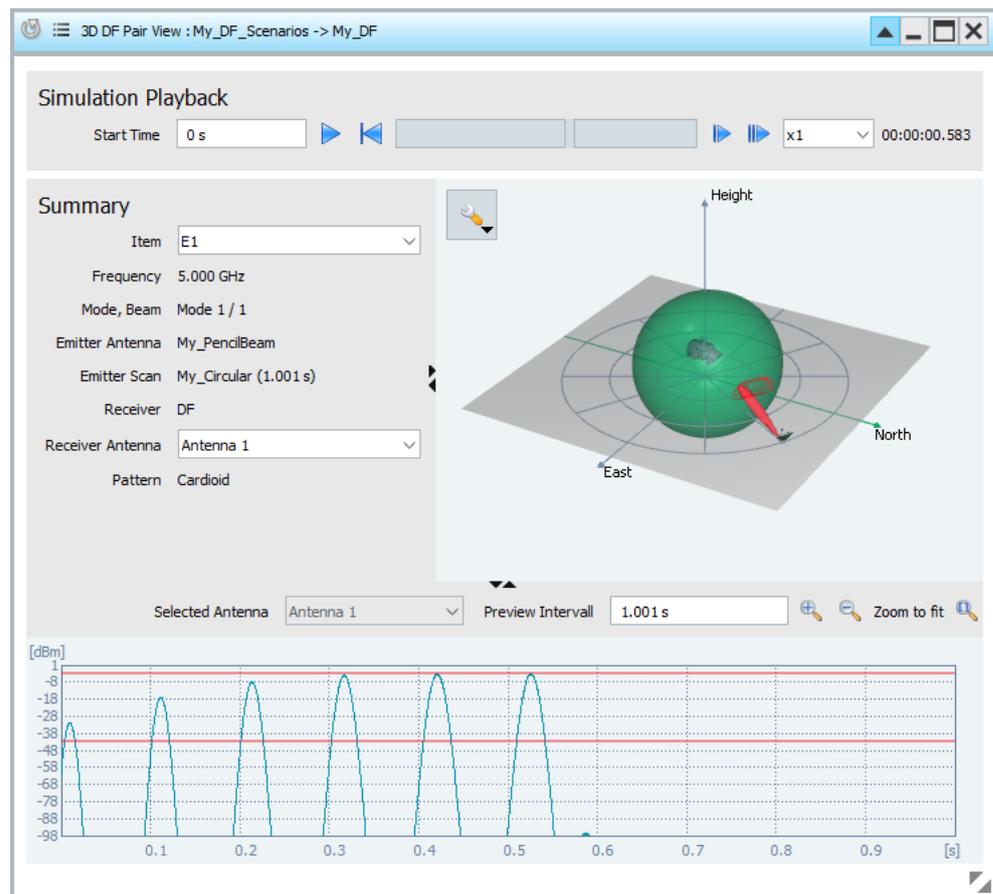
Figure 16-7: Example: 3D Scan Pair View display, Scenario Type = DF

This dialog displays:
- A live plot of the normalized signal power level at the receiver.
- A 3D view of the receiver and emitter antennas with their patterns and scans.
- In "Direction Finding" scenario, the plot and the 3D view correspond to the selected receiver antenna element.

Settings:

Simulation Playback

Controls the timing of the simulation.

For example, with these settings you can analyze when/where/how often the beams overlap while simulating:

- A static scanning RX and a static scanning TX with different scan cycles.
- A moving RX and static TX
- A moving TX with different modes and a static RX

Figure 16-8: Simulation Playback

In addition to setting the overall duration, you can also observe a particular time-segment of the simulation by starting and stopping at particular times.

Note: The duration of a simulation depends on the scenario. An emitter with a long trajectory and low speed causes a long duration. A short trajectory and/or high speed cause a short duration.
The duration provides the maximum value for the "Stop Time" and "Preview Interval" settings.

"Start Time"　　　Set to *0 s* or *minimum*, to play the simulation from the beginning. Typically, set this value to begin the simulation at a particular point along the trajectory.

Buttons　　　Control the simulation using the following buttons:

- Pause/Play - use this toggle to pause the simulation and resume playing.
- Restart - use this button to restart the simulation at the configured "Start Time".
- Play slower/Play faster - decrease/increase the "Speed-up factor"

Summary (live plot and 3D view configuration)
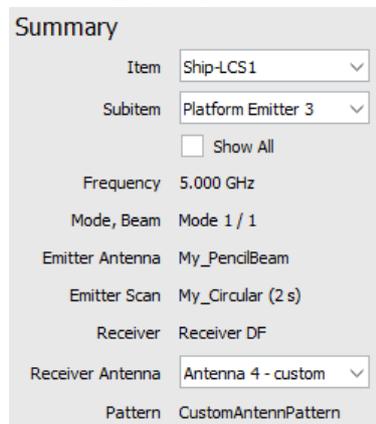
Figure 16-9: Summary

The "Summary" provides the following information fields and drop-down lists:

"Item"　　　Lists all emitters/interferers and platforms of the selected scenario. Select the item that you want to use for the simulation.
If you select a platform, see "Subitem" for a list of the emitters on the platform.

"Subitem"　　　Lists all emitters on the selected platform. Select the emitter that you want to use for the simulation.

"Frequency"　　　Shows the frequency used by the selected emitter.

"Mode, Beam"   Shows the active mode and beam combination (continuously updated) during the simulation.

This information is useful when:
- The selected emitter has more than 1 mode.
- The active mode has more than one beam.

"Emitter Antenna"
Shows the "Antenna Pattern" used by the selected emitter.

"Emitter Scan"   Shows the name of the "Antenna Scan" used by the selected emitter.

"Receiver"   Shows the name of the selected emitter.

"Receiver Antenna"
Lists all antennas configured for the selected receiver.
Select the antenna that you want to use for the simulation.

"Pattern"   Shows the "Antenna Pattern" used by the selected "Receiver Antenna".

Settings

Access: click the visualization settings icon to open the settings dialog.

Defines the way that the scan is represented:

"Scan Line"   The RX and TX beams are represented by lines.

"Pattern"   The RX and TX beams are represented by antenna patterns.

"HPBW"   The RX and TX beams are represented by colored eclipses. The size of an eclipse indicates the half power beam width HPBW.

"Show Line of Sight"
Displays the line of sight (LOS) between the emitter and the receiver.

"North"

When zoomed/zooming, the view is focused on the center of the "North" line.
Typically used with maps. Helps you maintain an awareness of the map's orientation.

"Track Emitter"

When zoomed/zooming, the view is focused on the emitter.
Typically used with moving emitters. The view stays focused on the
emitter as it moves.

"Mouse Click"

When zoomed/zooming, the view is focused on a user-defined
marker. Use the left mouse button to put a marker on the map. Ena-
bles you to decide the focus of the view when zoomed/zooming.

Preview graphic settings
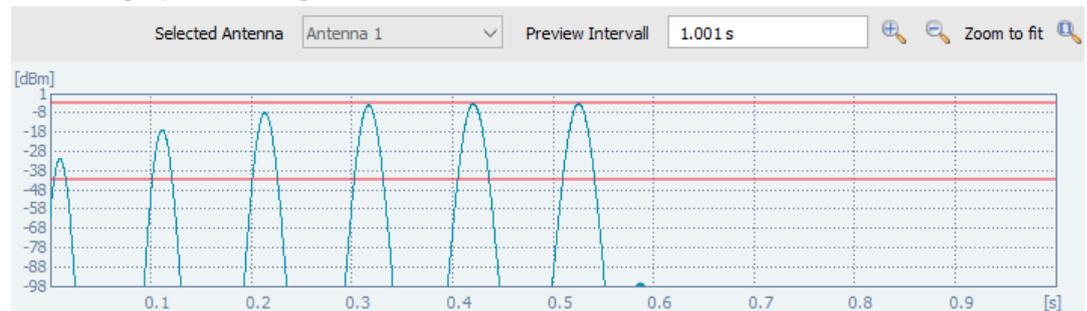
Figure 16-10: Settings for the preview graphic

Define which signal is visualized in the graphic.

See also "How to create the scenario and configure the receiver" on page 418.

"Selected Antenna"

In "Direction Finding" scenario and if "Receiver Antenna > Show that
All Antennas" is selected, switches between the receiver antenna ele-
ments.

"Preview Interval"

Sets the time interval shown on the preview at the bottom of the view.
The horizontal axis of the preview corresponds to the configured
"Preview Interval".
Typically, set to a multiple of the TX scan cycle time, to show several
sweeps in one preview. For example, for an emitter with a two-sec-
ond cycle, an interval of 10 seconds shows 5 sweeps.

Note: "Preview Interval" is the time represented by the horizontal axis
of the preview. It is not the time taken to fill the preview.
"Preview Duration" is the time taken to fill the preview, which is calcu-
lated from your settings, automatically.

"Zoom in", "Zoom out" and "Zoom to fit"

These icons refer to the preview graphic settings time axis:

16.4 Properties of TX items (emitters/interferers and platforms)

The "TX Items" dialog provides access to the properties of all the emitters/interferers and platforms in the current scenario.

Access:

1. Open a suitable scenario.

   Use any scenario that offers "Map" in the "Scenario Creation" block.
   ● "Localized Emitters"
   ● "Direction Finding"

2. In the block diagram, select "Map".

   The "2D" dialog displays a 2D view of the receiver and currently configured emitters/interferers, platforms, together with their main characteristics.

3. In the "2D" dialog, use one of the following to access the "TX Items Properties" dialog:

   ● "Properties" list > "TX Items" link
   ● Map
     – Double-click a platform/emitter/interferer icon
     – Right-click a platform/emitter/interferer icon > context menu > "Properties"
   ● "TX Items" list
     – Double-click an entry
     – Right-click an entry > context menu > "Properties"

The "TX Items Properties" dialog opens.
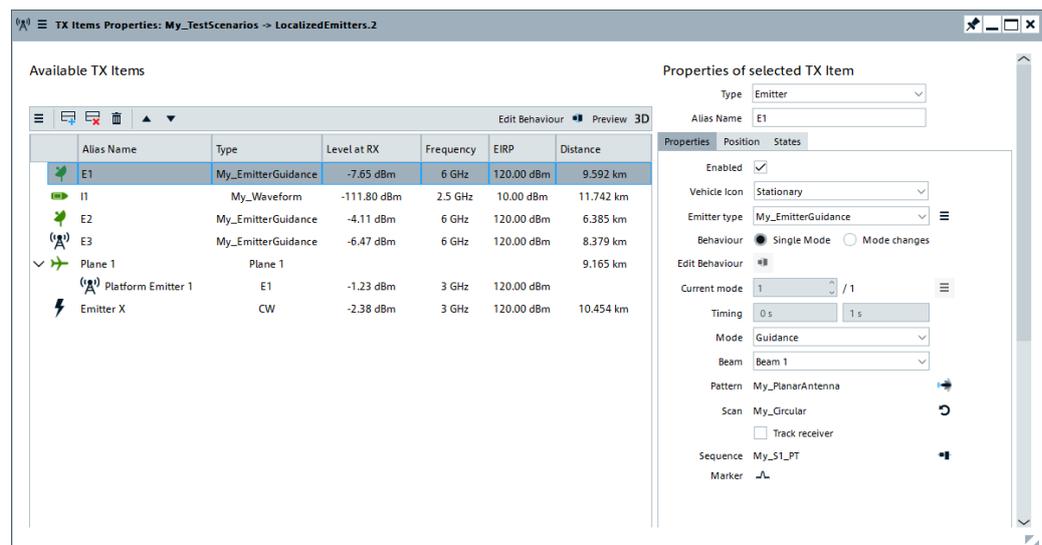
Figure 16-11: TX Items Properties dialog example

As Figure 16-11 shows, this dialog contains 2 parts:

- "Available TX Items" (left)
  Select the item that you want to configure.
- "Properties of selected TX Item" (right)
  This part contains 3 tabs. Use the appropriate tab to configure the "Properties", "Position" or "States" of the selected item.

16.4.1 Available TX items

Access:

► In the "2D" dialog, use one of the following to access the "TX Items Properties" dialog:

- "Properties" list > "TX Items"
- Map
  - Double-click a platform/emitter/interferer icon
  - Right-click a platform/emitter/interferer icon > context menu > "Properties"
- "TX Items" list
  - Double-click an entry
  - Right-click an entry > context menu > "Properties"

Append, Delete, Clear, Move Up/Down
Use the standard functions of these buttons to add, reorder, or remove items.

Preview 3D
Opens the "3D Preview" for the selected "Emitter".

See Chapter 16.3, "3D scan pair view settings", on page 429.

Edit Behaviour
Accesses the "Mode Editor" dialog.

The "Mode Editor" is a graphical editor for jointly configuring and defining the duration emitters are working in the different modes.

It works like the Mode Changes settings for each of the emitters but rather than defining the start and end time of a mode, you drag the start and end time on the time scale.

In the dialog, you can configure the mode changes of all emitters available in the scenario.

For a description, see Chapter 10.3.3, "Mode editor settings", on page 317.

List of available TX items
Overview information of the available TX items in table form.

"Alias Name"   Displays an alias name.

"Type"   Displays the type of TX item.

"Level at RX"

Indicates the maximum level at the receiver origin.
The calculation considers the receiver EIRP and the free-space
attenuation. Individual antenna attenuation due to positioning and
polarization is omitted.

Remote command:
SCENario:LOCalized:LEVel on page 759
SCENario:DF:LEVel on page 759

"Frequency"   As set with the parameter Frequency.

Remote command:
SCENario:LOCalized:FREQuency? on page 753
SCENario:DF:FREQuency? on page 753

"EIRP"   As set with the parameter EIRP or with the parameter "EIRP".

"Distance"   The distance from the TX item to the receiver.
As set with the East, North, Height/Distance, Azimuth, Elevation
parameters.

Remote command:
SCENario:LOCalized:ADD on page 591
SCENario:LOCalized:SELect on page 592
SCENario:LOCalized:CURRent on page 746
SCENario:LOCalized:DELete on page 595
SCENario:LOCalized:CLEar on page 596
SCENario:DF:ADD on page 591
SCENario:DF:SELect on page 592
SCENario:DF:CURRent on page 746
SCENario:DF:DELete on page 595
SCENario:DF:CLEar on page 596

16.4.2 Properties of the selected TX item

Access:

1. In the "2D" dialog, use one of the following to access the "TX Items Properties"
   dialog:

   ● "Properties" list > "TX Items"
   ● Map
     – Double-click a platform/emitter/interferer icon
     – Right-click a platform/emitter/interferer icon > context menu > "Properties"
   ● "TX Items" list
     – Double-click an entry
     – Right-click an entry > context menu > "Properties"

2. In the list of "Available TX Items", select a platform/emitter/interferer.
   For details on the "Properties of Selected TX Item" area, see the following sub-
   chapters.

In "Properties of selected TX Item", not all TX items support all tabs.

See Table 16-1.

Table 16-1: TX items and tabs

"TX Item"	"Properties" tab	"Position" tab	"States" tab	Remarks
Platform	●	●	○	States depend on emitters. See Chapter 16.4.2.3, "States", on page 448.
Platform emitter	●	○	●	Position depends on the plat-form. See Chapter 16.4, "Properties of TX items (emitters/interferers and platforms)", on page 434.
Emitter	●	●	●	Independent emitter. See Chapter 16.4, "Properties of TX items (emitters/interferers and platforms)", on page 434.
Interferer	●	●	●	Independent interferer. See Chapter 16.4, "Properties of TX items (emitters/interferers and platforms)", on page 434.

The selected "TX Item" is described with the following parameters:

16.4.2.1 Properties (type, alias name and configuration)

The content of the "Properties" tab depends on which TX item you select. Not all the
configuration options display for all TX items.

Access:

1. In the list of "Available TX Items", select a platform/emitter/interferer.

2. Select "Properties of selected TX Item" > "Properties" tab.

Type

Defines whether an emitter/interferer is configured.

Remote command:

SCENario:LOCalized:TYPE on page 752

SCENario:DF:TYPE on page 752

Alias Name

Enters an alias name.

Remote command:

SCENario:LOCalized:ALIas on page 746

SCENario:DF:ALIas on page 746

Emitter/Interferer Name

Selects a platform/emitter/interferer.

Remote command:

SCENario:LOCalized:EMITter on page 748

SCENario:LOCalized:WAVeform on page 753

SCENario:DF:EMITter on page 747

SCENario:DF:WAVeform on page 753

Enabled

Enables selected item for calculation.

Remote command:

SCENario:DF:EMITter:ENABle on page 748

SCENario:LOCalized:EMITter:ENABle on page 748

Platform type

Defines or edits the platform assigned to the selected "TX Item".

Remote command:

SCENario:LOCalized:EMITter on page 748

SCENario:DF:EMITter on page 747

Vehicle Icon

Assign different icons to the emitters/interferers on the 2D map.

The icons are merely for display; the function and the properties of the emitters do not depend on the icon.

Remote command:

SCENario:LOCalized:MOVement:VEHicle on page 777

SCENario:DF:MOVement:VEHicle on page 777

Behaviour

Enables emulation of emitters that change their mode over time.

For more information on emitter modes, see "Operational mode" on page 305.

To visualize the effect of the mode switching, select "2D map > Emitter (with several modes) > context menu > 3D View with Receiver".

You can define and configure the duration of the modes in one of the following ways:
● With the settings:
  – "▤ Append, Remove Current, Remove All" on page 440
    "Current Mode" on page 440
    "Timing" on page 440
● With the graphical editor "Edit Modes", see Chapter 10.3.3, "Mode editor settings", on page 317.

Single Mode ← Behaviour

The emitter works in one constant mode, as selected with the "Mode" and "Beam" parameters during the whole scenario.

This setting applies even for emitters for which you defined different modes, possibly each with several beams, in the Emitter dialog.

Remote command:

SCENario:CEMit:MCHG:STATe 0

SCENario:LOCalized:MCHG:STATe 0

SCENario:DF:MCHG:STATe 0

Mode Changes ← Behaviour

If selected, you can define different emitter modes and their timing.

Each mode lasts a defined duration. Once it elapses, the subsequent mode is used.

Use the "Mode changes" function to simulate more realistic scenarios where emitters do not use one fixed mode but change their beam and antenna settings over time.

Remote command:

SCENario:CEMit:MCHG:STATe on page 756

SCENario:LOCalized:MCHG:STATe on page 756

SCENario:DF:MCHG:STATe on page 756

Edit Behaviour ← Behaviour

Accesses the "Mode Editor" dialog.

The "Mode Editor" is a graphical configuration for defining the duration the *selected emitter* is working in the different modes.

Works like the Mode Changes settings but rather than defining the start and end time of a mode, you drag the start and end time on the time scale.

Current Mode ← Behaviour
Selects the entry (mode) for which the displayed and configured settings apply.

The second number indicates the number of enabled modes.

Remote command:
SCENario:CEMit:MCHG:SELect on page 593
SCENario:LOCalized:MCHG:SELect on page 593
SCENario:DF:MCHG:SELect on page 593
SCENario:CEMit:MCHG:COUNt? on page 592
SCENario:LOCalized:MCHG:COUNt? on page 592
SCENario:DF:MCHG:COUNt? on page 592

▤ **Append, Remove Current, Remove All ← Behaviour**
Standard functions for entry handling.

Remote command:
SCENario:CEMit:MCHG:ADD on page 591
SCENario:LOCalized:MCHG:ADD on page 591
SCENario:DF:MCHG:ADD on page 591
SCENario:CEMit:MCHG:DELete on page 595
SCENario:LOCalized:MCHG:DELete on page 595
SCENario:DF:MCHG:DELete on page 595
SCENario:CEMit:MCHG:CLEar on page 596
SCENario:LOCalized:MCHG:CLEar on page 756
SCENario:DF:MCHG:CLEar on page 596

Timing ← Behaviour
Sets the duration of the mode, defined as start and end time.

Where:
● The start value of one entry must be smaller than or equal to the end value.
● The start value of a subsequent entry has to be greater than the end value of the previous one.

Tip: Configure the modes backwards: configure the last mode first, set its end time first.

Remote command:
SCENario:CEMit:MCHG:STARt on page 756
SCENario:CEMit:MCHG:STOP on page 756
SCENario:LOCalized:MCHG:STARt on page 756
SCENario:LOCalized:MCHG:STOP on page 756
SCENario:DF:MCHG:STARt on page 756
SCENario:DF:MCHG:STOP on page 756

EIRP
Sets the EIRP.

Remote command:
SCENario:LOCalized:WAVeform:EIRP on page 753
SCENario:DF:WAVeform:EIRP on page 753

Frequency
Sets the frequency of the TX item.

Remote command:
SCENario:LOCalized:WAVeform:FREQuency on page 753
SCENario:DF:WAVeform:FREQuency on page 753

Mode
Selects the mode that the emitter is working in.

Remote command:
SCENario:LOCalized:EMITter:MODE on page 748
SCENario:DF:EMITter:MODE on page 748

Beam
Sets the number of currently used beams.

Remote command:
SCENario:LOCalized:EMITter:MODE:BEAM on page 748
SCENario:DF:EMITter:MODE:BEAM on page 748

Antenna Pattern, Scan, Sequence
Displays the currently selected antenna pattern, antenna scan and sequence.

To change any of them, select the corresponding icon.

Track Receiver
If enabled, the scan follows the receiver automatically.

Remote command:
SCENario:LOCalized:EMITter:MODE:TRACkrec on page 749
SCENario:DF:EMITter:MODE:TRACkrec on page 749

Emitter Marker
Available in map-based and emitter collection scenarios.

Enables the configuration of emitter markers in the "Emitter Marker Config" dialog.

You can assign up to 4 markers for an emitter, for example rise and fall time, pulse width and gate.

Select the required mode:
- "Unchanged"
  Leaves the marker unchanged, as defined in the pulses and sequences settings of this emitter.
- "Force Marker"
  Forces the selected marker type for every pulse from this emitter.

The gate marker has the same width as the pulse.

A restart marker indicates the first 10% of the entire PRI.

For pre and post markers, set their timing in the corresponding fields.

For details of other types of marker, see Chapter 21, "Defining and enabling marker signals", on page 534.

Remote command:

SCENario:CEMit:MARKer:FORCe on page 681

SCENario:CEMit:MARKer:RISE on page 682

SCENario:CEMit:MARKer:WIDTh on page 682

SCENario:CEMit:MARKer:FALL on page 682

SCENario:CEMit:MARKer:AUTO on page 683

SCENario:CEMit:MARKer:GATE on page 683

SCENario:CEMit:MARKer:PRE on page 683

SCENario:CEMit:MARKer:TIME:PRE on page 684

SCENario:CEMit:MARKer:POST on page 684

SCENario:CEMit:MARKer:TIME:POST on page 684

SCENario:DF:MARKer:FORCe on page 681

SCENario:DF:MARKer:RISE on page 682

SCENario:DF:MARKer:WIDTh on page 682

SCENario:DF:MARKer:FALL on page 682

SCENario:DF:MARKer:AUTO on page 683

SCENario:DF:MARKer:GATE on page 683

SCENario:DF:MARKer:PRE on page 683

SCENario:DF:MARKer:TIME:PRE on page 684

SCENario:DF:MARKer:POST on page 684

SCENario:DF:MARKer:TIME:POST on page 684

SCENario:LOCalized:MARKer:FORCe on page 681

SCENario:LOCalized:MARKer:RISE on page 682

SCENario:LOCalized:MARKer:WIDTh on page 682

SCENario:LOCalized:MARKer:FALL on page 682

SCENario:LOCalized:MARKer:AUTO on page 683

SCENario:LOCalized:MARKer:GATE on page 683

SCENario:LOCalized:MARKer:PRE on page 683

SCENario:LOCalized:MARKer:TIME:PRE on page 684

SCENario:LOCalized:MARKer:POST on page 684

SCENario:LOCalized:MARKer:TIME:POST on page 684

16.4.2.2 Position settings (static position and discrete position steps)

Access:

1. On the 2D map, select "Properties > TX Items".

2. In the list of "Available TX Items", select an emitter.

3. Select "Propertiers of the Selected Emitters/Interferers > Position"

   The emitters/interferers on the 2D map are static or moving elements with configurable coordinates.

The platform/emitter/interferer position can be defined as:

- East/North/Height coordinates relative to the origin
- Azimuth/Distance/Elevation angular positions relative to the origin
- Longitude/Latitude/Altitude absolute coordinates (requires georeferenced map)

The definitions are interdependent.

Mode
Sets if the emitter is static or moving.

"Static"	The emitter is static; its coordinates are fixed during the whole scenario. See East, North, Height/Distance, Azimuth, Elevation.
"Steps"	You can define several static positions for each emitter/interferer. See "Discrete position steps" on page 446.
"Moving"	Emitters/Interferers are moving objects, defined with their start position and trajectory. See "Fill Position Steps" on page 447. See Chapter 16.9, "Creating trajectories on a 2D map", on page 468.

Remote command:

SCENario:DF:LOCation:PMODe on page 771

SCENario:LOCalized:LOCation:PMODe on page 771

Longitude, Latitude, Altitude
Sets the position of the selected emitter using absolute coordinates (requires georeferenced map).

Remote command:

SCENario:DF:LOCation:LONGitude on page 772

SCENario:DF:LOCation:LATitude on page 772

SCENario:DF:LOCation:ALTitude on page 772

SCENario:LOCalized:LOCation:LONGitude on page 772

SCENario:LOCalized:LOCation:LATitude on page 772

SCENario:LOCalized:LOCation:ALTitude on page 772

East, North, Height/Distance, Azimuth, Elevation
To change the location of an element:
- Drag this element on the 2D map.
- Use the "Position" parameters to define the location more precisely.

Example:
- An emitter with the East/North coordinates "East = 0 m", "North = 7000 m" is placed on the North, i.e. the "Azimuth Angle = 0°".

Location parameters	Map view
East 0 m North 7000 m Height 0 m Distance 7000 m Azimuth 0 ° Elevation 0 °	
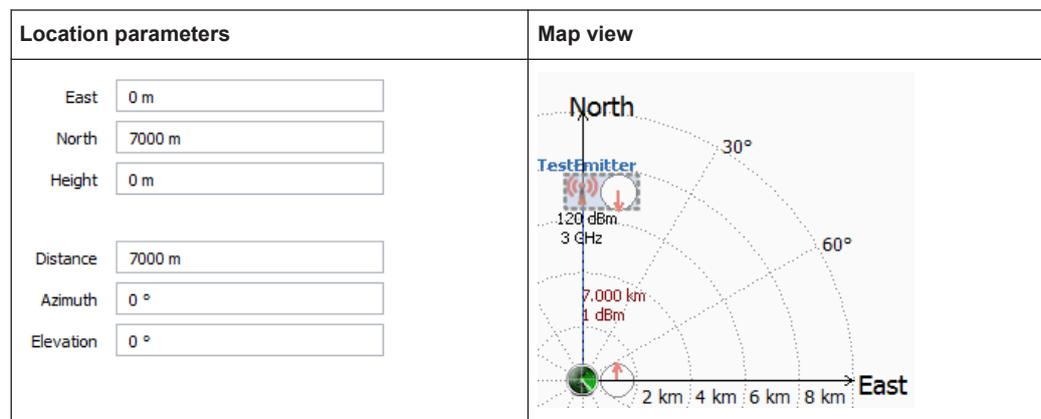
- If the "Azimuth Angle = 90°", the emitter is placed on the east axis, i.e. "North = 0 m" and "East = 7000 m".

Location parameters	Map view
East: 7000 m North: 0 m Height: 0 m Distance: 7000 m Azimuth: 90 ° Elevation: 0 °	

Remote command:

SCENario:LOCalized:LOCation:EAST on page 750

SCENario:LOCalized:LOCation:NORTh on page 750

SCENario:LOCalized:LOCation:HEIGht on page 751

SCENario:LOCalized:DISTance on page 749

SCENario:LOCalized:LOCation:AZIMuth on page 750

SCENario:LOCalized:LOCation:ELEVation on page 751

SCENario:DF:LOCation:EAST on page 749

SCENario:DF:LOCation:NORTh on page 749

SCENario:DF:LOCation:HEIGht on page 751

SCENario:DF:DISTance on page 749

SCENario:DF:LOCation:AZIMuth on page 750

SCENario:DF:LOCation:ELEVation on page 751

Attitude

Defines the orientation of the antennas as "Pitch", "Yaw", and "Roll".

Figure 16-12: Yaw: understanding the displayed information

1 = Yaw is the angle between the antenna's boresight and north

If "Point to Receiver" is enabled, the antenna is automatically turned in the direction of the receiver.

See also "Emitter attitude" on page 305.

Remote command:
SCENario:LOCalized:DIRection:TRACk on page 749
SCENario:LOCalized:DIRection:PITCh on page 747
SCENario:LOCalized:DIRection:YAW on page 746
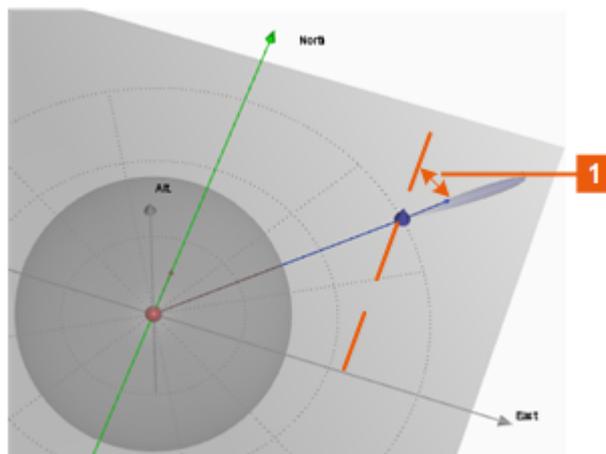SCENario:LOCalized:DIRection:ROLL on page 747
SCENario:DF:DIRection:TRACk on page 749
SCENario:DF:DIRection:PITCh on page 747
SCENario:DF:DIRection:YAW on page 746
SCENario:DF:DIRection:ROLL on page 747

Discrete position steps
In "Scenario Type > Localization/Direction Finding", you can define a sequence of **discrete static positions** (steps) to simulate the position change of the emitter/interferer. With **"Mode > Steps"**, the emitter is not moving but it changes its position in terms of **coordinates, height and attitude**.

The time duration that the emitter remains at a particular position is defined depending on the selected "Scenario > Signal Calculation > Config > Config > Signal Calculation Settings" > "Duration", see Table 16-2.

Table 16-2: Time duration per position

"Duration"	**"Antenna Scan"**	**"Mode Change"**	**Time duration (or waveform content) *per position***
"Auto"	Steady (no scan)	Static	One sequence
	Any scan type	Static	One antenna scan
	Any, incl. no scan	Static/Mode Change	All defined modes
"Fixed"	Any, incl. no scan	Static/Mode Change	Time specified with the parameter "Duration > Fixed" divided by the number of steps
			Longer sequences or scans are truncated; shorter repeated
			Longer modes are truncated; shorter are not repeated

Note: To prevent that transitions (or jumps) over large distances lead to pulses overtaking one another, distance-related signal propagation delays are not considered.

Option: R&S SMW-K501/K502: Required to play the signal. The output mode of the waveform generator is selected automatically, see Output.

Current Entry/Number of Entry ← Discrete position steps
Selects the position step for that the displayed position coordinates apply. This is also the position indicated on the 2D map.

The second number indicates the number of available static positions.

Remote command:
SCENario:DF:LOCation:PSTep:SELect on page 593
SCENario:DF:LOCation:PSTep:COUNt? on page 592

SCENario:LOCalized:LOCation:PSTep:SELect on page 593
SCENario:LOCalized:LOCation:PSTep:COUNt? on page 592

☰ Append, Remove Current, Remove All ← Discrete position steps
Standard functions for entry handling.

Remote command:
SCENario:DF:LOCation:PSTep:ADD on page 591
SCENario:DF:LOCation:PSTep:SELect on page 593
SCENario:DF:LOCation:PSTep:DELete on page 595
SCENario:LOCalized:LOCation:PSTep:ADD on page 591
SCENario:LOCalized:LOCation:PSTep:SELect on page 593
SCENario:LOCalized:LOCation:PSTep:DELete on page 595

Auto fill ← Discrete position steps
Opens the "Fill Position Steps" dialog.

See "Fill Position Steps" on page 447.

☰ Import list X,Y,Z / Dist., Azim., Elev. [Y, P, R] ← Discrete position steps
To define the static positions one by one, you can also import the positions from list
files. Lists are simple text files with extension *.txt. You can create the lists with any
text editor, for example, Notepad.

The list contains the coordinates of each position and (optionally) the yaw, pitch, roll [Y,
P, R] values. Position coordinates can be specified in one of the following formats:
- X, Y, Z: East, North, Height
- Dist., Azim., Elev.: Distance, Pitch, Roll

The coordinates of two subsequent positions are separated by line break.

Fill Position Steps
"Fill Position Steps" is a wizard, where you can define steps automatically, based on
start and end position and the number of static positions. Static positions can be
placed on a straight line or on a circle.

Position Steps > 2D Line ← Fill Position Steps
When distributed on a straight line, the static positions are defined with:
- "Start East/North" coordinates
- "Stop East/North" coordinates
- "Steps" defines the number of static positions between the start and the stop position; positions are spaced equidistant.

Position Steps > 2D Circle ← Fill Position Steps
When distributed on a circle, the static positions are defined with:
- "Distance" sets the circle radius, where the circle center is at the receiver.
- "Start Angle" sets the angle of the start position, where "Start Angel = 0°" corresponds to North.
- "Increment" defines the step size with that the angle is incremented.
- "Steps" defines the number of static positions between the start and the end position; positions are spaced equidistant, where the step size is defined with "Increment".

Attitude > Pointing, Yaw, Pitch, Roll ← Fill Position Steps
Defines the orientation of the antennas:
- "Pointing = Static", the orientation is given as "Pitch", "Yaw", and "Roll".
- "Pointing = to Receiver/ to Front": the antenna attitude is adapted automatically at the specified direction and depending on the current static position.
  This setting is useful if the static positions are distributed on a circle.

Append ← Fill Position Steps
Adds the automatically created positions at the end of the current positions' list.

Ok ← Fill Position Steps
Confirms the settings and apples them.
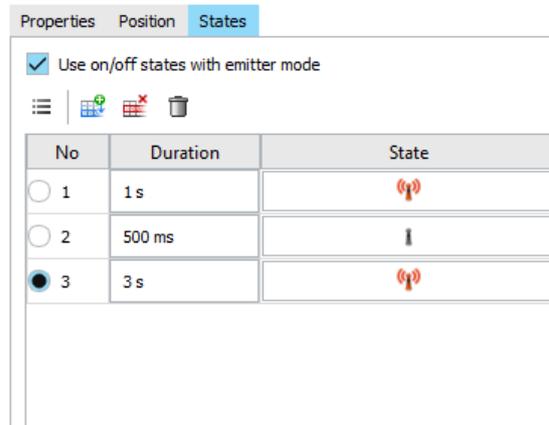
Cancel ← Fill Position Steps
Aborts the changes. Configurations in the "Fill Position Steps" dialog are lost.

16.4.2.3 States

Access:

1. On the 2D map, select "Emitter Properties".

2. In the list of "Available TX Items", select an emitter.

3. Select "Properties of Selected TX Item > Properties > Configuration > Static".

4. Select "States".

No	Duration	State
1	1 s	
2	500 ms	
3	3 s	

Emitters with constant operation mode can be configured to interrupt their signal transmission. You can define the duration of the intervals during that the emitters transmit signal and during which they are muted.

Settings:

Use on/off states with emitter mode
Enables that an emitter can use on and off states.

Remote command:
SCENario:DF:EMITter:STATe:ENABle on page 756
SCENario:LOCalized:EMITter:STATe:ENABle on page 756

▤ Append, Remove Current, Remove All
Standard functions for entry handling.

Remote command:
SCENario:DF:EMITter:STATe:ADD on page 591
SCENario:DF:EMITter:STATe:INSert on page 594
SCENario:DF:EMITter:STATe:COUNt? on page 592
SCENario:DF:EMITter:STATe:DELete on page 595
SCENario:DF:EMITter:STATe:CLEar on page 596
SCENario:LOCalized:EMITter:STATe:ADD on page 591
SCENario:LOCalized:EMITter:STATe:INSert on page 594
SCENario:LOCalized:EMITter:STATe:COUNt? on page 592
SCENario:LOCalized:EMITter:STATe:DELete on page 595
SCENario:LOCalized:EMITter:STATe:CLEar on page 596

No
Subsequent number indicating the state.

Remote command:
SCENario:DF:EMITter:STATe:SELect on page 593
SCENario:LOCalized:EMITter:STATe:SELect on page 593

Duration
Sets the duration during that the emitter remains in the current state.

Remote command:
SCENario:DF:EMITter:STATe:DURation on page 757
SCENario:LOCalized:EMITter:STATe:DURation on page 757

State
Select the icon to toggle the state.

Remote command:
SCENario:DF:EMITter:STATe:VALue on page 757
SCENario:LOCalized:EMITter:STATe:VALue on page 757

Execute States in Loop
Repeats the state definition cyclically.

Remote command:
SCENario:DF:EMITter:STATe:LOOP on page 757
SCENario:LOCalized:EMITter:STATe:LOOP on page 757

16.5 Moving emitters/interferers and platforms

In addition to the static emitters, and the possibility to define emitters that change their position at discrete steps, you can also define emitters that move along a trajectory with a defined shape. Emitters can move along simple trajectories, such as a straight line, a circle or a part of a circle (arc). Emitters can follow complex trajectories defined by a sequence of map coordinates.

A sequence of map coordinates can be defined in two ways.

- **"Trace points"**

  You can use the "Trace" function to set a sequence of coordinates (known as "Trace points") manually:

  - **In the map**
    Select an emitter > right-click > context menu > "Trajectory" > "Trace"
    Left-click to set trace points. Right-click to apply the trace.

  - **In the position settings of an emitter**
    Select an emitter > right-click > "Properties" > "Position" tab
    Select and set/edit the position of each "Trace point" in turn.

- **"Waypoints"**

You can load a waypoint file which configures a sequence of coordinates automatically. Waypoint files are simple text files with predefined file format and extension. For a description, see Chapter A.3, "Movement files", on page 850.

To improve the visualization, you can assign different icons to the emitters/interferers on the 2D map. The icons are merely used for visualization purposes; the function and the properties of the emitters do not depend on the icon.

If at least one moving emitter is configured and its trajectory is defined, you can visualize it on the 2D map. The 2D map also displays the movement of the emitter along the trajectory. In the 3D combined views of the receiver and emitter, you observe the effect of all events influencing the received signal strength. These are: antenna pattern and scan at the emitter and receiver, distance between the emitter and receiver, emitter movement and antenna attitude.

ⓘ Positions are configured relative to the receiver's origin using the "East", "North" and "Height" settings.

If a georeferenced map is loaded (see Chapter 16.10, "Using georeferenced maps", on page 469), absolute positions can also be defined using the "Longitude", "Latitude" and "Altitude" settings.

Similar to the emitters/interferers, the receiver can also be a moving object. The receiver configuration is similar to the configuration of the moving emitters/interferers. For settings description, see Chapter 13, "Emulating receivers", on page 348.

Access:

1. On the 2D map, select "Emitter Properties".

2. In the list of "Available TX Items", select an emitter.

3. Select "Properties of the Selected TX Item > Position".

4. Select "Mode > Moving".

5. Configure the "Trajectory" and further settings.

For step-by-step description, see "How to configure moving emitter and receiver" on page 420.
For information trajectories, see Chapter 16.9, "Creating trajectories on a 2D map", on page 468.

Settings:

Trajectory

Defines the trajectory shape.

"Line"	The emitter is moving on a straight line, starting from the current position of the emitter (East, North, Height/Distance, Azimuth, Elevation) and ending at the selected "End Position".
"Arc"	The emitter moves on a part of a circle, where the center of the circle is defined with the "Center Position".
"Waypoint"	Emitter with varying coordinates, defined in waypoints files. You find a subset of example files in the installation package but you can also create and load your own files.
"Trace"	Emitter with varying coordinates, defined as a set of "Trace points".

Remote command:

SCENario:LOCalized:MOVement:TYPE on page 772

SCENario:DF:MOVement:TYPE on page 772

Line trajectory definition

A line is defined with:

Speed ← Line trajectory definition

Sets the speed of the moving emitter.

Remote command:

SCENario:LOCalized:MOVement:SPEed on page 774

SCENario:DF:MOVement:SPEed on page 774

Acceleration ← Line trajectory definition

Sets the acceleration of the moving emitter.

Remote command:

SCENario:LOCalized:MOVement:ACCeleration on page 774

SCENario:DF:MOVement:ACCeleration on page 774

End Position ← Line trajectory definition

Sets the "East/North" coordinates and the "Height" of the emitter at the end of the movement.

If "Maps" are enabled, the endpoint can also be defined using "Latitude", "Longitude" and "Altitude".

Remote command:

SCENario:LOCalized:MOVement:EAST on page 775

SCENario:LOCalized:MOVement:NORTh on page 775

SCENario:LOCalized:MOVement:HEIGht on page 775

SCENario:DF:MOVement:EAST on page 775

SCENario:DF:MOVement:NORTh on page 775

SCENario:DF:MOVement:HEIGht on page 775

SCENario:LOCalized:MOVement:ALTitude on page 773

SCENario:LOCalized:MOVement:LATitude on page 773

SCENario:LOCalized:MOVement:LONGitude on page 773

SCENario:DF:MOVement:ALTitude on page 773

SCENario:DF:MOVement:LATitude on page 773

SCENario:DF:MOVement:LONGitude on page 773

Mode ← Line trajectory definition
Defines the behavior of the moving object when the end of the trajectory is reached.

"Cyclic"   The trajectory is repeated cyclically. Once the "End Position" is
                   reached, file movement starts again from the beginning.

                   Using this mode is recommended in the following cases:
                   - A circle trajectory
                   - A trajectory in which the start and the end positions are close to
                     each other.

"Round Trip"   The emitter moves back and forth on the defined trajectory.

"One Way"   The movement is executed once.
                   When the "End Position" is reached, this position is assumed to be a
                   static one.

Remote command:
SCENario:DF:MOVement:RMODe on page 778

SCENario:LOCalized:MOVement:RMODe on page 779

Arc trajectory definition
An arc is defined with:

Speed ← Arc trajectory definition
Sets the speed of the moving emitter.

Remote command:
SCENario:LOCalized:MOVement:SPEed on page 774

SCENario:DF:MOVement:SPEed on page 774

Angle ← Arc trajectory definition
Sets the arc angle and thus defines the arc length.

Remote command:
SCENario:LOCalized:MOVement:ANGLe on page 775

SCENario:DF:MOVement:ANGLe on page 775

Center Position ← Arc trajectory definition
Sets the "East/North" coordinates of the center of the circle on that the emitter is mov-
ing.

If "Maps" are enabled, the center point can also be defined using "Latitude" and "Longi-
tude".

Remote command:
SCENario:LOCalized:MOVement:EAST on page 775

SCENario:LOCalized:MOVement:NORTh on page 775

SCENario:LOCalized:MOVement:CLATitude on page 774

SCENario:LOCalized:MOVement:CLONgitude on page 774

SCENario:DF:MOVement:EAST on page 775

SCENario:DF:MOVement:NORTh on page 775

SCENario:DF:MOVement:CLATitude on page 773

SCENario:DF:MOVement:CLONgitude on page 773

Mode ← Arc trajectory definition
Defines the behavior of the moving object when the end of the movement is reached.

"Cyclic"   The trajectory is repeated cyclically. Once the "End Position" is
                   reached, file movement starts again from the beginning.

                   Using this mode is recommended in the following cases:
                   - A circle trajectory
                   - A trajectory in which the start and the end positions are close to
                     each other.

"Round Trip"   The emitter moves back and forth on the defined trajectory.

"One Way"   The movement is executed once.
                   When the "End Position" is reached, this position is assumed to be a
                   static one.

Remote command:
SCENario:DF:MOVement:RMODe on page 778
SCENario:LOCalized:MOVement:RMODe on page 779

Trace trajectory definition
You can use the "Trace" function to set a sequence of coordinates (known as "Trace
points"), manually.

A trace is defined with:

Trace points ← Trace trajectory definition
Selects the trace point that is configured by the following fields.

Remote command:
SCENario:LOCalized:LOCation:PSTep:SELect on page 593
SCENario:DF:LOCation:PSTep:SELect on page 593

Longitude ← Trace trajectory definition
Sets the longitude of the selected trace point.

Remote command:
SCENario:LOCalized:LOCation:LONGitude on page 772
SCENario:DF:LOCation:LONGitude on page 772

Latitude ← Trace trajectory definition
Sets the latitude of the selected trace point.

Remote command:
SCENario:LOCalized:LOCation:LATitude on page 772
SCENario:DF:LOCation:LATitude on page 772

Altitude ← Trace trajectory definition
Sets the altitude above sea-level of the selected trace point.

Remote command:
SCENario:LOCalized:LOCation:ALTitude on page 772
SCENario:DF:LOCation:ALTitude on page 772

East from RX ← Trace trajectory definition
Sets the number of meters East, from the receiver to the selected trace point.

Remote command:
`SCENario:LOCalized:LOCation:EAST` on page 750
`SCENario:DF:LOCation:EAST` on page 749

North from RX ← Trace trajectory definition
Sets the number of meters North from the receiver to the selected trace point.

Remote command:
`SCENario:LOCalized:LOCation:NORTh` on page 750
`SCENario:DF:LOCation:NORTh` on page 749

Height ← Trace trajectory definition
Sets the number of meters vertically, from the plane of the receiver to the selected trace point.

Remote command:
`SCENario:LOCalized:LOCation:HEIGht` on page 751
`SCENario:DF:LOCation:HEIGht` on page 751

Distance to RX ← Trace trajectory definition
Number of meters along the line of sight from the receiver to the selected trace point.

Remote command:
`SCENario:LOCalized:DISTance` on page 749
`SCENario:DF:DISTance` on page 749

Azimuth from RX ← Trace trajectory definition
Horizontal angle from the receiver to the selected trace point.

Remote command:
`SCENario:LOCalized:LOCation:AZIMuth` on page 750
`SCENario:DF:LOCation:AZIMuth` on page 750

Elevation from RX ← Trace trajectory definition
Vertical angle from the receiver to the selected trace point.

Remote command:
`SCENario:LOCalized:LOCation:ELEVation` on page 751
`SCENario:DF:LOCation:ELEVation` on page 751

Speed ← Trace trajectory definition
Sets the speed of the moving emitter.

Remote command:
`SCENario:LOCalized:MOVement:SPEed` on page 774`SCENario:LOCalized:MOVement:SPEed 100`
`SCENario:DF:MOVement:SPEed` on page 774

Mode ← Trace trajectory definition
Defines the behavior of the moving object when the end of the movement is reached.

| "Cyclic" | The trajectory is repeated cyclically. Once the "End Position" is reached, file movement starts again from the beginning. |
| | Using this mode is recommended in the following cases: |

- A circle trajectory
- A trajectory in which the start and the end positions are close to each other.

"Round Trip"	The emitter moves back and forth on the defined trajectory.
"One Way"	The movement is executed once.
	When the "End Position" is reached, this position is assumed to be a static one.

Remote command:

SCENario:LOCalized:MOVement:RMODe on page 779

SCENario:DF:MOVement:RMODe on page 778

Smoothing for traces

The discrete positions (trace points) defined by a "Trace" can cause abrupt changes in the movement direction. This function is identical to the smoothing function used with a waypoints file except that the waypoints are derived from the trace rather than a file (see "Smoothing for waypoints > State" on page 459).

Waypoints trajectory definition

A waypoint file is description of a movement with possibly attitude coordinates that can have different forms, like, for example, a sequence of positions or vector arc movement. A waypoint file must have the extension *.txt, *.kml or *.xtd. For description, see Chapter A.3, "Movement files", on page 850.

You find a subset of example files in the installation package but you can also create and load custom-specific files. Moreover, waypoint smoothing can be activated if vehicle description files are used.

If waypoint files are used, the emitter position and trajectory are retrieved from the file content; the traditional position coordinates parameters are disabled.

Waypoint File ← Waypoints trajectory definition
Indicates the name of the currently used waypoint file.

☰ Load, Delete ← Waypoints trajectory definition
Opens the Windows explorer so that you can navigate to and load waypoint file.

Discards the selected waypoint file.

Remote command:
SCENario:DF:MOVement:WAYPoint on page 776
SCENario:LOCalized:MOVement:WAYPoint on page 776

Read Out Mode ← Waypoints trajectory definition
Indicates the way the waypoint file is processed.

"Cyclic"   The waypoint file is processed cyclically. Once the last waypoint is reached, file processing starts again from the beginning.

Using this mode is recommended if the waypoint file describes one of the following:
- A circle trajectory
- A trajectory in which the start and the end positions are close to each other.

"Round Trip"   When the end of the file is reached, the file is processed backwards.

"One Way"   The file is processed once.
                   When the end of the file is reached, the last described position is
                   assumed to be a static one.

Remote command:
SCENario:DF:MOVement:RMODe on page 778
SCENario:LOCalized:MOVement:RMODe on page 779

Reference Frame ← Waypoints trajectory definition
Select the reference frame used to define the emitters coordinates.

The transformation between the reference frames is performed automatically.

The following applies:
- $X_{WGS84} = (1 - 0.008*10^{-6})*X_{PZ\,90} - 0.2041*10^{-7}*Y_{PZ\,90} + 0.1716*10^{-7}*Z_{PZ\,90} - 0.013$
- $Y_{WGS84} = (1 - 0.008*10^{-6})*Y_{PZ\,90} - 0.2041*10^{-7}*X_{PZ\,90} + 0.1115*10^{-7}*Z_{PZ\,90} + 0.106$
- $Z_{WGS84} = (1 - 0.008*10^{-6})*Z_{PZ\,90} - 0.1716*10^{-7}*X_{PZ\,90} - 0.1115*10^{-7}*Y_{PZ\,90} + 0.022$

Both reference frames are ECEF frames with a set of associated parameters.

"WGS-84"   The World Geodetic System WGS-84 is the reference frame used by
                   GPS.

"PZ 90.11 (GLONASS)"
                   Parametry Zemli PZ (parameters of the Earth) is a reference frame,
                   used, for example, by GLONASS.

Remote command:
SCENario:DF:MOVement:RFRame on page 778
SCENario:LOCalized:MOVement:RFRame on page 778

Smoothing for waypoints > State ← Waypoints trajectory definition
The discrete positions (waypoints) defined in the waypoints file can cause abrupt
changes in the movement direction.

The R&S Pulse Sequencer provides an internal interpolating algorithm that smooths
the movement or the trajectory. This algorithm evaluates the dedicated vehicle descrip-
tion (*.xvd) file, retrieves the velocity vector and the <proximity> value, and inserts
waypoints to smooth the trajectory. The resulting movement is more realistic.

To use the algorithm, enable "Smoothing", select "Vehicle File > Load" to load a *.xvd
file and select "Calculate". See Chapter A.4, "Vehicle description files (Used for
smoothening)", on page 858.

Remote command:
SCENario:DF:MOVement:SMOothening on page 779
SCENario:LOCalized:MOVement:SMOothening on page 779

Vehicle File ← Waypoints trajectory definition
Indicates the name of the currently used vehicle description (*.xvd) file.

☰ Load, Delete ← Waypoints trajectory definition
Opens the Windows explorer so that you can navigate to and load vehicle description
file.

Discards the selected vehicle description file.

Remote command:
SCENario:DF:MOVement:VFILe on page 777
SCENario:DF:MOVement:VFILe:CLEar on page 778
SCENario:LOCalized:MOVement:VFILe on page 777
SCENario:LOCalized:MOVement:VFILe:CLEar on page 778

Attitude ← Waypoints trajectory definition
Defines how the attitude information is defined.

"From Waypoint File"

The attitude parameters are extracted from the selected waypoint file. Further settings are not required.
This forces the attitude parameters towards motion direction even if the waypoint has attitude information, like, for example, in a `*.xtd` file with `<property waypointformat="position_attitude">`. For specific emitters like ships or land vehicles, it is realistic to set the yaw and pitch to the vehicle's motion direction. This is because the usual body axes angles of a land vehicle are in the direction of the velocity vector.
For other emitters, however, like landing planes, this parameter is not useful. As an example, the nose of the plane is in an upward direction at the time when the plane is moving downwards.

"Align to Motion"

Enables a constant rate of change of the roll.

"Constant"   Emitter's attitude is set as the combination of the "Yaw/Heading", "Pitch/Elevation", "Roll/Bank" values.
The resulting attitude is a constant value.

Remote command:
SCENario:DF:MOVement:ATTitude on page 776
SCENario:LOCalized:MOVement:ATTitude on page 776

Yaw, Pitch, Roll ← Waypoints trajectory definition
Sets the angles of rotation in the corresponding direction, i.e. the rotation around the respective yaw, pitch and roll axes. "Yaw, Pitch, Roll" are defined relative to the local horizon.

Remote command:
SCENario:DF:MOVement:YAW on page 778
SCENario:DF:MOVement:PITCh on page 778
SCENario:DF:MOVement:ROLL on page 778
SCENario:LOCalized:MOVement:YAW on page 778
SCENario:LOCalized:MOVement:PITCh on page 778
SCENario:LOCalized:MOVement:ROLL on page 778

Calculate ← Waypoints trajectory definition
Loads the selected waypoint and vehicle description file.

The "Movement Data" display indicates a summary of the major parameters:
● Start/End Position

- Duration of the movement
- Read Out Mode as defined in the file
- Smoothing state
- Attitude behavior

The status LED indicates the following states:
- **Red**: the waypoint file is not selected, the file is not imported and the movement is not calculated or the file is erroneous.
- **Green**: Movement is calculated

Remote command:

SCENario:DF:MOVement:IMPort on page 779

SCENario:LOCalized:MOVement:IMPort on page 779

▤ Clear ← Waypoints trajectory definition
Discards the waypoint and vehicle description file.

Remote command:

SCENario:DF:MOVement:CLEar on page 779

SCENario:LOCalized:MOVement:CLEar on page 779
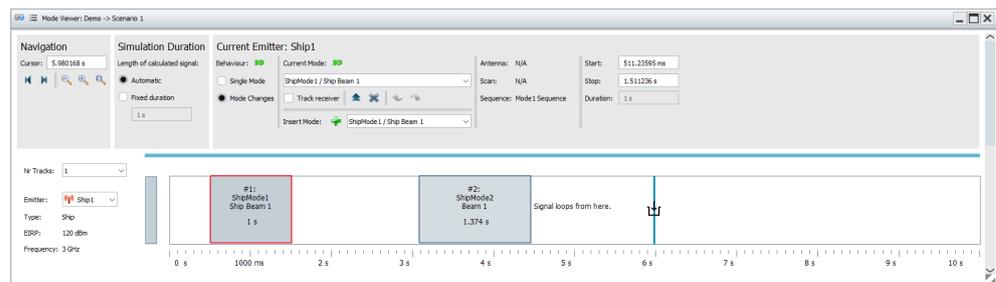

16.6 Mode editor

Access:

1. Perform one of the following:
   - In a "Scenario Type = Localized Emitters", select "Emitters > 2D".
   - In a "Scenario Type = Direction Finding", select "Map > 2D".

2. In the 2D map, select "Properties > Available TX Items".

3. Select "Edit Modes".

   You access the dialog for configuration of the mode changes of *all emitters* available in the scenario.
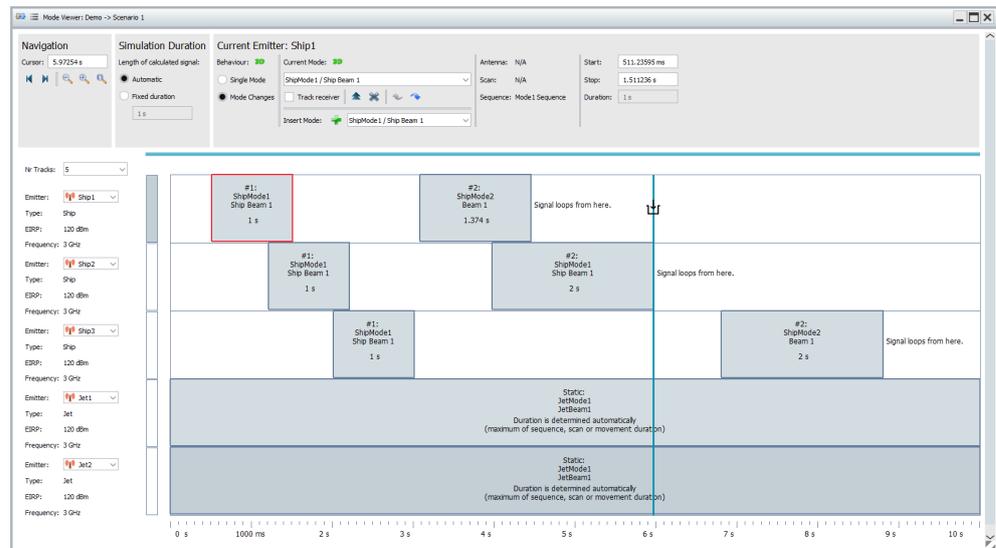
4. In the "Properties" of the selected emitter, select "Edit Modes".

   You access the "Mode Editor" dialog for the *current emitter*.

The "Mode Editor" is a graphical editor for joint configuration and defining the duration emitters are working in the different modes.

Operation is similar to Mode Changes settings for each of the emitters. However, rather than defining the start and end time of a mode, you drag the start and end time on the time scale.
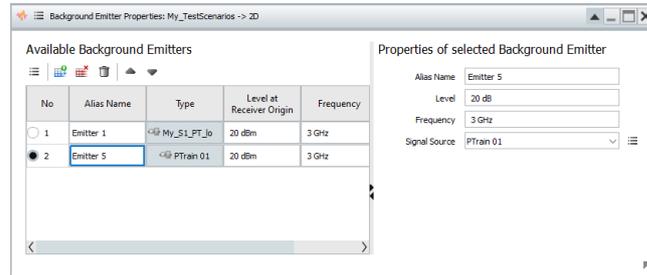
The provided settings and configuration possibilities are identical with the configuration of an individual emitter. For description, see Chapter 10.3.3, "Mode editor settings", on page 317.

16.7 Background emitters properties

Access:

1. Perform one of the following:
   ● In a "Scenario Type = Localized Emitters", select "Emitters > 2D".
   ● In a "Scenario Type = Direction Finding", select "Map > 2D".

2. In the 2D map, double-click the signal in the "Background Signals" block.

Settings:

Available Background Emitters
Displays a list of the sequences or waveforms used to create the individual background signals.

Use the standard functions in the context menu to add, reorder, or remove items.

Background emitters are identified by their "Alias Name", "Type", "Level" and used "Frequency".

Background emitters do not have antennas, specific positions on the 2D map or direction.

See also "General workflow for creating complex 2D scenarios" on page 416.

Remote command:
SCENario:LOCalized:ALIas on page 746
SCENario:DF:ALIas on page 746
SCENario:LOCalized:WAVeform:LEVel on page 753
SCENario:DF:WAVeform:LEVel on page 753
SCENario:LOCalized:WAVeform:FREQuency on page 753
SCENario:DF:WAVeform:FREQuency on page 753

Properties of selected Background Emitter
Displays and sets the properties for the selected background emitter.

Changing the properties here updates the values for the corresponding emitter in the "Available Background Emitters" list.

Alias Name ← Properties of selected Background Emitter
Sets the name of the selected emitter.

Remote command:
SCENario:LOCalized:ALIas on page 746
SCENario:DF:ALIas on page 746

Level ← Properties of selected Background Emitter
Sets the level at the receiver for the selected emitter.

The value is the maximum level at the receiver origin. The calculation considers only emitter EIRP and free-space attenuation and omits individual antenna attenuation due to position and polarization.

Remote command:
SCENario:LOCalized:WAVeform:LEVel on page 753
SCENario:DF:WAVeform:LEVel on page 753

Frequency ← Properties of selected Background Emitter
Sets the frequency of the selected emitter.

Remote command:
SCENario:LOCalized:WAVeform:FREQuency on page 753
SCENario:DF:WAVeform:FREQuency on page 753

Signal Source ← Properties of selected Background Emitter
Sets the signal source of the selected emitter.

To change the source, select one from the drop-down list.

To edit the current source or add a new one, select the icon and then select the required item from the menu.

Remote command:
SCENario:LOCalized:WAVeform on page 753
SCENario:DF:WAVeform on page 753

16.8 Platforms with multiple emitters

Access:

1. Open a suitable scenario.

   Use any scenario that offers "Map" in the "Scenario Creation" block.
   - "Localized Emitters"
   - "Direction Finding"

2. In the block diagram, select "Map".

   The "2D" dialog displays a two-dimensional view of the receiver and currently con-figured emitters/interferers and platforms.
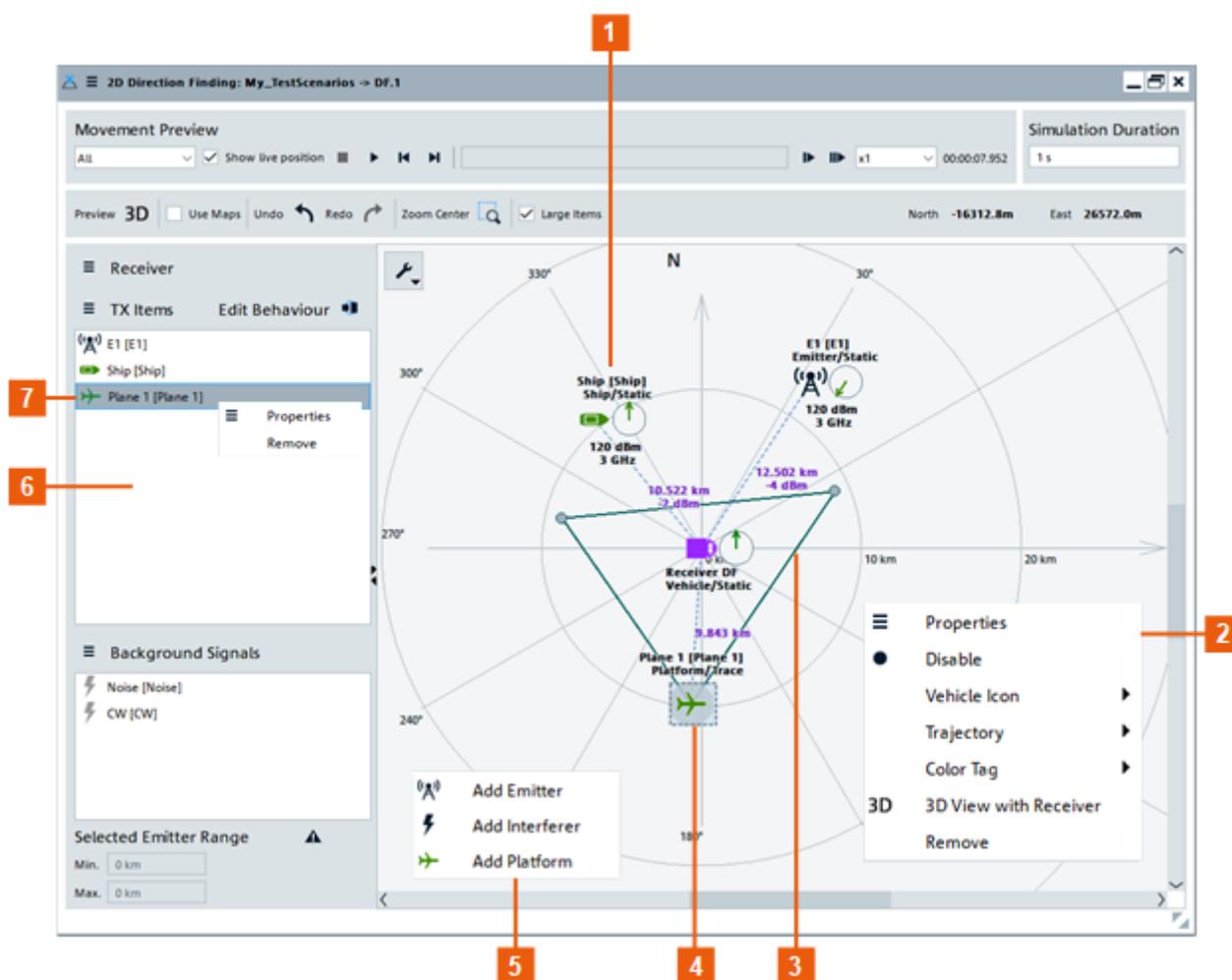
Figure 16-13: Platform-related functions of the 2D dialog

1 = Static platform
2 = Platform context menu
3 = Trajectory of moving platform
4 = Moving platform
5 = Map area context menu (right-click)
6 = "TX Items" list
7 = TX Items context Menu (right-click)

Figure 16-13 shows the platform-related elements of the "2D" dialog.

Adding and configuring platforms

Use the following procedure to add and configure platforms in a scenario:

1. Access the "2D" dialog (as described previously).

2. Add platforms to the list.

   Use one of the following methods:
   - Drag&drop a platform from the repository.

- Right-click in the map area and select "New Platform" in the context menu (see Chapter 16.8.2, "Map area context menu", on page 467).

A platform icon appears in the "Platforms" list and on the map.

3. Configure the platform.

Use one of the following methods:
- Right-click a platform icon on the map.
A context menu appears. Select "Properties".
See Chapter 16.8.3, "Platform context menu", on page 467.
- Right-click an entry in the "Platforms" list.
A context menu appears. Select "Properties".
See Chapter 16.8.1, "TX items context menu", on page 466.
- Double-click a platform icon or "Platforms" list entry.
Opens the "TX Items Properties" dialog.
See Figure 16-11.
- Click the "TX Items" icon in the list of "Properties" links.
Opens the "TX Items Properties" dialog.
SeeFigure 16-11.

Settings:

16.8.1   TX items context menu

To access this context menu, right-click an entry in the list.
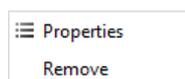
```
≔ Properties
  Remove
```

The following functions are available on the context menu:

Properties
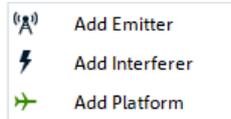Opens the "TX Items Properties" dialog.

See Figure 16-11.

Remove
Removes the selected platform from the list.

16.8.2 Map area context menu

To access this context menu, right-click anywhere in the map area that is not occupied by an object (e.g. an emitter).

```
((A))   Add Emitter
 ⚡     Add Interferer
 ✈     Add Platform
```

The following functions are available on the context menu:

Add Emitter
Adds an emitter icon to the map at the cursor position.

Adds a corresponding entry to the "Emitters/Interferers" list.

Note: Double-click the entry or the icon to configure it in the TX Items Properties dialog.

Add Interferer
Adds an interferer icon to the map at the cursor position.

Adds a corresponding entry to the "Emitters, Interferers" list.

Note: Double-click the entry or the icon to configure it in the TX Items Properties dialog.
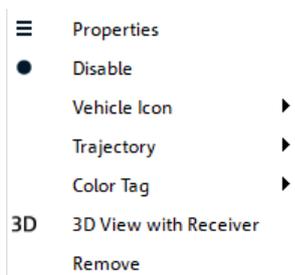
Add Platform
Adds a platform icon to the map at the cursor position.

Adds a corresponding entry to the "Platforms" list.

Note: Double-click the entry or the icon to configure it in the TX Items Properties dialog.

16.8.3 Platform context menu

To access this context menu, right-click a platform icon in the map area.

```
≡     Properties
●     Disable
      Vehicle Icon        ▶
      Trajectory          ▶
      Color Tag           ▶
3D    3D View with Receiver
      Remove
```

The following functions are available on the context menu:

Properties

Opens the "TX Items Properties" dialog.

See Chapter 16.4, "Properties of TX items (emitters/interferers and platforms)", on page 434.

Enable/Disable

Toggles the platform's emitters on/off within the scenario.

Determines whether the platform is considered in a calculation or not.

Vehicle Icon

Selects the icon for the platform on the 2D map.

The icons are merely indication; the function and the properties of the platform do not depend on the icon.

You can import and use custom icons, see Chapter 16.11, "Importing user icons", on page 473.

Trajectory

Defines the path followed by a moving platform.

R&S Pulse Sequencer supports several trajectory types, including: line, arc, trace and waypoint.

See Chapter 16.5, "Moving emitters/interferers and platforms", on page 450.

See Chapter 16.9, "Creating trajectories on a 2D map", on page 468.

Color Tag

A platform icon can have a colored background.

Set the color to: "None", "Green", "Blue", or "Red".

3D View with Receiver

Opens a "3D" view containing the selected platform and the receiver/DF.

See Chapter 16.3, "3D scan pair view settings", on page 429.

Remove

Removes the selected platform from the list.

16.9　Creating trajectories on a 2D map

Access:

1. In a "2D Map", right-click to open the context menu.

2. In the context menu, select "Trajectory" to open a list of trajectory types.

3. Select a trajectory type from the list.

4. Use the mouse-keys to create the trajectory on the map.

Settings

Line

A line appears between the current position of the emitter and the mouse pointer.

Move the mouse pointer to adjust the position and direction of the line. Left-click to fix the endpoint of the line trajectory.

Arc

A circle is drawn that centers on the mouse pointer and passes through the emitter position. Left-click to fix the center of the circle. Move the mouse pointer to define the length of the arc. Left-click to fix the endpoint of the arc trajectory.

Trace

A line appears between the current position of the emitter and the mouse pointer.

Move the mouse pointer to adjust the position and direction of the line. Left-click to set a trace point. Set the required number of trace points. Right-click to apply the trace (this action does not set an additional trace point).

Waypoints

This option only defines the type of trajectory. You must also define the waypoints file.

Select the emitter > right-click > "Properties" > "Position" tab > "Waypoint file"

16.10 Using georeferenced maps

The R&S SMW-K309 option enables you to use georeferenced maps with R&S Pulse Sequencer.

The key benefits of this option are that you can:

- Visualize a real-world geographical scenario.
- Accurately position emitters to emulate real-world radar installations and interfer-ers:
    - Drag and drop the icons on the map.
    - Use the actual coordinates (lat/lon) of the emitters.
- Create realistic movement paths:

–   Ships moving in shipping lanes.

–   Planes using air corridors and flight levels.

–   Vehicles driving on roads.

●   Easy handling:

–   Load and unload maps via toolbar.

–   Reposition map using left mouse-key.

–   Zoom in/out using mouse scroll-button.

ⓘ   Without the R&S SMW-K309 option - the "Cursor Position" is displayed in meters "North" and "East" of the receiver's origin. This enables accurate positioning relative to the receiver.

Without the R&S SMW-K309 option - the "Cursor Position" is displayed as "Longitude" and "Latitude" coordinates. This enables accurate positioning relative to a georeferenced map.

Access:

1.   Open a suitable scenario.

Use any scenario that offers "Map" in the "Scenario Creation" block.
●   "Localized Emitters"
●   "Direction Finding"

2.   In the block diagram, select "Map".

The "2D" dialog displays a 2D view of the receiver and currently configured emitters/interferers, together with their main characteristics.

☐ Use Maps   3.   In the "2D" dialog, select the "Use Maps" checkbox.

Note: If you enable this function, the scenario uses the R&S SMW-K309 option and it **cannot be removed** from the signal generation of the scenario.
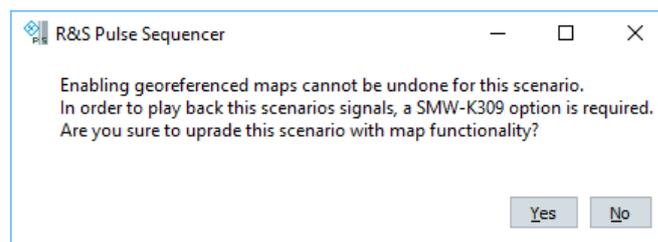For this reason, a popup message appears asking you to confirm.

Figure 16-14: Popup regarding R&S SMW-K309

Select "No" if you do not want the scenario to depend on the extended option.
Select "Yes" to proceed.

The "Map Options" toolbar appears.

Figure 16-15: Map Options toolbar

1 = "Open Map" file browser
2 = Unload map
3 = "Application Colors" dialog
4 = Set opacity of loaded map

Settings:

Use Maps
Option: R&S SMW-K309

Enables the scenario to use georeferenced maps.

Once enabled, it cannot be disabled (see the note in "Access:" on page 470).

Application Colors
Customizes the colors used in R&S Pulse Sequencer.

Alternative access: select "Configure" > "Colors". See also Changing Colors).

Use the "Map" tab to customize the map-specific colors.

Open Map
Use this dialog to select a georeferenced map file.

The file browser only displays acceptable file formats.

Supported formats:
- `.tif`
- `.tiff`

If the format of the selected file is supported, the map appears in the "2D" diagram and "3D Live Vsiaulization".
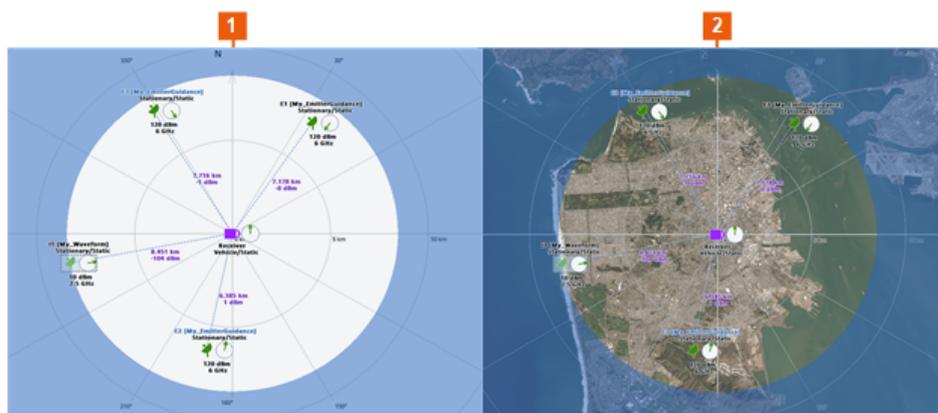
Figure 16-16: 2D diagram before and after loading a map

1 = Scenario with a plain background
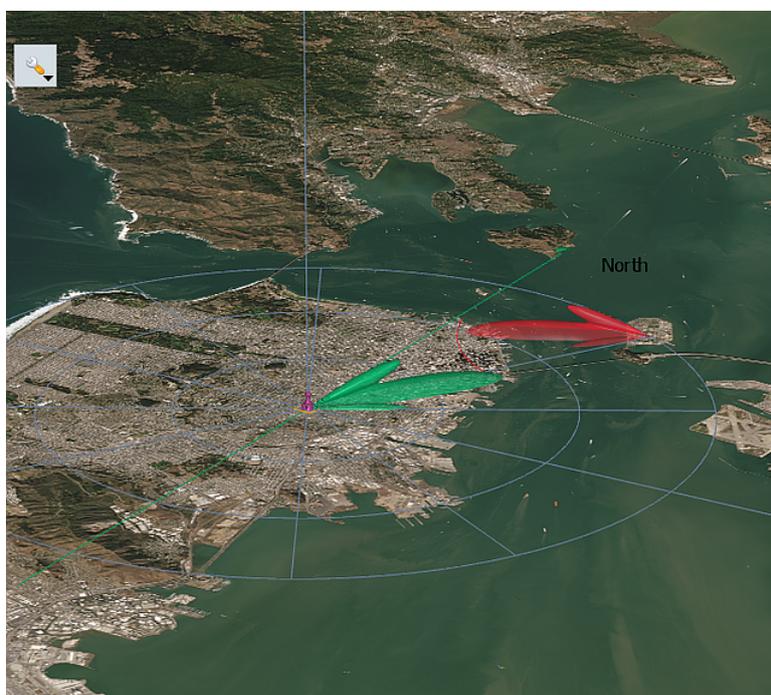2 = Scenario with a georeferenced map

Figure 16-17: 3D Live Visualization with map background

If the selected file does not contain the required georeferencing metadata, R&S Pulse Sequencer outputs error messages in the "Message Log".
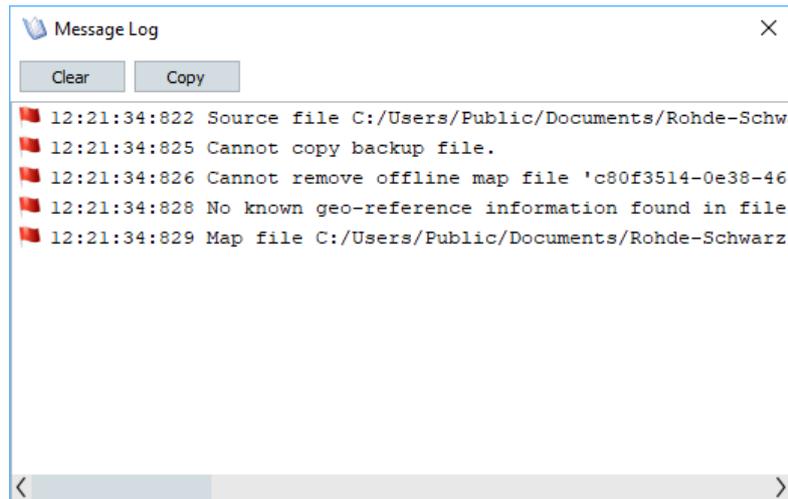
Figure 16-18: Typical errors caused by missing metadata

Unload map

Use this function to unload the current map.

The "2D" diagram is not affected by this action - e.g. the current map centering and zoom level stay the same.

Note: The map is unloaded immediately.
The application does not ask you to confirm the action.

There are no negative consequences if you accidentally unload a map. Use the "Open Map" dialog to load the map again.

Opacity

Use this function to make the map or the scenario more or less prominent.

Move the slider left/right to make the background color more/less visible through the map.

For example:
● To focus on the map, set the slider fully to the right.
● To focus on the scenario, set the slider fully to the left.
● To show the scenario in relation to the environment, set the slider in the middle.

16.11 Importing user icons

You can import custom icons to expand the list of available icons for displaying RX and TX items in a scenario.

1. In the menu bar, select "Configure > Settings > Icons".

   An empty list is shown.

2. Add icons to your list.

Use one of the following methods:
- Add single icons to the list by using the standard functions, see Table 2-4.
- Import an icon archive that was created by the pulse sequencer software.
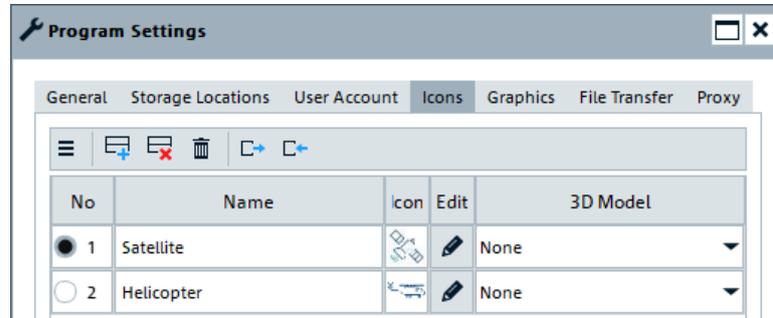
3. With the icons in the list you can do the following:

- Rename the icon.
- Edit the icon.
- Select a 3D model for it.
- Export the icons to create an archive.

4. Finish with "Apply".

   You can now select the user icons from the list of "Vehicle Icons" in a scenario, see Chapter 16.8.3, "Platform context menu", on page 467.

Settings:

Export/import icons as archive
Export the icons to create an archive or import an icon archive that was created by the pulse sequencer software.

Name
Enter the name of the icon.

Icon
Displays the imported user icon.

Edit
Change the displayed user icon.

3D Model
Select a 3D model for the icon from a list of predefined models. The selected 3D model is used to display the TX/RX item in the interactive 3D display.

"None": a cone is shown for the TX/RX item in the interactive 3D display.

17 Playing the generated waveform files

Complex pulse signals created with the R&S Pulse Sequencer can be played by:

● R&S SMW
● R&S SMBV
● R&S SMBV100B
● R&S SMM
● R&S SGT

Depending on the installed option and the capabilities of the vector signal generator, files and waveforms are processed as an ARB file or in real time.

Waveform types

The R&S Pulse Sequencer creates and outputs waveforms in the following types:

● **"Pure" ARB file**
  (R&S SMW/SMBV/SGT/SMM-K300/-K301)
  The R&S Pulse Sequencer calculates a *long waveform without sequencing*; the calculation is performed offline and in advance.

● **Multi-segment waveform (MSW)**
  (R&S SMW/SMBV/SGT/SMM-K300/-K301)
  The R&S Pulse Sequencer *pre-processes the sequence and splits it up into sequential* elements. These elements are then played using loops.

● **Extended sequencing**
  (R&S SMW-K300/-K301/-K501/-K502)
  The R&S Pulse Sequencer calculates the required data and creates the following files:

  – A waveform file describing one pulse
  – Two files in Rohde & Schwarz proprietary file format describing control data, like pulse amplitude, frequency or phase variation.

  The calculation is performed offline and in advance. The created files are small. The files can be automatically transferred to and *played* by an R&S SMW equipped as follows:

  – Option R&S SMW-K501
    Standard baseband generator (R&S SMW-B10) and baseband main module (R&S SMW-B13/B13T)
  – Option R&S SMW-K502
    Wideband baseband generator (R&S SMW-B9) and wideband baseband main module (R&S SMW-B13XT)

  For details on the extended sequencing option, see R&S®SMW-K501/-K502/-K503 Extended and Real-Time Sequencing, Real-Time Control Interface user manual

Waveform handling and impact of the instrument's hardware on the calculated waveform

The preferred output waveform format is specified in the Signal calculation settings dialog. The R&S Pulse Sequencer applies the preferred waveform format but, if necessary, automatically selects a more suitable format.

An automatic fall back applies in the following situations:

● If the signal generator does not support a particular waveform format
● If the signal properties or current scenarios are not suitable for the selected waveform format

A highly complex pulse sequence with MOP and possibly also IPM is best generated as a multi-segment waveform. Whereas the quickest way to process a long sequence of pulses without modulation is the extended sequencing option.

Waveform processing

Generated waveforms are transmitted directly to the assigned signal generator or stored in the repository. Once a waveform is stored in the repository, it can be loaded from there and sent to the assigned signal generator without a new start of the waveform generation process.

The examples in the following sections explain the principle and use R&S SMW to play the waveforms.

For step-by-step instructions, see:

● How to create and play the waveforms of simple scenarios
● How to play the waveforms of 2D scenarios
● How to use the extended sequencing option

For description of the related settings, see:

● Signal calculation settings
● Chapter 17.2, "Assign signal source or receiver signals to generator", on page 481
● Chapter 17.3, "File transfer settings", on page 485
● Chapter 17.4, "Control panel settings", on page 487
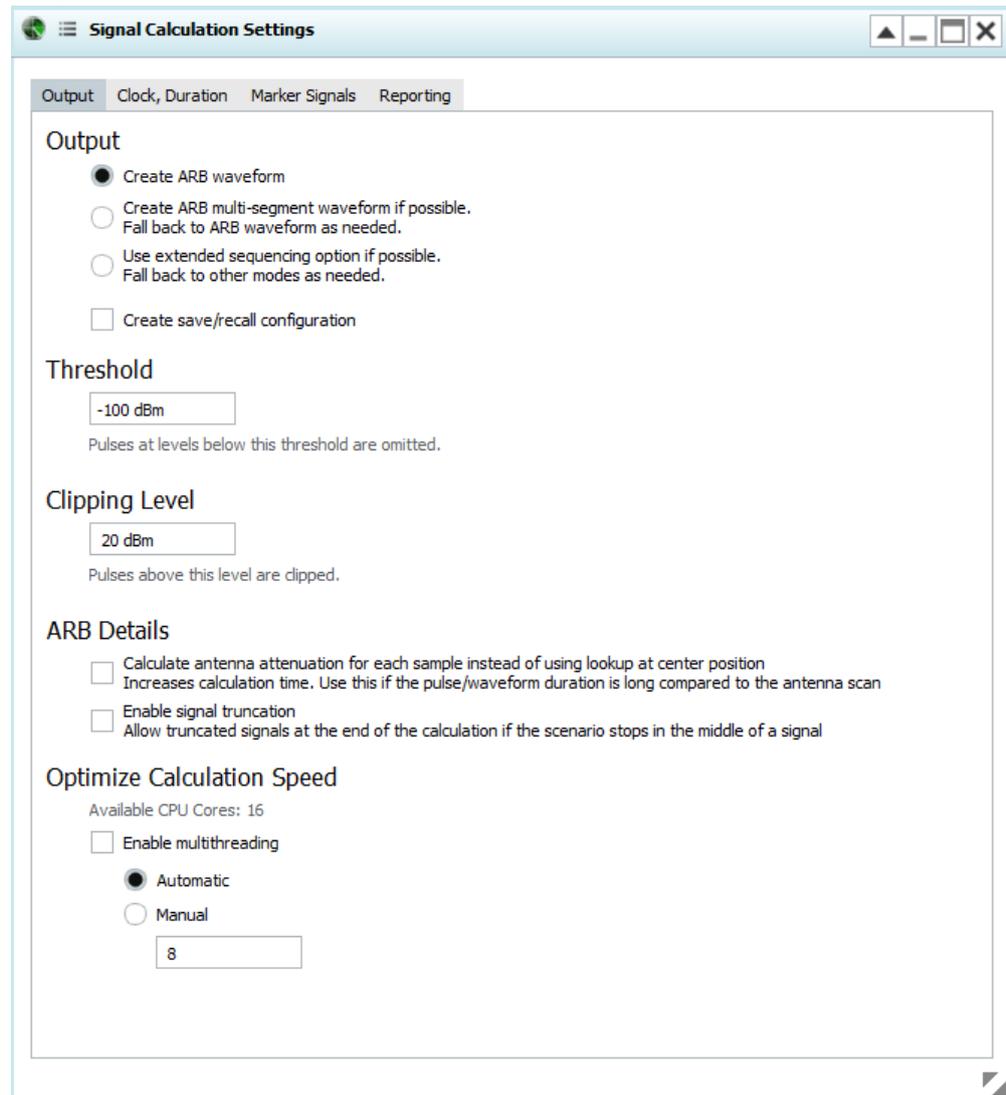
17.1 Signal calculation settings

Access:

► In the "Scenario" dialog, select "Signal Calculation > Config".

Settings:

Output

Output Format ← **Output**

Access: in the "Scenario" dialog, select "Signal Calculation > Config > Config... > Output".

Defines the type of the generated waveform file:
- A single ARB file
- An ARB multi-segment waveform file
- A waveform file describing one pulse and two files in Rohde & Schwarz proprietary file format describing control data.

The software calculates the most suitable waveform for the current situation, incl. current scenario, signal properties and used instrument.

If the preferred waveform format is evaluated as unsuitable, an automatic fallback mechanism selects the best waveform format.

See also:

"Waveform types" on page 475.

Remote command:
SCENario:OUTPut:FORMat on page 762

Create save/recall configuration ← Output
Creates the PS_Recall\<Repository Name> directory on the generator's hard drive and stores the current signal generator configuration as a save/recall file in it.

The save/recall file has the extension *.savrcltxt and its name reflects the name of the current scenario.

Remote command:
SCENario:OUTPut:RECall:ENABle on page 762

Threshold ← Output
Enters a level threshold to limit the dynamic range of the signal. Pulses at levels below this threshold are omitted.

The maximum "Threshold" is 0 dBm.

Remote command:
SCENario:OUTPut:THReshold on page 765

Clipping Level ← Output
Enters a maximum level to limit the dynamic range of the signal. Pulses at levels above this threshold are reduced (*clipped*) to the configured level.

Remote command:
SCENario:OUTPut:CLIPping on page 760

ARB Details ← Output
Access: in the "Scenario" dialog, select "Signal Calculation > Config > Config > Output".

You can enable the following functions:
● Calculate antenna attenuation.
   If enabled, calculates antenna attenuation for each sample. Otherwise a lookup at center position is used.
   Enable this setting by the calculation of imported waveforms and CW signals or if the signal comprises long pulses.
● Enable signal truncation.
   If enabled, the calculation process allows truncated signals at the end of the signal duration, for example, only a fraction of a pulse if the time ends within the last pulse.

See also Chapter 23.4, "Speeding up calculation", on page 561.

Remote command:
SCENario:OUTPut:ARB:DETails:ALBS on page 765
SCENario:OUTPut:ARB:DETails:TRUNcate on page 766

Optimize calculation speed ← Output
Enable this parameter to apply multithreading and decrease the calculation time.

Select one of the options:
- "Automatic"
  The system automatically applies the optimum number of threads for the scenario.
- "Manual"
  Define the number of threads to be used.
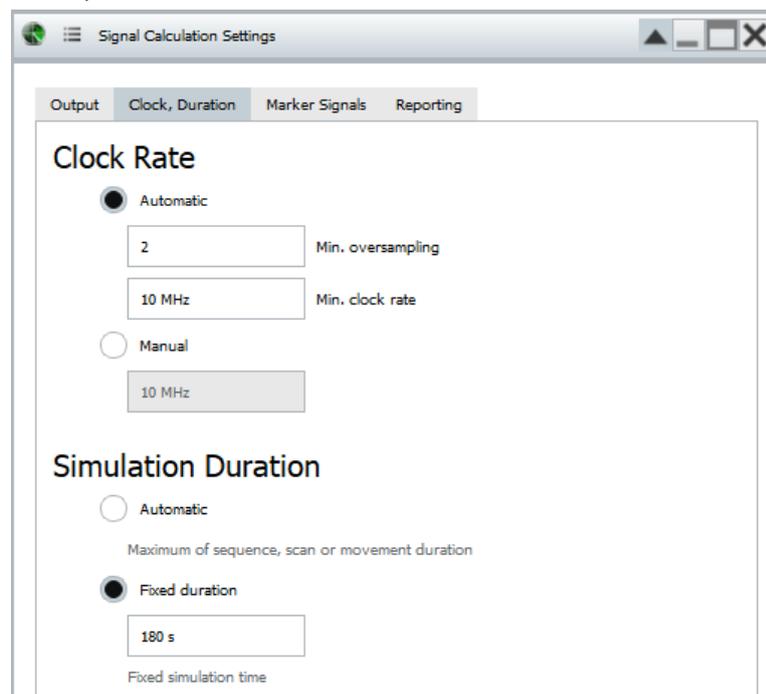
See also Chapter 23.4, "Speeding up calculation", on page 561.

Remote command:
SCENario:OUTPut:MULTithread on page 766
SCENario:OUTPut:MTMode on page 766
SCENario:OUTPut:MTTHreads on page 766

Clock, Duration

Provides clock-related settings:

"Clock Rate"   Retrieves the clock rate automatically from the generated waveform or sets a user-defined value.
The clock rate influences the size of the generated waveform.
Tip: To reduce the size of long waveforms, change the clock rate.
See "To reduce the file size of the created file by adjusting the clock rate" on page 563.

"Duration"   Defines the waveform content.
                       Works like the settings in "Scenario > Simulation". If you change the
                       settings here, they update automatically in the "Simulation" area in
                       the "Scenario" dialog. For a description, see "Duration" on page 125.

                       The generated signal can:
                       ● Last a specified duration.
                         Short sequences are repeated, longer once are truncated
                       ● Have an automatically determined content and duration, that best
                         fits the current scenario and configuration.
                         Short sequences result in waveform files with small size.
                         Short sequences are repeated, longer once are truncated.
                         See also Table 16-2.

Remote command:
SCENario:OUTPut:CLOCk:MODE on page 760
SCENario:OUTPut:CLOCk:AUTO:BORDer on page 761
SCENario:OUTPut:CLOCk:AUTO:OVERsampling on page 760
SCENario:OUTPut:CLOCk:USER on page 761
SCENario:OUTPut:DURation:MODE on page 761
SCENario:OUTPut:DURation:TIME on page 761

Marker Signals
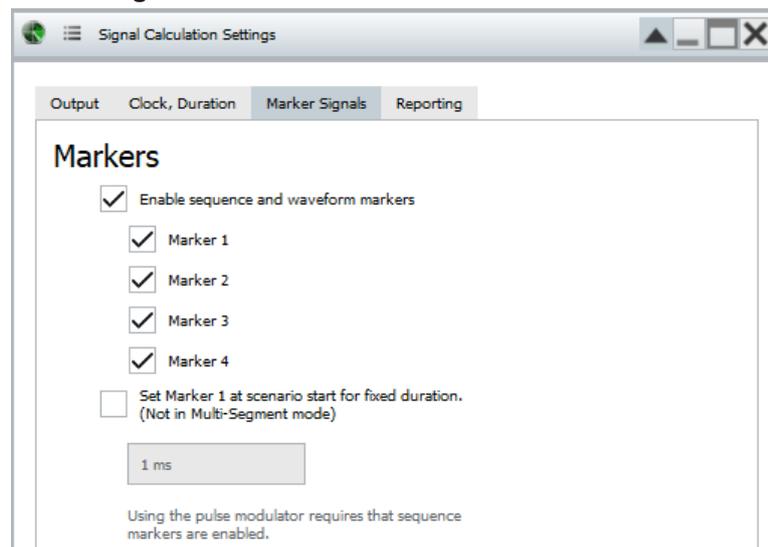
Enables configured markers to be considered in the generated file.

See "Global Marker Signals" on page 536.

"Set Marker 1 at scenario start for fixed duration"
                       You can also enable an additional marker, that is held high from the
                       scenario start for a selected duration. This marker signal is output as
                       marker 1. If other marker signals are defined, all markers are sum-
                       med.
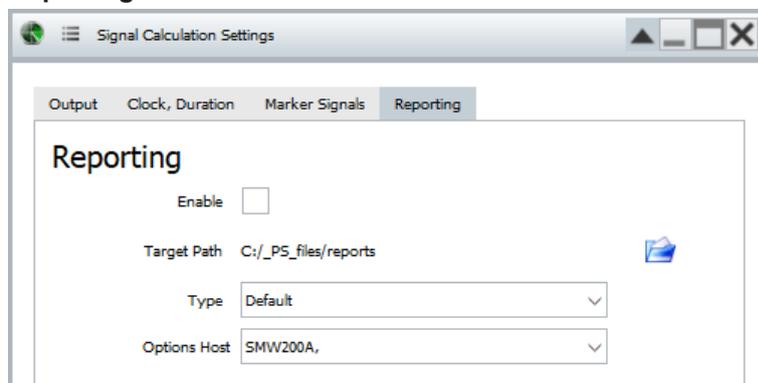
Remote command:
SCENario:OUTPut:MARKer:ENABle on page 680
SCENario:OUTPut:MARKer:FLAGs on page 680

SCENario:OUTPut:MARKer:SCENario:ENABle on page 680
SCENario:OUTPut:MARKer:SCENario:DURation on page 681
SCENario:OUTPut:MARKer:PMODulator:ENABle on page 681

Reporting

Enables and configures report file generation.

Note: Settings change depending on which "Type" option is selected.

See Chapter 22, "Creating reports and documenting measurement results",
on page 542.

17.2 Assign signal source or receiver signals to generator

In "Localized Emitters" and in "Direction Finding" scenarios, you have to select the signal of which emitter/interferer or receiver is transmitted to and played by the connected instrument.

Access:

► Perform one of the following:
- In the "Scenario" dialog, select "Target > Assign > Config".
- In the 2D map of a "Scenario Type = Localized Emitters", select "Assign Emitters".
- In the 2D map of a "Scenario Type = Direction Finding", select "Assign Receiver Signals".

The name of the dialog changes depending on the scenario but the settings and the workflow are similar.
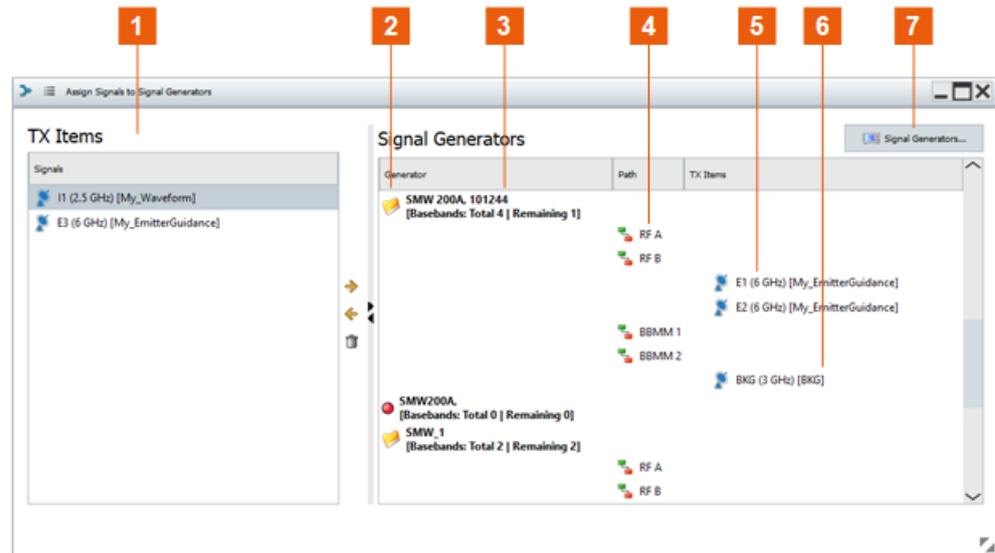
Figure 17-1: Assign Signal to Signal Generators: understanding the displayed information (Scenario > Target = Generator)

1 = Unassigned Tx items
2 = Icon folder: virtual generator profile, no physical generator is assigned
3 = List of generators (entries depend on the "Scenario > Target", see Available for assignment); indicated are generator name, physical instrument name and serial number, available and assigned resources (basebands and emitters, if interleaving is enabled)
4 = Indicates synchronized generator ("Primary/Secondary")
5 = Signals of two emitters, assigned to the same RF path
6 = Signal assigned to external RF, connected to and controlled from the R&S SMW
7 = Opens the "Signal Generators" dialog (see Chapter 3.6, "Signal Generators settings", on page 75)

In multi emitter scenarios with interleaving, the dialog contains one additional assailment level, the interleaving groups, see Interleaving Groups and Groups Content.

For step-by-step instructions, see:

● Chapter 16.1, "How to create scenarios with receiver and TX items", on page 416
● "To create, load and play the waveforms of a direction finding scenario automatically" on page 492

Settings:

TX Items

Common representation of the available signals.

Emitters/Interferers ← TX Items

Lists the alias names of all configured emitters and interferers, that are still not assigned to a generator.

Remote command:

ASSignment:EMITters:LIST? on page 638

ASSignment:EMITters:SELect on page 593

Receiver Signals ← TX Items

Lists all receiver signals, that are still not assigned to a generator.

There is one receivers signal per receiver antenna element and emitter combination. The name of the receiver signals follows the syntax:

"<Antenna Element Alias> - <Emitter Name Alias>", as selected with the parameters:
- Antenna Element Name
- Emitter/Interferer Alias Name

Remote command:

ASSignment:ANTennas:LIST? on page 639

ASSignment:ANTennas:SELect on page 593

Interleaving Groups ← TX Items

In scenarios that have interleaving enabled, the "Interleaving Groups" list shows all interleaving groups defined for the selected scenario.
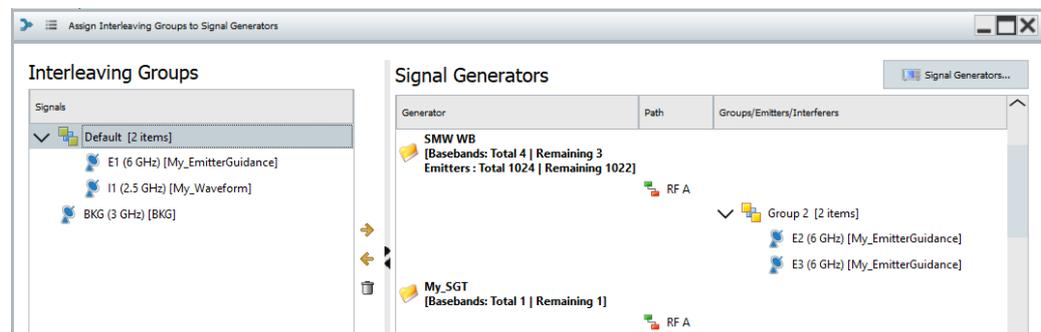
Click the group name to expand the group content. Indicated are all emitters assigned to the particular group.

Interleaving groups are assigned jointly. Select the group name and Drag&Drop it to one of paths.

The number of available and remaining emitters is updated automatically. The number of available resources depends on the option of the selected instrument. A total number of 1024 emitters for instance requires a two-path instrument with 2 x R&S SMW-K307 installed, see "Multiple Emitters" on page 92.

Remote command:

ASSignment:GROup:LIST? on page 640

ASSignment:GROup:SELect on page 640

Add, Remove, Clear

Adds (or removes) a selected emitter/interferer to the selected RF path, incl. connected external RF instruments (R&S SGT, R&S SGS also together with R&S SGU).

Alternatively, use the Drag&Drop method.

Note:

You can assign several signals to the same RF path. Assigning several signals is possible, if the RF frequency and the bandwidth of the resulting signal do not exceed the supported RF frequency and baseband bandwidth of the physical instrument.

Remote command:

ASSignment:GENerator:PATH:EMITter:ADD on page 591

ASSignment:GENerator:PATH:EMITter:SELect on page 593

ASSignment:GENerator:PATH:EMITter:DELete on page 594

ASSignment:GENerator:PATH:EMITter:CLEar on page 596

ASSignment:GENerator:PATH:ANTenna:ADD on page 591

ASSignment:GENerator:PATH:ANTenna:SELect on page 593

ASSignment:GENerator:PATH:ANTenna:DELete on page 594

ASSignment:GENerator:PATH:ANTenna:CLEar on page 596

Signal Generators

Lists the generators with their names, available paths and assigned emitters/interferers or interleaving groups with their group content.

Available for assignment ← Signal Generators

The number of listed generators depends on the "Scenario > Target":

- "Scenario > Target" > **"Generator"**
  Listed are the physical generators that are available in the current "Setup" in the "Signal Generators" dialog.
  The displayed information follows the syntax:
  "<Generator Name> (<Instrument Name>,<Serial Number>)"
- "Scenario > Target" > **"File"**
  Listed are all physical and virtual generators that are available in the repository.

The colored LEDs indicate the instrument status:

- **Green** - the instrument is online.
- **Red** - the instrument is currently not online.

Remote command:

ASSignment:GENerator:LIST? on page 639

ASSignment:GENerator:SELect on page 593

ASSignment:GENerator:CAPabilities? on page 639

ASSignment:GENerator:PATH:LIST? on page 639

ASSignment:GENerator:PATH:SELect on page 593

ASSignment:GENerator:PATH:EMITter:LIST? on page 640

ASSignment:GENerator:PATH:ANTenna:LIST? on page 640

Path ← Signal Generators

Indicates the available paths per generator.

Groups/Emitters/Interferers ← Signal Generators

Indicates the alias names of the assigned signal sources.

Groups Content ← Signal Generators

In scenarios with enabled interleaving, indicates the emitters or PDW lists that are assigned to the particular path of the generator.
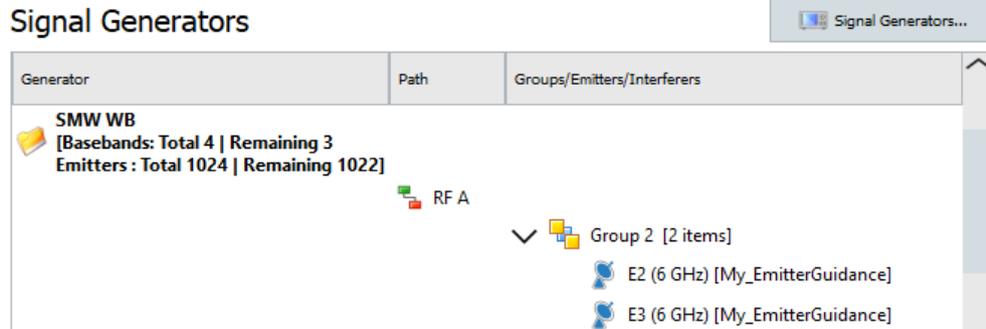

17.3 File transfer settings

Waveform files generated with the software are automatically uploaded to the connected signal generator in one of the following alternative ways:

- Via the SCPI interface
- Via FTP transfer
  If enabled, this second option is automatically applied when the calculated files are large.

Both methods require that the signal generator and the PC on that the software is installed are connected to the same LAN.

17.3.1 How to enable FTP transfer

1. In the menu bar, select "Configure > Settings > File Transfer".
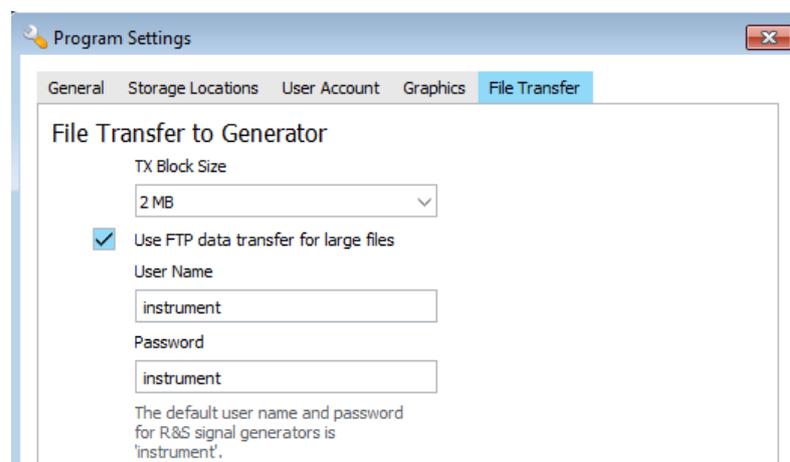
2. If necessary, change the size of the transport blocks transmitted via SCPI interface.

3. To use the FTP data transfer:

   a) *In the signal generator*, enable the FTP interface.
      You find the required settings in the "Setup > Security" dialog.
      For more information, see the signal generator's embedded help or refer to its user manual (see Chapter G, "References", on page 896).

   b) *In the R&S Pulse Sequencer*, enable an FTP data transfer to speed up the data upload to the connected signal generator.
      The FTP transmission requires "User Name" and "Password". The default "User Name" and "Password" is *instrument*.

   Note: The R&S Pulse Sequencer attempts to connect to *any* connected signal generator with the *same credentials* (i.e. "User Name" and "Password").

   Depending on the file size, the software uses either the SCPI interface or the FTP data transfer.

17.3.2 File transfer settings

Access:
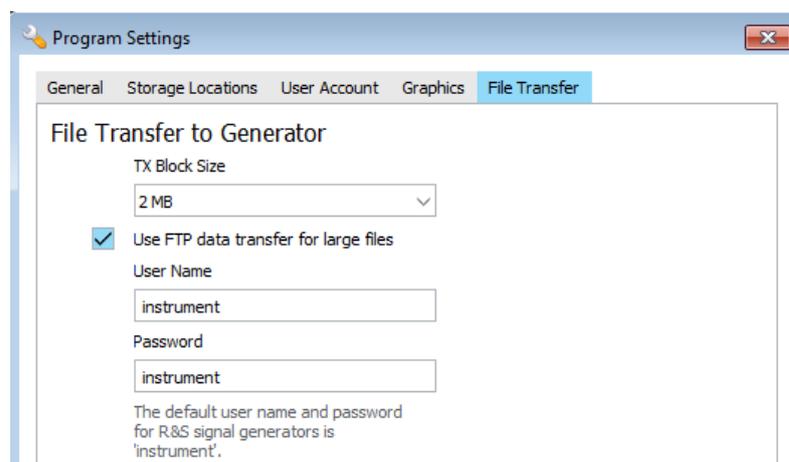
► In the menu bar, select "Configure > Settings > File Transfer".

TX Block Size
Sets the size of the transport blocks transmitted via SCPI interface.

Use the FTP data transfer for large files
Access: in the menu bar, select "Configure > Settings > File Transfer".

Select a "TX Block Size".

Enable an FTP data transfer to speed up the data upload to the connected signal generator.

Enter the "User Name" and "Password" of the processing instrument.

How to: Chapter 17.3.1, "How to enable FTP transfer", on page 485.

Remote command:
PROGram:TRANsfer:FTP:BLOCksize on page 599
PROGram:TRANsfer:FTP:ENABle on page 600
PROGram:TRANsfer:FTP:UNAMe on page 600
PROGram:TRANsfer:FTP:PASSwd on page 600

17.4 Control panel settings

Access:

► In the menu bar, select the "Tools > Control Panel".
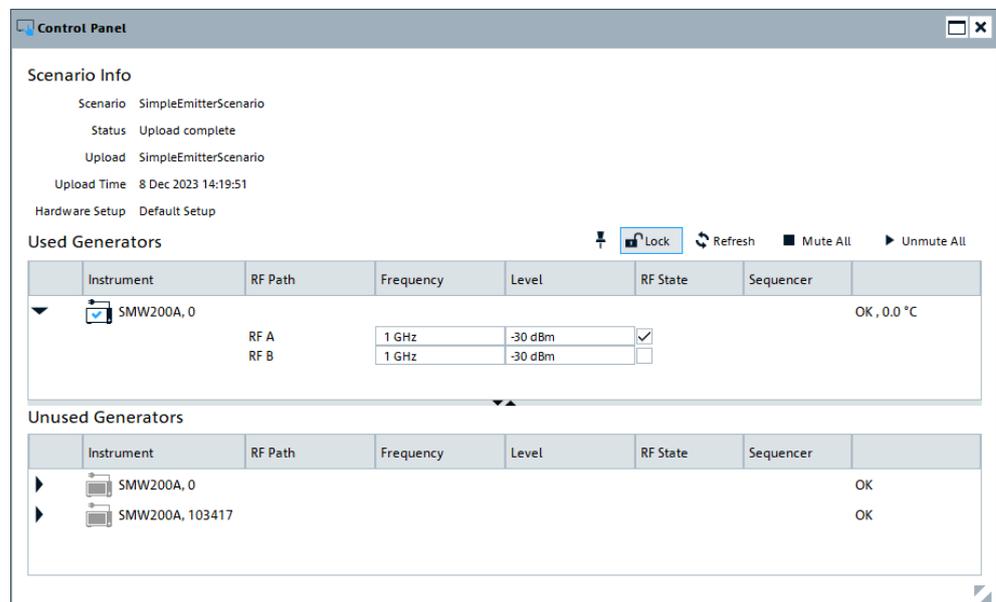
The "Control Panel" is a dashboard window. It displays key status information on the physical instruments that are configured in the selected tab of the "Signal Generators" dialog.

From the "Control Panel" dialog, you can change the level and frequency settings directly in these instruments. No need to install extra software, write dedicated remote control programs or change the settings manually.

This dialog is especially useful, if your test setup consists of several R&S SGT instruments.

Scenario

Indicates the name of the current used scenario.

Remote command:
CPANel:SCENario:CURRent:NAME? on page 632

Status

Status information, like "Calculated", "Upload complete", "Ready" or "Execution Failed".

"Idle" indicates that there is no loaded or calculated scenario, for example right after starting up the R&S Pulse Sequencer.

Remote command:
CPANel:SCENario:CURRent:STATus? on page 632

Upload

Indicates the name of the last uploaded scenario. That is, the scenario that has been calculated and stored (locally or on the hard drive of the instrument) last.

Remote command:
CPANel:SCENario:DEPLoyed:NAME? on page 632

Upload time

Indicates the date and time the scenario has been uploaded.

"Unknown" indicates that there is no loaded or calculated scenario, for example right after starting up the R&S Pulse Sequencer.

Remote command:
CPANel:SCENario:DEPLoyed:TIME? on page 632

Hardware

Indicates the name of the current selected setup. The current hardware setup is the tab name selected in the "Signal Generators" dialog, see Chapter 3.6, "Signal Generators settings", on page 75.

Remote command:
CPANel:SETup? on page 632

Pin/Unpin

Displays the window always on the top of all other windows.

Unlock/Lock

Unlocks the RF frequency, level, and output state settings for editing.

Refresh

Updates the displayed information.

Remote command:
CPANel:REFResh on page 631

Mute All

Deactivates the RF outputs of all signal generators.

Remote command:
CPANel:MUTE on page 631

Unmute All

Activates the RF outputs of all signal generators.

Remote command:
CPANel:UNMute on page 631

Used Generators, Unused generators
An overview of assigned (used) and the available but not used (unused) signal generators with their main characteristics:

"Generator"   Instrument name as listed in the list of instruments in the Signal Generators dialog.

Remote command:
CPANel:USED:LIST? on page 633
CPANel:USED:SELect on page 593
CPANel:UNUSed:LIST? on page 633
CPANel:UNUSed:SELect on page 593

"RF path, Frequency, Level, State"
Available RF paths, current RF frequency, level and output state, as configured for the last deployed scenario.
Indicated are also the RF frequency, level, and output states of the RF extensions. RF extensions is a term describing the instruments connected to the R&S SMW, like R&S SGT.
To edit the values and the states, select "Unlock".

Remote command:
CPANel:USED:PATH:LIST? on page 633
CPANel:UNUSed:PATH:LIST on page 633
CPANel:USED:PATH:SELect on page 593
CPANel:UNUSed:PATH:SELect on page 593
CPANel:USED:FREQuency on page 633
CPANel:UNUSed:FREQuency on page 633
CPANel:USED:LEVel on page 633
CPANel:UNUSed:LEVel on page 633
CPANel:USED:STATe on page 634
CPANel:UNUSed:STATe on page 634

Additional information
Displays additional information, like temperature and the presence of static errors due to missing reference, licenses, adjustment data.

17.5 How to create and play the waveforms of simple scenarios

This example uses the test setup as shown on Figure 3-1 (on page 56).

1. Open a scenario.

2. Configure the settings, for example, as follows:
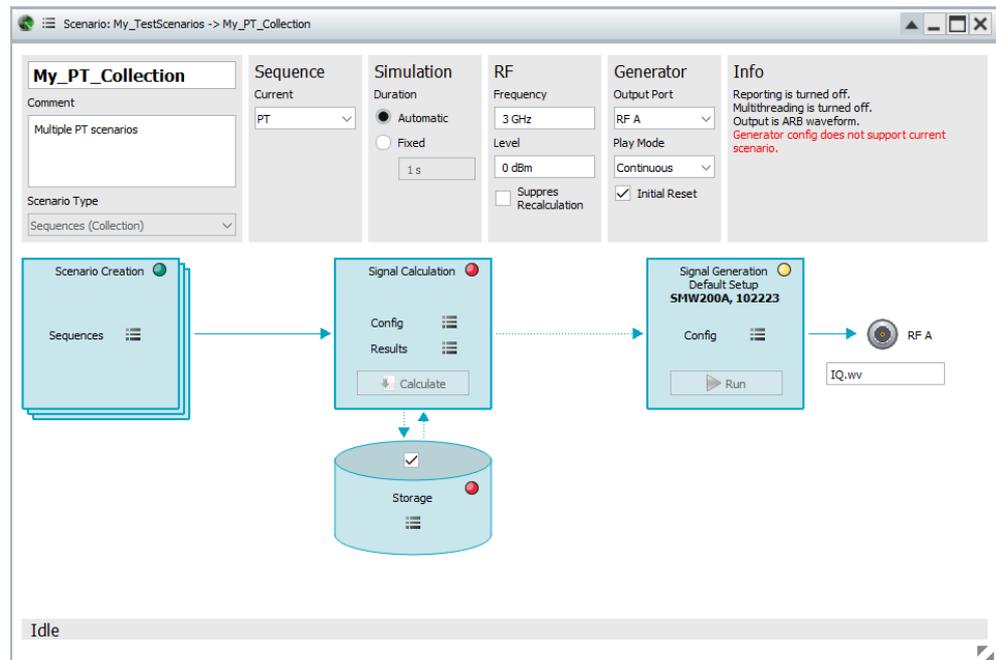
3. Select "Signal Generation > Config > context menu".
   Select:

   a) "Target" > "Generator" and select a generator, e.g. "SMW200A,102030".
   b) "Signal Generators" and select one from the Signal Generators dialog.

   The information displayed in the "Signal Generation" block confirms the selected generator.

4. Observe the status indication.

   - **Green LED**: indicates that the selected generator is suitable for the current scenario;
     If all other prerequisites are fulfilled, the "Run" function is active.
   - **Yellow LED**: indicates insufficient capabilities
   - **Red LED**: indicates that no generator is assigned

5. Use the default "Waveform Generator" settings.

   All prerequisites are fulfilled.
   The "Calculate" button is enabled.

6. Select "Calculate".

   The software calculates the waveform. Progress information indicates the calculation progress.

7. Select "Load" to transfer the waveform to the instrument.

   - The waveform is automatically loaded in the "ARB".
   - The output frequency and level of the signal generator are set accordingly.
   - Signal generation is started and the signal is output at the [RF A] connector.
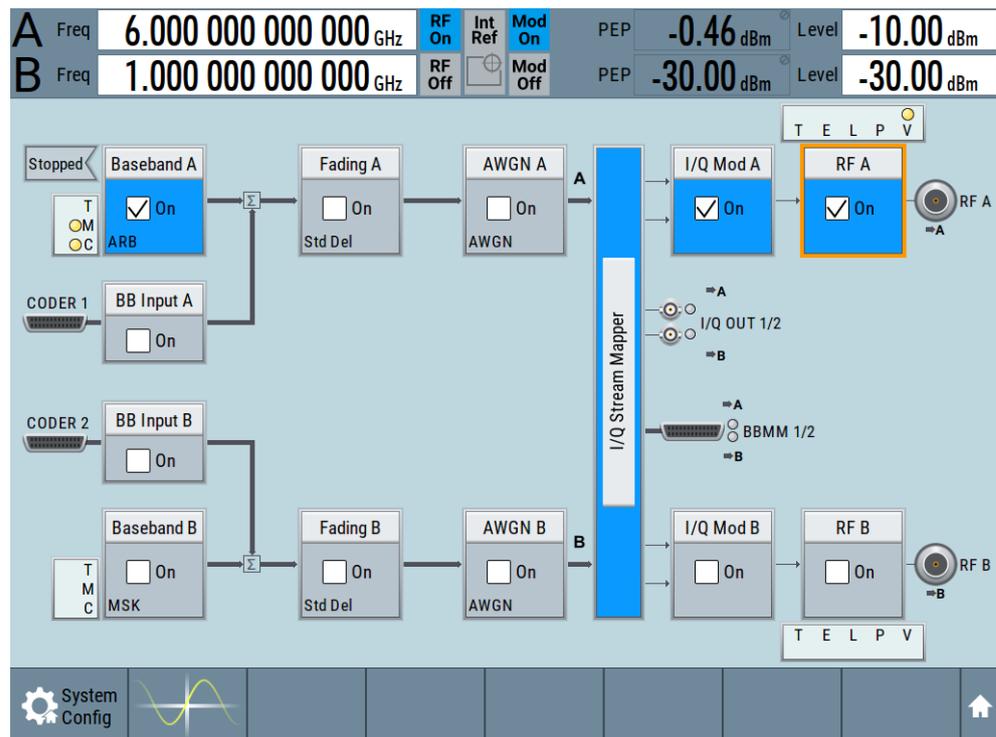
In the signal generator, perform further configurations, e.g.:

● Select "Baseband > ARB > General > Waveform Info" to confirm that the waveform generated with the R&S Pulse Sequencer.
  You retrieve also information on the waveform filename, used clock rate, storage location, etc.

Filename: IQ.wv
Path: /var/user/RS_PulseSequencer/0f0f28f6-
e800-45c9-8f8f-9db7a2c6d53f/2/
Date: 2018-08-07;16:53:27
File size 96605 Bytes
Comment: SimplePulseTrain
Clock: 2.4e+06 Hz
Marker 1 Available
Marker 2 Available
Marker 3 Available
Option String: SMx-K300 (Pulse Sequencer), SMx-K301
(Advanced Pulse Sequencer)
Peak Level: 0.000 dBFS
RMS Level: -9.542 dBFS
Samples: 24000

OK
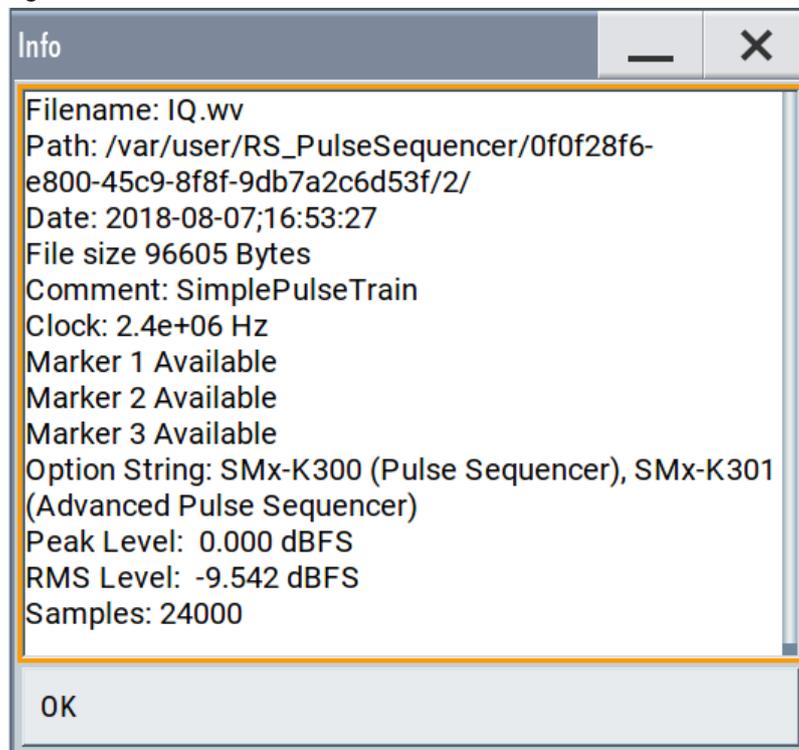
● The generated signal contains marker signals.
Use the default routing or change the mapping of the marker signals to the output connectors.

For details, see the user manual of the corresponding base unit.

In the R&S Pulse Sequencer:

● Select "Signal Calculation > Result > View" to visualize the generated waveform.
The display resembles the one on Figure 7-9.

Playing the waveforms of 2D scenarios

If your current scenario type is "Localized Emitters" or "Direction Finding", you also have to:

● Select the signal of which emitter/interferer/receiver is transmitted to the connected signal generator

● In a test setup with two or more signal generators, connect them as illustrated in the "Wiring Diagram".

For a detailed description, see Chapter 17.6, "How to play the waveforms of 2D scenarios", on page 492.

To delete waveforms from the hard disk of the signal generator

If the waveforms uploaded on the vector signal generator are no longer required, they can be deleted automatically from the R&S Pulse Sequencer:

1. Open the "Scenario" dialog.

2. In the context menu of the "Signal Calculation" block, select "Sanitize".
Select for example "This Scenario".

The waveforms belonging to the current scenario are deleted from the hard disk of the signal generator.

17.6 How to play the waveforms of 2D scenarios

The following example shows how to play a complex "Direction Finding" scenario. To play "Localized Emitter" scenarios, follow the same workflow.

To create, load and play the waveforms of a direction finding scenario automatically

In the following, we use R&S SMW and two R&S SGSs connected to its digital I/Q interfaces as external RF instruments.

The setup is similar to the test setup illustrated on Figure 3-2.

In complex 2D scenario ("Localized Emitters" and "Direction Finding"), you have to select the signal of which emitter/interferer or receive antenna is output on which signal generator RF path.

1. Connect the instruments.

2. Open the "Signal Generators" dialog.
   Configure your setup.
   Follow the description in Chapter 3.3, "Configuring the test setup in the R&S Pulse Sequencer", on page 60.
   Observe the "Wiring Diagram".
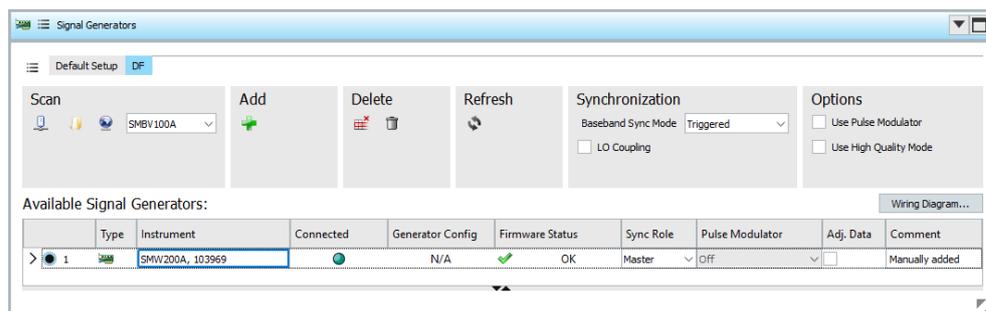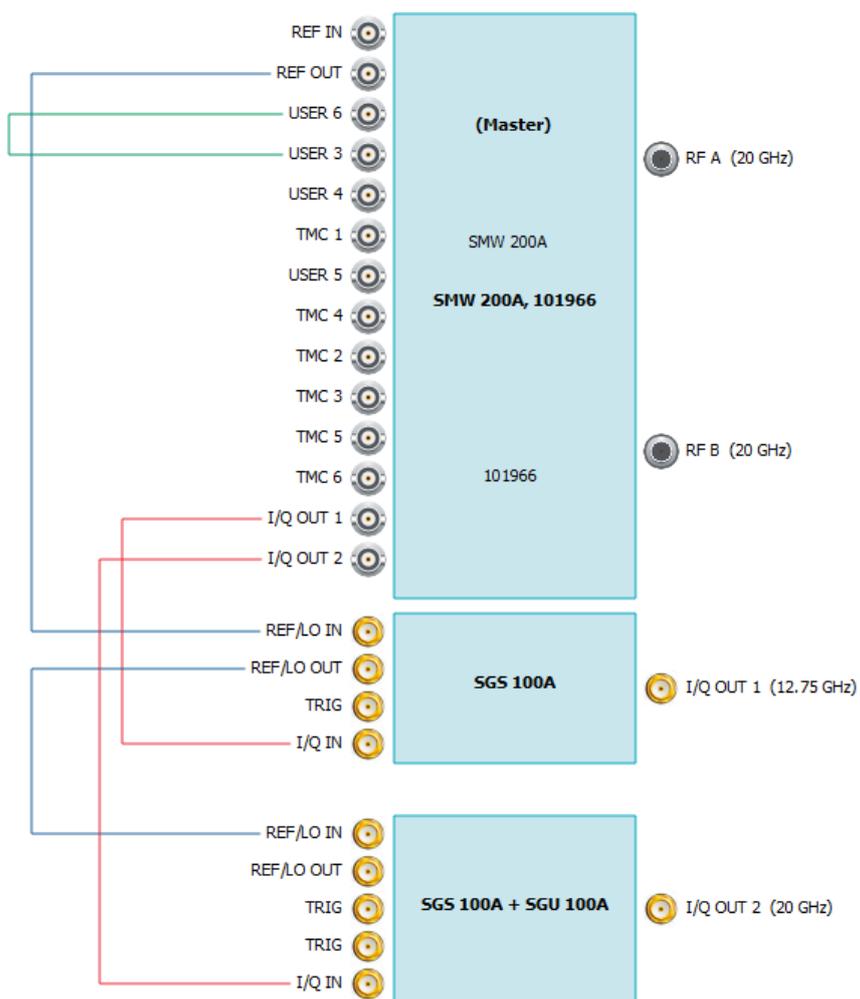
3. Open a test scenario with "Scenario Type" > "Direction Finding".
   Use, for example, the scenario created in "How to create the scenario and configure the receiver" on page 418.
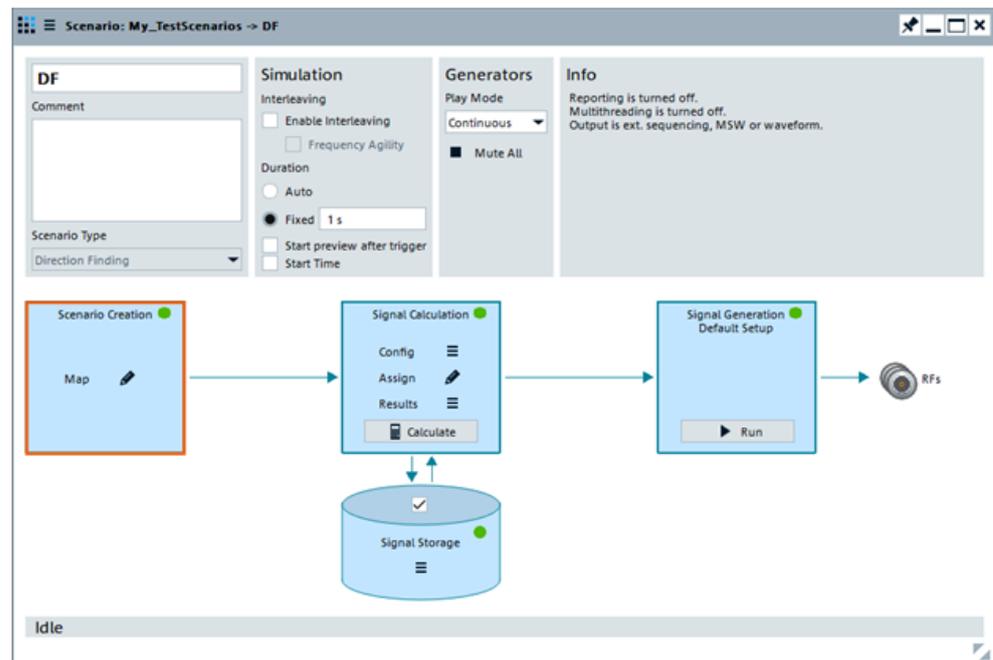
4. Select "Signal Generation > Config > Target > Generator".

5. If you want to, you can select the "Start preview after trigger" checkbox in the "Simulation" area.
   This option opens the 2D map of the scenario and shows the movement preview after the setup is triggered.

6. Select "Signal Generation" > "Assign".

   The "Assign Receiver Signals to Generators" dialog opens.

7. Drag and drop a signal to assign it to a generator.
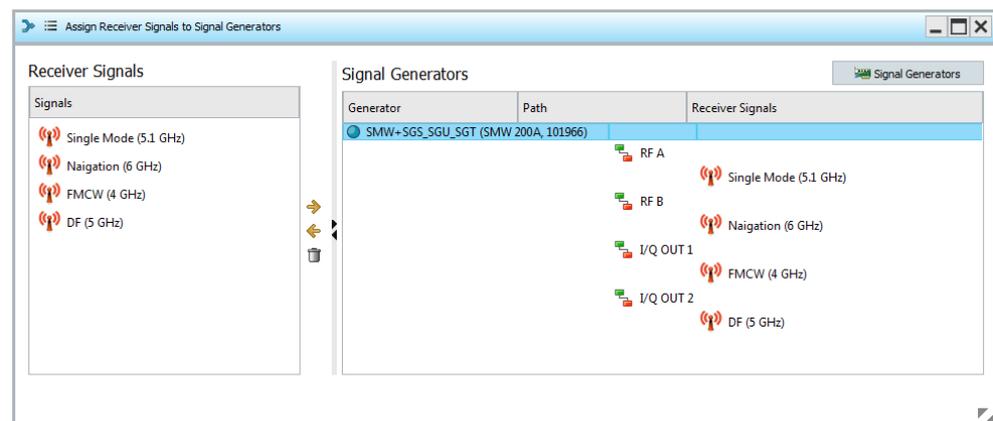
8. In the scenario dialog, select "Signal Calculation > Calculate".

   The software calculates the waveforms. Progress information indicates the calculation progress.

The waveforms are stored in the volatile memory and transferred to the instruments.

9. To confirm the file transfer to the R&S SMW:

   a) Press [SAVE/RCL] > "File Manager".
   b) Open the `RS_PulseSequencer/...` directory.

IQ_E1_R4_G1_P5.wv	30.03.2016 11:26	WV File
IQ_E1_R4_G1_P5.wvs	30.03.2016 11:26	WVS File
IQ_E1_R2_G1_P1.wvs	30.03.2016 11:26	WVS File
IQ_E1_R3_G1_P4.wv	30.03.2016 11:26	WV File
IQ_E1_R3_G1_P4.wvs	30.03.2016 11:26	WVS File
IQ_E1_R1_G1_P0.wv	30.03.2016 11:26	WV File
IQ_E1_R1_G1_P0.wvs	30.03.2016 11:26	WVS File
IQ_E1_R2_G1_P1.wv	30.03.2016 11:26	WV File
stat.ini	30.03.2016 11:26	INI File

10. In the R&S Pulse Sequencer, select "Signal Generation > Run".

   The waveforms are loaded in the "ARB" of the R&S SMW.
   The output frequency and level of the signal generators are set accordingly.
   Signal generation is started synchronous in all basebands; the signal is output at the [RF] connectors.

To create, transfer and play the waveforms of scenarios manually

During the initial tests, it is useful to create test waveforms and store them on a local or network drive. These waveforms can be later transferred to and played with a signal generator.

1. Open a test scenario with "Scenario Type > Direction Finding".

2. Select "Signal Generation > Config > Target" > "File".

3. Select "Local File > Config > Set Path".
   Select the storage location for the generated waveform.

4. Select "Repository > On".

5. Select "Calculate".

   The software calculates the waveform. Progress information indicates the calculation progress.
   The waveforms are created, automatically named and saved in the selected directory.

6. If your user rights are sufficient, you can store the generated file also in the repository.
   Select "Signal Calculation > Results > Save calculated results".

7. Select "Local File > Open Path in Explorer".

   In this example, the directory contains the `Setup_Info.txt` file and four waveforms and four multi-segment waveforms, one for each receiver antenna-emitter combination.

IQ_E1_R1_G1_P0.wv	WV File	99 KB
IQ_E1_R1_G1_P0.wvs	WVS File	1 KB
IQ_E1_R2_G1_P1.wv	WV File	94 KB
IQ_E1_R2_G1_P1.wvs	WVS File	1 KB
IQ_E1_R3_G1_P4.wv	WV File	54 KB
IQ_E1_R3_G1_P4.wvs	WVS File	1 KB
IQ_E1_R4_G1_P5.wv	WV File	94 KB
IQ_E1_R4_G1_P5.wvs	WVS File	1 KB
Setup_Info.txt	Text Document	1 KB

8. Open the `Setup_Info.txt` file.

```
Repository:    DF
Scenario:      DF
Creator:       R&S
Date:          Mi Mrz 30 10:45:31 2016

Setup Table

Generator   |  Path  |   Waveform File    | Frequency |    Level    | Trigger offset |
-----------------------------------------------------------------------------------------
SMW + 2xSGT | Path A | IQ_E1_R1_G1_P0.wv  |   5 GHz   |  -92.76 dBm |     0    s   |
SMW + 2xSGT | Path B | IQ_E1_R2_G1_P1.wv  |   5 GHz   |  -94.78 dBm |   1.523 ns  |
SMW + 2xSGT | Path C | IQ_E1_R3_G1_P4.wv  |   5 GHz   | -133.87 dBm |  27.893 ns  |
SMW + 2xSGT | Path D | IQ_E1_R4_G1_P5.wv  |   5 GHz   | -102.51 dBm |  26.371 ns  |
-----------------------------------------------------------------------------------------
```

To create, load and play the waveforms of a 2D scenario automatically

► Follow the instructions listed in "To create, load and play the waveforms of a direction finding scenario automatically" on page 492

17.7 How to use the extended sequencing option

This example shows how to create waveform that can be played with R&S SMW if equipped with the option R&S SMW-K501.

The example shows how to calculate an ARB waveform file, then change the output format and compare the size of the created output files.

To accelerate the calculation, we configure a long sequence of pulses without modulation.

1. Create a pulse with "Rise Time = Fall Time = 0 us", "Pulse Width = 10 us", "Standard Profile = 0/100" and "MOP = Off".

2. Create a sequence with this pulse. Set "PRI = 20 us" and "Rep Count = 9999".

3. Create a single sequence scenario.
   Assign the created sequence to it.

4. Select "Signal Generation Local File > Config > Target > File".
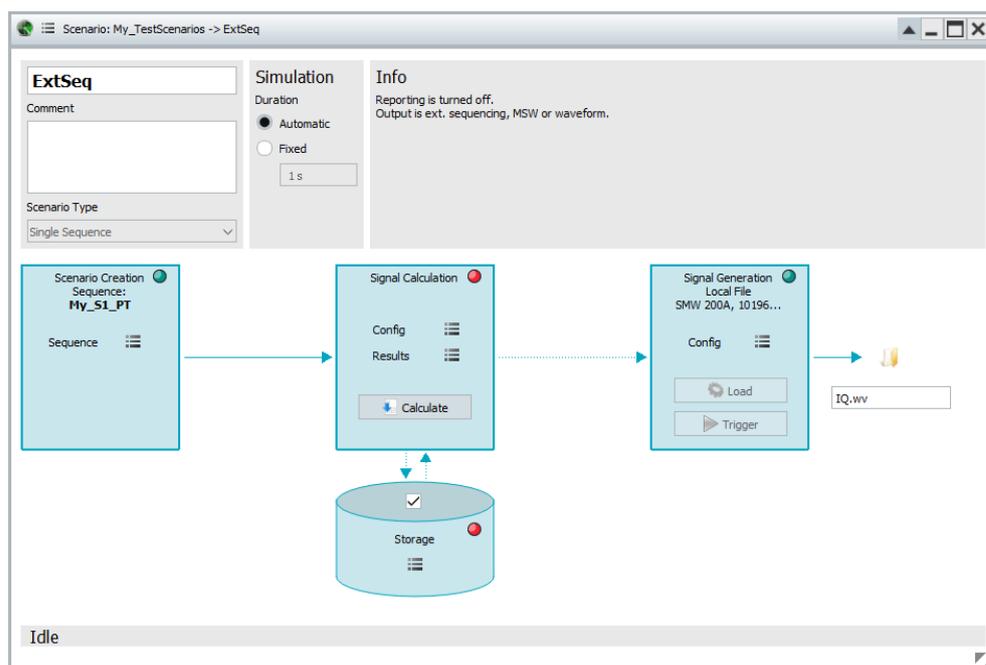
5. Select "Signal Calculation > Config > Config > Waveform Generation > Config > Output > Output Format" > "Create ARB waveform".
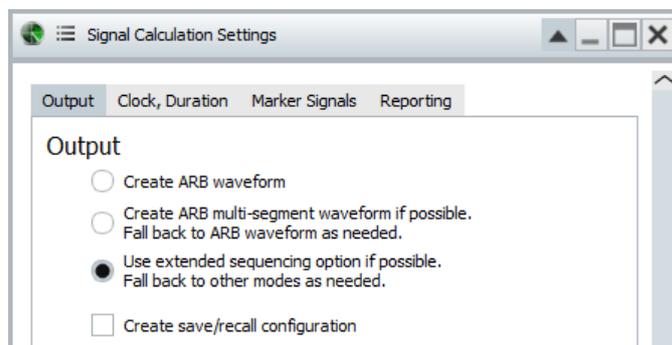
6. Select "Signal Calculation > Calculate".
   Wait for the calculation to complete.

7. Select the "Open File location" icon to reveal the file in the file system.
   Observe the file size.

8. Select "Signal Calculation > Config > Output > Output Format" > "Use extended sequencing option ...".

9. Select "Calculate".

10. Select the "Open File location" icon.
    Compare the file size.

IQ.ps_adr	PS_ADR File	1 KB
IQ.ps_def	PS_DEF File	129 KB
IQ.wv	WV File	2 KB

11. Transfer the files to the connected R&S SMW **automatically**.

    a) Select "Signal Generation > Config > Target > Generator".
    b) Select "Target > Select". Select a suitable generator profile from the list.
       The generator must exist and must be linked to a connected R&S SMW instru-ment.
    c) Select "Prepare".

    The files are stored on the hard disk of the instruments.

12. Select "Load".

    The files are loaded in the dedicated "Extended Sequencer" dialog and not in the ARB generator

13. To confirm, select "Baseband > Extended Sequencer".
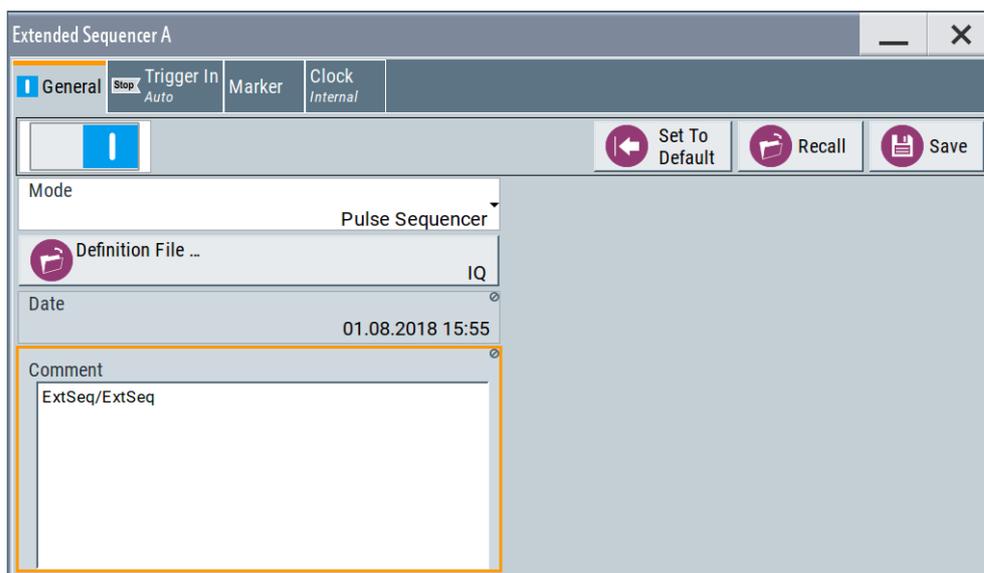
The display confirms current repository and scenario names.
For concise description of the provided settings, see user manual R&S SMW-K501/-K502/-K503/-K504 Extended Sequencer.

18 Working with PDWs

The pulse description word (PDW) is a commonly used file format that describes radar signals. The PDWs contain the radar signal parameters for each pulse together with a timestamp that defines the pulse start time. Sometimes the PDWs can originate from earlier simulations or live recordings. In other cases, you would like to generate the radar test signal directly in PDW format.

In R&S Pulse Sequencer, you have the following different possibilities to use PDWs:

● **Import custom PDW lists ("Imported Signals")**
  Instead of configuring pulses, MOPs, sequences and IPM effects, you can import your custom PDW lists.
  Based on files with mapping rules, the R&S Pulse Sequencer PDW import interface transforms the custom-specific PDW list into the Rohde & Schwarz proprietary PDW format.
  Waveform elements created in this way can be used as any other waveform element, for example, used as background emitters, interferes, included into sequences.
  Custom PDWs can be imported in any scenario type, but the created output file differs depending on the scenario type:

  – The "PDW List (Collection)" scenario is dedicated for importing custom PDWs. The generated files are compatible with the file format that can be loaded in and processed by R&S SMW equipped with the option extended sequencer (R&S SMW-K501/K502).

  – In scenario other than the "PDW List (Collection)", the created output file is a waveform.
    Waveforms describing long simulation time are usually large.

● **Create files for the extended sequencer option R&S SMW-K501/K502 automatically**
  Instead of creating single-segment or multi-segment waveform files, you can configure the R&S Pulse Sequencer to create PDW lists in Rohde & Schwarz proprietary file format.
  The created output files are smaller in size compared to the waveform files. A further advantage is the increased duration of the generated signal.
  PDW files can be created in two ways:

  – *Internally*, from the configured pulses, MOPs, sequences and IPM effects.

  – *Externally*, by importing custom PDW lists, as described above.

  For details on the file format, see R&S SMW-K501/-K502 Extended and Real-Time Sequencing user manual

● **Merging the signals of several emitters** (several PDW lists)
  Per default, you use the "PDW List (Collection)" scenario to configure several PDW lists and switch between them sequentially.
  However, you can also combine them into a single output file. During the interleaving process, overlapping PDWs with lower priority are dropped, where you define the priority of the individual PDW lists (see Figure 18-3).

The solution above introduce the possibilities to generate or regenerate test signals with the R&S Pulse Sequencer.

Alternatively to these solutions and without using the R&S Pulse Sequencer, you can **stream the radar scenario** (that is saved PDWs) to **the R&S SMW**. For details and interface description, see R&S SMW-K503/-K504 Real-Time Control Interface user manual

For step-by-step instructions, see:

● Chapter 18.1, "How to import PDW lists and create output files", on page 501.

For description of the related settings, see:

● Chapter 18.5, "PDW list (Collection) settings", on page 514.
● Chapter 18.7, "PDW interleaving progress statistics", on page 519.
● Chapter 18.8, "Signal preview with interleaved and dropping statistics", on page 520.

18.1 How to import PDW lists and create output files

To import PDW list data

The import of custom PDW list files requires a template that describes how to extract the information form the PDW list file.

The instruction shows you how to operate the software and import the PDW list files. We assume that the template file is created, as described in the document "R&S Pulse Sequencer PDW Import Interface Specification".

1. Select "Repository Tree > Waveform & Imported Signals > New".

2. Select "Type > PDW Data".

3. Select "Import".

4. In the "PDW Data Import" dialog, load the PDW list file and the PDW template.

   a) Select the "Select PDW File" icon.
      PDW list files are custom text-based PDW lists in ASCII or coma-separated file format (`*.txt` or `*.csv`).
   b) Select the "Import template file" icon.
      Template files are ASCII files with predefined file format and file extension `*.pdwt.`

5. Select "Process PDW File".

   The PDW list file is parsed. The "Import Status" dialog displays parsing process information.

6. Select "Imported Data" to observe detailed information on the imported file.

   The imported data is temporarily stored in memory.
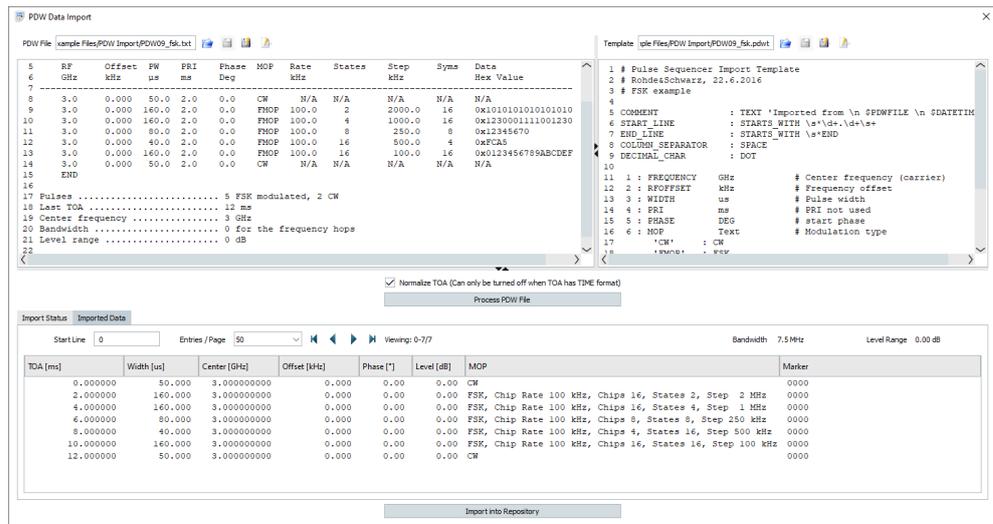
7. Select "Import into Repository" to store the data permanently in the repository.

The imported PDW information can be used as part of a sequence or as a background emitter.

8. Observe the overview information in the "Wave" dialog.
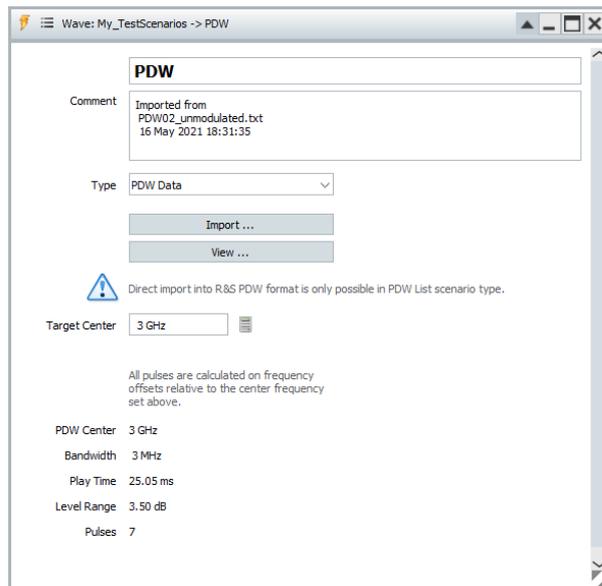
9. Select "View" to display more details on the waveform.

The dialog shows the waveform bandwidth, the level range and the waveform content as a sequence of pulses with their main characteristics.

The displayed information resembles the information in the "PDW Data Import > Imported Data" dialog.

10. Assign the waveform to a sequence.

11. Assign the sequence to a scenario.

   a) Calculate the signal.
   b) In the "Scenario" dialog, select "Signal Calculation > Results > View".

The "Signal Preview" dialog confirms that the calculated waveform consists of seven unmodulated pulses, each with the specified timing settings.
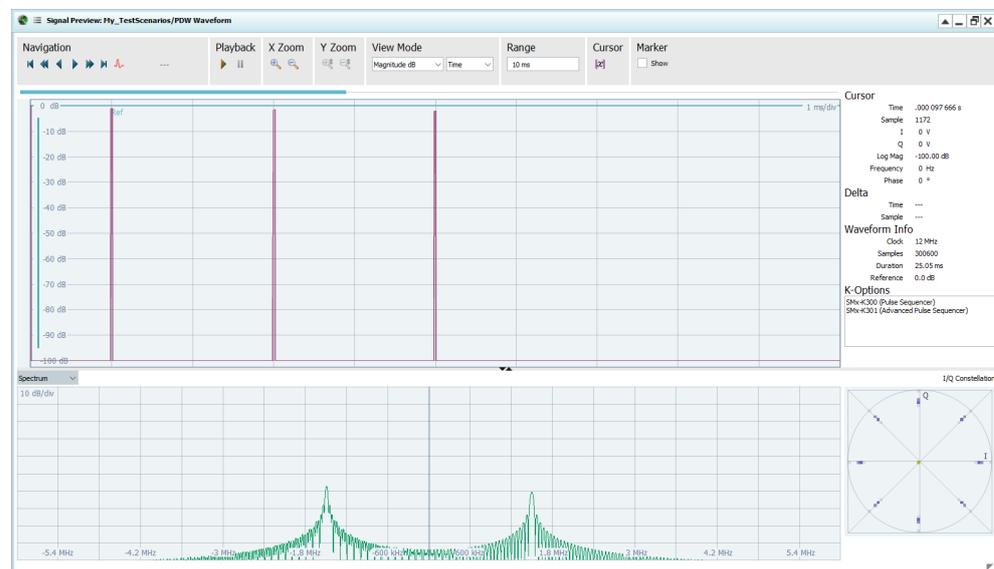
Figure 18-1: Signal preview of waveform calculated from imported PDW list file (first four pulses shown)

For more information, see Chapter 15, "Visualizing and analyzing signals", on page 400.

Create PDW list scenario

1. Select "Scenaros > New" > **"Scenario Type = PDW List (Collection)"**.
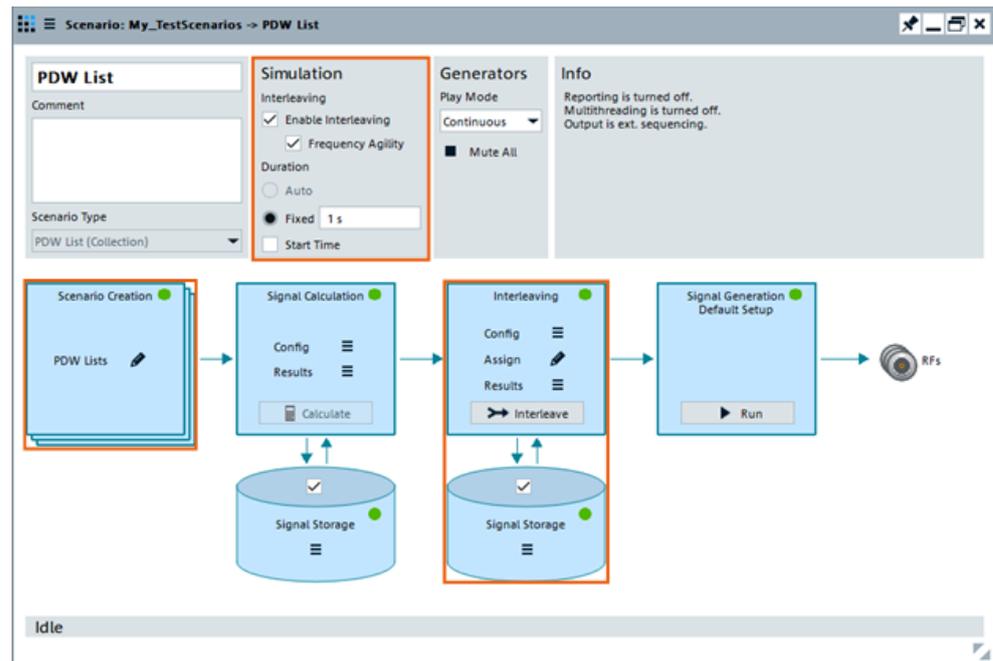
2. Select "Scenario Creation > PWD Lists" to open the "PDW List (Collection)" dialog.

3. Select "Append" to add PDWs.

4. Enter an alias name.
   Select "PDW List" and select an existing waveform element containing an imported PDW.

5. To merge the PDW lists into a single output file, in the block diagram select "Simulation > Interleaving > On".

   The "Interleaving" block appears on the block diagram.

6. Select "Interleaving > Config".

Figure 18-2: PDW List (Collection) Interleaving: understanding the displayed information

1 = Priority of the selected PDW lists
2 = Enables you to change settings for interleaving groups
3 = Enables you to select an interleaving group

7. For each PDW, define a "Priority".

   Multiple lists can be interleaved into a single output file using a priority-based drop-ping algorithm.
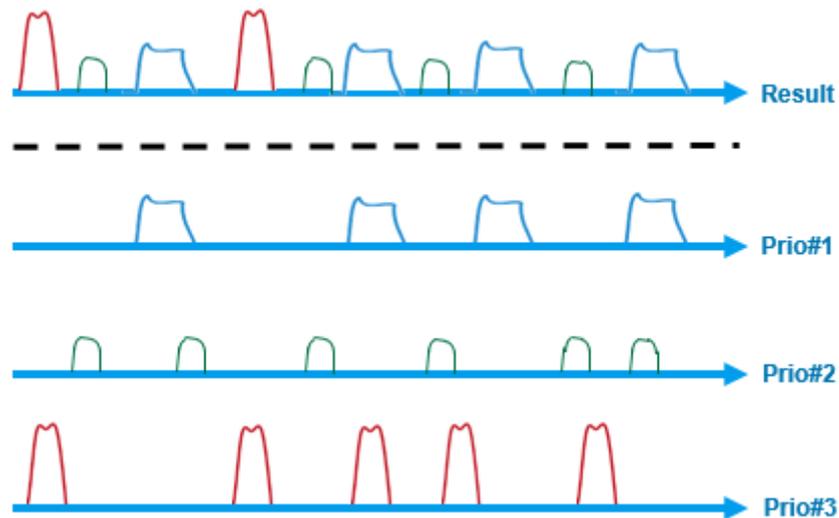
Figure 18-3: Principle of the priority-based interleaving algorithm

Prio#1 = PDW list with the highest priority (i.e. the priority value is 0 or smaller than the value for the other PDW lists)
Prio#3 = PDW list with the lowest priority (i.e. the highest priority value within the PDW lists)

8. Select a signal generator that supports the following **mandatory** options:
   ● Wideband baseband signal generation (R&S SMW-B9)
   ● Multiple emitters (R&S SMW-K306)
   ● Extended sequencing (R&S SMW-K501/-K502).

9. Select "Signal Calculation > Calculate".

   The signal is calculated and the "Signal Calculation" status indication is green.
   The signal calculation and interleaving steps are separated from each other. The interleaving can be applied optionally, as post-processing on the previously calcula-ted data. As long as the "Signal Calculation" settings are not changed, signal recal-culation is not required.

10. Select "Interleaving > Interleave"

    During the interleaving, the "Interleaving progress" display indicates the percentage of pulses being processed and dropped in each of the PDW lists.
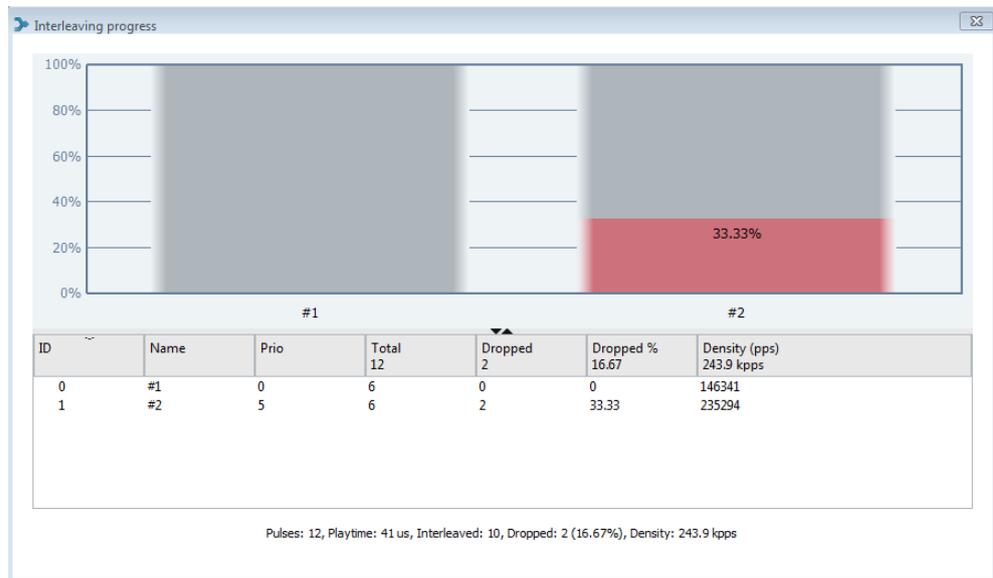
11. To observe more information on the created output files, in the "Scenario" dialog:

   a) Select "Interleaving > Results > View > Interleaved PDW/Dropped PDWs".
      Both views display summary information on the interleaved or dropped PDWs
      and a drop out statistic. The information is represented in a table and in a
      graphical form ("Table"/"Statistic" and "Graphics"/"Histogram").

   b) Select "Interleaving > Results > View > Analyze".

   The "Analyze Dropped PDWs" dialog indicates the PDWs that are and are not
   included in the resulting output file. PDWs displayed in **red are dropped out**.
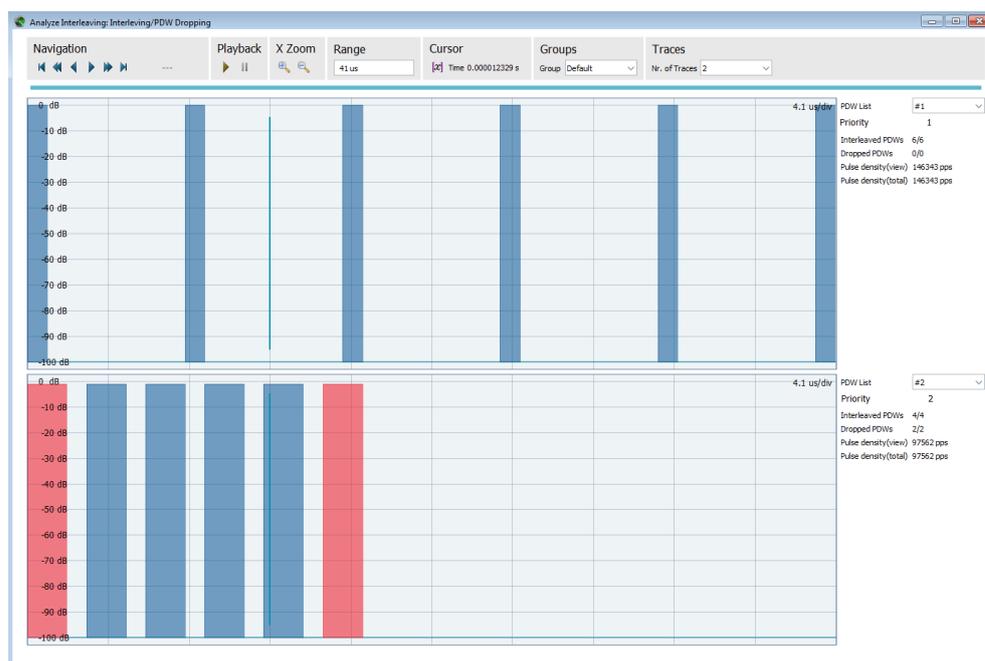
12. To increase the number of interleaved PDWs, add a delay to one of the PDW lists.
Set, for example, "Interleaving > Config > PDW_1 > Time Offset = 11 us".
Create the output signal in the same manner. Observe the resulting output signal.

Depending on the PDW list, multiple or all PDWs from all PDW lists are included in the output file.

18.2 PDW import mechanism

R&S Pulse Sequencer uses a template-based import mechanism for the PDW import. Import templates are human readable text files that describe how information is extracted from the PDW list file. The PDW list file is also a human readable text file using one single row per PDW. The columns contain the various parameters related to the PDW.

The diagram on Figure 18-4 shows the principal concept.
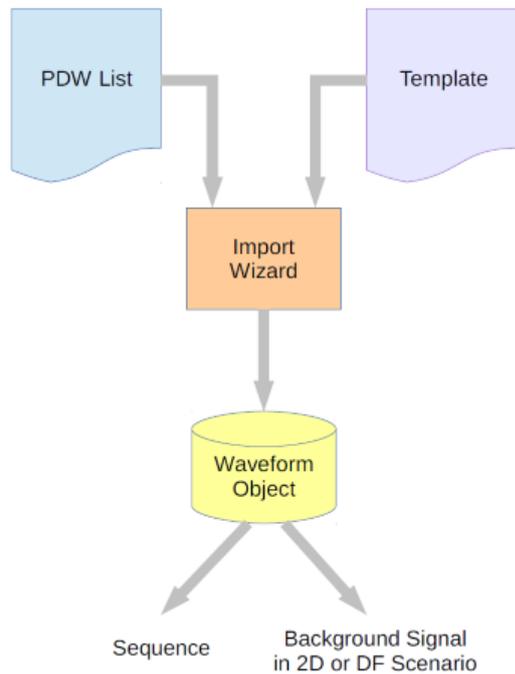
Figure 18-4: Import mechanism principle

The "PDW Data Import" wizard is a dialog that loads both files and shows their content. This dialog is also used to control the import process and accept the imported data.

The import process starts by parsing the text-based template file. Once parsing the template is complete, R&S Pulse Sequencer loads and analyzes the PDW list file. All imported data is temporarily stored in memory until you choose to store the data permanently in the repository.

Storing the data lets R&S Pulse Sequencer copy the extracted PDW data to a waveform object within the R&S Pulse Sequencer repository. The internal storage format is a proprietary binary stream. If future extensions to the data stream are required, R&S Pulse Sequencer automatically converts existing streams to the newer format.

The waveform container with the PDW information can be used as part of a sequence or as a background emitter, for example. Thus, using this signal is identical to using a captured RF signal, digital standard signals originating from R&S® WinIQSIM2™, or background emitter noise.

After the import process has completed the original template and PDW files are no longer required.

For description of the related settings, see:

● Chapter 18.4, "PDW data import wizard settings", on page 511.

For step-by-step instruction, see:

● "To import PDW list data" on page 501.

18.3 PDW data settings

Access:

1. Select "Repository Tree > Waveform & Imported Signals > New".

2. Select "Type = PDW Data".
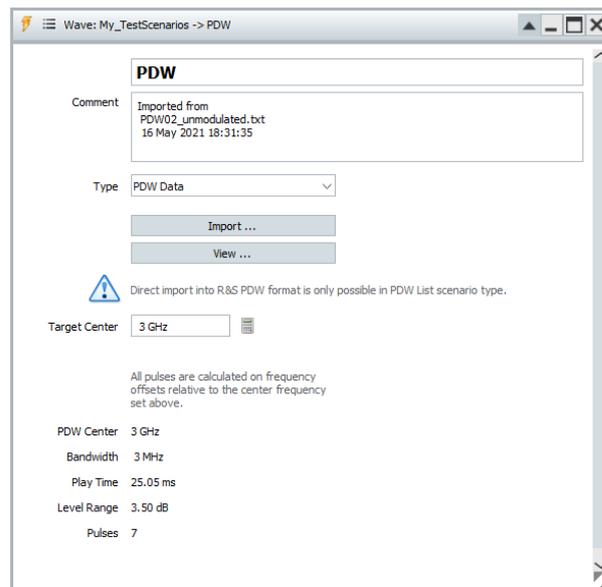
This section describes the available settings. For step-by-step instructions, see "To import PDW list data" on page 501.
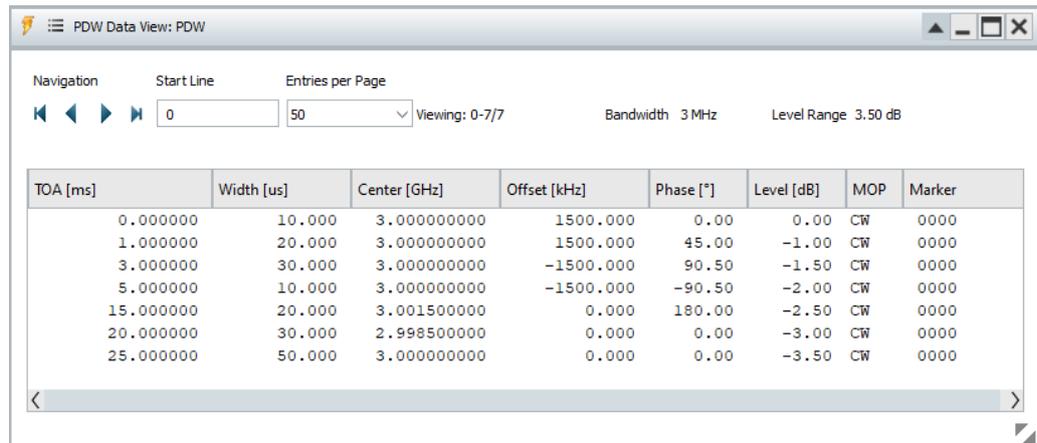
Settings:

Import
Accesses a file import wizard, see Chapter 18.4, "PDW data import wizard settings", on page 511.

View > PDW Data View
If the PDW list file is imported in the repository, it opens the "PDW Data View".

The dialog shows the waveform bandwidth, the level range and the waveform content as a sequence of pulses with their main characteristics.

The displayed information resembles the information in the "PDW Data Import > Imported Data" dialog.

"Start Line"   Sets the first line to be displayed.

Remote command:
IMPort:VIEW:TIME:STARt on page 674

"Entries per Page"

Sets the number of lines displayed per page.

Remote command:
IMPort:VIEW:COUNt on page 674

"Start, End, Forwads, Backwords"

Scrolls through the entries in the selected way.

Remote command:
IMPort:VIEW:MOVE:STARt on page 674
IMPort:VIEW:MOVE:FORWard on page 674
IMPort:VIEW:MOVE:BACKwards on page 674
IMPort:VIEW:MOVE:END on page 674

Target Center

Sets the center frequency for the output waveform.

When a PDW list file is imported, the value is set to the automatically calculated center frequency of the PDW file. The value resembles the value indicated as "PDW Center".

If the actual frequency of the generator differs from the calculated one, set the parameter "Target Center" to the *center frequency of the generator*.

The frequency offsets of the pulses are recalculated. The bandwidth increases.

Remote command:
WAVeform:PDW:CENTer on page 803

Calculate

The "Calculate" icon resets the "Target Center" field to the value displayed as "PDW Center".

**PDW Center, Bandwidth, Play Time, Level Range, Pulses**
If the PDW list file is imported in the repository, the dialog displays information on the waveform:

- "PDW Center": *Calculated center frequency*
  The indicated value is the center frequency the software uses during the import process. The pulses are calculated relatively to this value.
  The "PDW Center" is calculated as the middle frequency between the min and the max frequency values included in the PDW file.
  Chirp frequency deviations are considered.
- "Bandwidth": Calculated bandwidth
- "Play Time": Duration
- "Level Range": Calculated level range
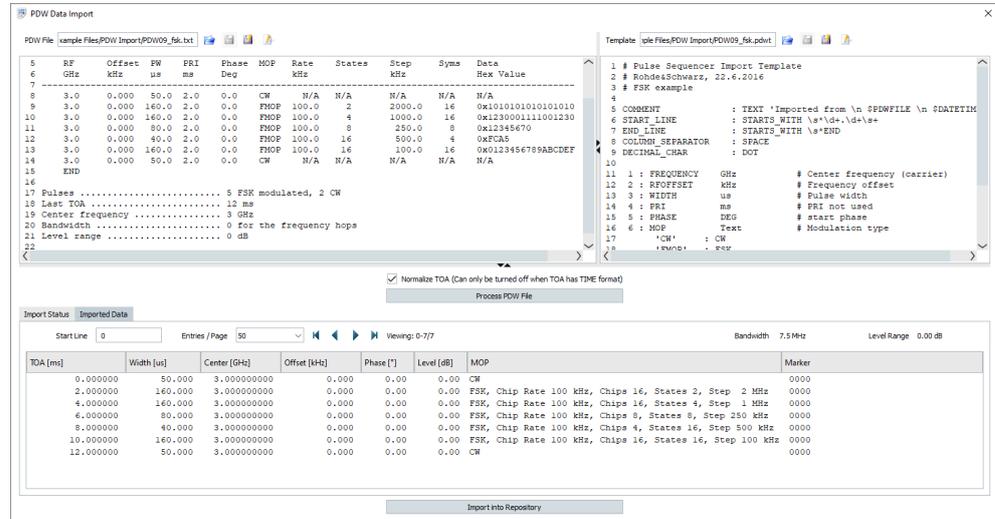- "Pulses": Number of pulses in the waveform.

18.4 PDW data import wizard settings

Access:

1. Select "Repository Tree > Waveform > New".

2. Select "Type = PDW Data".
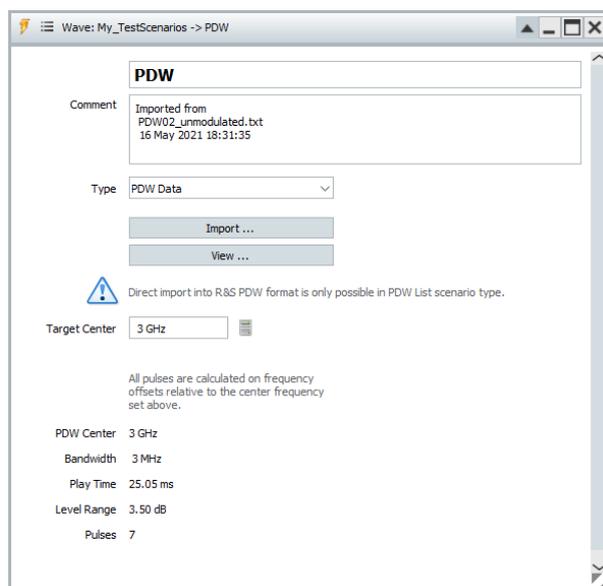
3. Select "Import".

If the required template file is provided, the "PDW Data Import" wizard converts custom PDW list files into a Rohde & Schwarz waveform file.

This section describes the available settings. For step-by-step instructions, see "To import PDW list data" on page 501.

Settings:

PDW File

File path and filename of the custom text-based PDW list file.

PDW list files are files in ASCII or coma-separated file format (`*.txt` or `*.csv`).

Remote command:
`IMPort:PDW:FILE:PDW` on page 670

Select PDW data file, Create new PDW data file, Save/Save as

Icons with standard file handling functions.

Note: The "Save/Save as" function stores changes in the PDW file or template itself. To parse the data, import it and store it into the repository, select "Process PDW File" and "Import into repository".

Remote command:

IMPort:PDW:FILE:PDW:LOAD on page 671

IMPort:PDW:FILE:PDW:SAVE on page 671

Template

File path and filename of the import template file.

Template files are ASCII files with predefined file format and file extension `*.pdwt`.

For more information, see document "R&S Pulse Sequencer PDW Import Interface Specification".

Remote command:

IMPort:PDW:FILE:TEMPlate on page 670

Select import template file, Create new import template, Save/Save as

Icons with standard file handling functions.

Remote command:

IMPort:PDW:FILE:TEMPlate:LOAD on page 671

IMPort:PDW:FILE:TEMPlate:SAVE on page 671

PDW and template files content editor

Displays the content of the loaded file in the editor.

If necessary, edit the files and store the changes.

Normalize TOA

If selected, it normalizes the time of arrival (TOA) of the first entry to zero. The following timestamps are considered relative to the first timestamp.

Remote command:

IMPort:PDW:NORM on page 671

Process PDW File

Starts parsing the PDW file.

Observe the parsing process information in the "Import Status" dialog.

The imported data is temporarily stored in memory.

Remote command:

IMPort:PDW:EXECute on page 671

Import Status

Indicates the parsing and import status.

Remote command:

IMPort:PDW:STATus? on page 671

Imported Data

Shows summary information retrieved from the imported waveform, such as the waveform bandwidth, the level range and the waveform content as a sequence of pulses with their main characteristics.

Remote command:
IMPort:PDW:DATA:SEL on page 672
IMPort:PDW:DATA:MOP? on page 672
IMPort:PDW:DATA:WIDTh? on page 673
etc.

Import into Repository
Stores the data permanently in the repository.

As any other waveform element, the imported PDW information can be used as part of a sequence or as a background emitter.

Remote command:
IMPort:PDW:STORe on page 673

18.5 PDW list (Collection) settings

Access:

► In a "Scenario Type = PDW List", select "PWD Lists > Edit".

Dialog displays overview information of the available PDW lists in table form.

No	Alias Name	PDW List	Absolute Level	Frequency
1	List 1	PDW	0 dBm	3.009 GHz
2	List 2	PDW	0 dBm	4 GHz

PDW List (Collection): My_TestScenarios -> PDW List
Collection of PDW Lists
Options

Settings:

Options
Joint configuration of all PDW lists at once.

● "Reset Frequency"

- "Reset Abs. Level"

Append, Delete, Clear, Move Up/Down, Copy and append
Use the standard functions in the context menu to add, reorder, or remove items.

Remote command:
SCENario:CPDW:ADD on page 591
SCENario:CPDW:DELete on page 595
SCENario:CPDW:CLEar on page 596

Search Aliases
Allows fast search of the PDW list; useful in scenarios with many PDW lists.

Filters and displays PDW lists in table rows according to the entered search string.

No
Subsequent number.

Remote command:
SCENario:CPDW:SELect on page 593

Alias Name
Enters an alias name.

Remote command:
SCENario:CPDW:ALIas on page 758

PDW List
Selects the waveform element, used to import the PDW list.

Remote command:
SCENario:CPDW:NAME on page 757

Absolute Level
Sets the absolute level for the selected PDW list.

Thus, you can combine PDW lists with different levels into one output signal.

Remote command:
SCENario:CPDW:LVABs on page 754

Frequency
Sets the frequency for the selected PDW list.

Thus, you can combine PDW lists with different frequencies into one output signal.

Remote command:
SCENario:CPDW:FREQ on page 753

18.6 PDW list (Collection) interleaving settings

Access:

1. In a "Scenario Type = PDW List", select "PWD Lists > Interleaving > On".

2. Select "Interleaving > Config".

   Dialog displays the available PDW lists as configured in the PDW List (Collection) dialog, together with interleaving related settings, like "Time Offset" and "Priority".

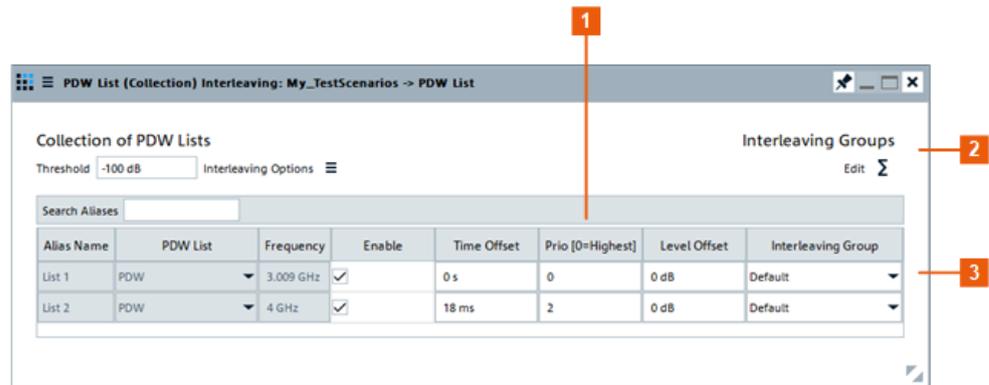*Figure 18-5: PDW List (Collection) Interleaving: understanding the displayed information*

1 = Priority of the selected PDW lists; applies only if PDW lists are merged, see Figure 18-3.
2 = Opens the "Interleaving Groups" dialog, where you can define groups within that the interleaving is applied
3 = Each PDW list is assigned to an interleaving group

Settings:

Threshold

Enters a level threshold to limit the dynamic range of the signal. Pulses at levels below this threshold are omitted.

Resembles the value set with the parameter "Scenario > Signal Calculation > Config > Config > Signal Calculation Settings > Output > Threshold".

Remote command:
SCENario:CPDW:THReshold on page 765

Search Aliases
Allows fast search of the PDW list; useful in scenarios with many PDW lists.

Filters and displays PDW lists in table rows according to the entered search string.

Interleaving Options
Joint configuration of all PDW lists at once.

- "Enable All"/"Disable All"
- "Reset Time offset"/"Reset Level offset"
- "Reset Prio"

Alias Name
Displays the alias name, as set with the parameter "Scenario > PDW Lists > Edit" > Alias Name.

Remote command:
SCENario:CPDW:ALIas on page 758

PDW List
Displays the waveform element, used to import the PDW list.

Remote command:
SCENario:CPDW:NAME on page 757

Frequency
Indicates the frequency of the particular PDW list, as set with the parameter "Scenario > PDW Lists > Edit" > Frequency.

Enable
Includes the PDW list in the output file.

Remote command:
SCENario:CPDW:ENABle on page 758

Time Offset
Shifts the processing of the PDW list in time.

Use this function, for example, to decrease the number of dropped PDWs.

Remote command:
SCENario:CPDW:LDELay on page 758

Priority [0 = Highest]
Sets the priority of the selected PDW list. Higher value means lower priority. If PDWs are overlapping, the PDWs from the PDW list with lower priority are discarded.

See Figure 18-3.

Remote command:
SCENario:CPDW:PRIority on page 758

Level Offset

Adds a level offset.

The value is set relative to the value set with the parameter "Scenario > PDW Lists > Edit" > Absolute Level.
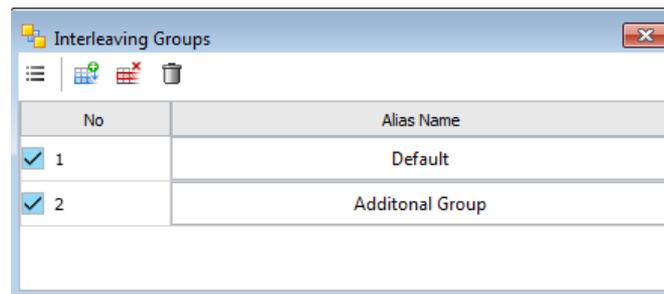
Remote command:
SCENario:CPDW:LEVel on page 759

Group

Assigns the emitter to one of the available interleaving groups, defined in the Edit Interleaving Groups dialog.

Remote command:
SCENario:CPDW:GROup on page 759

Edit Interleaving Groups

Select the icon to access the "Edit Interleaving Groups" dialog.

Interleaving groups allow you to combine the emitter signals into different output signals.

Those signals can in turn be generated by different signal generators or output at the different outputs of an instrument. See also Chapter 17.2, "Assign signal source or receiver signals to generator", on page 481.

Use the standard functions (icons) to add a new group, rearrange the groups, delete the existing one or all groups.

Select, Insert, Append/Prepend, Remove, Clear ← Edit Interleaving Groups

Standard functions for items handling, see "Standard function in the context menus" on page 33.

Select a row, for example, to delete it or to insert a row before it.

Remote command:
SCENario:CPDW:GROup:CATalog? on page 759
SCENario:CPDW:GROup:SELect on page 592
SCENario:CPDW:GROup:ADD on page 591
SCENario:CPDW:GROup:DELete on page 594
SCENario:CPDW:GROup:CLEar on page 595

No. ← Edit Interleaving Groups

Select a row to perform any row-based actions.

Interleaving groups are indicated by consecutive number; also used for indication in remote control.

Remote command:
SCENario:CPDW:GROup:COUNt? on page 592
SCENario:CPDW:GROup:SELect on page 592

Alias Name ← Edit Interleaving Groups

Enter an alias name for the interleaving group.

Remote command:
SCENario:CPDW:GROup:ALIas on page 760

18.7 PDW interleaving progress statistics

Access:

1. In a "Scenario Type = PDW List", select "Interleaving > On".

2. Select "Signal Control > Prepare".

3. Select "Signal Control > Interleave".

   The "Interleaving Progress" display opens automatically during the signal calculation, if PDW lists are merged. It indicates the percentage of pulses being processed and dropped in each of the PDW lists.
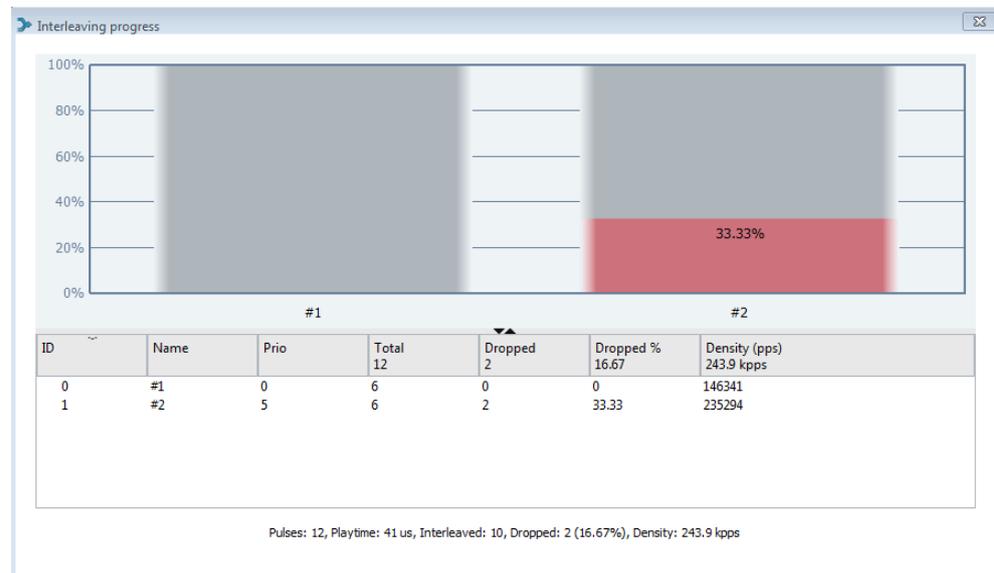
18.8 Signal preview with interleaved and dropping statistics

Access:

1. In a "Scenario Type = PDW List", select "Interleaving > On".

2. Select "Signal Control > Prepare".

3. Select "Signal Control > Interleave".

4. Select "Interleave > Results > Interleaved PDW/Dropped PDWs".

The dialog is similar to the "Signal Preview" dialog.

This section describes the settings dedicated to the "Interleaved/Dropped PDWs" dialogs. For description of all other settings, see Chapter 15.2, "Signal preview settings", on page 405.

Settings:

Display > Filter

If interleaving is used, the calculated output signal contains pulses originating for multiple PDW lists or emitters. By default, all PDWs are displayed. Use the "Filter" parameter to narrow down the displayed information to the PDWs of one of the PDW lists.

The filter does not change the zoom level or the scaling on both axis.

Display > Groups

If interleaving groups are used, selects the group for which pulses are displayed.

Table

Provides summary information on the number of processes PDWs and the content of the output file.

Graphics

Visualize the variation of the parameters over time.

For details, see Chapter 15.2, "Signal preview settings", on page 405.

Graphics > Highlight

Highlights pulses belonging to the selected PDW list.

Histogram

Resembles the information displayed during the signal processing, see Chapter 18.7, "PDW interleaving progress statistics", on page 519.

Use the filtering option to narrow down to the displayed information according to one of the predefined criteria.

Statistics

The drop out statistic is the table form representation of the same information as the Histogram.

18.9 Analyze interleaving

Access:

1. In a "Scenario Type = PDW List", select "Interleaving > On".

2. Select "Signal Control > Prepare".

3. Select "Signal Control > Interleave".

4. Select "Interleaving > Results > View > Analyze".

   The "Analyze Interleaving" dialog indicates the PDWs that are and are not included in the resulting output file. PDWs displayed in **red color are dropped out**.
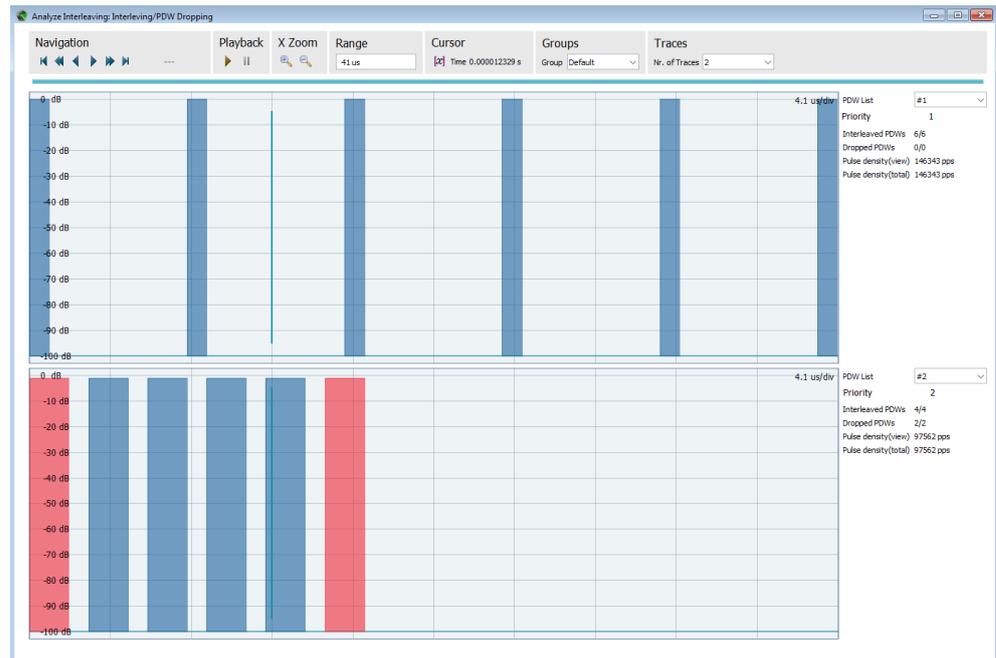
The provided settings are a subset of the settings available in the "Waveform View/ Signal Preview" displays and they have the same effect.
For description, see Chapter 15.2, "Signal preview settings", on page 405.
If frequency agility is enabled, the dialog displays the additional trace CNTRL PDWs. This trace is always displayed at the top and cannot be disabled. For example, see Figure 5-2.
The following settings are dedicated to this dialog.

Display > Groups
If interleaving groups are used, selects the group whose pulses are displayed.

Traces
Sets the number of traces to be displayed.

If the selected interleaving group comprises many PDW lists, this function can be used to narrow down the number of simultaneously displayed traces and hence increase the zoom level on the y-axis.

Information per trace
The following information is displayed:
● Trace name
● Priority
● Interleaved PDWs
● Dropped PDWs
● Pulse density (view)
● Pulse density (total)

19 Defining the data source

The R&S Pulse Sequencer provides a wide range of internally defined data sources, such as patterns or PRBS generators. Data sources are available globally within the project. Once created, they can be used in all pulses. Each pulse retrieves its data individually.

Per default, data is retrieved cyclically, starting from the first list entry. Once all bits are used, the data is retrieved from the second one. When the list end is reached, the data is retrieved from the first entry.

If the data entry contains fewer bits than the required, the data bits are repeated cyclically.

You can, however, define on pulse basis, if the data source should be restarted at the end of each pulse, see "Data Source" on page 184.

Example: Data source processing
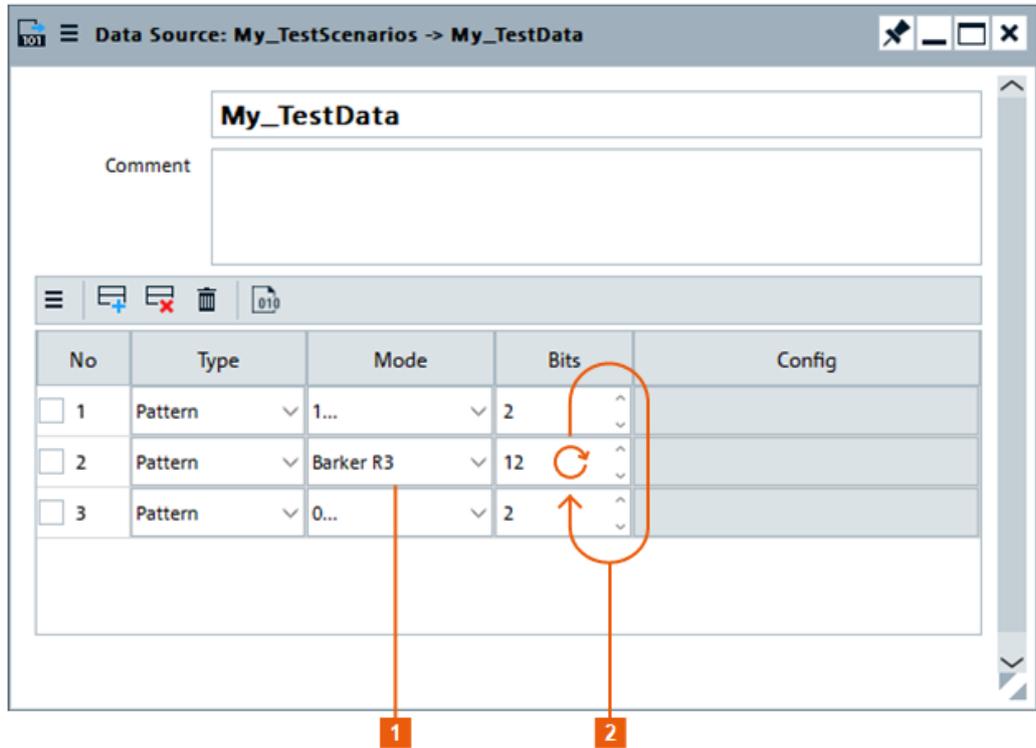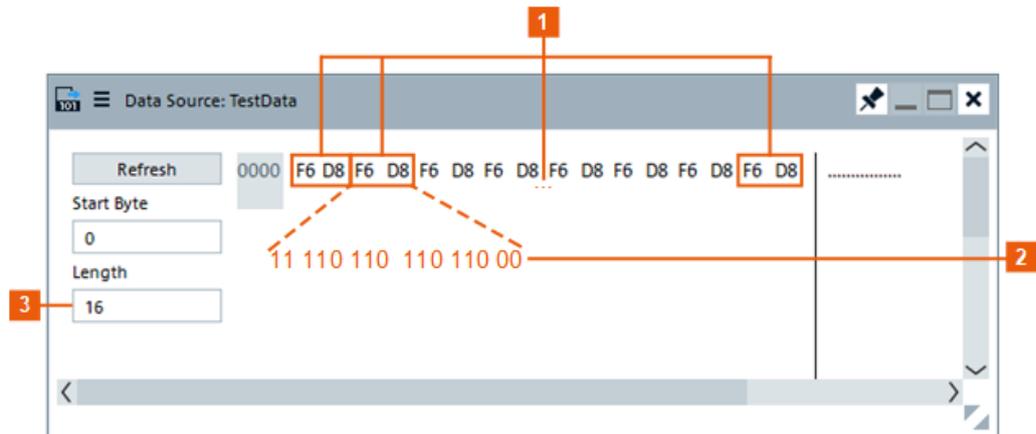
Configure the data source as shown on Figure 19-1.

Figure 19-1: Data Source: understanding the displayed information

1 = Barker R3 pattern = 110; required are 12 bits, i.e. the pattern is repeated 4 times
2 = If more than 16 bits are required, the 16 bits are repeated

Use the Preview Data Source to visualize the content.

1 = Configured 16 bits (2 bytes), repeated 8 times
2 = Content of each 2 byte: 2x"Pattern = 1" 4x"Pattern = Barker R3" 2x"Pattern = 0"
3 = Displayed length is 16 bytes

Supported types of data source

The R&S Pulse Sequencer provides the following types of data sources:

- *Pattern*
    - Simple data patterns such as binary 0 ("0...") strings or 1 ("1...") strings
    - Variable bit strings ("1010") with alternating 0 and 1 and a maximum length of 999 bits
    - Different Barker codes: Barker R3, Barker R4a, etc.
- *User*
  A user-defined sequence of 0 bits and 1 bits

  The following rules apply for user defined data source:
    - Accepted are 0 and 1, and numbers in hexadecimal format
    - Comments can be enclosed in slashes (/)
    - Blank characters are ignored
    - The sequence #x starts a hexadecimal input for the remainder of the line
    - A new line turns hexadecimal input and comments off
    - Ones and zeroes are evaluated as single bits
    - Strings can be enclosed in quotation marks (")
- *PRBS* (Pseudo-Random Binary Sequence)
  The PRBS generators deliver pseudo-random binary sequences of differing length and duration. They are known as maximum length sequences, and are generated with the aid of ring shift registers with feedback points determined by the polynomial.

  The pseudo-random sequence from a PRBS generator is uniquely defined by the register number and the feedback, see Table 19-1.

  Example:

  By way of example, the diagram on Figure 19-2 shows a 9-bit generator with feedback to registers 4 and 0 (output). The generated serial data is converted internally, e.g 2 Bit/Symbol for QPSK.
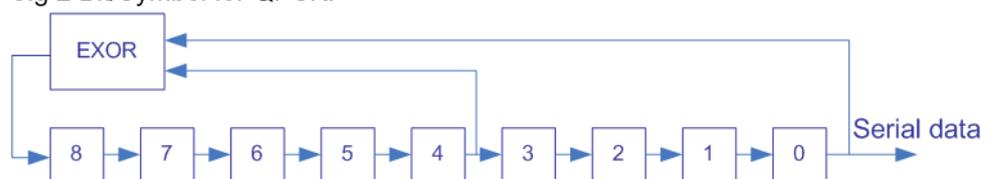

Figure 19-2: A 9-bit PRBS generator

Table 19-1: Overview of PRBS generators

PRBS generator	Length in bits	Feedback to
7-bit	$2^7 - 1 = 127$	Registers 1, 0
9-bit	$2^9 - 1 = 511$	Registers 4, 0
11-bit	$2^{11} - 1 = 2047$	Registers 2, 0
15-bit	$2^{15} - 1 = 32767$	Registers 1, 0

PRBS generator	Length in bits	Feedback to
16-bit	$2^{16} - 1 = 65535$	Registers 5, 3, 2, 0
20-bit	$2^{20} - 1 = 1048575$	Registers 3, 0
21-bit	$2^{21} - 1 = 2097151$	Registers 2, 0
23-bit	$2^{23} - 1 = 8388607$	Registers 5, 0

19.1 Data sources settings

Access:

► Select "Repository Tree > Data Source > New".
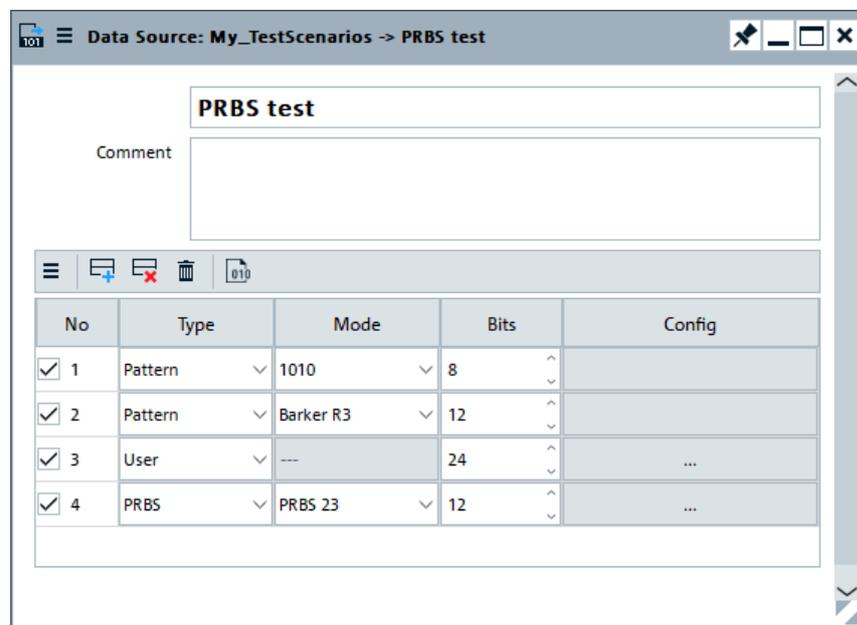
Available settings:

Data Source Name
Enter a name for the data source.

Remote command:
DSRC:NAME on page 589
DSRC:CREate on page 588

DSRC:CATalog? on page 587

DSRC:SELect on page 588

DSRC:REMove on page 590

DSRC:CLEar on page 595

Comment

Enter a short description.

Remote command:

DSRC:COMMent on page 589

Data Source Table

A table with one or more items that described the content of the data source.

"Item Select"   Selects an item for performing standard tasks, like inserting an item above the current one, or deleting an item.

Remote command:

DSRC:ITEM:SELect on page 593

DSRC:ITEM:ADD on page 591

DSRC:ITEM:DELete on page 594

"Data Source Type, Data Source Mode"

Sets the type of the data and the data content for the current item, see "Supported types of data source" on page 525.

See also:

● Chapter 19.2, "How to configure the bit stream used by the MOP", on page 529
● Example"Data source processing" on page 524

Remote command:

DSRC:ITEM:TYPE on page 636

DSRC:ITEM:PATTern on page 635

DSRC:ITEM:PRBS:TYPE on page 636

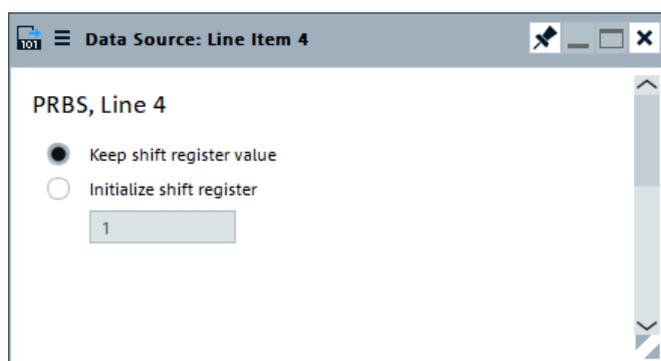"Bits"   Sets the length of the current item in bits.

Remote command:

DSRC:ITEM:BITS on page 634

"Config"   Accesses a dialog with more settings, see:
"Shift register" on page 527

Shift register

A PRBS sequence is defined by a length and a register shift number, see Table 19-1.

"Keep shift register value"

Use the default values.

"Initialize shift register"

Customize the shift register.
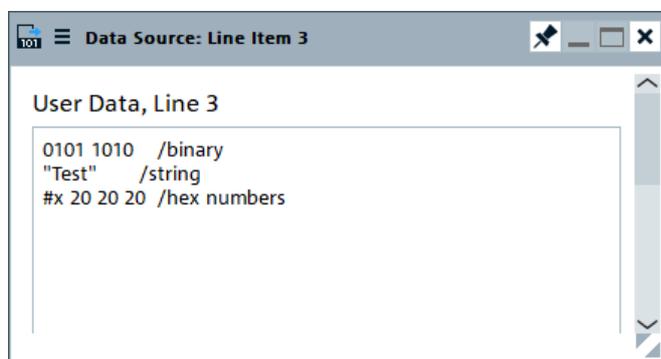
"Initialization value"

Set a new initialization value.

Remote command:

DSRC:ITEM:PRBS:INIT on page 635

DSRC:ITEM:PRBS:INIT:VALue on page 636

User Data

For "Data Source > Type > User", you can define your own data pattern, see "Supported types of data source" on page 525.

Remote command:

DSRC:ITEM:DATA on page 635

Preview Data Source

Displays the data content.

See also Example"Data source processing" on page 524.

"Refresh"   Perform refresh to display the current data content.

"Start Byte, Length"

Sets the data bits currently displayed.

## 19.2 How to configure the bit stream used by the MOP

See:
- "To configure the data source" on page 529
- "To assign the data source to a MOP" on page 529

To configure the data source

To access the "Data Source" dialog:

1. Select "Repository Tree > Data Source > New".

2. Select the "Append new item" icon.

3. Enter a name and a comment.

4. For each item, select the data type, the data mode, and the data length in bits.
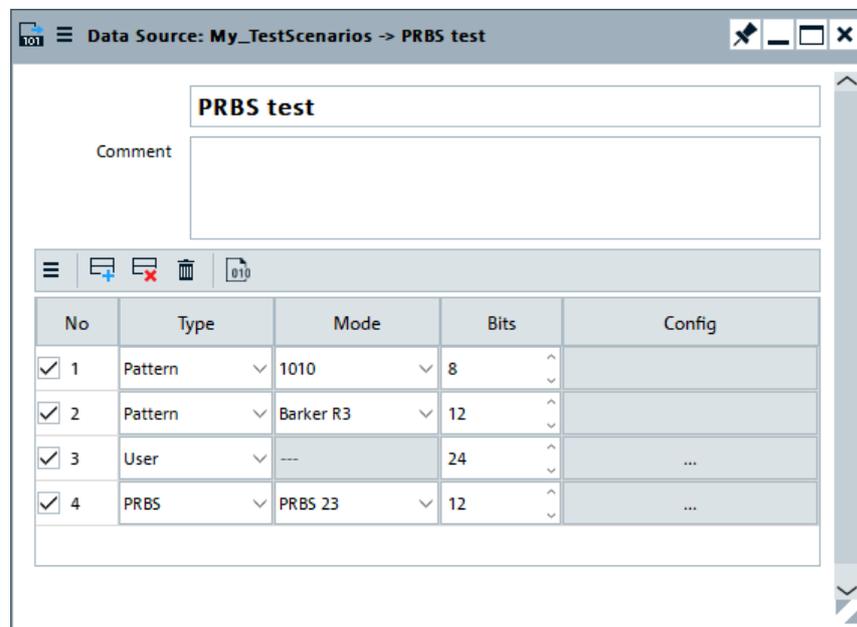
5. To view the data source content, select the "Preview" icon.

   Data is retrieved cyclically, starting from the first list entry, see Example"Data source processing" on page 524.

To assign the data source to a MOP

1. In the repository tree, select "Pulse > PulseName e.g. My_TestPulse > MOP".

2. Select "MOP > Settings" and select a MOP that requires a data source, e.g. "MOP Type > FSK"

3. Select "Data Source > e.g. My_TestData".

20 Defining complex modulation schemes and IPM profiles

Custom modulation schemes and custom envelope shapes are defined by an external plugin.

Plugins are Microsoft Windows DDL modules that contain the maths that is required for the envelope shaping and the modulation on pulse.

Some example plugins are provided with the software as binary and source code. These examples can serve as a starting point for your own applications. Once imported, the software handles the plugins automatically.

Each plugin:

● Has to provide a range of functions to identify itself
● Has to provide the functions necessary to perform the calculations required for the modulation on pulse
● Can register a set of configuration parameters that can be used as variables inside the plugin.

You can load plugins to:

● Specify your custom modulation scheme (MOP)
● Define a custom inter-pulse modulation profile (IPM)
● Create a report file during the signal calculation process
● Define the file format of the custom antenna pattern files.

How to import and assign user defined plugins

See:

● Chapter C, "Plug-in programming API", on page 863 for a description of the plugin programming API and instructions on how to load the plugin examples.
● "To import a plugin" on page 530
● "To assign the user defined modulation scheme to a pulse" on page 532
● "To assign the user defined IPM profile" on page 532

See also:

● "To import an antenna pattern file in custom file format" on page 302
● Chapter 22, "Creating reports and documenting measurement results", on page 542

To import a plugin

1. In the repository tree, select "Plugins > New".

2. In the "Plugin" dialog, enter a name and description.

3. Select "Load", navigate to the `*.dll` file, and select it.

Only files containing modulation data are listed and can be loaded.

The software loads the `*.dll` file, retrieves information from it and displays it in the "Plugin" dialog.
The "Variables" table lists the plugin variables. You can edit them when you load the plugin into a repository element.
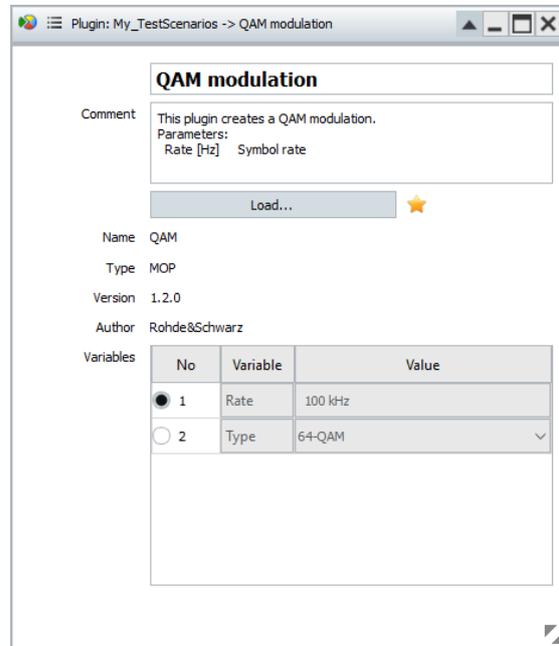
You can use the plugins to create:
● Create custom modulation schemes, see "To assign the user defined modulation scheme to a pulse" on page 532.
  See also Chapter 6.2.5.8, "Plug-ins", on page 182.
● Create custom IPM profile, see "To assign the user defined IPM profile" on page 532
● Import custom antenna pattern file formats, see "To import an antenna pattern file in custom file format" on page 302.

Remote Commands:
● `PLUGin:CREate` on page 588
● `PLUGin:CATalog?` on page 587
● `PLUGin:NAME` on page 589
● `PLUGin:COMMent` on page 590
● `PLUGin:LOAD` on page 687
● `PLUGin:SELect` on page 588
● `PLUGin:REMove` on page 590
● `PLUGin:MODule:NAME?` on page 589
● `PLUGin:MODule:TYPE?` on page 688
● `PLUGin:MODule:VERSion?` on page 688
● `PLUGin:MODule:AUTHor?` on page 688
● `PLUGin:MODule:COMMent?` on page 688
● `PLUGin:MODule:DATA?` on page 688

To assign the user defined modulation scheme to a pulse

1. In the repository tree, select "Pulse > PulseName > MOP".

2. Select "MOP > Settings".

3. Select "MOP > MOP Type > Plugin".

4. Select "Plugin" and select a loaded user-defined modulation scheme, e.g. `My_QAM_PlugIn`.
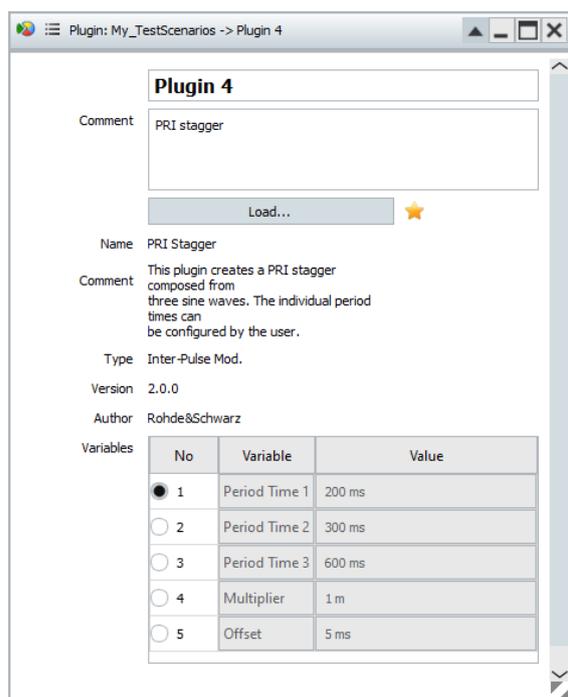

5. Select "Pulse > PulseName > MOP > Enable Modulation on Pulse (MOP) > On".

6. Use the Restrict MOP Range settings to determine the exact pulse part the modulation is applied to.

To assign the user defined IPM profile

1. Import a suitable plugin.

2. In the repository tree, select "Inter-Pulse Mods > IPM-Name".

3. Select "Profile > Plugin".

4. Select "Plugin" and select a loaded user-defined IPM profile, e.g. `Stagger PRI`.
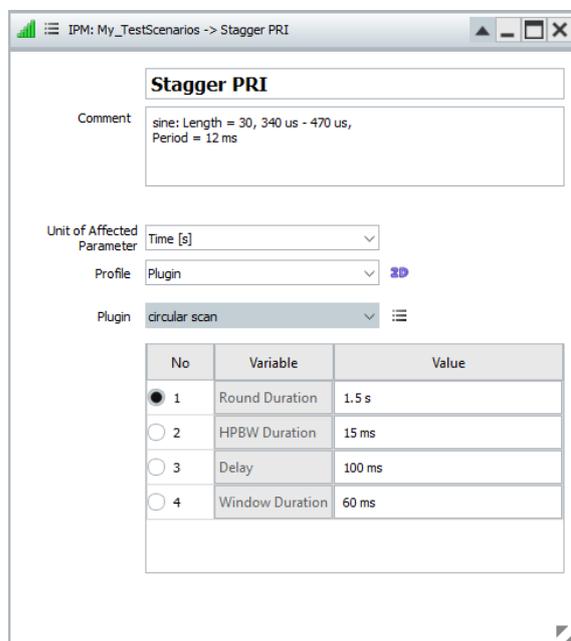
5. Select "2D" to visualize the IPM profile.

6. If required, change the values of the used variables and observe the effect on the "2D" diagram.

21 Defining and enabling marker signals

Markers signals or markers are binary signals that are additionally generated and added to the generated signal.

Markers are commonly used to synchronize a device under test (DUT) or a spectrum analyzer to the generated signal, for example to the beginning of a pulse. In the vector signal generator, generated marker signals can be routed to the corresponding output connectors.

For details, see the user manual of the corresponding base unit.
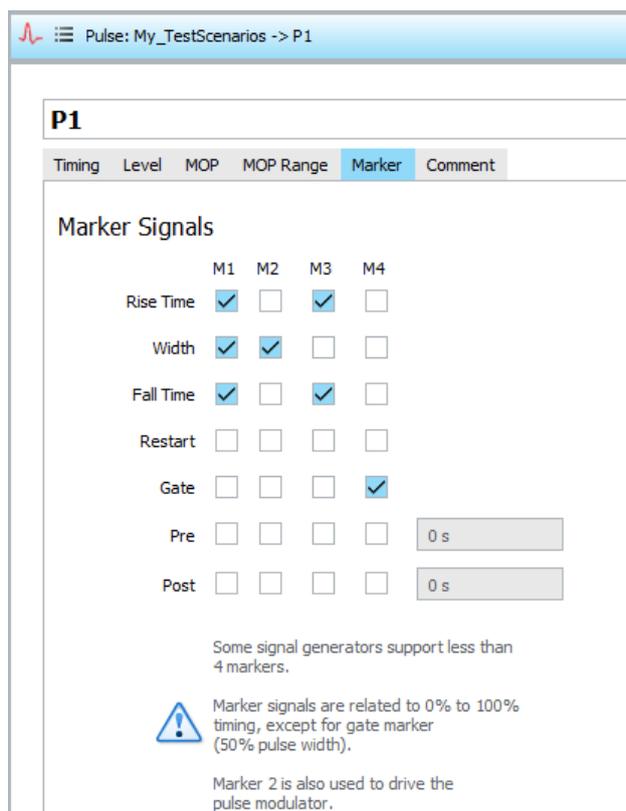
21.1 Marker settings

In the R&S Pulse Sequencer, you define markers on four levels: on a pulse basis, for emitters, in the sequence or define global conditions like the start of a scenario.

Access:

1. Select "Repository Tree > Pulse > Marker".

    See "Pulse Markers" on page 534.

2. Select "Repository Tree > Sequence > Sequence Description > Marker".

    See "Sequence Markers" on page 536.

3. Select "Map > TX Items > Properties > TX Items Properties".

    See "Emitter Marker" on page 441

4. Select "Repository Tree > Scenario > Signal Calculation > Config > Marker Signals > Enable sequence and waveform markers".
    See "Global Marker Signals" on page 536.

Pulse Markers
You can assign up to 4 markers to the different pulse phases, for example rise and fall time, pulse width. A restart marker indicates the first 10% of the entire PRI.

Marker information is directly added to the resulting waveform and the marker signal output is therefore synchronous with the waveform playback.

For pre and post markers, set their timing in the corresponding field. Enabled pre marker with timing different than zero shifts the start of the pulse rising edge and thus the entire pulse.

Remote command:

PULSe:MARKer:RISE on page 678

PULSe:MARKer:WIDTh on page 678

PULSe:MARKer:FALL on page 678

PULSe:MARKer:AUTO on page 678

PULSe:MARKer:GATE on page 678

PULSe:MARKer:PRE on page 678

PULSe:MARKer:POST on page 678

PULSe:TIME:PRE on page 678

PULSe:TIME:POST on page 679

Sequence Markers

"First, Last, All"   If pulse markers are defined, you can also define markers on multiple repeating pulses and mark the first, last or all pulses.

Remote command:
SEQuence:ITEM:MARKer:FIRSt on page 679
SEQuence:ITEM:MARKer:LAST on page 679
SEQuence:ITEM:MARKer:ALL on page 679

"Condition, Variable, Value"

A marker signal is generated if a defined condition is fulfilled. Conditions are defined as logical expressions, where the variable value ("Variable") is compared to a fixed value ("Value"). The variable must exist.

Example:

If a sequence item is included into a loop, you can create marker signals that mark for example one specific loop run number.

See:
- Chapter 7.2.3, "Loop settings", on page 207
- "To enable sequence markers" on page 539

Remote command:
SEQuence:ITEM:MARKer:CONDition on page 679
SEQuence:ITEM:MARKer:CONDition:VARiable on page 679
SEQuence:ITEM:MARKer:CONDition:TYPE on page 679
SEQuence:ITEM:MARKer:CONDition:VALue on page 680

Global Marker Signals

If the selected sequence in a scenario contains markers, you can define that these markers be considered in the output from the signal calculation.
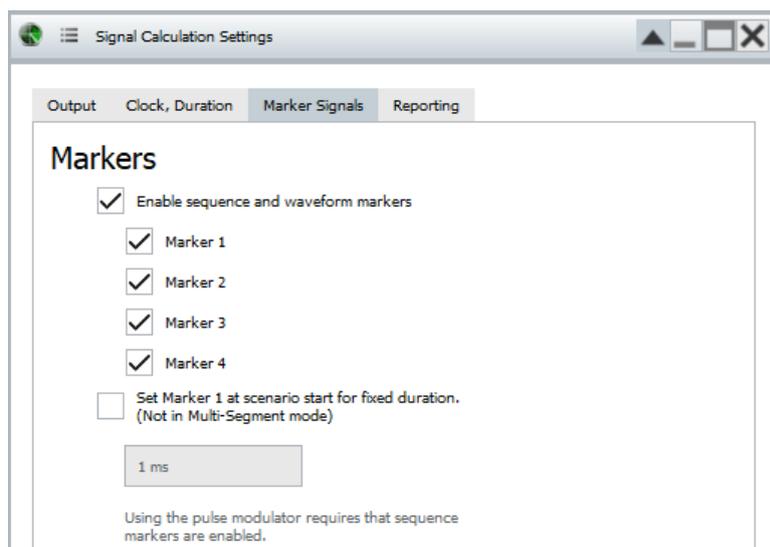
For example, to use marker M1 as trigger signal that marks the scenario start, enable M1 on the scenario level and disable it in the sequence.

Remote command:
SCENario:OUTPut:MARKer:ENABle on page 680
SCENario:OUTPut:MARKer:FLAGs on page 680
SCENario:OUTPut:MARKer:SCENario:ENABle on page 680
SCENario:OUTPut:MARKer:SCENario:DURation on page 681

21.2 How to configure and visualize markers

To enable pulse markers

1. Select "Repository Tree > Pulse > Marker".

2. Enable Marker 1 ("M1") for "Rise Time", "Width" and "Fall Time".
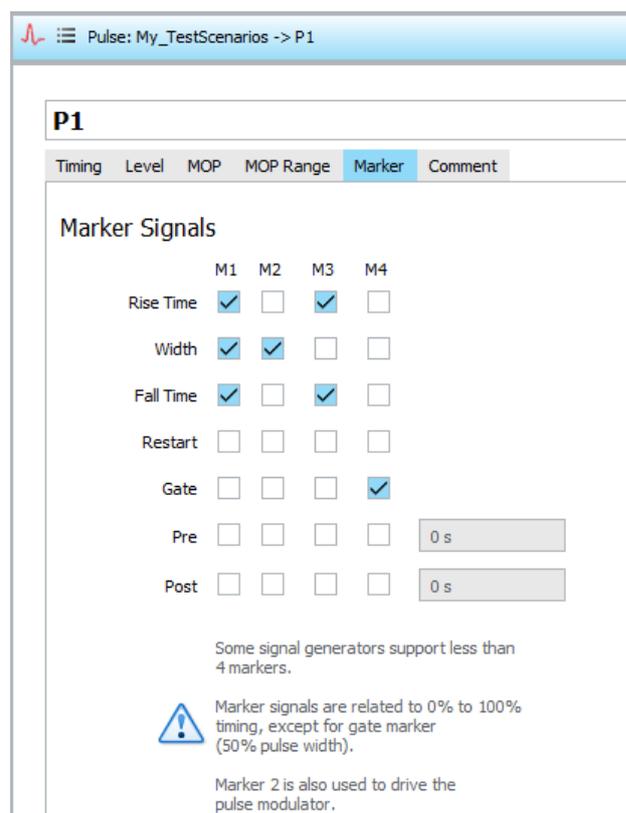
The Marker 1 is active for the entire number of samples. That is, the marker is held high for the whole time from beginning of the rising edge to the end of the falling edge.

3. Enable "M2 > Width", "M3 > Rise and Fall Time" and "M4 > Gate".

The envelope graph marks the time periods during that the markers are held high.
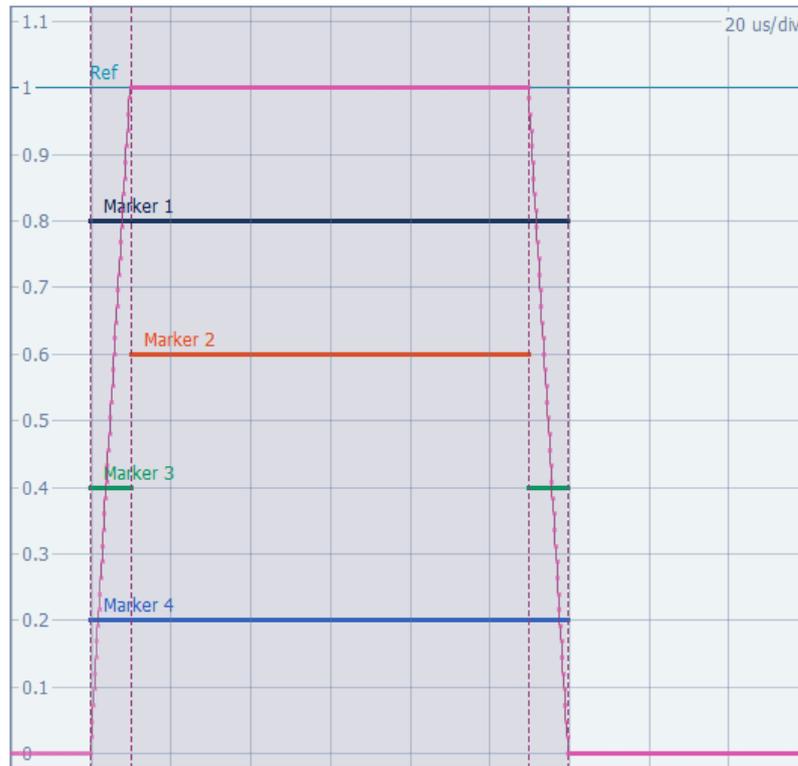
To enable sequence markers

1. Select "Repository Tree > Sequence".

Figure 21-1: Example of a sequence composed of two pulses included into a loop

2. Select "Loop > ..." and enter "Loop Variables > Prefix = LP".
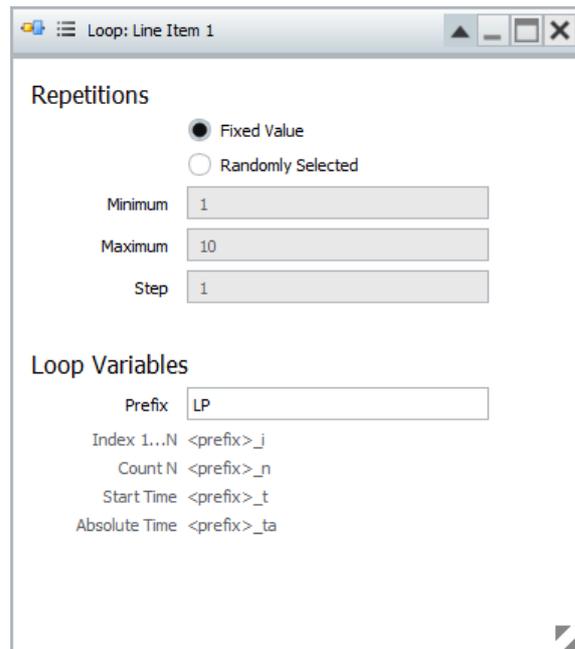
3. Select "Sequence Description table > Item#2 > Marker".

4. Select "M1 > All", "M2 > First", "M3 > Last".
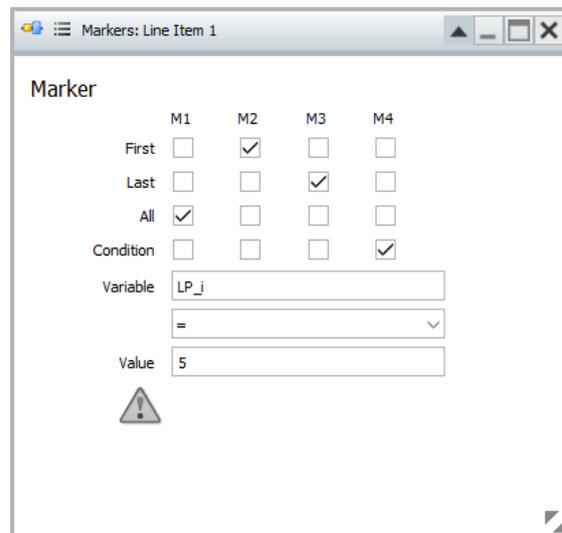
The Marker 2 is generated only for the first pulse out of all repetitions.

5. Select "M4 > Condition", "Variable = LP_i" and "Value = 5".

   If the condition LP_i = 5 is fulfilled, a marker is generated. That is, marker 4 is generated during the fifth loop run.
   See How to visualize the configured markers.

To enable marker signals in the output signal

1. Select "Repository Tree > Scenario > Signal Calculation > Config > Config > Signal Calculation Settings".

2. Select "Marker Signals > Enable sequence and waveform markers".
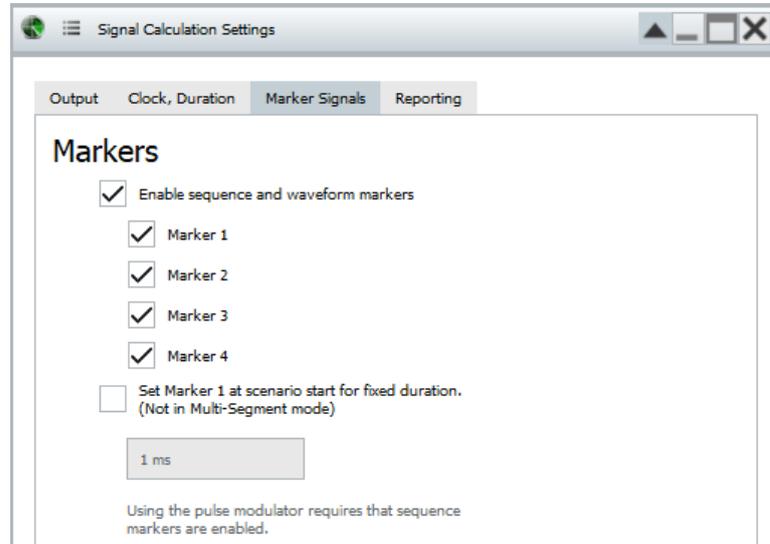
See also Signal calculation settings.

3. Optionally, select "Set at scenario start for fixed duration > On" and set "Duration = 1 ms", to enable an extra global signal.

   This marker signal is output as marker 1. If other marker signals, like for example pulse and sequence markers, are defined, all markers are summed.

How to visualize the configured markers

1. Open the "Waveform/Data Viewer", see "To access the "Signal Preview" dialog" on page 411.

2. Select "Marker > On".

   The "Waveform/Data Viewer" evaluates the marker information and displays the marker traces M1 to M4.

Figure 21-2: Example: Waveform View > Marker traces (signal portion)

M1 = All
M2 = First
M3 = Last
M4 = Condition LP_i = 5, i.e. M4 is generated for all P1 pulses during the fifth loop run

22 Creating reports and documenting measurement results

In R&S Pulse Sequencer, you can document your measurement results and software configuration in the following ways:

- Create and export a repository archive, so that you can repeat measurements under the same conditions
- Create automatically scaled hardcopy of current screens or dialogs
- Print antenna diagrams together with additional information
- Create and store reports in text files, in `HTML` format or in Excel files with several spreadsheets.

This section focuses on the built-in report generation function. For information on the other functions, see:

- "To create and export a repository archive" on page 117
- "To set the size and color scheme of the hardcopies" on page 51
- Chapter 9, "Defining antenna patterns and antenna scans", on page 257

Report generation

The R&S Pulse Sequencer provides a built-in report generation function so that all parameters used during the signal generation are automatically saved to a printable file. You can configure the target directory in which the generated reports are stored.

Report files can be formatted based on:

- **Template**
  The reporting uses a predefined but editable template. Generated is an ASCII text file or an `HTML` page, where data is formatted in columns, including header description. The reporting files in ASCII format use the predefined file extension `*.pwd`

  – ASCII text file

```
This is a template-based report.

User:       R&S
Rep Path:   C:\_PS_files\reports

Date:       28.10.2015 10:07:25
ISO Date:   2015-10-28T10:07:25

Repository: My_TestScenarios
  Comment:
  Version:   2.3
  Author:    Rohde&Schwarz
  Created:   25 Jun 2014 15:27:04
  Path:      C:\Users\Public\Documents\Rohde-Schwarz\Pulse Sequencer\Repositories\20141124_100351

Scenario:   SimplePulseTrain
  Comment:
```

TOA ns	RF GHz	PW us	PA dBm	MF	MOP	BW kHz	Rep
0	3.0	120.000	-30.0	0	NONE	0	1
2500000	3.0	120.000	-30.0	0	NONE	0	2
5000000	3.0	220.000	-30.0	0	NONE	0	1
10000000	3.0	220.000	-30.0	0	NONE	0	2
15000000	3.0	220.000	-30.0	0	NONE	0	3
22000000	3.0	120.000	-40.0	0	NONE	0	1
24500000	3.0	120.000	-40.0	0	NONE	0	2

Figure 22-1: Example of an ASCII report file ("Type > Template")

- HTML page

PDW Report Generated with PS Example

User	R&S
Repository Path	C:_PS_files\reports
Date	28.10.2015 08:15:18
ISO Date	2015-10-28T08:15:18

Repository	My_TestScenarios
Comment	
Version	2.3
Author	Rohde&Schwarz
Created	25 Jun 2014 15:27:04
Path	C:\Users\Public\Documents\Rohde-Schwarz\Pulse Sequencer\Repositories\20141124_100351

SimplePulseTrain

TOA	RF	PW	PA	MF	MOP	BW	Rep
ns	GHz	us	dBm			kHz	
0	3.0	120.000	-30.0	0	NONE	0	1
2500000	3.0	120.000	-30.0	0	NONE	0	2
5000000	3.0	220.000	-30.0	0	NONE	0	1
10000000	3.0	220.000	-30.0	0	NONE	0	2
15000000	3.0	220.000	-30.0	0	NONE	0	3
22000000	3.0	120.000	-40.0	0	NONE	0	1
24500000	3.0	120.000	-40.0	0	NONE	0	2

Figure 22-2: Example of an HTML report file ("Type > Template")

- **User-defined plugin**
  This reporting creates reports according to custom templates, e.g. DFS.
  You can, for example, create a Microsoft Excel plugin that retrieves the pulse parameters and enters them in a spreadsheet.

For a detailed description of the file formats, the template and the plugin, see:

- Chapter A.1, "File format of the reporting template", on page 838
- Chapter C, "Plug-in programming API", on page 863
- Chapter 20, "Defining complex modulation schemes and IPM profiles", on page 530.

Status information on whether report logging is enabled or not is displayed in the "Scenario" dialog, see Figure 5-1.

See also Chapter 22.2, "How to create test reports", on page 548.

22.1 Reporting settings

Access:

1. In the "Scenario" dialog, select "Signal Calculation > Config > Config".

2. Select "Reporting".
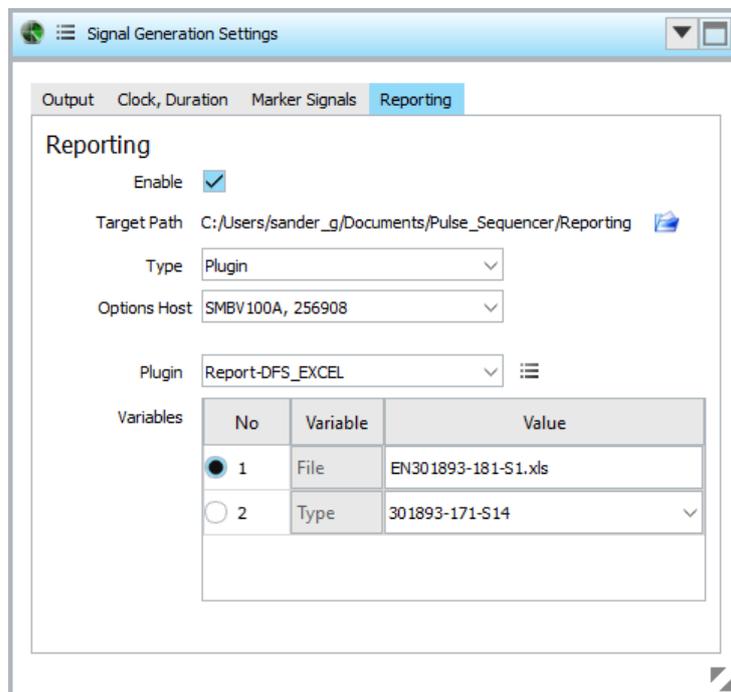
Available settings:

Reporting Enable
Enables generation of reports.

Remote command:
SCENario:PDW:ENABle on page 690

Target Path, Set Path
Indicates the current storage location.

Per default, report logs are stored in the user directory of the current user, that is the %HOMEPATH% directory.

To change the storage location, select the "Set the target path" icon.

See also "To set the storage location for the generated report files" on page 548.

Remote command:
SCENario:PDW:PATH on page 690

Type

Sets the template used be the reporting function.

"Default"　　　　Loads the predefined template.
The report is a plain text file. The format is simple, column-based and cannot be modified. Further settings are not required.

"Template"　　　　Enables the predefined template for editing (see "Edit Template" on page 545).

"Plugin"　　　　Generates report in custom format by using a plugin.
Custom templates are used in the same way as the predefined template (see "Plugin and plugin variables" on page 547).
For details on the plugin API, see Chapter C, "Plug-in programming API", on page 863.

"AMMOS PDW"

Writes PDW reports in the Rohde & Schwarz proprietary PDW format. This format is used by Rohde & Schwarz monitoring products.
See also"AMMOS IF and AMMOS PDW Import" on page 382.

Remote command:
SCENario:PDW:TYPE on page 690

Options Host

Provides a list of available devices that can provide the required options. To enable reporting, the selected device must be connected and already have the options installed that are required for this scenario.

Remote command:
SCENario:PDW:HOST on page 690

Edit Template

Opens the predefined template in edit mode.

Figure 22-3: Example: Template that creates text reports in ASCII format

Note: The software uses the last modified template version. Previous versions cannot be recalled.
Before you modify the template, store the template content as a text file.

See Chapter A.1, "File format of the reporting template", on page 838 for description of the file format.

Remote command:
SCENario:PDW:TEMPlate on page 692

Specific settings for Type > AMMOS PDW
The following settings are dedicated to "Type > AMMOS PDW":

Format ← Specific settings for Type > AMMOS PDW
Selects format of the AMMOS file.

"PDW"   Standard AMMOS PDWs, including header information.

"PPDW"   Shortened AMMOS PDWs, without header information.

Remote command:
SCENario:PDW:AMMos:PPDW on page 691

Frame Length ← Specific settings for Type > AMMOS PDW
Sets the frame length.

A frame can contain several PDWs but it can also be empty.

Remote command:
SCENario:PDW:AMMos:FRAMe on page 691

Start Time, User Set Value ← Specific settings for Type > AMMOS PDW
Defines how the report start time is set.

- "User Defined Value > Off"
  The reporting start time is time at that the scenario calculation starts.
- "User Defined Value > On"
  The reporting starts at user-defined moment. This setting is useful, for example, if particular event or moment of time is to be captured.

Remote command:
SCENario:PDW:AMMos:UTIMe:ENABle on page 691
SCENario:PDW:AMMos:UTIMe:ISO on page 691

Azimuth ← Specific settings for Type > AMMOS PDW
Defines whether the angle of the RX antenna or the bearing (i.e. the azimuth direction of the emitter) is reported.

The former is suitable if spinning dish antennas are used. Use the bearing for direction finding.

Remote command:
SCENario:PDW:AMMos:AZIMuth on page 692

Plugin and plugin variables
Selects and loads a reporting template form a *.dll file. This template must exist in the "plugin" library.

Report plugins can register a set of variables that can be used to control the plugin's algorithms.

See:
- Chapter C, "Plug-in programming API", on page 863
- "To import a plugin" on page 530.

Remote command:
SCENario:PDW:PLUGin:NAME on page 690
SCENario:PDW:PLUGin:VARiable:CATalog? on page 688
SCENario:PDW:PLUGin:VARiable:SELect on page 689
SCENario:PDW:PLUGin:VARiable:VALue on page 689

22.2 How to create test reports

This section shows how to:

To enable report file generation

The "Scenario" dialog provides information on the report logging status. The related information is displayed to the right of the "Comment" field.

To enable or disable the report file generation, proceed as following:

1. In the "Scenario" dialog, select "Waveform Generation > Config".

2. In the "Signal Calculation Settings" dialog, select "Reporting > Reporting Enable > On".

To set the storage location for the generated report files

Per default, report files are stored in the user home directory of the current user, that is the `%HOMEPATH%` directory.

1. Open the "Scenario" dialog to retrieve information on the current storage location.

   Information is displayed to the right of the "Comment" field.

2. Select "Signal Calculation > Config > Config".

3. In the "Signal Calculation Settings" dialog, select "Reporting".

   The parameter "Target Path" indicates the current storage location.
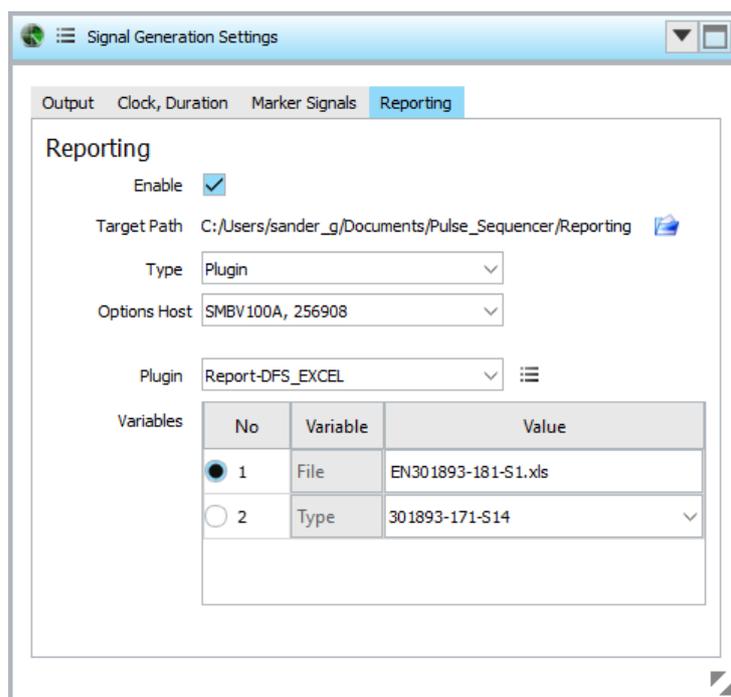
4. To change this location, select the "Set target path" icon, navigate to the directory, and confirm with "Select Folder".

To change the report type

You can also store the test reports in text form or as HTML pages (e.g. if Microsoft Excel is not installed on your PC).

To change the type of the generated reports, proceed as following:

1. In the "Scenario" dialog, select "Signal Calculation > Config > Config > Report".

2. Select "Type > Template".

3. Select "Edit" to display the current template.
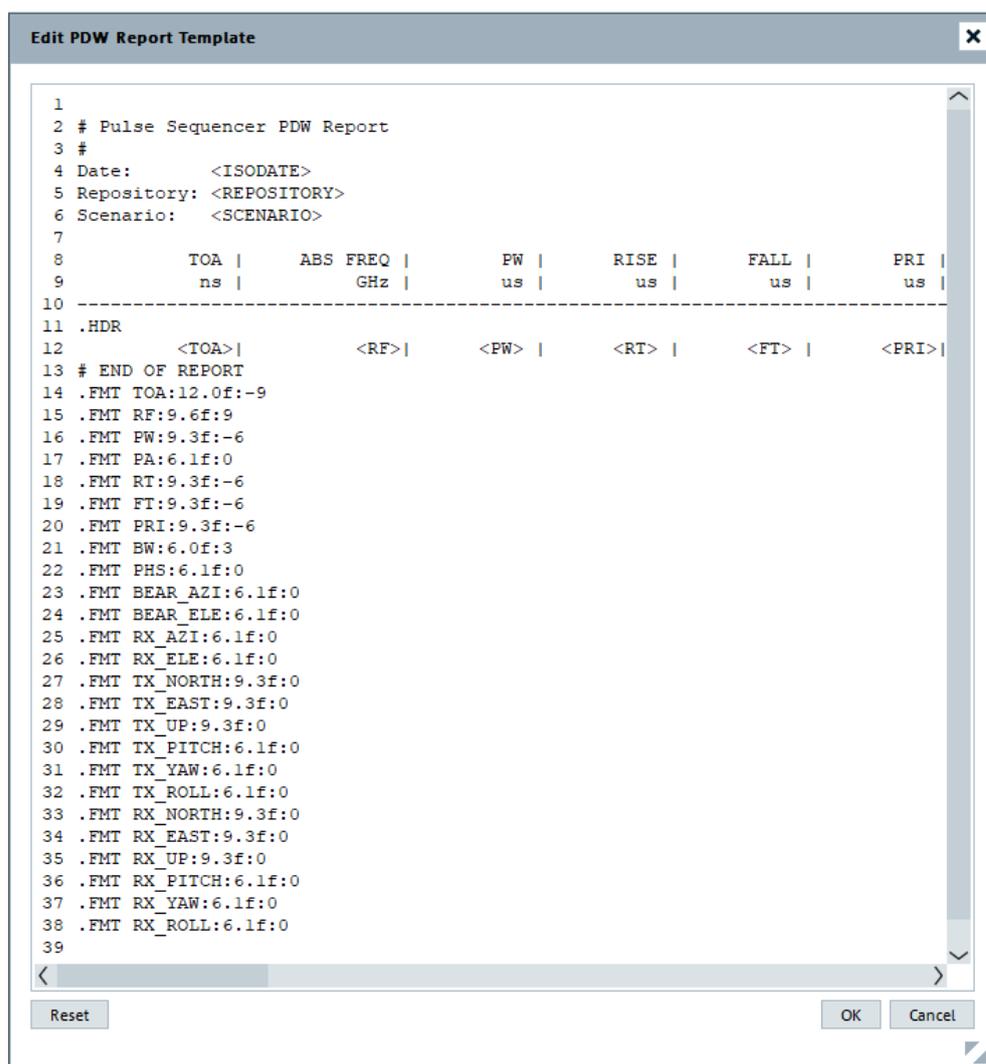
You can also change the used templates or create and load your plugin.

See:
- Chapter A.1, "File format of the reporting template", on page 838
- Chapter C, "Plug-in programming API", on page 863
- Chapter 20, "Defining complex modulation schemes and IPM profiles", on page 530.

23 Performing administration tasks

This section informs you about file handling, user management and settings you can use to speed up calculation.

23.1 Setting general program settings

Access:
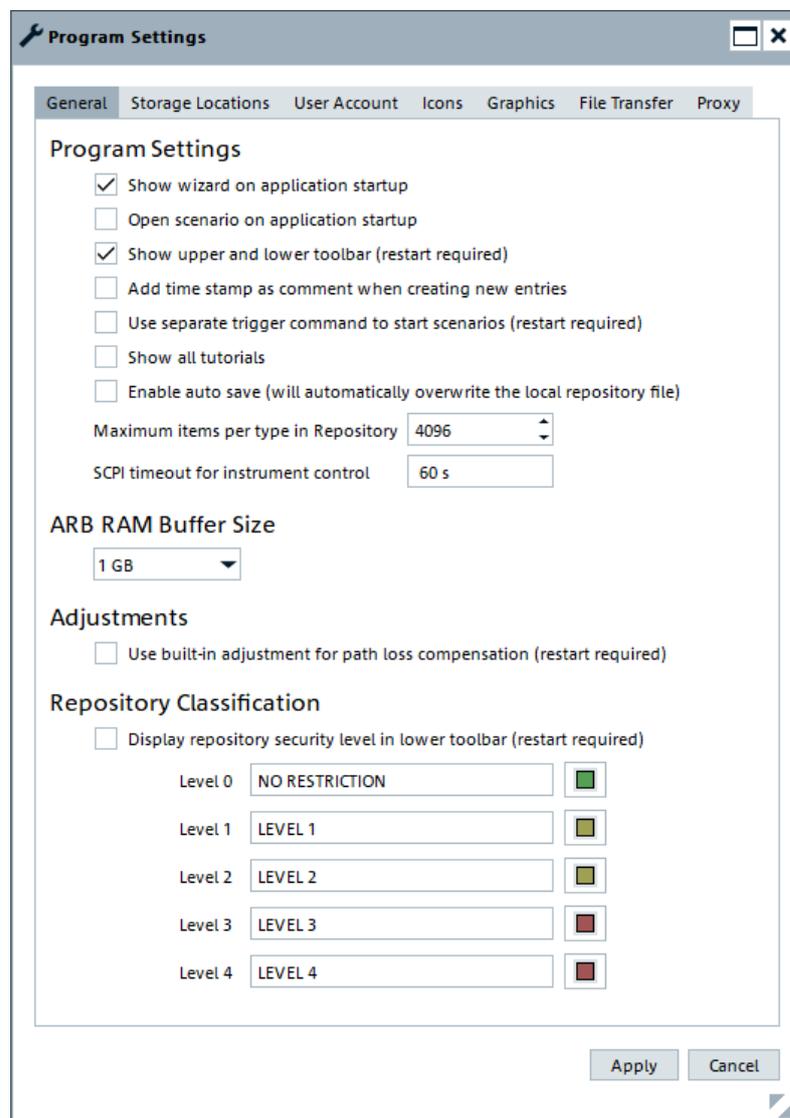
► In the menu bar, select "Configure > Settings".

Program Settings

Enable each of the optional configurations according to your needs.

Set the value for the maximum items per type in the repository. The limit for items per type is 4096.

Set the SCPI timeout for instrument control. The value defines the time span that the pulse sequencer software waits for a response after a SCPI command is sent to the signal generator. When the time span is elapsed, a VISA timeout exception is generated. The maximum value is 100 s.

Remote command:

PROGram:STARtup:WIZard:ENABle on page 599

PROGram:STARtup:LOAD:ENABle on page 599

PROGram:TOOLbar:ENABle on page 599

PROGram:COMMent:ENABle on page 597

PROGram:SCENario:XTRG:ENABle on page 598

PROGram:SCPI:TIMeout on page 598

PROGram:TUTorials:SHOW:ENABle on page 600

ARB RAM Buffer Size

Access: in the menu bar, select "Configure > Settings > General".

Sets the buffer size reserved in the RAM of your computer for the ARB waveforms calculation.

Larger buffer size speeds up waveform calculation; it is advantageous if larger ARB waveforms are calculated.

See also Chapter 23.4, "Speeding up calculation", on page 561.

Remote command:

PROGram:RAMBuff:SIZE on page 598

Adjustments

Enables using the compensation procedures, see Chapter 3.5, "Compensating path loss", on page 67.

Remote command:

PROGram:ADJustments:ENABle on page 597

Repository Classification

Define whether the classification level is displayed in the repository and sets the used colors, see "To change the general program settings" on page 50.

See also Chapter 2.8, "Customizing the software", on page 48.

Remote command:

PROGram:CLASs:ENABle on page 597

Apply, Cancel

Accept or reject the changed settings.

Remote command:
PROGram:SETTings:ACCept on page 599
PROGram:SETTings:REJect on page 599

23.2 User administration

The R&S Pulse Sequencer uses its own **user rights management** that is independent from the user administration of the operating system but follows the same principles: the current user cannot delete its own account; users with lower access rights cannot delete accounts with higher rights.

User roles

Repositories support simultaneous access from several users. Per default, all users are granted with the same (administrator) rights. The user rights can be restricted by assigning of a different user role, other than the creator role. At least one user with administrator rights must always exist.

To protect a repository from accidental changes, the repository data can be password protected. Passwords are then required to access a repository. If the password is lost, access to the repository is no longer possible.
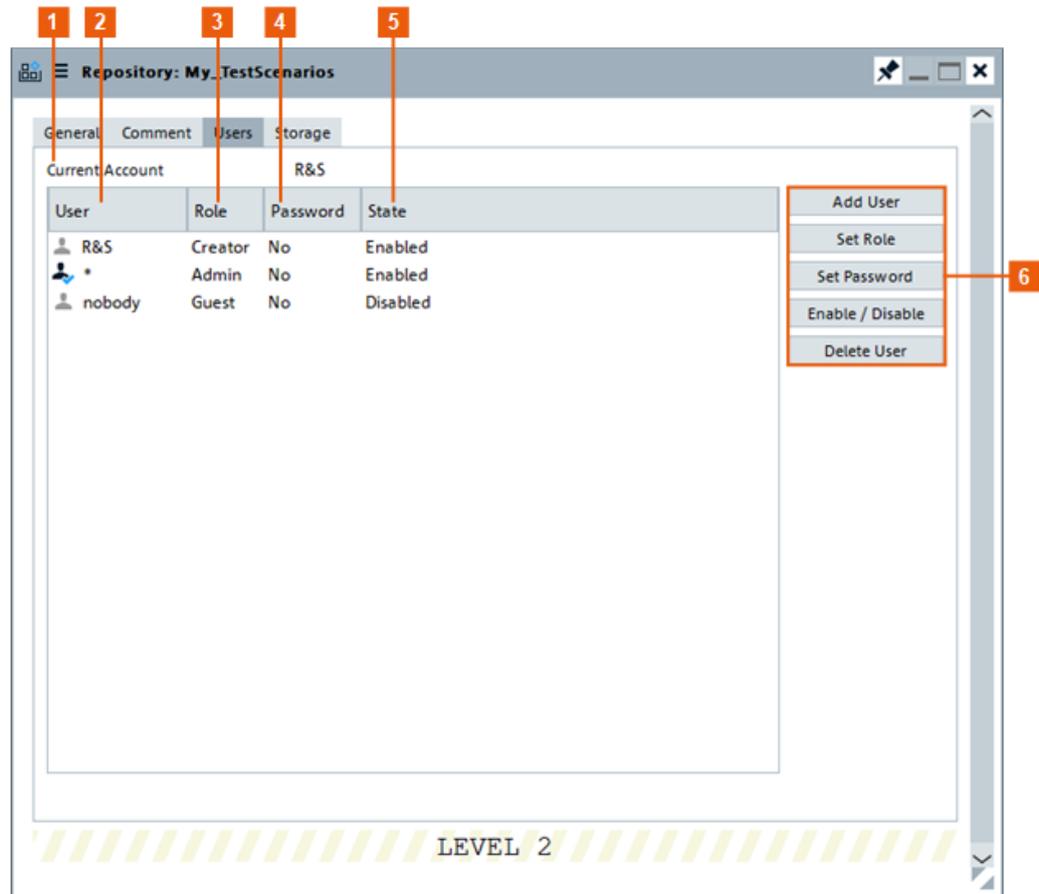
Figure 23-1: Repository > Users: understanding the displayed information

1 = Indicates the user name of the logged in user
2 = Login name or custom user name
3 = Assigned user role; the creator role is assigned automatically and cannot be changed (see also
     Table 23-1)
4 = Identifies if the repository is password protected or not
5 = Current user state
6 = Standard functions for user management

Table 23-1: Overview of the available user roles and their access rights

User role	Read access	Write access	Create a scenario from the existing elements	Delete elements	Copy repository elements
Creator	x	x	x	x	x
Admin	x	x	x	x	x
User	x	-	x	-	x
Guest	x	-	-	-	x

Per default, the R&S Pulse Sequencer uses your Windows login name to access the repositories but you can also use a custom name, see "User Account" on page 557.

See also Chapter 23.2.1, "How to set and change user passwords", on page 555.

23.2.1 How to set and change user passwords

See:
- "To protect a repository from accidental changes" on page 555
- "To remove the password protection on a repository" on page 556
- "To change the default user used to access repositories" on page 556

To protect a repository from accidental changes

1. In the project tree, double-click the repository name.

2. In the "Repository" dialog, select "Users".

3. To protect the repository with a password:

   a) Select "Users > <current Account>" and select "Set Password".
      The "Change Password" dialog opens.
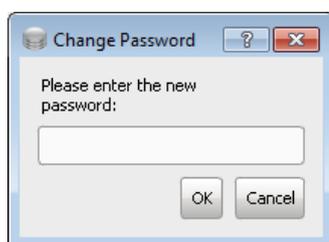
   b) Enter a password and click "Ok".

   On a new startup or when this repository is loaded, the R&S Pulse Sequencer requires the correct password to load the repository.

4. To allow read-only access:

   a) Select "Users > *" and select "Set Role".
      The "Change Account Type" dialog opens.

   b) Select a new role with read-only access, e.g. "Guest" and click "Ok".

To remove the password protection on a repository

1. In the project tree, double-click the repository name.

2. In the "Repository" dialog, select "Users".

3. Select "Users > <Current Account>" and select "Set Password".
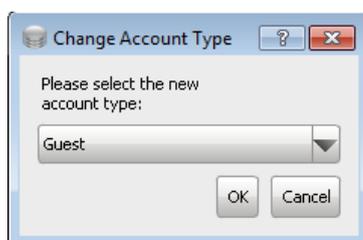
4. In the "Change Password" dialog, leave the password field blank.

5. Click"Ok".

    The "Users" dialog confirms that the repository is not protected with a password.

To change the default user used to access repositories

1. In the toolbar, select "Configure > Settings > User Account".
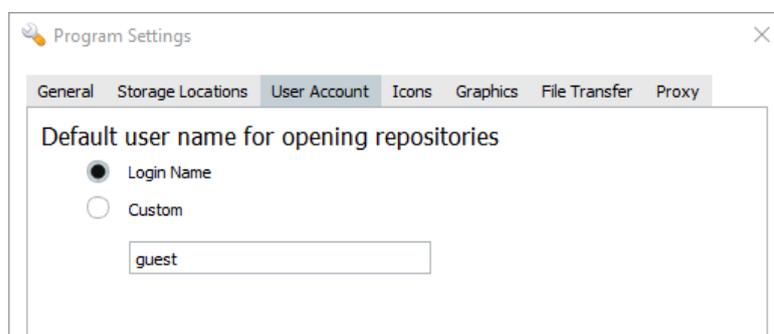
2. Select "Custom > On" and enter an existing user name.

23.2.2 Related settings

Access:

1. To open the "Repository" user management dialog:

    a) In the project tree, double-click a repository name.
        In the "Repository" dialog, select "Users".

    For example, see Figure 23-1.

2. To change the default account used to open repositories, perform the following:

    a) In the menu bar, select "Configure > Settings".
    b) Select "User Account".
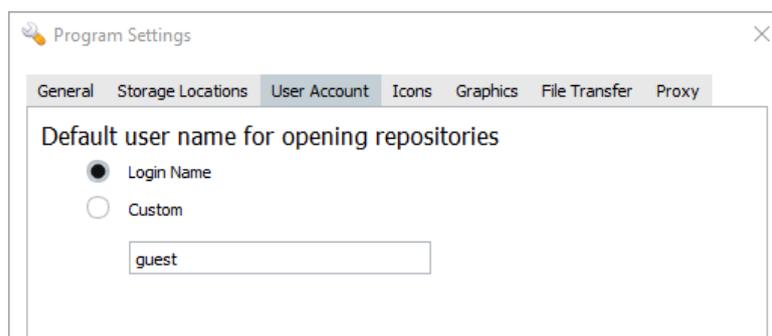
Users

Indicates the user name of the currently used account and an overview of all defined users together with their roles.

For more information, see Chapter 23.2, "User administration", on page 553.

Remote command:
REPository:ACCess? on page 722

User Account

Set the user name for opening the repositories. Per default, your Windows login name is used, but you can also define and use a custom user name.

For more information, see Chapter 23.2, "User administration", on page 553.

23.3 Defining storage locations

During software installation, the R&S Pulse Sequencer creates a predefined folder structure, concerning program and project data, report files, startup log file, calculated data, volatile data, etc. The default folders depend on how the software is installed, for all users or for a particular one, but you can change some of them, if necessary.

Table 23-2 resumes the information provided in Installing the software and extends it with information on whether and how file paths can be modified.

Table 23-2: Overview of file types, default storage locations and where to change the file paths

File type	Default location*	Modification	Information indicated in

Program data	`%PROGRAMFILES(X86)%\`
	`Rohde-Schwarz\`
	`Pulse Sequencer`
	`(Path)`
		not possible	

Project data (repository database)
SDK files
	`%PUBLIC%\Public\Documents\`
	`Rohde-Schwarz\`
	`Pulse Sequencer`
	`(DataPath)`
		"Repository Manager > Add Path/Add Home Path"
		
		See "To set the storage location for repositories" on page 559.
			• "Program Settings > Storage Location > Repository Databases"
			• For a particular repository: "Repository > Storage"
			See "To find out the current storage location for reports, program and project data" on page 559.

Report files	`%HOMEPATH%` `(HomePath or ReportPath)`	• "Program Settings > Storage Location > Reports Path" • "Scenario > Signal Calculation > Config > Reports > Target Path" See "To set the storage location for the generated report files" on page 548.	"Scenario > Status Information"
Calculated data	`%HOMEPATH%`	• "Program Settings > Storage Location > Calculated Storage > Path" • "Scenario > Signal Generation > Config > File" and "Config > Set Path"	"Scenario > Signal Generation > Config > File" and "Config > Set Path"
Volatile data	`%HOMEPATH%`	"Program Settings > Storage Location > Volatile Storage > Path"	"Scenario > Storage" block, no indication of the file path
Adjustment data	`%HOMEPATH%`	"Adjustment Configuration > Storage Location" See "How to define where adjustment data is saved" on page 74.	• "Adjustment Configuration > Storage Location" • "Signal Generators > Adj. Data", if data is saved on an instrument from the hardware setup
Startup log file	`%HOMEPATH%`	not possible	-

User settings (workspace, etc.)	`%HOMEPATH%\AppData\`
	`Roaming\Rohde-Schwarz\`
	`Pulse Sequencer`
		not possible	-

[*] Software installation for all users.

23.3.1 How to set and change storage locations

See:

- "To find out the current storage location for reports, program and project data" on page 559
- "To set the storage location for repositories" on page 559
- See also Chapter A, "Supported file types and file formats", on page 837.

To find out the current storage location for reports, program and project data

Use one of the following:

1. Select "Menu bar > Help > Debug Information".
   Observe the information in the R&S Pulse Sequencer section.

   Storage locations are indicated for *reports, program and project data*.

2. Select "Menu bar > Configure > Settings > Storage Locations".
   Observe the indication for "Repository Databases > Install Path" and "Reports".

   Storage locations are indicated for *reports and project data*.

3. Double-click a repository name and select "Repository > Storage".

   The storage location of the *particular repository* is indicated.

4. Open a "Scenario" and observe the displayed "Info".

   The *reports* storage location is indicated.

To set the storage location for repositories

Per default, repositories are saved in the `DataPath` directory (see Table 23-2). Alternatively, you can define a specific path and, for example, save repositories in a network folder. The latter allows for repository sharing among different users or access from different computers.

1. Select "Menu bar > Configure > Settings > Storage Locations".

   Select the respective storage locations, e.g. for reports.

2. Select "Menu bar > Repository > Load + Manage".

   The "Repository Manager" opens.

3. In the "Repository Manager", select "Add Home Path".

4. In the "Repository Manager", select "Add Path".

5. Navigate to the required directory. Select it.

   Found repository files are displayed in the list.

To set the storage location for adjustment data

► See "How to define where adjustment data is saved" on page 74.

23.3.2 Storage locations settings

Access:

1. In the menu bar, select "Configure > Settings".

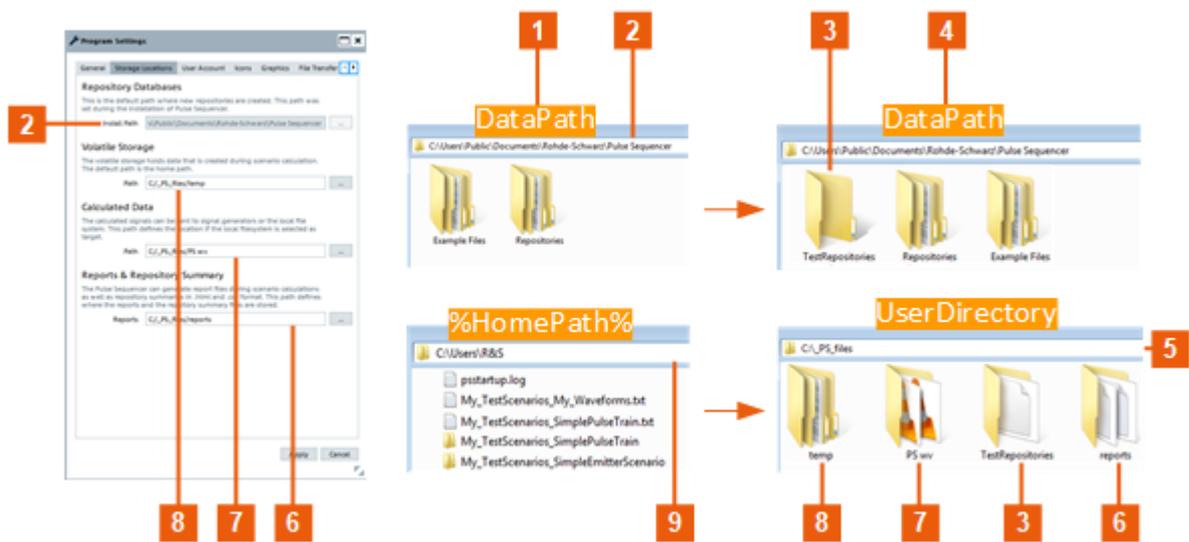2. In the "Program Settings" dialog, select "Storage Location".

Figure 23-2: Storage Location: Understanding the displayed information

1 = Default storage location
2 = Default DataPath - cannot be changed; subdirectories are permitted (see "To set the storage location for repositories" on page 559)
3 = User-defined repository storage location, as a subdirectory of the default DataPath/HomePath directory or in a freely defined location (e.g. in the UserDirectory)
4 = User-defined storage location or subdirectories
5 = User-defined storage location (UserDirectory)
6 = Reports, default and user-defined location
7 = Calculated data, default and user-defined location
8 = Volatile data, default and user-defined location
9 = Default HomePath; per default holds repository files, calculated data, etc.

For step-by-step descriptions, see Chapter 23.3.1, "How to set and change storage locations", on page 558.

Settings:

Repository Databases

Indicates information concerning the repository storage place.

"Install Path"   Storage location for repository files.
Resembles the information displayed in the "Repository Manager" (see Figure 4-1).

Remote command:
`PROGram:PATH:INSTall?` on page 597

Volatile Storage

Defines the directory that holds volatile data, created during scenario preparation.

Remote command:
PROGram:PATH:VOLatile on page 598

Calculated Data

Defines the directory that holds the calculated waveforms, if "Target > File" (see "Signal Calculation" on page 130).

Works like "Scenario > Signal Generation > Config > Set Path".

Remote command:
PROGram:PATH:CALCulated on page 598

Reports

Defines the directory that holds generated reports.

Works like "Signal Calculation > Config > Config > Reporting > Target Path".

Remote command:
PROGram:PATH:REPort on page 598

23.4 Speeding up calculation

In R&S Pulse Sequencer, you have the different possibilities to speed up the calculation process. Use them exclusively or simultaneously.

Overview

To improve performance, especially if movements are simulated and large waveforms are calculated, we recommend that you:

- Optimize calculation speed by enabling multiple threads.
- Use a larger ARB RAM buffer size.
- Enable GPU-based antenna pattern calculation
- Enable FTP transfer for large files to the instruments
- Use the extended sequencer option with the possibility to fall back automatically to multi-segment waveform generation
- Reduce clock rate and oversampling
- If the radar signal is composed of short pulses, disable the calculation of antenna attenuation for each sample. You can also disable this setting if a constant pulse level is acceptable for imported waveforms or CW signals.

Step-by-step instructions

See Chapter 23.4.1, "How to optimize performance", on page 562.

Settings

See Chapter 23.4.2, "Related settings", on page 563.

23.4.1 How to optimize performance

To speed up calculation by enabling multiple threads

1. In the "Scenario" dialog, select "Signal Calculation > Config > Config > Output".

2. Select "Optimize calculation speed > Enable multithreading > On".

To speed up calculation by using a larger ARB RAM buffer size

1. In the menu bar, select "Configure > Settings > General".

2. If your computer is equipped with enough RAM memory, set "ARB RAM Buffer Size = 1 GB".

   Smaller buffer sizes result in longer calculation times.

To accelerate the antenna pattern calculation and display

The calculation of antenna patterns can be a time-consuming operation. To accelerate this operation and the 2D and 3D display of the antenna patterns, proceed as follows:

1. In the menu bar, select "Configure > Settings > Graphics".

2. Select "GPU Support > Use GPU based antenna pattern calculation > On".

   GPU Support
   ☑ Use GPU based antenna pattern calculation (restart required)

3. Click "Apply".

4. Restart the software.

   The GPU (graphics processing unit) takes over the calculation. The calculation is accelerated around 800 times.

To enable FTP for faster file transfer to instruments

► See Chapter 17.3.1, "How to enable FTP transfer", on page 485.

To define the format of the calculated file

1. In the "Scenario" dialog, select "Signal Calculation > Config > Config > Output".

2. If the signal generator is equipped with the required option, it is best to select "Use extended sequencing option if possible... > On".

3. Otherwise, select "Create ARB multi-segment waveform if possible... > On".

   The software calculates a waveform, most suitable for the current situation, incl. current scenario, signal properties and used instrument.
   If the preferred waveform format is evaluated as unsuitable, an automatic fallback mechanism selects the best waveform format.

To reduce the file size of the created file by adjusting the clock rate

1. In the "Scenario" dialog, select "Signal Calculation > Config > Config > Clock, Duration".

2. To reduce the clock rate, use one of the following:
   a) Select "Clock Rate > Manual".
      Enter a smaller clock rate value manually.
   b) Select "Clock Rate > Automatic".
      Change the minimum clock rate value and the minimum oversampling factor.

   The size of the waveform is reduced.

To speed up calculation for radar signals with short pulses

1. In the "Scenario" dialog, select "Signal Calculation > Config > Config > Output".

2. Set "ARB Details > Calculate antenna attenuation for each sample ... > Off".

23.4.2 Related settings

Optimize calculation speed
Enable this parameter to apply multithreading and decrease the calculation time.

Select one of the options:
- "Automatic"
  The system automatically applies the optimum number of threads for the scenario.
- "Manual"
  Define the number of threads to be used.

See also Chapter 23.4, "Speeding up calculation", on page 561.

Remote command:
SCENario:OUTPut:MULTithread on page 766
SCENario:OUTPut:MTMode on page 766
SCENario:OUTPut:MTTHreads on page 766

ARB RAM Buffer Size
Access: in the menu bar, select "Configure > Settings > General".

Sets the buffer size reserved in the RAM of your computer for the ARB waveforms calculation.

Larger buffer size speeds up waveform calculation; it is advantageous if larger ARB waveforms are calculated.

See also Chapter 23.4, "Speeding up calculation", on page 561.

Remote command:
PROGram:RAMBuff:SIZE on page 598

GPU Support
Access: "Configure > Settings > Graphics".

If enabled, this accelerates the antenna pattern calculation and display.

See also Chapter 23.4, "Speeding up calculation", on page 561.

Remote command:
PROGram:GPU:ENABle on page 597

Use the FTP data transfer for large files
Access: in the menu bar, select "Configure > Settings > File Transfer".

Select a "TX Block Size".

Enable an FTP data transfer to speed up the data upload to the connected signal generator.

Enter the "User Name" and "Password" of the processing instrument.

How to: Chapter 17.3.1, "How to enable FTP transfer", on page 485.

Remote command:
PROGram:TRANsfer:FTP:BLOCksize on page 599
PROGram:TRANsfer:FTP:ENABle on page 600
PROGram:TRANsfer:FTP:UNAMe on page 600
PROGram:TRANsfer:FTP:PASSwd on page 600

Output Format
Access: in the "Scenario" dialog, select "Signal Calculation > Config > Config... > Output".

Defines the type of the generated waveform file:
● A single ARB file
● An ARB multi-segment waveform file
● A waveform file describing one pulse and two files in Rohde & Schwarz proprietary file format describing control data.

The software calculates the most suitable waveform for the current situation, incl. current scenario, signal properties and used instrument.

If the preferred waveform format is evaluated as unsuitable, an automatic fallback mechanism selects the best waveform format.

See also:

"Waveform types" on page 475.

Remote command:
SCENario:OUTPut:FORMat on page 762

Clock, Duration
See "Clock, Duration" on page 479.

ARB Details

Access: in the "Scenario" dialog, select "Signal Calculation > Config > Config > Output".

You can enable the following functions:

- Calculate antenna attenuation.
  If enabled, calculates antenna attenuation for each sample. Otherwise a lookup at center position is used.
  Enable this setting by the calculation of imported waveforms and CW signals or if the signal comprises long pulses.
- Enable signal truncation.
  If enabled, the calculation process allows truncated signals at the end of the signal duration, for example, only a fraction of a pulse if the time ends within the last pulse.

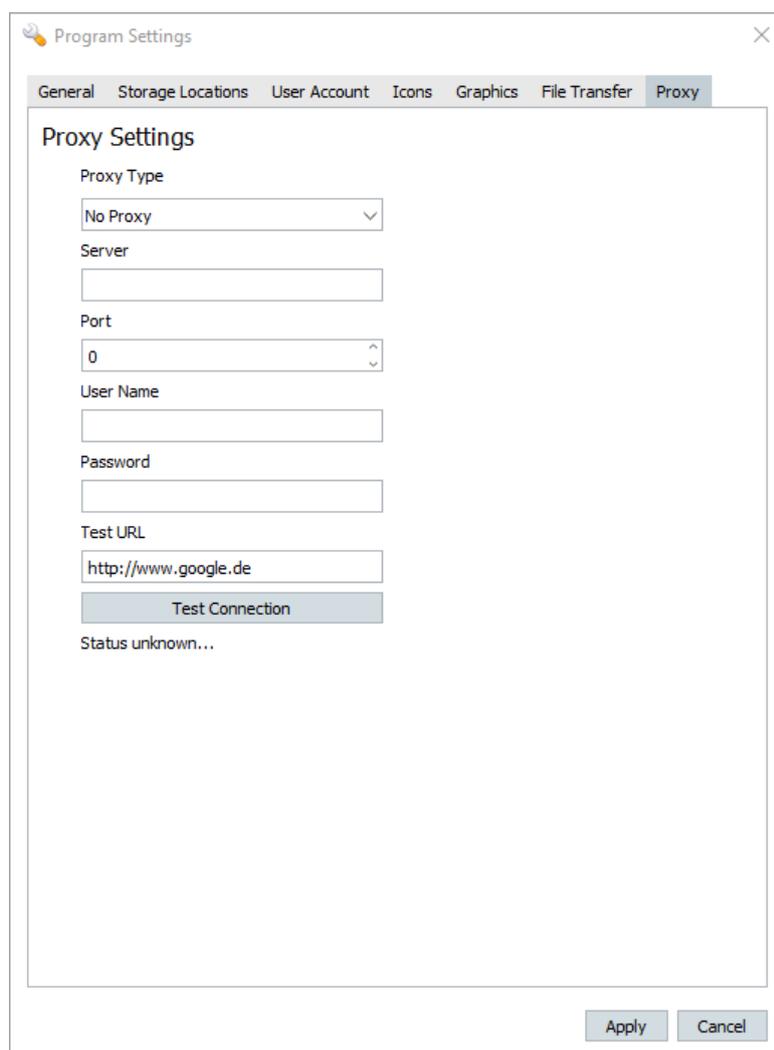See also Chapter 23.4, "Speeding up calculation", on page 561.

Remote command:
SCENario:OUTPut:ARB:DETails:ALBS on page 765
SCENario:OUTPut:ARB:DETails:TRUNcate on page 766

23.5 Configuring a proxy server

Access: in the menu bar, select "Configure > Settings > Proxy".

A proxy server is useful when a PC running the R&S Pulse Sequencer in a local network needs secure access to the internet, for example, to download georeferenced maps.

To configure a proxy server

1. Set "Proxy Type" = "HTTP Proxy".

2. Enter the IP and port of the proxy server.

3. Enter the user name and password to access the proxy server.

4. Enter a test URL of your choice.

5. Select "Test Connection" to attempt connection to the specified test URL.

   The dialog displays the status of the connection test.

6. Select "Apply".

7. To reject the changes and close the dialog, select "Cancel".
   The settings remain saved in the dialog fields if you do not clear them.

Select "Proxy Type = No Proxy" to deactivate the proxy settings.

23.6 Updating the software

You find the last released software version and the release notes on the R&S Pulse Sequencer product page at:

https://www.rohde-schwarz.com/software/pulse-sequencer/.

To update the software version

1. Download the software and the release notes.

2. Follow the installation instructions provided in the release notes.

24 Automation of R&S Pulse Sequencer

In addition to working with the R&S Pulse Sequencer software interactively, it is also possible to operate and control it from a remote PC. Remote control operation allows you to automate the configuration process and is useful when a higher configuration speed is required.

Remote control is an operation by which the software is operated remotely from a so called controller. A controller can be any PC on which some required software is installed and configured. R&S Pulse Sequencer can be but not necessarily is installed on the same PC. In case R&S Pulse Sequencer is not physically located on the controller's PC, a LAN connection between both PCs is required.

The following figures depict the following three possible cases:

- The controller is a script in the R&S Pulse Sequencer itself and controls the software and possibly also other instruments.
- The controller and R&S Pulse Sequencer are on the same PC.
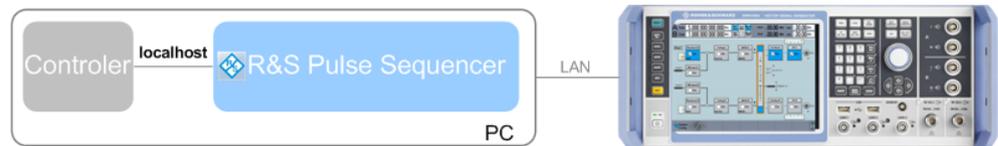
- The controller and the software are installed on two separate PCs.
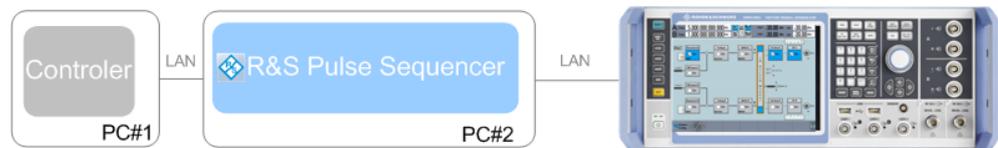
In remote control operation, the software itself is operated by remote control commands. For description of specific SCPI commands, refer to Chapter 25, "Remote control commands", on page 577.

ⓘ We assume that the controller and the signal generator have already been set up for remote operation in a network as described in the documentation of the instrument. A knowledge about the remote control operation and the SCPI command syntax is assumed.

Supported interfaces and protocols

R&S Pulse Sequencer support remote control over the LAN interface and with the socket protocol. The socket controller is sufficient; a VISA (Virtual Instrument Software Architecture) library is not required.

Socket communication requires the specification of the port (commonly referred to as port number); the registered default port is 5025 but it can be changed if necessary.

Socket communication

Socket communication is a simple network communication and is also referred to as "Raw Ethernet communication". It is available by default on all operating systems.

The simplest way to establish socket communication is to use the built-in telnet program. The telnet program is part of every operating system and supports a communication with the software on a command-by-command basis. For more convenience and to enable automation by programs, user-defined sockets can be programmed.

24.1 Remote settings

Access:

► In the menu bar, select "Configure > Remote Control".
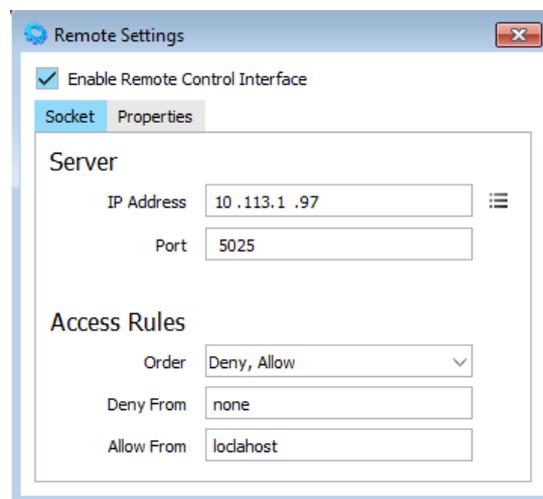
Settings:

Enable Remote Control Interface
Activates the remote control interface.

If enabled, the "IP Address" of the server and the used port are indicated in the taskbar.

Socket
Configures the remote control interface.

Server ← Socket
Configures the TCP/IP server to be used for remote control:

"IP Address"   Enter the IP address of the TCP/IP server or select it from the list of automatically retrieved values.

"Port"   Sets the port number for the remote control interface.

Access Rules ← Socket
Defines the access rules:

"Order"   Sets the order of the deny/allow rules.

"Deny from/Allow from"

Deny or allow remote access.

Allowed values:
- none
- all
- localhost i.e. 127.0.0.1
- If DNS supported, the hostname of a particular computer
- IPv4 address
- Address area, e.g. 192.168.10

Properties
Set the serial number of the software. The serial number is a unique 6-digit number, returned as a result of the $*IDN?$ query.

See

24.2 Script manager settings

The R&S Pulse Sequencer provides also a built-in java script editor. With this function, you can write scripts that control the software.

You can also use scripts to control the instruments, that are part of the test setup. Examples for the latter are one or more signal generators, or an oscilloscope.
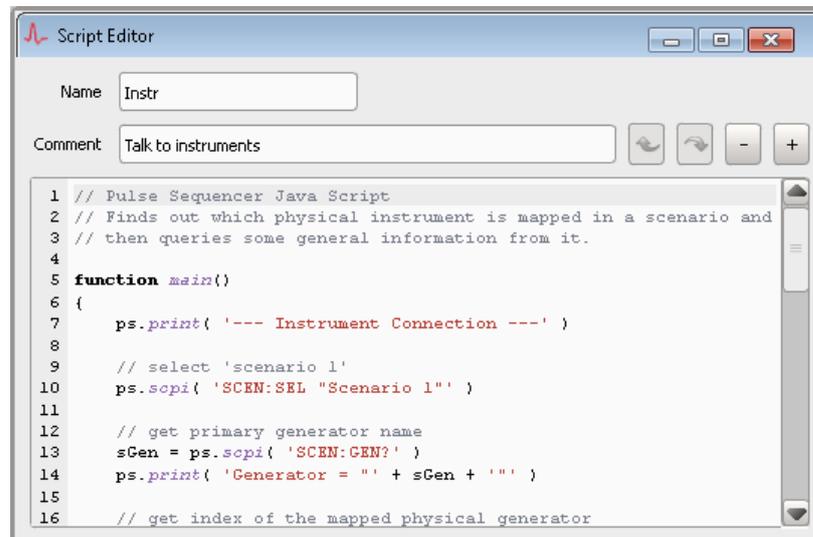
To access the script manager

1. In the menu bar, select "Tools > Script".

    The "Script Manager" lists the available scripts with their name and comment.
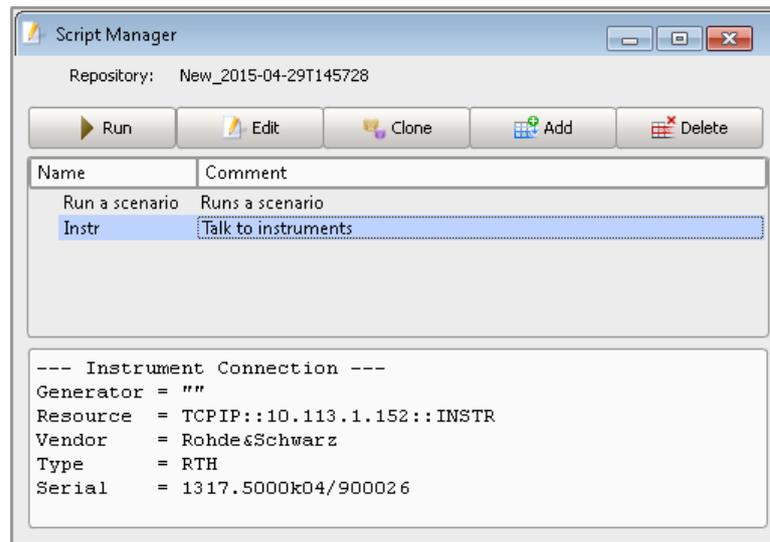
2. To execute a script, select "Run".
   The script output from the `ps.print` function is displayed in the "Script Manager".

3. To create a script, select "Add" or use the "Clone" function.

4. To edit a script, double-click a script in the script list or select a script and select "Edit".

You can write or copy your java script file here.

For description of the script syntax and script examples, see:

● Chapter D, "Scripting API", on page 882
● Chapter D.3, "Script examples", on page 890

Script Manager

The dialog indicates the current repository and lists the available scripts with their name and comment.

If a script is executed, the script output, i.e. the output from the `ps.print` function, is displayed in the lower part of the dialog.

"Run"   Executes the selected script.

"Edit"   Opens the selected script for editing in the "Script Editor".

"Clone, Add, Delete"
                 Performs the corresponding action.

Script Editor

The script editor is a dialog, where you can change the name of a script, add a comment and create or edit the script code.

The icons provide common "Undo/Redo" and "Zoom In/Out" functions.

24.3 SCPI log settings

The "SCPI Log" dialog provides a sequential register of the SCPI commands associated with actions performed on the user interface. Copy the list of recorded SCPI commands to the clipboard and create scripts to automate the usage of the R&S Pulse Sequencer.

To start logging SCPI commands

1. Select the "SCPI Log" dialog icon ⬤ on the taskbar.

   The "SCPI Log" dialog opens.
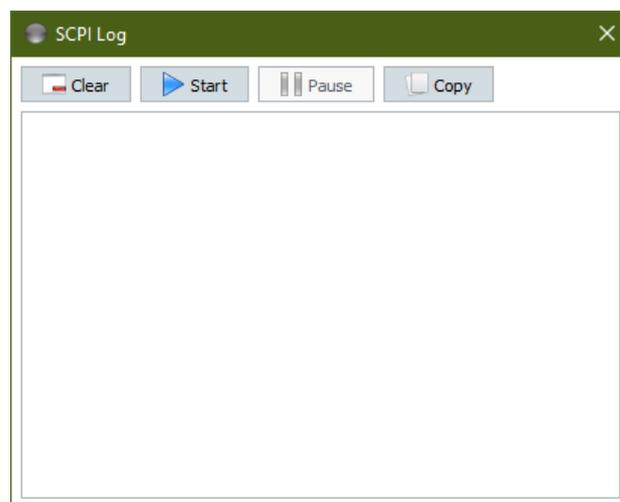
2. Select the "Start" button.

   The "SCPI Log" dialog button on the taskbar changes to 🔴.
   The "Start" button is grayed out and the "Pause" button becomes available.

3. To skip logging, select "Pause".

4. To resume logging, select "Start".

See also Chapter 24.5, "How to log SCPI commands", on page 575.

Copy
Copies the copy the content of the "SCPI Log" dialog to the clipboard.

Clear
Deletes the content of the "SCPI Log" dialog.

Context menu
In the "SCPI Log" dialog, open the context menu (right click) to access a context menu with basic text editing capabilities.

24.4 How to configure and enable remote control of R&S Pulse Sequencer

See:
- "To access the required settings" on page 573
- "To use the console to test SCPI commands" on page 574
- "To find the SCPI command corresponding to a parameter on the user interface" on page 575

To access the required settings

1. In the menu bar, select "Configure > Remote Control".

2. Select "Enable Remote Control Interface".

3. Do not change the default socket port.
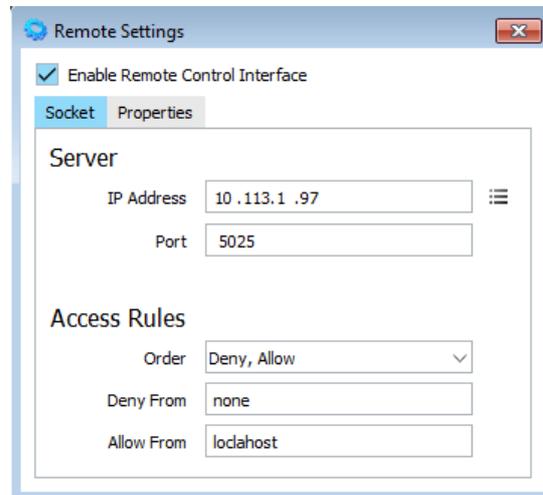
To identify the R&S Pulse Sequencer by its serial number

If you remotely control several R&S Pulse Sequencer from the same controller, it is useful to use serial numbers to distinguish between the different installations.

To set the serial number of the software:

1. In the menu bar, select "Configure > Remote Control".

2. Open the "Properties" tab.

3. In the "Serial" field, enter a unique 6-digit number.
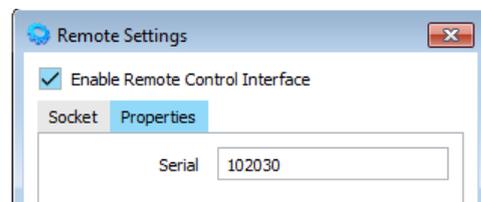
To query the serial number, use the command `*IDN?`.
See also "To use the console to test SCPI commands" on page 574.

To use the console to test SCPI commands

R&S Pulse Sequencer provides a built-in console window that allows you to test SCPI commands or commands sequences.

1. To access this window, in the menu bar, select "Tools > Console"

2. In the "Command Console" window, type `SCPI` to start the remote control mode.

3. Enter any SCPI command.

See "To use the console to test SCPI commands" on page 574.
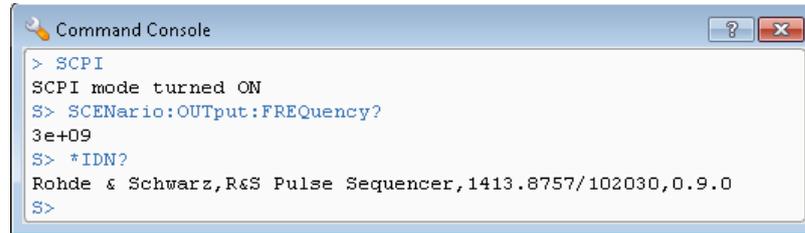
```
Command Console                                    ?  ×
> SCPI
SCPI mode turned ON
S> SCENario:OUTput:FREQuency?
3e+09
S> *IDN?
Rohde & Schwarz,R&S Pulse Sequencer,1413.8757/102030,0.9.0
S>
```

To find the SCPI command corresponding to a parameter on the user interface

Perform one of the following:

1. Right mouse click to open the context menu of the parameter.

2. Press F1 to open the "Help" window. You find a link to the description of the SCPI command right after the parameter description.

   For a concise description of the available SCPI commands, see Chapter 25, "Remote control commands", on page 577.

24.5 How to log SCPI commands

To start logging, see To start logging SCPI commands.

1. Create the emitter.

   For details on creating scenarios and setting emitter properties, see:
   - Chapter 16, "Creating complex 2D scenarios with receiver and TX items", on page 415
   - Chapter 16.7, "Background emitters properties", on page 462

   The "SCPI Log" dialog displays:
   ```
   SCENario:CEMit:ADD
   SCENario:CEMit:SELect 4
   SCENario:LOCalized:TYPE BACKGROUND
   ```
   A new emitter is added.

2. Rename the emitter.

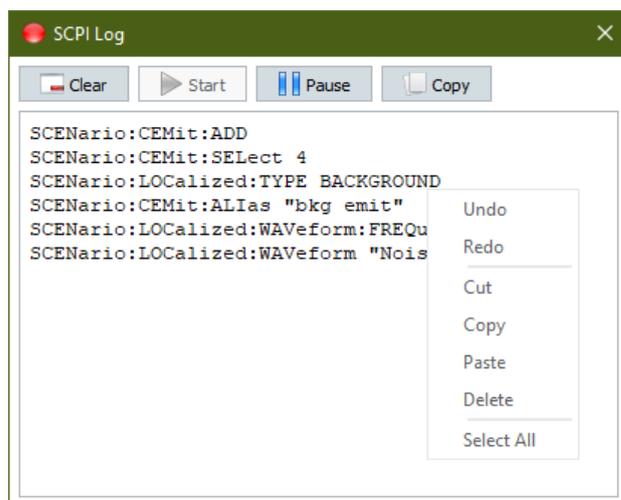   A new line appears in the "SCPI Log" dialog:
   ```
   SCENario:CEMit:ALIas "bkg emit"
   ```

3. To edit the logged SCPI command list, open the context menu (right-click). The provided functions are self-explanatory.

   The figure below shows the final result.

ⓘ   The context menu only edits the item or items selected in the SCPI log list. It does not change the GUI.

25 Remote control commands

The following commands are required to perform signal generation with the R&S Pulse Sequencer option in a remote environment. We assume that the R&S Pulse Sequencer has already been set up for remote operation in a network as described in the R&S Pulse Sequencer documentation. Knowledge about the remote control operation and the SCPI command syntax is assumed.

25.1 Conventions used in SCPI command descriptions

The following conventions are used in the remote command descriptions:

- **Command usage**
  If not specified otherwise, commands can be used both for setting and for querying parameters.
  If a command can be used for setting or querying only, or if it initiates an event, the usage is stated explicitly.

- **Parameter usage**

If not specified otherwise, a parameter can be used to set a value, and it is the result of a query.
Parameters required only for setting are indicated as **Setting parameters**.
Parameters required only to refine a query are indicated as **Query parameters**.
Parameters that are only returned as the result of a query are indicated as **Return values**.

- **Conformity**
  Commands that are taken from the SCPI standard are indicated as **SCPI confirmed**. All commands used by the R&S Pulse Sequencer follow the SCPI syntax rules.

- **Asynchronous commands**
  A command which does not automatically finish executing before the next command starts executing (overlapping command) is indicated as an **Asynchronous command**.

- **Reset values (*RST)**
  Default parameter values that are used directly after resetting the instrument (`*RST` command) are indicated as ***RST** values, if available.

- **Factory preset values**
  Default parameter values that are reset only by factory preset.

- **Default unit**
  The default unit is used for numeric values if no other unit is provided with the parameter.

- **Manual operation**
  If the result of a remote command can also be achieved in manual operation, a link to the description is inserted.

25.2　Programming examples

The following sections provide simple programming examples for the R&S Pulse Sequencer. The purpose of the examples is to present **all** commands for a given task. In real applications, one would rather reduce the examples to an appropriate subset of commands.

The programming examples have been tested with the built-in console which provides an environment for the development and execution of remote tests. To keep the examples as simple as possible, only the "clean" SCPI-like syntax elements are reported. The // character combination ("double-slash") indicates a non-executable command line (e.g. a comment).

25.3　Common commands

Common commands are described in the IEEE 488.2 (IEC 625-2) standard. These commands have the same effect and are employed in the same way on different devices. The headers of these commands consist of "*" followed by three letters. Many common commands are related to the Status Reporting System.

Available common commands:

---

*IDN?

Identification

Returns the instrument identification.

Return values:

<ID>	"Rohde&Schwarz,<device type>,<part number>/<serial number>,<firmware version>"
Example:	`Rohde & Schwarz,R&S Pulse Sequencer,` `1413.8757/102030,1.5.6320`
Usage:	Query only

---

*OPT?

Option identification query

Queries the options included in the instrument. For a list of all available options and their description, refer to the specifications document.

Return values:

<Options>	The query returns a list of options. The options are returned at fixed positions in a comma-separated string. A zero is returned for options that are not installed.
Usage:	Query only
Manual operation:	See "Instrument" on page 81

---

*RST

Reset

Sets the software to a defined default status. The default settings are indicated in the description of commands.

Unsaved settings are lost.

Usage:	Setting only

25.4 Commands with similar syntax

This section describes commands, that follow similar syntax and are common to the command groups listed in the following sections.

- `<COMMAND>:ADD`

- `<COMMAND>:CATalog`
- `<COMMAND>:CLEar`
- `<COMMAND>:COMMent`
- `<COMMAND>:COUNt`
- `<COMMAND>:CREate`
- `<COMMAND>:DELete`
- `<COMMAND>:INSert`
- `<COMMAND>:NAME`
- `<COMMAND>:REMove`
- `<COMMAND>:SELect`

Commands for handling of repository elements

Repository elements are referenced by their *unique name*. The name can contain empty spaces. In each of the command groups, there is a `...:SELect` command. Use this command to select one repository element to which the subsequent commands apply.

A typical remote control sequence would look like in the Example"Handling repository elements" on page 580.

Example: Handling repository elements

The following is a simple example that explains the principle of repository elements handling by using the common commands.

```
// activate the remote control mode
SCPI
// create new empty repository
REPository:CREate "Repository for tests"
REPository:CATalog?
// "Repository for tests"
REPository:SAVE

// create two new scenarios
SCENario:CREate "New 1"
SCENario:CREate "New 2"
SCENario:SELect "New 1"
SCENario:COMMent "simple test scenario"
// rename the scenario
SCENario:NAME "New"
// remove it from the repository
SCENario:CATalog?
// "New", "New 2"
SCENario:REMove "New 2"
SCENario:CATalog?
// "New"

// create new pulse
```

```
// PULSe:CREate "New"
PULSe:CATalog?
// "New"
// although the pulse and the scenario use the same name,
// they are different and unique elements for the repository
```

Commands for handling of list and table items

Several repository elements are described in table form or as lists. For example, the sequence description table, the FM and AM Step modulations, IPM profiles in list form, or lists of multiple emitters.

Lists and tables are composed of items, where items are referenced by their *number*. Use the `...:SELect` command to select one item to which the subsequent commands apply.

A typical remote control sequence would look like in the Example"Handling items" on page 581.

Example: Handling items

The following is a simple example that explains the principle of items handling by using the common commands.

```
// activate the remote control mode
SCPI
// creates a pulse-based sequence with 2 items (i.e. segments)
SEQuence:CREate "Seq1"
SEQuence:SELect "Seq1"
SEQuence:TYPE PULSe
SEQuence:ITEM:ADD
SEQuence:ITEM:COUNt?
// 1
SEQuence:ITEM:SELect 1; SEQuence:ITEM:TYPE PULSe
SEQuence:ITEM:PULSe "P1"
SEQuence:ITEM:ADD
SEQuence:ITEM:COUNt?
// 2
SEQuence:ITEM:DELete 2
SEQuence:ITEM:COUNt?
// 1
SEQuence:ITEM:CLEar
SEQuence:ITEM:COUNt?
// 0
```

ANTenna:CATalog?
DSRC:CATalog?
EMITter:CATalog?
RECeiver:CATalog?
GENerator:CATalog?
IPM:CATalog?
PLATform:CATalog?
PLUGin:CATalog?
PULSe:CATalog?
SCAN:CATalog?
SCENario:CATalog?
SEQuence:CATalog?
WAVeform:CATalog?
REPository:CATalog?

Queries the available repository elements in the database.

Return values:

<Catalog>   string

Example: See Example"Working with repositories" on page 721

Usage: Query only

Manual operation: See "Info" on page 107

ANTenna:CREate <Create>
DSRC:CREate <Create>
EMITter:CREate <Create>
RECeiver:CREate <Create>
GENerator:CREate <Create>
IPM:CREate <Create>
PLATform:CREate <Create>
PLUGin:CREate <Create>
PULSe:CREate <Create>
SCAN:CREate <Create>
SCENario:CREate <Create>
SEQuence:CREate <Create>
WAVeform:CREate <Create>
REPository:CREate <Create>

Creates a repository element with the selected name.

Setting parameters:

<Create>   string

                        Must be unique for the particular type of repository elements.
                        May contain empty spaces.

Example: See Example"Working with repositories" on page 721

Usage: Setting only

Manual operation: See "Info" on page 107

ANTenna:SELect <Select>
DSRC:SELect <Select>
EMITter:SELect <Select>
PLATform:EMITter:SELect <Select>
PLATform:SELect <Select>
RECeiver:SELect <Select>
GENerator:SELect <Select>
IPM:SELect <Select>
PLUGin:SELect <Select>
PULSe:SELect <Select>
SCAN:SELect <Select>
SCENario:SELect <Select>
SEQuence:SELect <Select>
WAVeform:SELect <Select>
REPository:SELect <Select>

Selects the repository element to which the subsequent commands apply.

Parameters:

<Select>   string

Element name, as defined with the `...:CREate` or `...:NAME` command.
To query the existing elements, use the `...:CATalog?` command.
For example, `REPository:CATalog?`.

Example: See Example"Working with repositories" on page 721.

Manual operation: See "Info" on page 107

ANTenna:NAME <Name>
DSRC:NAME <Name>
EMITter:MODE:NAME <Name>
EMITter:NAME <Name>
EMITter:MODE:BEAM:NAME <Name>
PLATform:NAME <Name>
RECeiver:NAME <Name>
GENerator:NAME <Name>
IPM:NAME <Name>
IPM:PLUGin:NAME <Name>
PLUGin:MODule:NAME?
PLUGin:NAME <Name>
PULSe:MOP:PLUGin:NAME <Name>
PULSe:NAME <Name>
SCAN:NAME <Name>
SCENario:NAME <Name>
SEQuence:NAME <Name>
WAVeform:NAME <Name>

Renames the selected repository element.

Parameters:

<Name>   string

Must be unique for the particular type of repository elements.
May contain empty spaces.

Example: See Example"Handling repository elements" on page 580

Manual operation: See "Waveform Name and Comment" on page 374

ANTenna:COMMent <Comment>
DSRC:COMMent <Comment>
EMITter:COMMent <Comment>
INSTrument:COMMent <Comment>
PLATform:COMMent <Comment>
RECeiver:COMMent <Comment>
GENerator:COMMent <Comment>
IPM:COMMent <Comment>

PLATform:COMMent <Comment>
PLUGin:COMMent <Comment>
PULSe:COMMent <Comment>
PULSe:MOP:COMMent <Comment>
SCAN:COMMent <Comment>
SCENario:COMMent <Comment>
SEQuence:COMMent <Comment>
WAVeform:COMMent <Comment>
REPository:COMMent <Comment>

Adds a description to the selected repository element.

Parameters:

<Comment>   string

Example: See Example"Working with repositories" on page 721

Manual operation: See "Comment" on page 108

ANTenna:REMove <Remove>
DSRC:REMove <Remove>
EMITter:REMove <Remove>
PLATform:REMove <Remove>
RECeiver:REMove <Remove>
GENerator:REMove <Remove>
IPM:REMove <Remove>
PLUGin:REMove <Remove>
PULSe:REMove <Remove>
SCAN:REMove <Remove>
SCENario:REMove <Remove>
SEQuence:REMove <Remove>
WAVeform:REMove <Remove>
REPository:REMove <File>[,<Username>,<Passwd>]

Removes the selected element from the workspace.

The element must not reference any child elements. Remove the referenced elements first.

Setting parameters:

<File>   string
                      Element name, as defined with the `...:CREate` or `...:NAME` command.

<Username>   string
                      Required if the repository is password protected

<Passwd>   string
                      Required if the repository is password protected

Example: See Example"Working with repositories" on page 721

Usage: Setting only

Manual operation: See "Info" on page 107

SCENario:CPDW:GROup:ADD
SCENario:CEMit:GROup:ADD
SCENario:DF:GROup:ADD
SCENario:LOCalized:GROup:ADD
ASSignment:GENerator:PATH:EMITter:ADD
ASSignment:GENerator:PATH:ANTenna:ADD
EMITter:MODE:ADD
EMITter:MODE:BEAM:ADD
PLATform:EMITter:ADD
PLATform:EMITter:BLANkranges:ADD
RECeiver:ANTenna:ADD
PULSe:MOP:AMSTep:ADD
PULSe:MOP:FMSTep:ADD
PULSe:MOP:PCHirp:ADD
PULSe:MOP:PIECewise:ADD
PULSe:MOP:CCHirp:ADD
PULSe:MOP:PLISt:ADD
SCENario:LOCalized:ADD
SCENario:DF:ADD
SCENario:CSEQuence:ADD
SCENario:CEMit:ADD
SCENario:CEMit:MCHG:ADD
SCENario:DF:MCHG:ADD
SCENario:DF:EMITter:STATe:ADD
SCENario:LOCalized:MCHG:ADD
SCENario:LOCalized:EMITter:STATe:ADD
SCENario:DF:LOCation:PSTep:ADD
SCENario:LOCalized:LOCation:PSTep:ADD
SCENario:DF:RECeiver:MOVement:PSTep:ADD
SCENario:LOCalized:RECeiver:MOVement:PSTep:ADD
SCAN:CUSTom:ENTRy:ADD
SCENario:CPDW:ADD
DSRC:ITEM:ADD
IPM:LIST:ITEM:ADD
PULSe:ENVelope:DATA:ITEM:ADD
SEQuence:ITEM:IPM:ADD
SEQuence:ITEM:ADD

Appends new item.

Example: See Example"Handling items" on page 581

Usage: Event

Manual operation: See "New, Insert, Append, Remove, Clear Items, Undo/Redo" on page 203

PLATform:EMITter:BLANkranges:COUNt?
SCENario:CPDW:GROup:COUNt?
SCENario:CEMit:GROup:COUNt?
SCENario:DF:GROup:COUNt?
SCENario:LOCalized:GROup:COUNt?
INSTrument:COUNt?
EMITter:MODE:BEAM:COUNt?
EMITter:MODE:COUNt?
PULSe:ENVelope:DATA:ITEM:COUNt?
PULSe:MOP:AMSTep:COUNt?
PULSe:MOP:FMSTep:COUNt?
PULSe:MOP:PCHirp:COUNt?
PULSe:MOP:PLISt:COUNt?
PULSe:MOP:CCHirp:COUNt?
PULSe:MOP:PIECewise:COUNt?
IPM:LIST:ITEM:COUNt?
SCENario:CEMit:MCHG:COUNt?
SCENario:DF:MCHG:COUNt?
SCENario:LOCalized:MCHG:COUNt?
SCENario:LOCalized:EMITter:STATe:COUNt?
SCENario:DF:EMITter:STATe:COUNt?
SCENario:DF:LOCation:PSTep:COUNt?
SCENario:LOCalized:LOCation:PSTep:COUNt?
SCAN:CUSTom:ENTRy:COUNt?
SETup:COUNt?
SEQuence:ITEM:IPM:COUNt?
SEQuence:ITEM:COUNt?

Queries the number of existing items.

Return values:

<Count>   integer

                            *RST: 0

Example: See Example"Handling items" on page 581

Usage: Query only

Manual operation: See "Select, No." on page 203

PLATform:EMITter:BLANkranges:SELect <Select>
PLATform:EMITter:SELect <Select>
PLATform:SELect <Select>
SCENario:CPDW:GROup:SELect <Select>
SCENario:CEMit:GROup:SELect <Select>
SCENario:DF:GROup:SELect <Select>
SCENario:LOCalized:GROup:SELect <Select>
SCENario:LOCalized:SELect <Select>
SCENario:DF:SELect <Select>
SCENario:CSEQuence:SELect <Select>
SCENario:CEMit:SELect <Select>

EMITter:MODE:BEAM:SELect <Select>
EMITter:MODE:SELect <Select>
RECeiver:ANTenna:SELect <Select>
PULSe:ENVelope:DATA:ITEM:SELect <Select>
PULSe:MOP:AMSTep:SELect <Select>
PULSe:MOP:FMSTep:SELect <Select>
PULSe:MOP:PCHirp:SELect <Select>
PULSe:MOP:PLISt:SELect <Select>
PULSe:MOP:CCHirp:SELect <Select>
PULSe:MOP:PIECewise:SELect <Select>
SCAN:CUSTom:ENTRy:SELect <Select>
DSRC:ITEM:SELect <Select>
IPM:LIST:ITEM:SELect <Select>
SCENario:CEMit:MCHG:SELect <Select>
SCENario:DF:MCHG:SELect <Select>
SCENario:DF:LOCation:PSTep:SELect <Select>
SCENario:DF:EMITter:STATe:SELect <Select>
SCENario:LOCalized:LOCation:PSTep:SELect <Select>
SCENario:LOCalized:EMITter:STATe:SELect <Select>
SCENario:LOCalized:MCHG:SELect <Select>
SCENario:CPDW:SELect <Select>
SETup:SELect <Select>
SEQuence:ITEM:IPM:SELect <Select>
SEQuence:ITEM:SELect <Select>

Selects the item to which the subsequent commands apply.

Parameters:

<Select>	float
	Item number within the range 1 to ...:COUNt.
	For example, SEQuence:ITEM:COUNt?.
	Range: 1 to 4096
Example:	See Example"Handling items" on page 581
Manual operation:	See "Select, No." on page 203

INSTrument:SELect <Select>
CPANel:USED:SELect <Select>
CPANel:UNUSed:SELECT <Select>
CPANel:USED:PATH:SELect <Select>
CPANel:UNUSed:PATH:SELect <Select>
ASSignment:ANTennas:SELect <Select>
ASSignment:EMITters:SELect <Select>
ASSignment:GENerator:SELect <Select>
ASSignment:GENerator:PATH:SELect <Select>
ASSignment:GENerator:PATH:EMITter:SELect <Select>
ASSignment:GENerator:PATH:ANTenna:SELect <Select>

Selects the element to which the subsequent commands apply.

Parameters:

<Select>   string

Available element as queried with the corresponding `...:LIST` command.

For example, `ASSignment:GENerator:PATH:ANTenna: LIST?`

Example: See Example"Receiver signals to generator mapping in direction finding scenario" on page 638.

Manual operation: See "Add, Remove, Clear" on page 484

SCAN:CUSTom:ENTRy:INSert <Insert>
SCENario:DF:EMITter:STATe:INSert <Insert>
SCENario:LOCalized:EMITter:STATe:INSert <Insert>
PULSe:MOP:AMSTep:INSert <Insert>
PULSe:MOP:FMSTep:INSert <Insert>
PULSe:MOP:PCHirp:INSert <Insert>
PULSe:MOP:CCHirp:INSert <Insert>
PULSe:MOP:PIECewise:INSert <Insert>
PULSe:MOP:PLISt:INSert <Insert>

Inserts a new item before the selected one.

Setting parameters:

<Insert>   float

*RST: 0

Usage: Setting only

Manual operation: See "Custom Phase" on page 178

PLATform:EMITter:BLANkranges:DELete <Delete>
PLATform:EMITter:DELete <Delete>
SCENario:CPDW:GROup:DELete <Delete>
SCENario:CEMit:GROup:DELete <Delete>
SCENario:DF:GROup:DELete <Delete>
SCENario:LOCalized:GROup:DELete <Delete>
INSTrument:DELete <Delete>
ASSignment:GENerator:PATH:EMITter:DELete
ASSignment:GENerator:PATH:ANTenna:DELete
SCAN:CUSTom:ENTRy:DELete <Delete>
DSRC:ITEM:DELete <Delete>
EMITter:MODE:BEAM:DELete <Delete>
EMITter:MODE:DELete <Delete>
RECeiver:ANTenna:DELete
IPM:LIST:ITEM:DELete <Delete>
PULSe:ENVelope:DATA:ITEM:DELete <Delete>
PULSe:MOP:AMSTep:DELete <Delete>
PULSe:MOP:FMSTep:DELete <Delete>

PULSe:MOP:PCHirp:DELete \<Delete\>
PULSe:MOP:CCHirp:DELete \<Delete\>
PULSe:MOP:PLISt:DELete \<Delete\>
PULSe:MOP:PIECewise:DELete \<Delete\>
SCENario:LOCalized:DELete \<Delete\>
SCENario:CEMit:MCHG:DELete \<Delete\>
SCENario:DF:MCHG:DELete \<Delete\>
SCENario:LOCalized:MCHG:DELete \<Delete\>
SCENario:LOCalized:EMITter:STATe:DELete \<Delete\>
SCENario:DF:EMITter:STATe:DELete \<Delete\>
SCENario:DF:DELete \<Delete\>
SCENario:CSEQuence:DELete \<Delete\>
SCENario:CEMit:DELete \<Delete\>
SCENario:DF:LOCation:PSTep:DELete \<Delete\>
SCENario:LOCalized:LOCation:PSTep:DELete \<Delete\>
SCENario:DF:RECeiver:MOVement:PSTep:DELete \<Delete\>
SCENario:LOCalized:RECeiver:MOVement:PSTep:DELete \<Delete\>
SCENario:CPDW:DELete \<Delete\>
SEQuence:ITEM:IPM:DELete \<Delete\>
SEQuence:ITEM:DELete \<Delete\>

Deletes the particular item.

Setting parameters:

\<Delete\>	float
	*RST: 0

Example:	See Example"Handling items" on page 581
Usage:	Setting only
Manual operation:	See "New, Insert, Append, Remove, Clear Items, Undo/Redo" on page 203

PLATform:EMITter:BLANkranges:CLEar
PLATform:EMITter:CLEar
SCENario:CPDW:GROup:CLEar
SCENario:CEMit:GROup:CLEar
SCENario:DF:GROup:CLEar
SCENario:LOCalized:GROup:CLEar
SCENario:GENerator:CLEar
INSTrument:CLEar
ANTenna:MODel:USER:CLEar
DSRC:CLEar
SCAN:CUSTom:ENTRy:CLEar
EMITter:MODE:ANTenna:CLEar
EMITter:MODE:BEAM:CLEar
EMITter:MODE:CLEar
EMITter:MODE:SCAN:CLEar
RECeiver:ANTenna:CLEar
IPM:LIST:CLEar

PULSe:ENVelope:DATA:CLEar
PULSe:MOP:AMSTep:CLEar
PULSe:MOP:FMSTep:CLEar
PULSe:MOP:PCHirp:CLEar
PULSe:MOP:PLISt:CLEar
PULSe:MOP:CCHirp:CLEar
PULSe:MOP:PIECewise:CLEar
ASSignment:GENerator:PATH:EMITter:CLEar
ASSignment:GENerator:PATH:ANTenna:CLEar
SCENario:EMITter:CLEar
SCENario:CSEQuence:CLEar
SCENario:CEMit:CLEar
SCENario:CEMit:MCHG:CLEar
SCENario:LOCalized:CLEar
SCENario:DF:MCHG:CLEar
SCENario:DF:EMITter:STATe:CLEar
SCENario:LOCalized:EMITter:STATe:CLEar
SCENario:CPDW:CLEar
SCENario:DF:CLEar
SEQuence:ITEM:CLEar

Deletes all items from the list or the table.

Example:	See Example"Handling items" on page 581
Usage:	Event
Manual operation:	See "New, Insert, Append, Remove, Clear Items, Undo/Redo" on page 203

25.5 Program settings

PROGram:ADJustments:ENABle <Enable>

Enables using the path loss compensation function.

Parameters:

<Enable>   ON | OFF | 1 | 0

Manual operation: See "Adjustments" on page 552

PROGram:CLASs:ENABle <Enable>

Enables whether the workspace classification level appears in the lower window (restart required).

Parameters:

<Enable>   ON | OFF | 1 | 0

Manual operation: See "Repository Classification" on page 552

PROGram:COMMent:ENABle <Enable>

Add timestamp as comment when creating entries.

Parameters:

<Enable>   ON | OFF | 1 | 0

Manual operation: See "Program Settings" on page 552

PROGram:GPU:ENABle <Enable>

Enables the GPU (Graphics Processing Unit) to be used for antenna pattern calculations.

Using the GPU accelerates the calculation.

Requires a restart.

Parameters:

<Enable>   ON | OFF | 1 | 0

Manual operation: See "GPU Support" on page 564

PROGram:PATH:INSTall?

Queries the storage location for repository files.

Return values:

<Install>   string

Usage: Query only

Manual operation: See "Repository Databases" on page 560

PROGram:PATH:VOLatile <Volatile>

Sets the directory that holds volatile data.

Parameters:
<Volatile>   string

Manual operation: See "Volatile Storage" on page 560

PROGram:PATH:CALCulated <Calculated>

Sets the directory that holds the calculated waveforms.

Parameters:
<Calculated>   string

Manual operation: See "Calculated Data" on page 561

PROGram:PATH:REPort <Report>

Sets the directory that holds generated reports.

Parameters:
<Report>   string

Manual operation: See "Reports" on page 561

PROGram:RAMBuff:SIZE <Size>

Sets the ARB RAM buffer size.

Parameters:
<Size>   16M | 64M | 128M | 256M | 512M | 1G

Manual operation: See "ARB RAM Buffer Size" on page 552

PROGram:SCENario:XTRG:ENABle <Enable>

Enables using a separate trigger for scenario start, see `SCENario:TRIGger`.

Parameters:
<Enable>   ON | OFF | 1 | 0

Manual operation: See "Program Settings" on page 552

PROGram:SCPI:TIMeout <Timeout>

Sets the SCPI timeout for instrument control. The maximum value is 100 s.

The value defines the time span that the pulse sequencer software waits for a response after a SCPI command is sent to the signal generator. When the time span is elapsed, a VISA timeout exception is generated.

Parameters:
<Timeout>   float

Manual operation: See "Program Settings" on page 552

PROGram:SETTings:ACCept
PROGram:SETTings:REJect

Reject changes to program settings.

Usage: Event

Manual operation: See "Apply, Cancel" on page 553

PROGram:STARtup:LOAD:ENABle <Enable>

Sets if a scenario is opened each time the software is started up.

Parameters:
<Enable>   ON | OFF | 1 | 0

Manual operation: See "Program Settings" on page 552

PROGram:STARtup:WIZard:ENABle <Enable>

Enable this command, if you wish the wizard to open when the software starts.

Parameters:
<Enable>   ON | OFF | 1 | 0

Manual operation: See "Program Settings" on page 552

PROGram:TOOLbar:ENABle <Enable>

Parameters:
<Enable>   ON | OFF | 1 | 0

Manual operation: See "Program Settings" on page 552

PROGram:TRANsfer:FTP:BLOCksize <Blocksize>

Block size used for data transfer.

Parameters:
<Blocksize>   16K | 32K | 64K | 1M | 2M | 4M

Manual operation: See "Use the FTP data transfer for large files" on page 486

PROGram:TRANsfer:FTP:ENABle <Enable>

Enables an FTP data transfer for transfer of large files to the processing instrument.

Parameters:
<Enable>   ON | OFF | 1 | 0

Manual operation: See "Use the FTP data transfer for large files" on page 486

PROGram:TRANsfer:FTP:PASSwd <Passwd>

Parameters:
<Passwd>   string

Manual operation: See "Use the FTP data transfer for large files" on page 486

PROGram:TRANsfer:FTP:UNAMe <Uname>

Sets the user name and password of the processing instrument.

Parameters:
<Uname>   string

Manual operation: See "Use the FTP data transfer for large files" on page 486

PROGram:TUTorials:SHOW:ENABle <Enable>

This setting re-enables all tutorials.

Tutorials are shown upon opening certain dialogs for the first time (e.g. 2D Map). When the tutorial has been viewed, it is then disabled.

Parameters:
<Enable>   ON | OFF | 1 | 0

Example: PROG:TUT:SHOW:ENAB ON

Manual operation: See "Program Settings" on page 552

25.6 Antenna pattern commands

Example: Configuring antenna patterns

```
SCPI
ANTenna:CREate "My_PlanarAntenna"
ANTenna:COMMent "planar phased antenna with parabolic aperture distribution"
ANTenna:MODel:TYPE ARRay
ANTenna:MODel:FREQuency 1e+10
ANTenna:MODel:BANDwidth 1e+9
ANTenna:MODel:POLarization HORizontal
ANTenna:MODel:ARRay:DISTribution PARabolic
ANTenna:MODel:ARRay:NX 12
ANTenna:MODel:ARRay:NZ 12
ANTenna:MODel:ARRay:XDIStance 0.6
ANTenna:MODel:ARRay:ZDIStance 0.3
ANTenna:MODel:ARRay:ELEMent:COSine 0
ANTenna:MODel:ARRay:PEDestal 0.1

ANTenna:CREate "Test"
ANTenna:MODel:TYPE HORN
ANTenna:MODel:ROTation:Z 0
ANTenna:MODel:ROTation:X 0
ANTenna:MODel:FREQuency 1e+10
ANTenna:MODel:POLarization HORizontal
ANTenna:MODel:HORN:LX 0.076
ANTenna:MODel:HORN:LZ 0.05
ANTenna:MODel:HORN:RESolution 5.E-01

ANTenna:CREate "My_PencilBeam"
ANTenna:MODel:TYPE SINC
ANTenna:MODel:ROTation:Z 0
ANTenna:MODel:ROTation:X 0
ANTenna:MODel:POLarization HORizontal
ANTenna:MODel:SINC:HPBW 3

ANTenna:CREate "Test CSC"
ANTenna:MODel:TYPE COSecant
ANTenna:MODel:POLarization HORizontal
ANTenna:MODel:COSecant:HPBW 2
ANTenna:MODel:COSecant:T1 5
ANTenna:MODel:COSecant:T2 70

ANTenna:CREate "User"
ANTenna:MODel:TYPE USER
ANTenna:MODel:USER:LOAD "C:/_PS_files/antenna.ant_pat"
ANTenna:MODel:ROTation:Z -90
ANTenna:MODel:ROTation:X 0
ANTenna:MODel:POLarization HORizontal
```

```
ANTenna:MODel:USER:CLEAr

ANTenna:CREate "Custom"
ANTenna:MODel:TYPE CUSTom
ANTenna:MODel:POLarization HORizontal
ANTenna:MODel:CUSTom:HPBW:XY 10
ANTenna:MODel:CUSTom:HPBW:YZ 3
ANTenna:MODel:CUSTom:SLSTart 30
ANTenna:MODel:CUSTom:SLRolloff 10
ANTenna:MODel:CUSTom:SLSCale 0.5
ANTenna:MODel:BACKlobe:ENABle 1
ANTenna:MODel:BACKlobe:TYPE MIRRor
ANTenna:MODel:BACKlobe:ATTenuation 40

ANTenna:NAME "Custom Phased Array"
ANTenna:MODel:TYPE CARRAY
ANTenna:MODel:POLarization VERTical
ANTenna:MODel:CARRay:GEOMetry RECTANGULAR
ANTenna:MODel:CARRay:RECTangular:NX 25
ANTenna:MODel:CARRay:RECTangular:NZ 25
ANTenna:MODel:CARRay:RECTangular:XDIStance 0.015
ANTenna:MODel:CARRay:RECTangular:ZDIStance 0.015
ANTenna:MODel:CARRay:ELEMent:COSine 1
ANTenna:MODel:CARRay:RECTangular:LATTice RECTANGULAR
ANTenna:MODel:CARRay:DISTribution COSN
ANTenna:MODel:CARRay:COSN 4
```

ANTenna:MODel:TYPE <Type>

Sets the antenna pattern.

Parameters:

<Type>	DIPole | PARabolic | GAUSsian | SINC | HORN | COSecant | ARRay | USER | CUSTom | CARRay | CARDoid | PLUGin
Example:	See Example"Configuring antenna patterns" on page 601.
Manual operation:	See "Type" on page 262

ANTenna:MODel:POLarization <Polarization>

Sets the antenna polarization.

Parameters:

<Polarization>	VERTical | HORizontal | CRIGht | CLEFt | SRIGht | SLEFt
Example:	Example"Configuring antenna patterns" on page 601
Manual operation:	See "Polarization" on page 263

ANTenna:MODel:CARRay:COSN <Cosn>
ANTenna:MODel:ARRay:COSN <Cosn>

Sets the value of the coefficient N in the $\cos^N$ distribution.

Parameters:

<Cosn>	float
	Range: 2 to 10
Example:	Example"Configuring antenna patterns" on page 601
Manual operation:	See "Planar Phased Array Antenna Settings" on page 269

ANTenna:MODel:ARRay:COSN:Z <Z>
ANTenna:MODel:ARRay:COSN:X <X>

Requires `ANTenna:MODel:ARRay:DISTribution:TYPE 1`.

Sets the individual value of the coefficient N in the $\cos^N$ distribution for X and Z direction.

Parameters:

<X>   float

                               Range: 2 to 10

Manual operation: See "Planar Phased Array Antenna Settings" on page 269

ANTenna:MODel:CARRay:COSN:Z <Z>
ANTenna:MODel:CARRay:COSN:X <X>

Requires `ANTenna:MODel:CARRay:DISTribution:TYPE 1`.

Sets the individual value of the coefficient N in the $\cos^N$ distribution for X and Z direction.

Parameters:

<X>   float

                               Range: 2 to 10

Manual operation: See "Custom Phased Array Settings" on page 272

ANTenna:MODel:CARRay:DISTribution:TYPE <Type>
ANTenna:MODel:ARRay:DISTribution:TYPE <Type>

Enables using the individual distribution function for X and Z direction.

Parameters:

<Type>   ON | OFF | 1 | 0

Manual operation: See "Planar Phased Array Antenna Settings" on page 269

ANTenna:MODel:ARRay:DISTribution <Distribution>

Sets the aperture distribution of the Planar Phased Array antenna.

Parameters:

<Distribution>   UNIForm | PARabolic | COSine | CSQuared | COSN |
                               TRIangular | HAMMing | HANN

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Planar Phased Array Antenna Settings" on page 269

ANTenna:MODel:CARRay:DISTribution <Distribution>

Sets the aperture distribution of the Custom Phased Array antenna.

Parameters:

<Distribution>   UNIForm | PARabolic | COSine | CSQuared | COSN |
                               TRIangular | HAMMing | HANN

Example: See Example"Configuring antenna patterns" on page 601

Manual operation: See "Custom Phased Array Settings" on page 272

ANTenna:MODel:ARRay:DISTribution:Z <Z>
ANTenna:MODel:ARRay:DISTribution:X <X>

Requires `ANTenna:MODel:ARRay:DISTribution:TYPE 1`.

Sets the individual aperture distribution function for X and Z direction.

Parameters:

<X>   UNIForm | PARabolic | COSine | CSQuared | COSN |
                TRIangular | HAMMing | HANN

Manual operation: See "Planar Phased Array Antenna Settings" on page 269

ANTenna:MODel:CARRay:DISTribution:Z <Z>
ANTenna:MODel:CARRay:DISTribution:X <X>

Requires `ANTenna:MODel:ARRay:DISTribution:TYPE 1`.

Sets the individual aperture distribution function for X and Z direction.

Parameters:

<X>   UNIForm | PARabolic | COSine | CSQuared | COSN |
                TRIangular | HAMMing | HANN

Manual operation: See "Custom Phased Array Settings" on page 272

ANTenna:MODel:ARRay:ELEMent:COSine <Cosine>
ANTenna:MODel:CARRay:ELEMent:COSine <Cosine>

Sets the characteristic of individual antenna elements.

Parameters:

<Cosine>   ON | OFF | 1 | 0

                0|OFF
                Omnidirectional characteristic

                1|ON
                Cosine characteristic

Example: See Example"Configuring antenna patterns" on page 601

Manual operation: See "Custom Phased Array Settings" on page 272

ANTenna:MODel:CARRay:LINear:N <N>
ANTenna:MODel:CARRay:HEXagonal:N <N>
ANTenna:MODel:CARRay:RECTangular:NX <Nx>
ANTenna:MODel:CARRay:RECTangular:NZ <Nz>
ANTenna:MODel:ARRay:NX <Nx>
ANTenna:MODel:ARRay:NZ <Nz>

Sets the number of elements of the antenna array.

Parameters:

<Nz>	float
	Range: 2 to 1000 (planar phased array; linear phase array), 100 (rectangular phase array), 50 (hexagonal phase array)

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Planar Phased Array Antenna Settings" on page 269

ANTenna:MODel:CARRay:PEDestal <Pedestal>
ANTenna:MODel:ARRay:PEDestal <Pedestal>

Sets the pedestal level of the antenna array.

Parameters:

<Pedestal>	float
	Range: 0 to 1

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Planar Phased Array Antenna Settings" on page 269

ANTenna:MODel:ARRay:PEDestal:Z <Z>
ANTenna:MODel:ARRay:PEDestal:X <X>

Requires `ANTenna:MODel:ARRay:DISTribution:TYPE 1`.

Sets the individual pedestal level of the antenna array in X or Z direction.

Parameters:

<X>	float
	Range: 0 to 1

Manual operation: See "Planar Phased Array Antenna Settings" on page 269

ANTenna:MODel:CARRay:PEDestal:Z <Z>
ANTenna:MODel:CARRay:PEDestal:X <X>

Requires `ANTenna:MODel:ARRay:DISTribution:TYPE 1`.

Sets the individual pedestal level of the antenna array in X or Z direction.

Parameters:

<X>	float
	Range: 0 to 1

Manual operation: See "Custom Phased Array Settings" on page 272

ANTenna:MODel:CARRay:CIRCular:DISTance <Distance>
ANTenna:MODel:CARRay:HEXagonal:DISTance <Distance>
ANTenna:MODel:CARRay:LINear:DISTance <Distance>

ANTenna:MODel:CARRay:RECTangular:XDIStance \<Xdistance\>
ANTenna:MODel:CARRay:RECTangular:ZDIStance \<Zdistance\>
ANTenna:MODel:ARRay:XDIStance \<Xdistance\>
ANTenna:MODel:ARRay:ZDIStance \<Zdistance\>

Sets the spacing between the elements of the array antenna.

Parameters:

\<Zdistance\>	float
	Range: 0.0001 to 1

Example:	Example"Configuring antenna patterns" on page 601
Manual operation:	See "Planar Phased Array Antenna Settings" on page 269

ANTenna:MODel:BACKlobe:ENABle \<Enable\>

Enables the simulation of a back lobe.

Parameters:

\<Enable\>	ON	OFF	1	0

Example:	Example"Configuring antenna patterns" on page 601
Manual operation:	See "Simulate Back Lobe, Attenuation, Type" on page 264

ANTenna:MODel:BACKlobe:TYPE \<Type\>

Sets the shape of the back lobe pattern.

Parameters:

\<Type\>	MIRRor	OMNidirect

Example:	Example"Configuring antenna patterns" on page 601
Manual operation:	See "Simulate Back Lobe, Attenuation, Type" on page 264

ANTenna:MODel:BACKlobe:ATTenuation \<Attenuation\>

Sets the attenuation of the back lobe.

Parameters:

\<Attenuation\>	float
	Range: 0 to 100

Example:	Example"Configuring antenna patterns" on page 601
Manual operation:	See "Simulate Back Lobe, Attenuation, Type" on page 264

ANTenna:MODel:COSecant:T1 \<T1\>
ANTenna:MODel:COSecant:T2 \<T2\>

Sets the Theta parameters.

Parameters:

<T2>   float

                         Range: 1 to 90

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Cosecant Squared Antenna Settings" on page 268

ANTenna:MODel:CARRay:CIRCular:LATTice <Lattice>
ANTenna:MODel:CARRay:RECTangular:LATTice <Lattice>

Sets the lattice.

Parameters:

<Lattice>   RECTangular | TRIangular

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Custom Phased Array Settings" on page 272

ANTenna:MODel:CARRay:CIRCular:RADius <Radius>

Set the radius of the circular phased array antenna.

Parameters:

<Radius>   float

                         Range: 1 to 50

Manual operation: See "Custom Phased Array Settings" on page 272

ANTenna:MODel:CARRay:GEOMetry <Geometry>

Sets the geometry of the custom phased array antenna.

Parameters:

<Geometry>   RECTangular | LINear | HEXagonal | CIRCular

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Custom Phased Array Settings" on page 272

ANTenna:MODel:CARRay:ELEMent <Element>

Set the state of one or more antenna elements.

Antenna elements are indicated by their subsequent number, as it is displayed in the "Antenna Geometry" dialog (Figure 9-3).

Parameters:

<Element>	numeric list
	A comma-separated list of numbers or of range of numbers enclosed in brackets. In the case of a range, the end points are inclusive.
	The listed numbers indicate the *active* antenna elements; if a number is omitted, the antenna element is disabled.

Example:

```
ANTenna:MODel:CARRay:GEOMetry LINEAR
ANTenna:MODel:CARRay:LINear:N 7
ANTenna:MODel:CARRay:ELEMent? (1,3:7)
// element 1 and 3 through 7 are active
ANTenna:MODel:CARRay:ELEMent (1,3,5,7)
// antenna elements 1, 3, 5 and 7 are active
// antenna elements 2, 4 and 6 are deactivated
```

Manual operation: See "Custom Phased Array Settings" on page 272

ANTenna:MODel:CUSTom:HPBW:XY <Xy>
ANTenna:MODel:CUSTom:HPBW:YZ <Yz>

Sets the required HPBW of the custom antenna.

Parameters:

<Yz>	float
	Range: 0.1 to 45

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Custom Antenna Settings" on page 271

ANTenna:MODel:CUSTom:SLSTart <Slstart>

Sets the power level of the first pairs of side lobes.

Parameters:

<Slstart>	float
	Range: 1 to 90

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Custom Antenna Settings" on page 271

ANTenna:MODel:CUSTom:SLRolloff <Slrolloff>

Sets the factor used to calculate the HPBW of the side lobes.

Parameters:

<Slrolloff>	float
	Range: 1 to 45

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Custom Antenna Settings" on page 271

ANTenna:MODel:CUSTom:SLSCale <Slscale>

Sets the step size to calculate the power level of the side lobes.

Parameters:
<Slscale>   float

                       Range: 0.01 to 10

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Custom Antenna Settings" on page 271

ANTenna:MODel:BANDwidth <Bandwidth>

Sets the antenna bandwidth.

Parameters:
<Bandwidth>   float

                       Range: 1e+06 to 1e+11

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Frequency, Bandwidth" on page 263

ANTenna:MODel:FREQuency <Frequency>

Sets the frequency.

Parameters:
<Frequency>   float

                       Range: 1e+06 to 1e+11
                       Default unit: Hz

Example: See Example"Configuring antenna patterns" on page 601

Manual operation: See "Frequency, Bandwidth" on page 263

ANTenna:MODel:COSecant:HPBW <Hpbw>

Sets the Half-Power Beam Width Cosecant Squared antenna.

Parameters:
<Hpbw>   float

                       Range: 0.01 to 30

Example: See Example"Configuring antenna patterns" on page 601.

Manual operation: See "Cosecant Squared Antenna Settings" on page 268

ANTenna:MODel:GAUSsian:HPBW:AZIMuth \<Azimuth\>
ANTenna:MODel:GAUSsian:HPBW:ELEVation \<Elevation\>
ANTenna:MODel:SINC:HPBW:AZIMuth \<Azimuth\>
ANTenna:MODel:SINC:HPBW:ELEVation \<Elevation\>

Sets the Half-Power Beam Width in azimuth and elevation direction for the Gaussian and Sin(x)/x antennas.

Parameters:

\<Elevation\>   float

Range: 0.1 to 45
Default unit: degree

Example: See Example"Configuring antenna patterns" on page 601.

Manual operation: See "Sin(x)/x Antenna Settings" on page 267

ANTenna:MODel:ARRay:RESolution \<Resolution\>
ANTenna:MODel:CARRay:RESolution \<Resolution\>
ANTenna:MODel:CARDoid:RESolution \<Resolution\>
ANTenna:MODel:COSecant:RESolution \<Resolution\>
ANTenna:MODel:CUSTom:RESolution \<Resolution\>
ANTenna:MODel:DIPole:RESolution \<Resolution\>
ANTenna:MODel:GAUSsian:RESolution \<Resolution\>
ANTenna:MODel:HORN:RESolution \<Resolution\>
ANTenna:MODel:PARabolic:RESolution \<Resolution\>
ANTenna:MODel:SINC:RESolution \<Resolution\>

Sets a custom resolution for the antenna pattern simulation.

Parameters:

\<Resolution\>   float

Range: 0.1 to 1

Example: See Example"Configuring antenna patterns" on page 601.

Manual operation: See "Resolution" on page 265

ANTenna:MODel:HORN:LX \<Lx\>
ANTenna:MODel:HORN:LZ \<Lz\>

Sets the length of the rectangular sides of the Pyramidal Horn antenna.

Parameters:

\<Lz\>   float

Range: 0.01 to 100
Default unit: m

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Pyramidal Horn Antenna Settings" on page 268

ANTenna:MODel:PARabolic:DIAMeter <Diameter>

Sets the diameter of the parabolic dish antenna.

Parameters:

<Diameter>   float

        Range: 0.05 to 100
        Default unit: m

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Parabolic Antenna Settings" on page 266

ANTenna:MODel:ROTation:X <X>
ANTenna:MODel:ROTation:Z <Z>

Sets the X and Z antenna rotation.

Parameters:

<Z>   float

        Range: -180 to 180
        Default unit: degree

Example: Example"Configuring antenna patterns" on page 601

Manual operation: See "Z-Rotation, X-Rotation" on page 263

ANTenna:MODel:CARDoid:EXPonent <Exponent>

Use values greater than 1 to narrow the antenna beam.

Parameters:

<Exponent>   float

        Range: 1 to 20

Manual operation: See "Cardioid Antenna Settings" on page 277

ANTenna:MODel:USER:LOAD <Load>

Loads a custom antenna pattern file.

Setting parameters:

<Load>   string

Example: Example"Configuring antenna patterns" on page 601

Usage: Setting only

Manual operation: See "Load/Purge" on page 278

ANTenna:MODel:USER:CSV:FORMat
   `<start_row>,<col_sep>,<decimal>,<col>{,<col>}`

Defines how the data in the selected `*.csv` file is interpreted.

The settings in this command are not permanent. The command affects only the currently selected antenna.

For description of the `*.csv` file format, see Chapter A.2, "Antenna pattern file formats", on page 843.

Parameters:

<col_sep>	COMMA | SEMICOLON | WHITESPACE
	Column separator
<decimal>	POINT | COMMA
	Decimal delimiter
<col>	THETA | PHI | GAIN | ETHRE | ETHIM | EPHRE | EPHIM
	Data content of the column: Theta, Phi, Gain, Re(E_Theta), Im(E_Theta), Re(E_Phi), Im(E_Phi)
	At least one out of maximum 6 columns must exist

Setting parameters:

<start_row>	First row with data
	Range: 1 to max

Example:	`ANTenna:MODel:TYPE USER`
	`ANTenna:MODel:USER:CSV:FORMat "2","COMMA","POINT","PHI","THETA","GAIN"`
	`ANTenna:MODel:USER:LOAD "D:/PS/ansys.csv"`
Usage:	Setting only
Manual operation:	See "Import wizard" on page 278

PLOT:POLar:TYPE <Type>

Sets the coordinates of the 2D antenna pattern diagram.

Setting parameters:

<Type>	POLar | CARTesian

Example:	`PLOT:POLar:TYPE CARTesian`
	`PLOT:POLar:CUT XY`
	`PLOT:POLar:LOG:MIN -90`
Usage:	Setting only
Manual operation:	See "2D" on page 265

PLOT:POLar:CUT <Cut>

Sets the diagram cut.

Setting parameters:

<Cut>   XY | YZ

Example: See PLOT:POLar:TYPE on page 614

Usage: Setting only

Manual operation: See "2D" on page 265

PLOT:POLar:LOG:MIN <Min>

Sets the minimum value displayed on the y axis.

Setting parameters:

<Min>   float

Example: See PLOT:POLar:TYPE on page 614

Usage: Setting only

Manual operation: See "2D" on page 265

25.7 Antenna scan commands

Example: Defining antenna scans

```
SCPI
SCAN:CREate "My_RasterScan"
SCAN:TYPE RASTer
SCAN:RASTer:WIDTh 70
SCAN:RASTer:BARWidth 3
SCAN:RASTer:RATE 15
SCAN:RASTer:BARS 10
SCAN:RASTer:DIRection HORizontal
SCAN:RASTer:RETRace 0.001
SCAN:RASTer:UNIDirection 1
SCAN:RASTer:FLYBack 0.001
SCAN:RASTer:PALMer 1
SCAN:RASTer:PRATe 5
SCAN:RASTer:PSQuint 1.5

SCAN:CREate "Test Antenna Scan"
SCAN:TYPE HELical
SCAN:HELical:RPM 45
SCAN:HELical:TURNs 8
SCAN:HELical:ELEVation:STEP 3
SCAN:HELical:RETRace 1
SCAN:HELical:ROTation CW

// circular scan
SCAN:CIRCular:MODE RPM
SCAN:CIRCular:RPM 6.E+01
SCAN:CIRCular:PERiod?
// 1.E+00
```

SCAN:CIRCular:ROTation <Rotation>
SCAN:CONical:ROTation <Rotation>
SCAN:HELical:ROTation <Rotation>
SCAN:LSW:ROTation <Rotation>
SCAN:SIN:ROTation <Rotation>
SCAN:SPIRal:ROTation <Rotation>

Sets the rotation direction of the antenna.

Parameters:
<Rotation>   CW | CCW

Example: See Example"Defining antenna scans" on page 616

Manual operation: See "Spiral Scan" on page 287

SCAN:CIRCular:MODE <Mode>

Sets if the scan turning speed is set as a scans rate or as a period.

Parameters:
<Mode>   RPM | SEC

                          RPM
                          Scan rate, set with the command `SCAN:CIRCular:RPM`.

                          SEC
                          Scan period, set with the command `SCAN:CIRCular:PERiod`.

Example: See Chapter 25.7, "Antenna scan commands", on page 616.

Manual operation: See "Circular Scan" on page 282

SCAN:CIRCular:PERiod <Period>

Sets the time it takes for the antenna to turn once.

Parameters:
<Period>   float
                          Range: 0.006 to 6000

Example: See Chapter 25.7, "Antenna scan commands", on page 616.

Manual operation: See "Circular Scan" on page 282

SCAN:CIRCular:RPM <Rpm>
SCAN:HELical:RPM <Rpm>

Sets the rotation speed of the antenna.

Parameters:

<Rpm>　　　　　　　　　float

　　　　　　　　　　　　Range:　　　0.01　to　1000
　　　　　　　　　　　　Default unit: degree/s

Example:　　　　　　　Example"Defining antenna scans" on page 616

Manual operation:　　See "Helical Scan" on page 286

SCAN:CONical:RATE <Rate>
SCAN:SECTor:RATE <Rate>
SCAN:SIN:RATE <Rate>
SCAN:RASTer:RATE <Rate>

Sets the turning speed.

Parameters:

<Rate>　　　　　　　　　float

　　　　　　　　　　　　Range:　　　0.01　to　100000
　　　　　　　　　　　　Default unit: degree/s

Example:　　　　　　　Example"Defining antenna scans" on page 616

Manual operation:　　See "Raster Scan" on page 285

SCAN:HELical:ELEVation:STEP <Step>

Sets the step width with that the antenna changes its elevation.

Parameters:

<Step>　　　　　　　　　float

　　　　　　　　　　　　Range:　　　0.01　to　11.25
　　　　　　　　　　　　Default unit: degree

Example:　　　　　　　See Example"Defining antenna scans" on page 616

Manual operation:　　See "Helical Scan" on page 286

SCAN:HELical:TURNs <Turns>

Sets the number of turns.

Parameters:

<Turns>　　　　　　　　float

　　　　　　　　　　　　Range:　　　1　to　30

Example:　　　　　　　Example"Defining antenna scans" on page 616

Manual operation:　　See "Helical Scan" on page 286

SCAN:LSW:DIRection <Direction>

Sets the horizontal or vertical switching direction.

Parameters:

<Direction>   H | V

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Lobe Switching Scan" on page 288

SCAN:LSW:DWELl <Dwell>

Sets the speed with that the antenna switches between the lobes.

Parameters:

<Dwell>   float

                     Range: 1e-06 to 1

Example: See Example"Defining antenna scans" on page 616

Manual operation: See "Lobe Switching Scan" on page 288

SCAN:LSW:LOBes <Lobes>

Set the number of lobes.

Parameters:

<Lobes>   2 | 4

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Lobe Switching Scan" on page 288

SCAN:CONical:SQUint <Squint>
SCAN:LSW:SQUint <Squint>

Sets the offset angle of the antenna beam, that means for the conical antenna the parameter sets the radius of the scanned circle.

Parameters:

<Squint>   float

                     Range: 0.05 to 15
                     Default unit: degree

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Lobe Switching Scan" on page 288

SCAN:RASTer:BARS <Bars>

Sets the number of scanned bars (sectors).

Parameters:

<Bars>	float
	Range: 1 to 30

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Raster Scan" on page 285

SCAN:RASTer:BARTranstime <Bartranstime>

Transition time between two bars in bidirectional scan mode.

Parameters:

<Bartranstime>	float
	Range: 0 to 1
	Increment: 1 ms
	Default unit: seconds

SCAN:RASTer:BARWidth <Barwidth>

Sets the distance between two consecutive scanned bars (sectors).

Parameters:

<Barwidth>	float
	Range: 0.1 to 9
	Default unit: m

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Raster Scan" on page 285

SCAN:RASTer:REWind <Rewind>

If enabled, the antenna scans forwards and backwards.

Parameters:

<Rewind>	ON	OFF	1	0

Example: See Example"Defining antenna scans" on page 616.

Manual operation: See "Raster Scan" on page 285

SCAN:RASTer:FLYBack <Flyback>
SCAN:SECTor:FLYBack <Flyback>

Sets the Flyback time for the antenna working in unidirectional mode.

Parameters:

<Flyback>	float
	Range: 0 to 1
	Default unit: s

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Sector Scan" on page 284

SCAN:HELical:RETRace \<Retrace>
SCAN:RASTer:RETRace \<Retrace>
SCAN:SPIRal:RETRace \<Retrace>

Sets the speed for the antenna to return to the initial orientation.

Parameters:

\<Retrace>   float

                     Range: 0 to 1

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Spiral Scan" on page 287

SCAN:SPIRal:UNIDirection \<Unidirection>
SCAN:RASTer:UNIDirection \<Unidirection>
SCAN:SIN:UNIDirection \<Unidirection>
SCAN:SECTor:UNIDirection \<Unidirection>

Enables a unidirectional scan mode.

Parameters:

\<Unidirection>   ON | OFF | 1 | 0

Example: See Example"Defining antenna scans" on page 616

Manual operation: See "Sector Scan" on page 284

SCAN:RASTer:WIDTh \<Width>
SCAN:SECTor:WIDTh \<Width>

Sets the width of the sector to be scanned.

Parameters:

\<Width>   float

                     Range: 0.1 to 360
                     Default unit: degree

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Sector Scan" on page 284

SCAN:RASTer:DIRection \<Direction>

Sets the scanning direction.

Parameters:

\<Direction>   HORizontal | VERTical

Manual operation: See "Raster Scan" on page 285

see Example"Defining antenna scans" on page 616

SCAN:SPIRal:ROUNds <Rounds>

Sets the number of rounds the antenna performs.

Parameters:

<Rounds>   float

                  Range: 0.1 to 15

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Spiral Scan" on page 287

SCAN:SPIRal:RTIMe <Rtime>

Sets the turning speed of the antenna.

Parameters:

<Rtime>   float

                  Range: 0.01 to 100
                  Default unit: degree/s

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Spiral Scan" on page 287

SCAN:SPIRal:STEP <Step>

Determines the step size to increase the scan radius.

Parameters:

<Step>   float

                  Range: 1 to 11.25
                  Default unit: degree

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Spiral Scan" on page 287

SCAN:TYPE <Type>

Sets the scan type.

Parameters:

<Type>   CIRCular | SECTor | RASTer | CONical | HELical | SPIRal |
                 LSW | SIN | CUSTom | LISSajous

Example: Example"Defining antenna scans" on page 616

Manual operation: See "Scan Type" on page 281

SCAN:CIRCular:NODDing <Nodding>
SCAN:SECTor:NODDing <Nodding>

Enables superimposing a horizontal nodding on the scan.

Parameters:

<Nodding>   ON | OFF | 1 | 0

Example: SCAN:SECTor:NODDing 1

                        SCAN:SECTor:NRATe 500

                        SCAN:SECTor:NELevation 15

Manual operation: See "Sector Scan" on page 284

SCAN:CIRCular:NELevation <Nelevation>
SCAN:SECTor:NELevation <Nelevation>

Sets the elevation angle.

Parameters:

<Nelevation>   float

                        Range: 0.01 to 90

Example: See SCAN:SECTor:NODDing on page 624

Manual operation: See "Sector Scan" on page 284

SCAN:CIRCular:NRATe <Nrate>
SCAN:SECTor:NRATe <Nrate>

Sets the elevation rate.

Parameters:

<Nrate>   float

                        Range: 0.01 to 2000

Example: See SCAN:SECTor:NODDing on page 624

Manual operation: See "Sector Scan" on page 284

SCAN:CIRCular:PALMer <Palmer>
SCAN:RASTer:PALMer <Palmer>
SCAN:SPIRal:PALMer <Palmer>
SCAN:SECTor:PALMer <Palmer>

Enables superimposing a conical scan on the current scan.

Parameters:

<Palmer>   ON | OFF | 1 | 0

Manual operation: See "Sector Scan" on page 284

see Example"Defining antenna scans" on page 616

SCAN:CIRCular:PRATe <Prate>
SCAN:RASTer:PRATe <Prate>
SCAN:SPIRal:PRATe <Prate>
SCAN:SECTor:PRATe <Prate>

Sets the scan rate.

Parameters:

<Prate>   float

                           Range: 0.1 to 1000

Manual operation: See "Sector Scan" on page 284

see Example"Defining antenna scans" on page 616

SCAN:CIRCular:PSQuint <Psquint>
SCAN:RASTer:PSQuint <Psquint>
SCAN:SPIRal:PSQuint <Psquint>
SCAN:SECTor:PSQuint <Psquint>

Sets the squint angle.

Parameters:

<Psquint>   float

                           Range: 0.05 to 45

Manual operation: See "Sector Scan" on page 284

see Example"Defining antenna scans" on page 616

SCAN:CUSTom:IMPort:FILE <File>

Sets the file to import.

Parameters:

<File>   string

Example: SCAN:CUSTom:IMPort:FILE "D:/PS/scan1.csv"

Manual operation: See "Custom Scan" on page 290

SCAN:CUSTom:IMPort:EXEC

Starts importings the file.

Usage: Event

Manual operation: See "Custom Scan" on page 290

SCAN:CUSTom:ENTRy:AZIMuth <Azimuth>

Sets the azimuth of the scan position.

Parameters:

<Azimuth>　　　　　　　　float

　　　　　　　　　　　　　Range:　　　-180　to　180

Example:　　　　　　　`SCAN:TYPE CUSTOM`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:ADD`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:COUNt?`
　　　　　　　　　　　　　`// 4`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:SELect 1`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:AZimuth 90`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:ELevation 25`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:DWELl 10`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:JUMPtype JUMP`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:SELect 2`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:AZimuth -120`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:ELevation 5`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:DWELl 25`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:JUMPtype MOVE`
　　　　　　　　　　　　　`SCAN:CUSTom:ENTRy:TRANstime 40`
　　　　　　　　　　　　　`...`

Manual operation:　　See "Custom Scan" on page 290

SCAN:CUSTom:ENTRy:ELEVation <Elevation>

Sets the elevation of the scan position.

Parameters:

<Elevation>　　　　　　　float

　　　　　　　　　　　　　Range:　　　-90　to　90

Example:　　　　　　　See `SCAN:CUSTom:ENTRy:AZIMuth` on page 625

Manual operation:　　See "Custom Scan" on page 290

SCAN:CUSTom:ENTRy:DWELl <Dwell>

Sets how long the scan stays in a position.

Parameters:

<Dwell>　　　　　　　　　float

　　　　　　　　　　　　　Range:　　　0　to　3600

Example:　　　　　　　See `SCAN:CUSTom:ENTRy:AZIMuth` on page 625

Manual operation:　　See "Custom Scan" on page 290

SCAN:CUSTom:ENTRy:JUMPtype <Jumptype>

Defines how to move to the next position, either with a jump or with a transition.

For transitions, you need to define a transition time.

Parameters:

<Jumptype>	ON | OFF | 1 | 0
	ON | 1 Jump enabled.
	OFF | 0 Transition enabled.

Example: See `SCAN:CUSTom:ENTRy:AZIMuth` on page 625

Manual operation: See "Custom Scan" on page 290

SCAN:CUSTom:ENTRy:TRANstime <Transtime>

Sets the time for the transition between two positions.

Parameters:

<Transtime>	float
	Range: 0 to 3600

Example: See `SCAN:CUSTom:ENTRy:AZIMuth` on page 625

Manual operation: See "Custom Scan" on page 290

SCAN:LISSajous:AMPX <Ampx>
SCAN:LISSajous:AMPZ <Ampz>

Sets the magnitudes of two harmonic vibrations.

Parameters:

<Ampz>	float
	Range: 0.01 to 45

Example:
```
SCAN:LISSajous:AMPX 45
SCAN:LISSajous:AMPZ 30
SCAN:LISSajous:FREQ 10
SCAN:LISSajous:XFACtor 9
SCAN:LISSajous:ZFACtor 3
SCAN:LISSajous:PHIX 15
SCAN:LISSajous:PHIZ 30
```

Manual operation: See "Lissajous" on page 292

SCAN:LISSajous:FREQ <Freq>

Sets the base frequency.

Parameters:

<Freq>	float
	Range: 0.01 to 1000

Example: See `SCAN:LISSajous:AMPZ` on page 627

Manual operation: See "Lissajous" on page 292

SCAN:LISSajous:PHIX <Phix>
SCAN:LISSajous:PHIZ <Phiz>

Sets the phases of the two harmonic vibrations.

Parameters:

<Phiz>   float

                      Range: 0 to 360

Example: See SCAN:LISSajous:AMPZ on page 627

Manual operation: See "Lissajous" on page 292

SCAN:LISSajous:XFACtor <Xfactor>
SCAN:LISSajous:ZFACtor <Zfactor>

Sets the ratio between the two angular frequencies.

Parameters:

<Zfactor>   float

                      Range: 1 to 10

Example: See SCAN:LISSajous:AMPZ on page 627

Manual operation: See "Lissajous" on page 292

SCAN:SIN:WIDTh <Width>

Sets the angle on the XY plane between the origin and the end of the scan.

Parameters:

<Width>   float

                      Range: 1 to 180

Example: ```
                      SCAN:TYPE SIN
                      SCAN:SIN:WIDTh 90
                      SCAN:SIN:HEIGht 30
                      SCAN:SIN:RATE 100
                      SCAN:SIN:ROTation CW
                      SCAN:SIN:UNIDirection 0
                      SCAN:SIN:INVersion 0
                      ```

**Manual operation:** See "Sine Scan" on page 289

---

**SCAN:SIN:HEIGht** <Height>

Sets the amplitude of the sine wave.

**Parameters:**

| | |
|---|---|
| <Height> | float |
| | Range: 1 to 90 |

| | |
|---|---|
| **Example:** | See SCAN:SIN:WIDTh on page 628 |
| **Manual operation:** | See "Sine Scan" on page 289 |

---

**SCAN:SIN:INVersion** <Inversion>

Sets whether the upper or the down (mirrored) sine wave is used first.

**Parameters:**

| | |
|---|---|
| <Inversion> | ON | OFF | 1 | 0 |
| | **OFF** |
| | Upper sine first |
| | **ON** |
| | Down sine first |

| | |
|---|---|
| **Example:** | See SCAN:SIN:WIDTh on page 628 |
| **Manual operation:** | See "Sine Scan" on page 289 |

---

**SCAN:STEering** <Steering>

Defines whether electronic steering is used.

Electronic steering is only available for scan types that use phased array antennas.

**Parameters:**

| | |
|---|---|
| <Steering> | ON | OFF | 1 | 0 |
| **Example:** | SCAN:STEering ON |
| **Manual operation:** | See "Electronic Scan" on page 282 |

## 25.8 Control panel commands

**Example: Configuring connected instruments**

```
SCPI
INSTrument:COUNt?
// 3

SCENario:NAME "DF"
SCENario:CALCulate
SCENario:NAME "LocalizedEmitters"
SCENario:CALCulate
```

```
// activate the remote control of the control panel
CPANel:ACTivate
CPANel:SETup?
// DF
CPANel:SCENario:CURRent:NAME?
// LocalizedEmitters
CPANel:SCENario:CURRent:STATe?
// Calculated
CPANel:SCENario:DEPLoyed:NAME?
// DF
CPANel:SCENario:DEPLoyed:TIME?
// 10 Dec 2015 08:38:34

CPANel:REFResh
INSTrument:LIST?
// 'SMW 200A, 0','SMW 200A, 100025','SMBV 100A, 100012'
CPANel:UNUSed:ALL 1
CPANel:UNUSed:LIST?
// 'SMW 200A, 100025','SMBV 100A, 100012'

CPANel:USED:LIST?
// 'SMW 200A, 0'
CPANel:USED:SELect 'SMW 200A, 0'
CPANel:USED:PATH:LIST?
// 'RF A', 'RF B'
CPANel:USED:PATH:SELect 'RF B'
CPANel:USED:FREQuency 1e+09
CPANel:USED:LEVel -30
CPANel:USED:STATe 1


CPANel:USED:LIST?
//'SMW 200A, 0'
CPANel:USED:SELect 'SMW 200A, 0'
CPANel:MUTE
CPANel:USED:STATe?
// 0
CPANel:UNMute


CPANel:DEACtivate
// always deactivate the remote control of the control panel
```

CPANel:ACTivate
CPANel:DEACtivate

Activates and deactivates the remote control of the control panel.

Further `CPANel:...` commands cannot be executed, if the remote control is not active.

After configuration, **always deactivate** the remote control of the control panel.

Example:	See Example"Configuring connected instruments" on page 629.
Usage:	Event

CPANel:REFResh

Refreshes the displayed information.

Example:	See Example"Configuring connected instruments" on page 629.
Usage:	Event
Manual operation:	See "Refresh" on page 488

CPANel:MUTE

Deactivates the RF outputs of all signal generators.

Example:	See Example"Configuring connected instruments" on page 629.
Usage:	Event
Manual operation:	See "Mute All" on page 488

CPANel:UNMute

Activates the RF outputs of all signal generators.

Example:	See Example"Configuring connected instruments" on page 629.
Usage:	Event

Manual operation: See "Unmute All" on page 488

CPANel:SETup?

Queries the name of the setup selected in the "Signal Generators" dialog.

Return values:

<Setup>	string

Example: See Example"Configuring connected instruments" on page 629.

Usage: Query only

Manual operation: See "Hardware" on page 488

CPANel:SCENario:DEPLoyed:NAME?
CPANel:SCENario:CURRent:NAME?

Queries the name of the current/last deployed scenario.

Return values:

<Name>	string

Example: See Example"Configuring connected instruments" on page 629.

Usage: Query only

Manual operation: See "Scenario" on page 487

CPANel:SCENario:CURRent:STATus?

Queries the scenario status.

Return values:

<Status>	string

Example: See Example"Configuring connected instruments" on page 629.

Usage: Query only

Manual operation: See "Status" on page 488

CPANel:SCENario:DEPLoyed:TIME?

Queries the date and time the scenario has been deployed.

Return values:

<Time>	<DD MMM YYY hh:mm:ss>

Example: See Example"Configuring connected instruments" on page 629.

Usage: Query only

Manual operation: See "Upload time" on page 488

CPANel:UNUSed:ALL <All>

Shows all unused signal generators.

Parameters:

<All>   ON | OFF | 1 | 0

Example: See Example"Configuring connected instruments" on page 629.

CPANel:UNUSed:LIST?
CPANel:USED:LIST?

Queries the names of the used/unused signal generators.

Return values:

<List>   "<Instr#1>","<Instr#2>",...

Example: See Example"Configuring connected instruments" on page 629.

Usage: Query only

Manual operation: See "Used Generators, Unused generators" on page 489

CPANel:UNUSed:PATH:LIST <List>
CPANel:USED:PATH:LIST?

Queries the available RF paths of the selected signal generators.

Return values:

<List>   "<Path#1>","<Paht#2>",...

Example: See Example"Configuring connected instruments" on page 629.

Usage: Query only

Manual operation: See "Used Generators, Unused generators" on page 489

CPANel:UNUSed:FREQuency <Frequency>
CPANel:USED:FREQuency <Frequency>

Sets the RF frequency.

Parameters:

<Frequency>   float

Example: See Example"Configuring connected instruments" on page 629.

Manual operation: See "Used Generators, Unused generators" on page 489

CPANel:UNUSed:LEVel <Level>
CPANel:USED:LEVel <Level>

Sets the RF level.

Parameters:

<Level>   float

Example:	See Example"Configuring connected instruments" on page 629.
Manual operation:	See "Used Generators, Unused generators" on page 489

CPANel:UNUSed:STATe <State>
CPANel:USED:STATe <State>

Activates the RF output of the selected signal generator.

Parameters:

<State>	ON	OFF	1	0
Example:	See Example"Configuring connected instruments" on page 629.
Manual operation:	See "Used Generators, Unused generators" on page 489

25.9 Data source commands

Example: Creating data source

```
SCPI
DSRC:CREate "DataSource"
DSRC:SELect "DataSource"
DSRC:ITEM:ADD
DSRC:ITEM:SELect 1
DSRC:ITEM:TYPE PATTern
DSRC:ITEM:PATTern ONE
DSRC:ITEM:BITS 2

DSRC:ITEM:ADD
DSRC:ITEM:SELect 2
DSRC:ITEM:TYPE PRBS
DSRC:ITEM:PRBS:TYPE P15
DSRC:ITEM:PRBS:INIT 1
DSRC:ITEM:PRBS:INIT:VALue 10
DSRC:ITEM:BITS 12
```

DSRC:ITEM:BITS <Bits>

Sets the length of the selected item in bits.

Parameters:

<Bits>   float

                             Range: 0 to 4096

Example: See Example"Creating data source" on page 634

Manual operation: See "Data Source Table" on page 527

DSRC:ITEM:DATA <Data>

Sets the user defined data pattern.

Parameters:

<Data>   string

Example: `DSRC:ITEM:TYPE USER`
                             `DSRC:ITEM:DATA?`
                             `// 0101 1010     /binary`
                             `// "Test"        /string`
                             `// #x 20 20 20   /hex numbers`

Manual operation: See "User Data" on page 528

DSRC:ITEM:PATTern <Pattern>

Sets the data pattern of the selected item.

Parameters:

<Pattern>   ZERO | ONE | ALT | R2A | R2B | R3 | R4A | R4B | R5 | R7 |
                             R11 | R13

                             ZERO|ONE
                             Binary 0 and 1

                             ALT
                             Variable bit strings ("1010") with alternating 0 and 1 and a maxi-
                             mum length of 999 bits

                             R2A|R2B|R3|R4A|R4B|R5|R7|R11|R13
                             Barker codes

Example: See Example"Creating data source" on page 634.

Manual operation: See "Data Source Table" on page 527

DSRC:ITEM:PRBS:INIT <Init>

Enables/disables initialization of the shift register with a user-defined value.

Parameters:

<Init>   ON | OFF

Example: See Example"Creating data source" on page 634

Manual operation: See "Shift register" on page 527

DSRC:ITEM:PRBS:INIT:VALue <Value>

Set a new initialization value.

Parameters:

<Value>   float

                  Range: 1 to 511

Example: See Example"Creating data source" on page 634

Manual operation: See "Shift register" on page 527

DSRC:ITEM:PRBS:TYPE <Type>

Sets the PRBS type for the selected item.

Parameters:

<Type>   P9 | P11 | P15 | P16 | P20 | P21 | P23 | P7

Example: See Example"Creating data source" on page 634

Manual operation: See "Data Source Table" on page 527

DSRC:ITEM:TYPE <Type>

Sets the data type of selected item.

Parameters:

<Type>   PATTern | PRBS | USER

Example: See Example"Creating data source" on page 634

Manual operation: See "Data Source Table" on page 527

25.10 Signal mapping commands

Example: Performing signal to generator mapping in 2D scenario

```
SCPI
ASSignment:GENerator:LIST?
// "My_RS_SMW","My_RS_SMBV","RS_SMW","My_SGT","SMW"

ASSignment:GENerator:SELect "My_RS_SMW"
ASSignment:GENerator:PRIMary "My_RS_SMW"
ASSignment:GENerator:CAPabilities?
// 1K76|1B106|1B206|2K300|2K301|1K308|2K511|2K512|2K522|
// 2B10|1B13T|#0|6e+09|0|0|
ASSignment:GENerator:PATH:LIST?
// "RF A","RF B","BBMM 1"
ASSignment:GENerator:PATH:SELect "RF A"
ASSignment:GENerator:PATH:EMITter:LIST?
// "E1","E2"
ASSignment:EMITters:LIST?
// "I1","Bkg. Emitter 1"

ASSignment:EMITters:SELect "Bkg. Emitter 1"
ASSignment:GENerator:PATH:SELect "BBMM 1"
ASSignment:GENerator:PATH:EMITter:ADD
ASSignment:GENerator:PATH:EMITter:LIST?
// "Bkg. Emitter 1"

ASSignment:EMITters:LIST?
// "I1"
```

Example: Receiver signals to generator mapping in direction finding scenario

```
SCPI
ASSignment:GENerator:LIST?
// "My_RS_SMW","My_RS_SMBV","RS_SMW","My_SGT","SMW"
ASSignment:GENerator:SELect "My_RS_SMW"

ASSignment:GENerator:PATH:LIST?
// "RF A","RF B","I/Q OUT 2"
ASSignment:GENerator:PATH:SELect "RF A"
ASSignment:GENerator:PATH:ANTenna:LIST?
// "Antenna 1 - TestEmitter","Antenna 2 - TestEmitter"
ASSignment:GENerator:PATH:SELect "RF B"
ASSignment:GENerator:PATH:ANTenna:LIST?
// "Antenna 3 - TestEmitter"
ASSignment:ANTennas:LIST?
// "Antenna 4 - TestEmitter"
ASSignment:ANTennas:SELect "Antenna 4 - TestEmitter"
ASSignment:GENerator:PATH:ANTenna:ADD
ASSignment:GENerator:PATH:ANTenna:LIST?
// "Antenna 3 - TestEmitter","Antenna 4 - TestEmitter"
ASSignment:GENerator:PATH:ANTenna:SELect "Antenna 3 - TestEmitter"
ASSignment:GENerator:PATH:ANTenna:DELete
ASSignment:GENerator:PATH:ANTenna:LIST?
// "Antenna 4 - TestEmitter"
```

ASSignment:EMITters:LIST?

Queries the alias names of the unassigned emitters/interferers.

Return values:

<List>   "<Emitter/Inter#1>","<Emitter/Inter#2>",...

Example: See Example"Performing signal to generator mapping in 2D scenario" on page 637

Usage: Query only

Manual operation: See "Emitters/Interferers" on page 483

ASSignment:ANTennas:LIST?

Queries the alias names of the unassigned receiver signals.

Return values:

<List>	"<ReceiverSignal#1>","<ReceiverSignal#2>",...
Example:	See Example"Receiver signals to generator mapping in direction finding scenario" on page 638.
Usage:	Query only
Manual operation:	See "Receiver Signals" on page 483

ASSignment:GENerator:LIST?

Queries a list of the available generators.

Return values:

<List>	"<GenName#1>","<GenName2>",...
Example:	See Example"Performing signal to generator mapping in 2D scenario" on page 637
Usage:	Query only
Manual operation:	See "Available for assignment" on page 484

ASSignment:GENerator:CAPabilities?

Queries the capabilities of the selected generator.

Return values:

<Capabilities>	<RFA>, <BWA>, <MEMA>, <SWA>, [<RFB>, <BWB>, <MEMB>, <SWB>]
	String that lists the maximum RF and BW, available memory size, and installed options per path.
Example:	See Example"Performing signal to generator mapping in 2D scenario" on page 637
Usage:	Query only
Manual operation:	See "Available for assignment" on page 484

ASSignment:GENerator:PATH:LIST?

Queries the available paths.

Return values:

<List>	"<Path#1>","<Path#2>",...
	List of available paths.
Example:	See Example"Performing signal to generator mapping in 2D scenario" on page 637

Usage:	Query only
Manual operation:	See "Available for assignment" on page 484

ASSignment:GENerator:PATH:ANTenna:LIST?

Queries the list of assigned receiver signals to the selected path.

Return values:

<List>	"<ReceiverSignal#1>","<ReceiverSignal#2>",...
Example:	See Example"Receiver signals to generator mapping in direction finding scenario" on page 638.
Usage:	Query only
Manual operation:	See "Available for assignment" on page 484

ASSignment:GENerator:PATH:EMITter:LIST?

Queries the list of assigned emitters/interferes to the selected path.

Return values:

<List>	"<Emitter/Inter#1>","<Emitter/Inter#2>",...
Example:	See Example"Performing signal to generator mapping in 2D scenario" on page 637.
Usage:	Query only
Manual operation:	See "Available for assignment" on page 484

ASSignment:GROup:LIST?

If interleaving groups are defined, queries the alias names of the unassigned interleaving groups.

Return values:

<List>	string
Example:	See ASSignment:GROup:SELect on page 640.
Usage:	Query only
Manual operation:	See "Interleaving Groups" on page 483

ASSignment:GROup:SELect <Select>

Assigns the selected group to the generator and path selected with the commands ASSignment:GENerator:SELect and ASSignment:GENerator:PATH:SELect.

Parameters:

<Select>	string

Example:
```
ASSignment:GENerator:LIST?
// "Gen 1"
ASSignment:GENerator:SELect "Gen 1"
ASSignment:GENerator:PATH:LIST?
// "RF A"
ASSignment:GENerator:PATH:SELect "RF A"
ASSignment:GROup:LIST?
// "Default","Group 2"
ASSignment:GROup:SELect "Default"
```

Manual operation: See "Interleaving Groups" on page 483

25.11 Emitter commands

Example: Creating emitters

```
EMITter:CREate "My_EmitterGuidance"
EMITter:COMMent "guidance&survaillance"
EMITter:EIRP 120
EMITter:FREQuency 3e+09
EMITter:MODE:ADD
EMITter:MODE:SELect 1
EMITter:MODE:NAME "Guidance"
ANTenna:CATalog?
// "Isotropic","My_PencilBeam","Test CSC","My_PlanarAntenna","Test","Custom"
EMITter:MODE:ANTenna "My_PencilBeam"
SCAN:CATalog?
// "Test Antenna Scan","My_RasterScan"
EMITter:MODE:SCAN "My_RasterScan"
EMITter:MODE:BEAM:ADD
EMITter:MODE:BEAM:SELect 1
EMITter:MODE:BEAM:STATe 1
SEQuence:CATalog?
// "Test Sequence","My_S2_S1-F1_S1-F2","PT"
EMITter:MODE:BEAM:SEQuence "My_S2_S1-F1_S1-F2"
EMITter:MODE:BEAM:OFFSet:FREQuency 0
EMITter:MODE:BEAM:OFFSet:ELEVation 0
EMITter:MODE:BEAM:OFFSet:AZIMuth 0
```

EMITter:EIRP <Eirp>

Sets the EIRP of the emitter.

Parameters:

<Eirp>   float

Range: -100 to 200
Default unit: dBW

Example: See Example"Creating emitters" on page 641

Manual operation: See "EIRP" on page 308

EMITter:FREQuency <Frequency>

Sets the operating frequency.

Parameters:

<Frequency>   float

Range: 1000 to 1e+11
Default unit: Hz

Example: Example"Creating emitters" on page 641

Manual operation: See "Frequency" on page 308

EMITter:MODE:ANTenna <Antenna>

Assigns an existing antenna pattern, see `ANTenna:CATalog?`.

Parameters:

<Antenna>   string

Example: Example"Creating emitters" on page 641

Manual operation: See "Emitter Modes" on page 308

EMITter:MODE:BEAM:OFFSet:AZIMuth <Azimuth>

Sets the Azimuth value for the beam offset.

Parameters:

<Azimuth>   float

Range: 0 to 360

Example: Example"Creating emitters" on page 641

Manual operation: See "Emitter Beams Definition" on page 309

EMITter:MODE:BEAM:OFFSet:ELEVation <Elevation>

Offsets the position of the beam in both the azimuth or elevation.

Parameters:

<Elevation>	float
	Range:　　　-90 to 90

Example:	Example"Creating emitters" on page 641
Manual operation:	See "Emitter Beams Definition" on page 309

EMITter:MODE:BEAM:OFFSet:FREQuency <Frequency>

Offsets the frequency of the beam.

Parameters:

<Frequency>	float
	Range:　　　-1e+09 to 1e+09

Example:	Example"Creating emitters" on page 641
Manual operation:	See "Emitter Beams Definition" on page 309

EMITter:MODE:BEAM:SEQuence <Sequence>

Assigns a pulse sequence, see `SEQuence:CREate`.

Parameters:

<Sequence>	string

Example:	Example"Creating emitters" on page 641
Manual operation:	See "Emitter Beams Definition" on page 309

EMITter:MODE:BEAM:STATe <State>

Activates a beam.

Parameters:

<State>	ON	OFF	1	0

Example:	Example"Creating emitters" on page 641
Manual operation:	See "Emitter Beams Definition" on page 309

EMITter:MODE:SCAN <Scan>

Assigns an antenna scan, see `SCAN:CREate`.

Parameters:

<Scan>	string

Example:	Example"Creating emitters" on page 641
Manual operation:	See "Emitter Modes" on page 308

25.12 Adjustment commands

ADJustment:RELoad

Reinitializes the adjustment database.

You can create the level adjustment files not only automatically with the build-in Run Level Adjustment function but also externally (or manually), by performing your own specific measurements. With this command, you can (re)load the level adjustment files, irrespectively of way they are created.

Example: ADJustment:RELoad

Usage: Event

Manual operation: See "Accept, Reject" on page 98

25.13 Generator and instrument commands

Example: Creating generators

```
SCPI
SETup:LIST?
// 'My_TestSetup','DF','2D'
SETup:SELect 'My_TestSetup'
SETup:BBSYnc CTRigger
SETup:LO 1

INSTrument:COUNt?
// 1
INSTrument:LIST?
// 'SMBV 100A, 100012'
INSTrument:CLEar
// deletes all instruments from the list
**********************************************************************
* Connecting to a physical instrument
**********************************************************************
INSTrument:ADD "10.113.0.104"
INSTrument:ADD "smw200a-101244"
INSTrument:LIST?
// 'SMW 200A, 102030','SMW 200A, 101244'
INSTrument:SELect "SMW 200A, 0"
INSTrument:NAME?
// "SMW 200A, 0"
INSTrument:PSEC PRIMary
INSTrument:CREate
INSTrument:CAPabilities?
// 1K76|1B106|1B206|2K300|2K301|1K308|2K350|2K511|2K512|2K522|2B10|1B13T|#0|0|0|0|
GENerator:CATalog?
```

```
// "My_RS_SMBV","My_SGT","SMW 200A, 102030","SMW 200A, 101963","SMW 200A, 101966","SMW 200A,
INSTrument:MAP "SMW 200A, 102030"
INSTrument:SUPPorted?
// 1
GENerator:SELEct "SMW 200A, 102030"
GENerator:TYPE?
// SMW

INSTrument:LINK:STATus?
// 1
INSTrument:FIND? "SMW 200A, 102030"
INSTrument:FIRMware?
// 3.1.18.2-3.20.012.88
INSTrument:RESource?
// TCPIP::10.113.0.104::INSTR




******************************************************************
* Creating a virtual generator manually and mapping it to an instrument
******************************************************************
GENerator:CREate "My_RS_SMW"
GENerator:TYPE SMW
... etc.
```

SETup:LIST?

Queries the name of the available hardware setups.

Return values:

<List>	"<Setup#1>","<Setup#2>",...

Example:	See Example"Creating generators" on page 644.
Usage:	Query only
Manual operation:	See "Compatibility" on page 80

GENerator:TYPE <Type>

Sets the generator type.

Parameters:

<Type>	SMU | SMBV | SMJ | SGT | SMBB | SW | SMM | SMW
	`SMBB` designates R&S SMBV100B.

Example:	See Example"Creating generators" on page 644.
Manual operation:	See "Model" on page 89

INSTrument:CAPabilities <Capabilities>
GENerator:CAPabilities <Capabilities>

Queries the generator capabilities regarding supported scenario types and processing of waveforms and multi-segment files.

Parameters:

<Capabilities>	string

Example:	See Example"Creating generators" on page 644.
Manual operation:	See "Show Config" on page 89

INSTrument:CONNect <Connect>

Connects to the instrument defined by `INSTrument:SELect` on page 593

Setting parameters:

<Connect>	string

Usage:	Setting only
Manual operation:	See "Remote Connection" on page 81

INSTrument:LIST?

Queries the names of the signal generators that are connected to the R&S Pulse Sequencer in the current setup.

See `SETup:LIST?` on page 646.

Return values:

<List>	"<Instr#1>","<Instr#2>",...

Example: See Example"Creating generators" on page 644.

Usage: Query only

Manual operation: See "Instrument" on page 81

INSTrument:ADD <Add>

Adds an instrument with the specified IP address, computer name, or the VISA resource string.

Setting parameters:

<Add>	string
	"<IP_Address>" or "<Computer_Name>" or "<GPIB_Address>"

Example: See Example"Creating generators" on page 644

Usage: Setting only

Manual operation: See "Add" on page 77

INSTrument:NAME <Name>

Queries the name of the selected instrument.

Parameters:

<Name>	string

Example: See Example"Creating generators" on page 644.

Manual operation: See "Instrument" on page 81

INSTrument:FIRMware?

Queries the firmware version of the selected instrument.

Return values:

<Firmware>	string

Example: See Example"Creating generators" on page 644

Usage: Query only

Manual operation: See "Instrument" on page 81

INSTrument:ONLine <Online>

Queries the connection status of a physical signal generator.

Parameters:

<Online>	float

0
Offline

1
Online

Manual operation: See "Instrument" on page 81

INSTrument:PSEC <Psec>

Sets the Primary/Secondary order in synchronized setups.

Parameters:

<Psec>   NONE | PRIMary | SEC1 | SEC2 | SEC3 | SEC4 | SEC5 |
                    SEC6 | SEC7 | SEC8 | SEC9 | SEC10 | SEC11 | SEC12 |
                    SEC13 | SEC14 | SEC15 | SEC16

                    NONE
                    Unsynchronized instrument.

                    PRIMary
                    The primary instrument.

                    SEC1
                    Denotes the secondary instruments.

Example: See Example"Creating generators" on page 644.

Manual operation: See "Instrument" on page 81

INSTrument:RESource?

Queries the resource string of the instrument selected with the command
INSTrument:SELect.

Return values:

<Resource>   string

Example: See Example"Creating generators" on page 644

Usage: Query only

INSTrument:SUPPorted?

Queries the supported status.

Return values:

<Supported>   0 | 1

                    1
                    Supported

                    0
                    Not supported

Example: See Example"Creating generators" on page 644.

Usage: Query only

Manual operation: See "Firmware Status" on page 81

INSTrument:TYPE <Type>

Sets the instrument type.

Parameters:

<Type>   SMU | SMBV | SMJ | SGT | SMBB | SW | SMM | SMW

INSTrument:VIRTual?

Queries the state of the virtual instrument.

Return values:

<Virtual>   ON | OFF | 1 | 0

Usage: Query only

INSTrument:ADB:STATe <State>

Defines the instrument that holds the adjustment data of hardware setup.

Parameters:

<State>   ON | OFF | 1 | 0

Example: `INSTrument:ADB:STATe 1`

Manual operation: See "Adj. Data" on page 83

SETup:HIGHq:MODE <Mode>

Defines which high-quality mode is used.

Requires that `SETup:HIGHq:ENABle` is set to ON (see `SETup:HIGHq:ENABle` on page 649).

Parameters:

<Mode>   NORMal | TABLe

                          NORM
                          Enables compensation for I/Q skew and frequency response correction.
                          This mode generates a flat signal over a large bandwidth but requires longer setting time and can lead to signal interruption.

                          TABL
                          This mode provides optimization while maintaining settling time.

Manual operation: See "Use High Quality mode" on page 78

SETup:HIGHq:ENABle <Enable>

Sets the I/Q modulator of signal generator to work in a high quality mode.

Parameters:

<Enable>   ON | OFF | 1 | 0

Example: `SETup:HIGHq:ENABle 1`

Manual operation: See "Use High Quality mode" on page 78

SETup:PMOD:ENABle <Enable>

Enables the activation of the pulse modulator for improving the on/off ratio.

Parameters:

<Enable>   ON | OFF | 1 | 0

Example: `INSTrument:PMOD INTernal`
                     `SETup:PMOD:ENABle 1`

Example: ```
****
// using the pulse modulator with external marker requires that:
// 1) the instruments are cabled according to the wiring diagram
// 2) high the clock rate is selected
// 3) pulse marker M2 is set to pulse width
// 4) sequence marker M2 is enabled
// 5) the generation of the dedicated marker signal is enabled
****
INSTrument:PMOD EXTernal
SCENario:OUTPut:CLOCk:USER 2e+08
PULSe:MARKer:WIDTh 4
SEQuence:ITEM:MARKer:ALL 65535
SCENario:OUTPut:MARKer:PMODulator:ENABle 1
SETup:PMOD:ENABle 1
```

Manual operation: See "Use Pulse Modulator" on page 78

INSTrument:PMOD <Pmod>

Select the marker source if pulse modulator is used, see `SETup:PMOD:ENABle` on page 650.

Parameters:

<Pmod>   OFF | INTernal | EXTernal

                   OFF
                   Disables the function for the selected instrument.

                   INTernal
                   No additional cabling or configuration of marker signals is
                   required.

	EXTernal
	Requires that:
	1) the instruments are cabled according to the wiring diagram
	2) high the clock rate is selected
	3) pulse marker M2 is set to pulse width
	4) sequence marker M2 is enabled
	5) the generation of the dedicated marker signal is enabled

Example:	See `SETup:PMOD:ENABle`.
Manual operation:	See "Pulse Modulator" on page 82

SETup:LOCPI:ENABle <Enable>

Couples the LO signals.

Parameters:

<Enable>	ON	OFF	1	0

Manual operation:	See "LO Coupling" on page 78

SETup:BBSYnc <Bbsync>

Sets if and which method the signal generator uses to synchronize the signals in the baseband domain.

Relevant in multi-instrument setups where the signals of the different emitters are generated in different paths and different signal generators.

Parameters:

<Bbsync>	UNSYnc	TRIGger	CTRigger
	UNSYnc
	Unsynchronized baseband generators
	TRIGger
	Synchronized setup, where the instruments are connected in a primary/secondary chain.
	General-purpose trigger signal is used.
	CTRigger
	Synchronized primary/secondary setup, that uses a dedicated common trigger signal.

Example:	See Example"Creating generators" on page 644.
Manual operation:	See "Baseband Sync Mode" on page 77

25.14 Inter-pulse modulation commands

Example: Using list and waveform type IPM profiles

```
SCPI
IPM:CREate "My_PRI-Stagger"
IPM:COMMent "List: 600 us, 750 us, 910 us"
IPM:UNIT SEConds
IPM:TYPE LIST
IPM:LIST:ITEM:ADD
IPM:LIST:ITEM:SELect 1
IPM:LIST:ITEM:BASE PULSe
IPM:LIST:ITEM:VALue 0.0006
IPM:LIST:ITEM:REPetitions 1
IPM:LIST:ITEM:COUNt 1
IPM:LIST:ITEM:ADD
IPM:LIST:ITEM:SELect 2
IPM:LIST:ITEM:VALue 0.00075
IPM:LIST:ITEM:COUNt 2
...
IPM:LIST:SAVE "C:\_PS_files\IPM_PRI_Stagger.txt"
IPM:LIST:CLEar
// load an existing ASCII file
IPM:LIST:LOAD "C:\_PS_files\IPM_PRList.txt"

IPM:CREate "Stagger PRI"
IPM:UNIT SEConds
IPM:TYPE WAVeform
IPM:WAVeform:TYPE SINE
IPM:WAVeform:OFFSet 0.0004
IPM:WAVeform:PHASe 0
IPM:WAVeform:PKPK 0.00014
IPM:WAVeform:BASE PULSe
IPM:WAVeform:COUNt 30
```

Example: Assigning an IPM profile to a sequence

```
SCPI
SEQuence:SELect "Test Sequence"
SEQuence:ITEM:SELect 1
SEQuence:ITEM:IPM:ADD
SEQuence:ITEM:IPM:SOURce:TYPE PROFile
SEQuence:ITEM:IPM:SOURce "My_PRI-Stagger"
SEQuence:ITEM:IPM:TARGet:TYPE PARameter
SEQuence:ITEM:IPM:TARGet:PARameter PRI
SEQuence:ITEM:IPM:EQUation "0.5*x+1"
SEQuence:ITEM:IPM:MODE INDividual
SEQuence:ITEM:IPM:RESTart 0
SEQuence:ITEM:IPM:RANDom:RESet 0
```

Example: Using the step IPM profile

```
SCPI
IPM:CREate "FHOP"
IPM:COMMent "Frequency Hops"
IPM:UNIT HERTz
IPM:TYPE STEPs
IPM:STEP:STARt 0
IPM:STEP:INCRement 1e+06
IPM:STEP:STEPs 5
IPM:STEP:BASE LENGth
IPM:STEP:BURSt 2
```

Example: Using PRI profiles of random type

```
SCPI
IPM:CREate "My_PRI-Jitter"
IPM:UNIT SEConds
IPM:TYPE RANDom
IPM:RANDom:DISTribution UNIForm
IPM:RANDom:UNIForm:MINimum 0.0011
IPM:RANDom:UNIForm:MAXimum 0.00115
IPM:RANDom:UNIForm:STEP 2.5e-06
DIALog:IPMPlot:VIEW TIMeseries
DIALog:IPMPlot:SAMPles 30

IPM:CREate "Test"
IPM:UNIT SEConds
IPM:TYPE RSTEp
IPM:RSTep:MINimum 0.0005
IPM:RSTep:MAXimum 0.005
IPM:RSTep:STEP:MINimum 0.0001
IPM:RSTep:STEP:MAXimum 0.0005
IPM:RSTep:PERiod 10

IPM:CREate "Test2"
IPM:UNIT HERTz
IPM:TYPE RLISt
// create a List (see the "My_PRI-Stagger" example)
IPM:RLISt:BASE LENGth
IPM:RLISt:BURSt 3
IPM:RLISt:REUSe 0
```

Example: Using the interpolated shape IPM profile

```
SCPI
IPM:CREate "Custom IPM"
IPM:UNIT DB
IPM:TYPE SHAPe
// create a List (see the "My_PRI-Stagger" example)
IPM:SHAPe:BASE PULSE
IPM:SHAPe:COUNt 10
// IPM:SHAPe:BASE TIME
// IPM:SHAPe:PERiod 0.5
IPM:SHAPe:INTerpol NONE
```

SEQuence:ITEM:IPM:SOURce:TYPE <Type>

Sets whether the variation is defined as a profile or as a variable.

Parameters:

<Type>   PROFile | VARiable

Example: See Example"Assigning an IPM profile to a sequence"
                           on page 652

Manual operation: See "Source" on page 243

SEQuence:ITEM:IPM:SOURce <Source>

Selects the profile source.

Use the command `IPM:CATalog?` to querry the existing profiles.

Parameters:

<Source>   string

Example: See Assigning an IPM profile to a sequence

Manual operation: See "Source" on page 243

SEQuence:ITEM:IPM:SOURce:VARiable <Variable>

Sets the variable that defines the variation.

Parameters:

<Variable>   string

Manual operation: See "Source" on page 243

SEQuence:ITEM:IPM:TARGet:TYPE <Type>

Sets whether the profile is assigned to a parameter or to a variable.

Parameters:

<Type>   PARameter | VARiable

Example: See Assigning an IPM profile to a sequence

Manual operation: See "Target" on page 243

SEQuence:ITEM:IPM:TARGet:PARameter <Parameter>

Sets the pulse parameter to that the IPM variation is assigned.

Parameters:

<Parameter>   LEVel | RLEVel | SRATe | FREQuency | PRI | WIDTh | FALL |
                            AMFRequency | FMDeviation | DELay | FSKDeviation | PRF |
                            FMFRequency | CDEViation | PHASe | RISE | AMDepth |
                            DROop | RFRequency | OVERshoot

Example: See Assigning an IPM profile to a sequence

Manual operation: See "Target" on page 243

SEQuence:ITEM:IPM:TARGet:VARiable <Variable>

Sets the variable to that the variation is assigned.

Parameters:

<Variable>   string

Manual operation: See "Target" on page 243

SEQuence:ITEM:IPM:EQUation <Equation>

Defines output value of the IPM mathematically.

Parameters:

<Equation>   string

Example: See Assigning an IPM profile to a sequence

Manual operation: See "Transformation f(x)" on page 244

SEQuence:ITEM:IPM:MODE <Mode>

Defines the way the variations are applied on repeating pulses.

Parameters:

\<Mode\>	INDividual | SAME

Example: See Assigning an IPM profile to a sequence

Manual operation: See "Repetition" on page 244

SEQuence:ITEM:IPM:RESTart \<Restart\>

Restarts the IPM for this sequence line item.

Parameters:

\<Restart\>	ON | OFF | 1 | 0

Example: See Assigning an IPM profile to a sequence

Manual operation: See "Restart" on page 244

SEQuence:ITEM:IPM:RANDom:RESet \<Reset\>

Resets the start seed of random generator.

Parameters:

\<Reset\>	ON | OFF | 1 | 0

Example: See Assigning an IPM profile to a sequence

Manual operation: See "Restart" on page 244

IPM:TYPE \<Type\>

Sets the shape of the profile.

Parameters:

\<Type\>	STEPs | WAVeform | RLISt | LIST | SHAPe | RANDom | EQUation | PLUGin | RSTep | BINomial

Example: See Using list and waveform type IPM profiles

Manual operation: See "Profile" on page 228

IPM:UNIT \<Unit\>

Sets the units of the profile.

Parameters:

\<Unit\>	NONE | SEConds | HERTz | DB | DEGRees | PERCent

Example: See Using list and waveform type IPM profiles

Manual operation: See "Unit of Affected Parameter" on page 228

IPM:EQUation \<Equation\>

Defines the IPM shape as a function.

Parameters:

<Equation>	string

Example: `IPM:EQUation "(i<10) ? 5 : (i*0.5)"`

Manual operation: See "Equation" on page 234

IPM:LIST:BASE <Base>

Sets the IPM profile base and defines how the steps repetition is defined.

Parameters:

<Base>   PULSe | TIME

PULSe
Steps are repeated several times, as set with the command
`IPM:LIST:ITEM:REPetition`.

TIME
Steps are repeated for the defined time duration, as set with the
command `IPM:LIST:ITEM:TIME`.

Example: See Using list and waveform type IPM profiles

Manual operation: See "List" on page 230

IPM:LIST:ITEM:VALue <Value>

Sets the value of the selected list item.

Parameters:

<Value>   float

Range: -1e+11 to 1e+11

Example: See Example"Using list and waveform type IPM profiles"
on page 652

Manual operation: See "List" on page 230

IPM:LIST:ITEM:REPetition <Repetition>

Sets the number of times a list item is repeated.

Parameters:

<Repetition>   float

Range: 1 to 1e+09

Example: See Using list and waveform type IPM profiles

Manual operation: See "List" on page 230

IPM:LIST:ITEM:TIME <Time>

Sets how long a list item is repeated.

Parameters:

<Time>   float

                          Range: 0 to 1e+09

Example: IPM:TYPE LIST

                          IPM:LIST:ITEM:SELect 3

                          IPM:LIST:ITEM:BASE TIME

                          IPM:LIST:ITEM:VALue 0.0006

                          IPM:LIST:ITEM:TIME 0.1

Manual operation: See "List" on page 230

IPM:LIST:FIRing:ENABle <Enable>

Enables using firing order for list-based IPM profiles.

Parameters:

<Enable>   ON | OFF | 1 | 0

Example: See IPM:LIST:FIRing:SEQuence on page 659.

Manual operation: See "List" on page 230

IPM:LIST:FIRing:SEQuence <Sequence>

Sets the firing order sequence.

Parameters:

<Sequence>   string

Example: IPM:LIST:FIRing:ENABle 1

                          IPM:LIST:FIRing:SEQuence "1x1,2x2,3x3,4x4"

Manual operation: See "List" on page 230

IPM:LIST:LOAD <Load>

Loads an IPM profile form an ASCII file.

Parameters:

<Load>   string

                          File path, file name, and file extension

Example: See Using list and waveform type IPM profiles

Manual operation: See "Edit List" on page 240

IPM:LIST:SAVE <Save>

Stores the IPM profile as a file.

Parameters:

<Save>   string

File path incl. file name and extension.

Example: See Using list and waveform type IPM profiles

Manual operation: See "Edit List" on page 240

IPM:RANDom:DISTribution <Distribution>

Sets the distribution function.

Parameters:

<Distribution>   UNIForm | NORMal | U

Example: See Example"Using PRI profiles of random type" on page 653

Manual operation: See "Random" on page 237

IPM:RANDom:NORMal:LIMit <Limit>

Sets the limit parameter of the normal distribution function.

Parameters:

<Limit>   float

Range: -1e+09 to 1e+09

Example: See `IPM:RANDom:NORMal:MEAN` on page 660

Manual operation: See "Random" on page 237

IPM:RANDom:NORMal:MEAN <Mean>

Sets the mean parameter of the normal distribution function.

Parameters:

<Mean>   float

Range: -1e+09 to 1e+09

Example: `IPM:RANDom:DISTribution NORMal`
`IPM:RANDom:NORMal:MEAN 0`
`IPM:RANDom:NORMal:STD 1`
`IPM:RANDom:NORMal:LIMit 3`

Manual operation: See "Random" on page 237

IPM:RANDom:NORMal:STD <Std>

Sets the standard deviation parameter of the normal distribution function.

Parameters:

<Std>   float

Range: 1e-09 to 1e+06

Example: See `IPM:RANDom:NORMal:MEAN` on page 660

Manual operation: See "Random" on page 237

IPM:RANDom:U:CENTer <Center>

Sets the center parameter of the U distribution.

Parameters:

<Center>   float

Range: -1e+09 to 1e+09

Manual operation: See "Random" on page 237

IPM:RANDom:U:RANGe <Range>

Sets the range parameter of the U distribution.

Parameters:

<Range>   float

Range: 1e-09 to 1e+09

Manual operation: See "Random" on page 237

IPM:RANDom:UNIForm:MINimum <Minimum>
IPM:RANDom:UNIForm:MAXimum <Maximum>

Sets the range of the uniform distribution function.

Parameters:

<Maximum>   float

Range: -1e+09 to 1e+09

Example: See Example"Using PRI profiles of random type" on page 653

Manual operation: See "Random" on page 237

IPM:RANDom:UNIForm:STEP <Step>

Sets the granularity of the uniform distribution function.

Parameters:

<Step>   float

Range: 1e-09 to 1e+09

Example: See Example"Using PRI profiles of random type" on page 653

Manual operation: See "Random" on page 237

IPM:RLISt:BASE <Base>

Sets the IPM profile base and defines how the increments repetition is defined.

Parameters:

<table>
<tr><td><Base></td><td>LENGth | TIME</td></tr>
<tr><td></td><td>LENGth
Increments are repeated several times, as set with the command <code>IPM:RLISt:BURSt</code>.</td></tr>
<tr><td></td><td>TIME
Increments are repeated for the defined time duration, as set with the command <code>IPM:RLISt:PERiod</code>.</td></tr>
<tr><td>Example:</td><td>See Example"Using PRI profiles of random type" on page 653.</td></tr>
<tr><td>Manual operation:</td><td>See "Random List" on page 235</td></tr>
</table>

IPM:RLISt:BURSt <Burst>

Defines how many times an increment is repeated.

Parameters:

<table>
<tr><td><Burst></td><td>float</td></tr>
<tr><td></td><td>Range: 1   to   8192</td></tr>
<tr><td>Example:</td><td>See Example"Using PRI profiles of random type" on page 653.</td></tr>
<tr><td>Manual operation:</td><td>See "Random List" on page 235</td></tr>
</table>

IPM:RLISt:PERiod <Period>

Sets how long an increment is repeated.

Parameters:

<table>
<tr><td><Period></td><td>float</td></tr>
<tr><td></td><td>Range: 1e-09   to   1e+09</td></tr>
<tr><td>Example:</td><td><code>IPM:RLISt:BASE TIME</code>
<code>IPM:RLISt:PERiod 0.01</code></td></tr>
<tr><td>Manual operation:</td><td>See "Random List" on page 235</td></tr>
</table>

IPM:RLISt:REUSe <Reuse>

If disabled, each value is used only once.

Parameters:

<table>
<tr><td><Reuse></td><td>ON | OFF | 1 | 0</td></tr>
<tr><td>Example:</td><td><code>IPM:RLISt:REUSe 0</code></td></tr>
<tr><td>Manual operation:</td><td>See "Random List" on page 235</td></tr>
</table>

IPM:SHAPe:BASE <Base>

Defines the way the list items are processed.

Parameters:

<Base>   PULSe | TIME

Example: See Using the interpolated shape IPM profile

Manual operation: See "Interpolated Shape" on page 233

IPM:SHAPe:COUNt <Count>

Sets the number of pulses for that the data from the list is used.

Parameters:

<Count>   integer

                                        Range: 1 to 1e+09

Example: See Using the interpolated shape IPM profile

Manual operation: See "Interpolated Shape" on page 233

IPM:SHAPe:INTerpol <Interpol>

Enables a linear transition between the increments.

Parameters:

<Interpol>   LINear | NONE

Example: See Using the interpolated shape IPM profile

Manual operation: See "Interpolated Shape" on page 233

IPM:SHAPe:PERiod <Period>

Sets the period of time over that the list items are equally distributed.

Parameters:

<Period>   float

                                        Range: 1e-09 to 1e+09

Example: See Using the interpolated shape IPM profile

Manual operation: See "Interpolated Shape" on page 233

IPM:STEP:BASE <Base>

Sets the IPM profile base and defines how the increments repetition is defined.

Parameters:

<Base>   LENGth | TIME

LENGth

Steps are repeated several times, as set with the command `IPM:STEP:BURSt`.

TIME

Steps are repeated for the defined time duration, as set with the command `IPM:STEP:PERiod`.

Example: See Example"Using the step IPM profile" on page 653

Manual operation: See "Steps" on page 229

IPM:STEP:BURSt <Burst>

Sets the number of times an increment is repeated.

Parameters:

<Burst>   float

                           Range: 1 to 1000

Example: See Example"Using the step IPM profile" on page 653

Manual operation: See "Steps" on page 229

IPM:STEP:PERiod <Period>

Sets how long an increment is repeated.

Parameters:

<Period>   float

                           Range: 1e-09 to 1e+09

Example: IPM:STEP:BASE TIME
                           IPM:STEP:PERiod 0.01

Manual operation: See "Steps" on page 229

IPM:STEP:INCRement <Increment>

Sets the step size.

Parameters:

<Increment>   float

                           Range: -1e+09 to 1e+09

Example: See Example"Using the step IPM profile" on page 653

Manual operation: See "Steps" on page 229

IPM:STEP:STARt <Start>

Sets the start value.

Parameters:

<Start>   float

                         Range: -1e+09 to 1e+09

Example: See Example"Using the step IPM profile" on page 653

Manual operation: See "Steps" on page 229

IPM:STEP:STEPs <Steps>

Sets the number of steps.

Parameters:

<Steps>   float

                         Range: 1 to 10000

Example: See Example"Using the step IPM profile" on page 653

Manual operation: See "Steps" on page 229

IPM:RSTep:MINimum <Minimum>
IPM:RSTep:MAXimum <Maximum>

Sets the value range.

Parameters:

<Maximum>   float

                         Range: 0 to 1e+11

Example: See Example"Using PRI profiles of random type" on page 653

Manual operation: See "Random Steps" on page 236

IPM:RSTep:STEP:MINimum <Minimum>
IPM:RSTep:STEP:MAXimum <Maximum>

Sets the step size range.

Parameters:

<Maximum>   float

                         Range: 0.1 to 0.5

Example: See Example"Using PRI profiles of random type" on page 653

Manual operation: See "Random Steps" on page 236

IPM:RSTep:PERiod <Period>

Sets the pattern length.

Parameters:

<Period>   float

                         Range: 0 to 4096

Example: See Example"Using PRI profiles of random type" on page 653

Manual operation: See "Random Steps" on page 236

IPM:WAVeform:TYPE <Type>

Sets the profile shape.

Parameters:

<Type>   RAMP | SINE | TRIangular

Example: See Using list and waveform type IPM profiles

Manual operation: See "Waveform" on page 232

IPM:WAVeform:BASE <Base>

Defines how the waveform period is defined, as a time duration or as a number of pulses.

Parameters:

<Base>   PULSe | TIME

Example: See Using list and waveform type IPM profiles

Manual operation: See "Waveform" on page 232

IPM:WAVeform:COUNt <Count>

Sets the waveform period as number of pulses.

Parameters:

<Count>   integer

                       Range: 1 to 1e+09

Example: See Using list and waveform type IPM profiles

Manual operation: See "Waveform" on page 232

IPM:WAVeform:OFFSet <Offset>

Shifts the profile by the selected offset.

Parameters:

<Offset>   float

                       Range: -1e+09 to 1e+09

Example: See Using list and waveform type IPM profiles

Manual operation: See "Waveform" on page 232

IPM:WAVeform:PHASe <Phase>

Enables a phase offset to change the start phase of the sine wave.

Parameters:

<Phase>   float

Range: -1e+09 to 1e+09
Default unit: sec

Example: See Using list and waveform type IPM profiles

Manual operation: See "Waveform" on page 232

IPM:WAVeform:PERiod <Period>

Sets the waveform period.

Parameters:

<Period>   float

Range: 1e-09 to 1e+09
Default unit: sec

Example: See Using list and waveform type IPM profiles

Manual operation: See "Waveform" on page 232

IPM:WAVeform:PKPK <Pkpk>

Sets the value range of the linear ramp profile or the period of the sine profile.

Parameters:

<Pkpk>   float

Range: 1e-09 to 1e+09
Default unit: sec

Example: See Using list and waveform type IPM profiles

Manual operation: See "Waveform" on page 232

IPM:BINomial:VAL1 <Val1>
IPM:BINomial:VAL2 <Val2>

Sets the values of the binomial distribution function.

Parameters:

<Val2>   float

Range: -1e+09 to 1e+09

Example:
```
IPM:TYPE BINOMIAL
IPM:BINomial:VAL1 100
IPM:BINomial:PVAL1 80
IPM:BINomial:VAL2 0
```

Manual operation: See "Binomial" on page 239

IPM:BINomial:PVAL1 <Pval1>

Sets the probability of occurrence of value 1 in the binomial distribution function.

Parameters:

<Pval1>	float
	Range: 0 to 100
	Default unit: PCT

Example:	See `IPM:BINomial:VAL2` on page 667
Manual operation:	See "Binomial" on page 239

DIALog:IPMPlot:VIEW <View>

Defines what kind of information is represented in the IPM profile diagram.

Setting parameters:

<View>	TIMeseries | HISTogram
	TIMeseries
	Visualization of the profile variation over time
	HISTogram
	Statistical representation of the relative frequency density

Example:	See Example"Using PRI profiles of random type" on page 653.
Usage:	Setting only
Manual operation:	See "2D" on page 228

DIALog:IPMPlot:SAMPles <Samples>

Sets the number of values to be displayed in the preview diagram of the IPM profile.

Setting parameters:

<Samples>	float

Example:	See Example"Using PRI profiles of random type" on page 653.
Usage:	Setting only
Manual operation:	See "2D" on page 228

25.15 Import interface commands

Example: PDW import (unmodulated pulses)

```
IMPort:PDW:FILE:PDW:LOAD "C:/_PS_files/PDW import/PDW01_unmodulated.txt"
IMPort:PDW:FILE:PDW?
// C:/_PS_files/PDW import/PDW01_unmodulated.txt
IMPort:PDW:FILE:TEMPlate:LOAD "C:/_PS_files/PDW import/PDW01_unmodulated.pdwt"
IMPort:PDW:FILE:TEMPlate?
// C:/_PS_files/PDW import/PDW01_unmodulated.pdwt

IMPort:PDW:NORM 1
IMPort:PDW:EXECute
IMPort:PDW:STATus?
// 1
IMPort:PDW:DATA:SEL 1
IMPort:PDW:DATA:TOA?
// 0
IMPort:PDW:DATA:WIDTh?
// 3e-05
IMPort:PDW:DATA:FREQuency?
// 3e+09
IMPort:PDW:DATA:OFFSet?
// 0
IMPort:PDW:DATA:PHASe?
// 0
IMPort:PDW:DATA:LEVel?
// 0
IMPort:PDW:DATA:MOP?
// CW
```

IMPort:PDW:FILE:PDW <Pdw>

Sets or queries the name of the used PDW list file.

Parameters:

<Pdw>   absolute file path and filename, incl. file extension

Example: See Example"PDW import (unmodulated pulses)" on page 669.

Manual operation: See "PDW File" on page 512

IMPort:PDW:FILE:TEMPlate <Template>

Sets or queries the name of the used import template file.

Parameters:

<Template>   absolute file path and filename, incl. file extension

Example: See Example"PDW import (unmodulated pulses)" on page 669.

Manual operation:　　　See "Template" on page 513

IMPort:PDW:FILE:PDW:LOAD
IMPort:PDW:FILE:TEMPlate:LOAD

Loads the selected file.

Example:　　　　　　　See Example"PDW import (unmodulated pulses)" on page 669.

Usage:　　　　　　　　Event

Manual operation:　　　See "Select import template file, Create new import template, Save/Save as" on page 513

IMPort:PDW:FILE:PDW:SAVE
IMPort:PDW:FILE:TEMPlate:SAVE

Stores the selected file.

Example:　　　　　　　See Example"PDW import (unmodulated pulses)" on page 669.

Usage:　　　　　　　　Event

Manual operation:　　　See "Select import template file, Create new import template, Save/Save as" on page 513

IMPort:PDW:NORM <Norm>

Normalizes the TOA (time of arrival) of the first pulse to 0. Subsequent TOAs are relative.

Parameters:

<Norm>　　　　　　　　ON | OFF | 1 | 0

Example:　　　　　　　See Example"PDW import (unmodulated pulses)" on page 669.

Manual operation:　　　See "Normalize TOA" on page 513

IMPort:PDW:EXECute

Starts parsing the PDW list file.

Example:　　　　　　　See Example"PDW import (unmodulated pulses)" on page 669.

Usage:　　　　　　　　Event

Manual operation:　　　See "Process PDW File" on page 513

IMPort:PDW:STATus?

Queries the parsing status.

Return values:

\<Status\>	ON | OFF | 1 | 0
	1
	Import completed
Example:	See Example"PDW import (unmodulated pulses)" on page 669.
Usage:	Query only
Manual operation:	See "Import Status" on page 513

IMPort:PDW:DATA:SEL \<Sel\>

Selects the pulse for that the further queries apply.

Parameters:

\<Sel\>	float
	Range: 1 to max
Example:	See Example"PDW import (unmodulated pulses)" on page 669.
Manual operation:	See "Imported Data" on page 513

IMPort:PDW:DATA:MOP?

Queries the used modulation on pulse (MOP).

Use the corresponding command to query further pulse and modulation parameter for the respective MOP.

Return values:

\<Mop\>	CW | AM | FM | ASK | FSK | PSK | LFM | NLFM | TFM | BKR2a | BKR2b | BKR3 | BKR4a | BKR4b | BKR5 | BKR7 | BKR11 | BKR13 | CPH | PLFM
Example:	See Example"PDW import (unmodulated pulses)" on page 669.
Usage:	Query only
Manual operation:	See "Imported Data" on page 513

IMPort:PDW:DATA:AM:DEPTh?
IMPort:PDW:DATA:AM:MODFreq?
IMPort:PDW:DATA:ASK:CHIPcount?
IMPort:PDW:DATA:ASK:RATE?
IMPort:PDW:DATA:ASK:STATes?
IMPort:PDW:DATA:ASK:STEP?
IMPort:PDW:DATA:CPH:CHIPcount?
IMPort:PDW:DATA:CPH:VALues \<Values\>
IMPort:PDW:DATA:FM:DEViation?
IMPort:PDW:DATA:FM:MODFreq?
IMPort:PDW:DATA:FREQuency?
IMPort:PDW:DATA:FSK:CHIPcount?

IMPort:PDW:DATA:FSK:RATE?
IMPort:PDW:DATA:FSK:STATes?
IMPort:PDW:DATA:FSK:STEP?
IMPort:PDW:DATA:LEVel?
IMPort:PDW:DATA:OFFSet?
IMPort:PDW:DATA:PHASe?
IMPort:PDW:DATA:PLFM:VALues <Values>
IMPort:PDW:DATA:PSK:CHIPcount?
IMPort:PDW:DATA:PSK:RATE?
IMPort:PDW:DATA:PSK:STATes?
IMPort:PDW:DATA:PSK:STEP?
IMPort:PDW:DATA:LFM:RATE?
IMPort:PDW:DATA:NLFM:CUBic?
IMPort:PDW:DATA:NLFM:LINear?
IMPort:PDW:DATA:NLFM:QUADratic?
IMPort:PDW:DATA:TOA?
IMPort:PDW:DATA:WIDTh?

Queries the pulse parameter.

Return values:

<Width>	float

Example:	See Example"PDW import (unmodulated pulses)" on page 669.
Usage:	Query only
Manual operation:	See "Imported Data" on page 513

IMPort:PDW:DATA:ASK:PATTern?
IMPort:PDW:DATA:FSK:PATTern?
IMPort:PDW:DATA:PSK:PATTern?

Queries the pulse parameter.

Return values:

<Pattern>	string

Example:	See Example"PDW import (unmodulated pulses)" on page 669.
Usage:	Query only

IMPort:PDW:STORe

Stores the imported PDW list file as waveform element in the repository.

Example:	See Example"PDW import (unmodulated pulses)" on page 669.
Usage:	Event
Manual operation:	See "Import into Repository" on page 514

IMPort:VIEW:COUNt <Count>

Sets the entries per page to be displayed.

Parameters:

<Count>   50 | 100 | 500 | 1000 | 5000 | 10000 | 50000 | 100000

Manual operation: See "View > PDW Data View" on page 509

IMPort:VIEW:TIME:STARt <Start>

Sets the start line displayed on the page.

Parameters:

<Start>   float

Manual operation: See "View > PDW Data View" on page 509

IMPort:VIEW:COUNt <Count>

Sets the entries per page to be displayed.

Parameters:

<Count>   50 | 100 | 500 | 1000 | 5000 | 10000 | 50000 | 100000

Manual operation: See "View > PDW Data View" on page 509

IMPort:VIEW:MOVE:BACKwards
IMPort:VIEW:MOVE:FORWard
IMPort:VIEW:MOVE:STARt
IMPort:VIEW:MOVE:END

Goes to the first/next/previous/last page.

Usage: Event

Manual operation: See "View > PDW Data View" on page 509

25.16 Marker commands

Marker commands enable or up to four markers in any combination at once.

Markers signals are binary signals. Each marker signal is represented by a single bit within a marker byte. The Table 25-1 explains the assailment.

Table 25-1: Setting parameter as function of the marker states

Marker	Marker byte	Value of the settings parameter in the remote commands
M1	0000 0001	0x01
M2	0000 0010	0x02

Marker	Marker byte	Value of the settings parameter in the remote commands
M3	0000 0100	0x04
M4	0000 1000	0x08

In the following examples, we assume that a pulse-base sequence "Test Sequence" and the required Pulses have been created.

See also:

- Example"Handling items" on page 581
- Example"Creating an unmodulated pulse" on page 693

Example: Defining pulse and sequence markers and enabling the global markers

```
SCPI
PULSe:SELect "P1"
// activate Marker 1 for the whole pulse on time (rise, fall, with time)
PULSe:MARKer:RISE 1
PULSe:MARKer:FALL 1
PULSe:MARKer:WIDTh 1
// enable Marker 2 on restart
PULSe:MARKer:AUTO 2
// enable Marker 3 als gate marker
PULSe:MARKer:GATE 4

// enable marker 3 as pre and marker 4 as post markers
// set pre and post marker duration to 1 us
// PULSe:MARKer:PRE 4
// PULSe:MARKer:POST 8
// PULSe:TIME:PRE 1.E-06
// PULSe:TIME:POST 1.E-06

SEQuence:SELect "Test Sequence"
SEQuence:ITEM:SELect 1
SEQuence:ITEM:MARKer:FIRSt 2
SEQuence:ITEM:MARKer:LAST 4
SEQuence:ITEM:MARKer:ALL 1

// enable Marker 4 to mark the fifth loop run, i.e.
// enable the condition $loop_i = 5 for Marker 4
// the used loop variable $loop must exist
SEQuence:ITEM:MARKer:CONDition 8
SEQuence:ITEM:MARKer:CONDition:VARiable "$loop_i"
SEQuence:ITEM:MARKer:CONDition:TYPE EQUAL
SEQuence:ITEM:MARKer:CONDition:VALue "5"

SCENario:OUTPut:MARKer:ENABle 1
SCENario:OUTPut:MARKer:FLAGs 14
SCENario:OUTPut:MARKer:SCENario:ENABle 0
// SCENario:OUTPut:MARKer:SCENario:DURation 0.01
```

Example: Defining emitter markers

```
SCPI
SCENario:CEMit:SELect 3
SCENario:CEMit:MARKer:FORCe 1
SCENario:CEMit:MARKer:RISE 1
SCENario:CEMit:MARKer:FALL 3
SCENario:CEMit:MARKer:WIDTh 2
SCENario:CEMit:MARKer:AUTO 4
SCENario:CEMit:MARKer:GATE 3
SCENario:CEMit:MARKer:PRE 1
SCENario:CEMit:MARKer:TIME:PRE 0.E+00
SCENario:CEMit:MARKer:POST 1
SCENario:CEMit:MARKer:TIME:PRE 0.E+00
SCENario:CEMit:MARKer:TIME:POST 0.E+00
```

PULSe:MARKer:PRE <Pre>
PULSe:MARKer:POST <Post>
PULSe:MARKer:RISE <Rise>
PULSe:MARKer:WIDTh <Width>
PULSe:MARKer:FALL <Fall>
PULSe:MARKer:GATE <Gate>
PULSe:MARKer:AUTO <Auto>

Enables up to four restart markers.

Parameters:

<Auto>	float
	See Table 25-1.
	Range: 0 to 65535

Example:	See Example"Defining pulse and sequence markers and enabling the global markers" on page 676.

Manual operation:	See "Pulse Markers" on page 534

PULSe:TIME:PRE <Pre>

Sets the marker's duration.

Value different than zero shifts the start of the pulse rising edge and the entire pulse.

Parameters:

<Pre>	float
	Range: 0 to 3600

Example:	See Example"Defining pulse and sequence markers and enabling the global markers" on page 676.

Manual operation:	See "Pulse Markers" on page 534

PULSe:TIME:POST <Post>

Sets the marker's duration.

Parameters:

<Post>   float

                          Range: 0 to 3600

Example: See Example"Defining pulse and sequence markers and ena-
                          bling the global markers" on page 676.

Manual operation: See "Pulse Markers" on page 534

SEQuence:ITEM:MARKer:CONDition <Condition>
SEQuence:ITEM:MARKer:FIRSt <First>
SEQuence:ITEM:MARKer:LAST <Last>
SEQuence:ITEM:MARKer:ALL <All>

Enables up to four markers of the corresponding type.

Parameters:

<All>   float

                          See Table 25-1.

                          Range: 0 to 65535

Example: See Example"Defining pulse and sequence markers and ena-
                          bling the global markers" on page 676.

Manual operation: See "Sequence Markers" on page 536

SEQuence:ITEM:MARKer:CONDition:TYPE <Type>

Sets the sign in the logical condition.

Parameters:

<Type>   SMALler | GREater | EQUal | NOTequal

Example: See Example"Defining pulse and sequence markers and ena-
                          bling the global markers" on page 676

Manual operation: See "Sequence Markers" on page 536

SEQuence:ITEM:MARKer:CONDition:VARiable <Variable>

Defines the value that is compared with the fixed values set with the command
`SEQuence:ITEM:MARKer:CONDition:VALue` on page 680.

Parameters:

<Variable>   string

Example: See Example"Defining pulse and sequence markers and ena-
                          bling the global markers" on page 676

Manual operation: See "Sequence Markers" on page 536

SEQuence:ITEM:MARKer:CONDition:VALue <Value>

Sets the numerical value used with the comparison.

Parameters:

<Value>   string

Example: See Example"Defining pulse and sequence markers and ena-
                      bling the global markers" on page 676

Manual operation: See "Sequence Markers" on page 536

SCENario:OUTPut:MARKer:ENABle <Enable>

Enables that markers are considered by the generation of the output waveform file.

Parameters:

<Enable>   ON | OFF | 1 | 0

Example: See Example"Defining pulse and sequence markers and ena-
                      bling the global markers" on page 676

Manual operation: See "Marker Signals" on page 480

SCENario:OUTPut:MARKer:FLAGs <Flags>

Enables up to four markers.

Parameters:

<Flags>   int

                      Binary value, where:
                      M1 = 1
                      M1 = 2
                      M1 = 4
                      M1 = 8

                      Range: 0 to 15
                      *RST: 15

Example: See Example"Defining pulse and sequence markers and ena-
                      bling the global markers" on page 676

Manual operation: See "Marker Signals" on page 480

SCENario:OUTPut:MARKer:SCENario:ENABle <Enable>

Enables an additional marker, that is held high from the scenario start until the dura-
tion, selected with the command `SCENario:OUTPut:MARKer:SCENario:
DURation`.

Parameters:

<Enable>   ON | OFF | 1 | 0

Example: See Example"Defining pulse and sequence markers and enabling the global markers" on page 676

Manual operation: See "Marker Signals" on page 480

SCENario:OUTPut:MARKer:SCENario:DURation <Duration>

Sets the duration of the scenario marker.

Parameters:

<Duration>   float

Range: 0 to 1
Increment: 1 ms
Default unit: sec

Example: See Example"Defining pulse and sequence markers and enabling the global markers" on page 676

Manual operation: See "Marker Signals" on page 480

SCENario:OUTPut:MARKer:PMODulator:ENABle <Enable>

Activates the generation and use of the dedicated marker signal M3 that marks the pulse width.

Parameters:

<Enable>   ON | OFF | 1 | 0

Firmware/software: See `SETup:PMOD:ENABle` on page 650.

Manual operation: See "Marker Signals" on page 480

SCENario:CEMit:MARKer:FORCe <Force>
SCENario:DF:MARKer:FORCe <Force>
SCENario:LOCalized:MARKer:FORCe <Force>

Determines how the marker is handled.

Parameters:

<Force>   ON | OFF | 1 | 0

ON | 1
Forces the selected marker type for every pulse of the selected emitter

OFF | 0
Leaves the marker unchanged, as defined in the pulses and sequences of this emitter.

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

SCENario:CEMit:MARKer:RISE <Rise>
SCENario:DF:MARKer:RISE <Rise>
SCENario:LOCalized:MARKer:RISE <Rise>

Enables marker for rise time.

Parameters:

<Rise>   float

                                    Binary value, where:
                                    M1 = 1
                                    M1 = 2
                                    M1 = 4
                                    M1 = 8

                                    Range: 0 to 15

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

SCENario:CEMit:MARKer:WIDTh <Width>
SCENario:DF:MARKer:WIDTh <Width>
SCENario:LOCalized:MARKer:WIDTh <Width>

Sets marker for the pulse width.

Parameters:

<Width>   float

                                    Binary value, where:
                                    M1 = 1
                                    M1 = 2
                                    M1 = 4
                                    M1 = 8

                                    Range: 0 to 15

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

SCENario:CEMit:MARKer:FALL <Fall>
SCENario:DF:MARKer:FALL <Fall>
SCENario:LOCalized:MARKer:FALL <Fall>

Enables the marker for fall time.

Parameters:

<Fall>   float

                                    Binary value, where:
                                    M1 = 1
                                    M1 = 2
                                    M1 = 4
                                    M1 = 8

                                    Range: 0 to 15

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

SCENario:CEMit:MARKer:AUTO <Auto>
SCENario:DF:MARKer:AUTO <Auto>
SCENario:LOCalized:MARKer:AUTO <Auto>

Enables the marker for restart.

Parameters:

<Auto>   float

Binary value, where:
M1 = 1
M1 = 2
M1 = 4
M1 = 8
Range: 0 to 15

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

SCENario:CEMit:MARKer:GATE <Gate>
SCENario:DF:MARKer:GATE <Gate>
SCENario:LOCalized:MARKer:GATE <Gate>

Enables marker for gate.

Parameters:

<Gate>   float

Binary value, where:
M1 = 1
M1 = 2
M1 = 4
M1 = 8
Range: 0 to 15

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

SCENario:CEMit:MARKer:PRE <Pre>
SCENario:DF:MARKer:PRE <Pre>
SCENario:LOCalized:MARKer:PRE <Pre>

Enables marker for pre time.

Parameters:

<Pre>   float

Binary value, where:
M1 = 1

M1 = 2
M1 = 4
M1 = 8

Range: 0 to 15

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

SCENario:CEMit:MARKer:TIME:PRE <Pre>
SCENario:DF:MARKer:TIME:PRE <Pre>
SCENario:LOCalized:MARKer:TIME:PRE <Pre>

Specifies pre marker time.

Parameters:

<Pre>   float

Range: 0 to 3600

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

SCENario:CEMit:MARKer:POST <Post>
SCENario:DF:MARKer:POST <Post>
SCENario:LOCalized:MARKer:POST <Post>

Enables marker for post time.

Parameters:

<Post>   float

Binary value, where:
M1 = 1
M1 = 2
M1 = 4
M1 = 8

Range: 0 to 15

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

SCENario:CEMit:MARKer:TIME:POST <Post>
SCENario:DF:MARKer:TIME:POST <Post>
SCENario:LOCalized:MARKer:TIME:POST <Post>

Specifies post marker time.

Parameters:

<Post>   float

Range: 0 to 3600

Example: See Example"Defining emitter markers" on page 677.

Manual operation: See "Emitter Marker" on page 441

25.17 Plugin and reporting commands

Example: Loading plugins in the repository

```
PLUGin:CREate "My_QAM_PlugIn"
PLUGin:LOAD "C:\_PS_files\QAM.dll"
PLUGin:MODule:AUTHor?
// Rohde&Schwarz
PLUGin:MODule:NAME?
// QAM
PLUGin:MODule:TYPE?
// MOP
PLUGin:MODule:VERSion?
// 2.0.0
PLUGin:MODule:COMMent?
// This plugin creates a QAM modulation.
// Parameters:
// Rate [Hz]     symbol rate
PLUGin:MODule:DATA?
// 0
```

Example: Using a plugin as a modulation source

```
SCPI
PULSe:CREate "Custom QAM"
PULSe:MOP:ENABle 1
PULSe:MOP:TYPE PLUGin
PLUGin:CATalog?
// "My_QAM_PlugIn","CustomIPM"
PULSe:MOP:PLUGin:NAME "My_QAM_PlugIn"
PULSe:MOP:PLUGin:VARiable:CATalog?
// "Rate","Type"
PULSe:MOP:PLUGin:VARiable:SELect "Rate"
PULSe:MOP:PLUGin:VARiable:VALue?
// 100000
PULSe:MOP:PLUGin:VARiable:SELect "Type"
PULSe:MOP:PLUGin:VARiable:VALue?
// 2
// selected is the third value from the list of variables
DSRC:CATalog?
// "DataSource"
PULSe:MOP:DATA:DSRC "DataSource"
PULSe:MOP:DATA:CODing NONE
PULSe:MOP:FILTer:TYPE COS
PULSe:MOP:FILTer:LENGth 128
PULSe:MOP:FILTer:BT 0.3
PULSe:MOP:FILTer:BWIDth 1000
```

Example: Using a plugin as an IPM profile

```
SCPI
IPM:CREate "Custom IPM"
IPM:UNIT PERCent
IPM:TYPE PLUGin
PLUGin:CATalog?
// "My_QAM_PlugIn","CustomIPM"
IPM:PLUGin:NAME "CustomIPM"
IPM:PLUGin:VARiable:CATalog?
// "$step","$pw"
IPM:PLUGin:VARiable:SELect "$pw"
IPM:PLUGin:VARiable:VALue 0.00001
```

Example: Generating reports

```
SCPI
SCENario:PDW:ENABle 1
SCENario:PDW:PATH "C:\_My_Report_Files"
SCENario:PDW:TYPE PLUGin
PLUGin:CATalog?
// "My_QAM_PlugIn","CustomIPM","Report PDW","Reporting DFS"
SCENario:PDW:PLUGin:NAME "Report PDW"
SCENario:PDW:PLUGin:VARiable:CATalog?
// "File","Header"
SCENario:PDW:PLUGin:VARiable:SELect "File"
SCENario:PDW:PLUGin:VARiable:VALue?

// "pdw_out.txt"
```

PLUGin:LOAD <Load>

Loads the selected DLL file, see also Chapter C, "Plug-in programming API", on page 863.

Setting parameters:

<Load>	string
	File path incl. file name and extension
Example:	See Example"Loading plugins in the repository" on page 685
Usage:	Setting only

PLUGin:MODule:AUTHor?
PLUGin:MODule:COMMent?
PLUGin:MODule:VERSion?
PLUGin:MODule:TYPE?

Queries information on the loaded file.

The query returns information as specified in the description of the corresponding function in Chapter C, "Plug-in programming API", on page 863.

The following are the possible values for the type query.

Return values:

<Type>	REPort	IPM	MOP
	MOP
	Plugin for MOP
	IPM
	Plugin for IPM
	REPort
	Plugin for reports created during the waveform generation
Example:	See Example"Loading plugins in the repository" on page 685
Usage:	Query only

PLUGin:MODule:DATA?

Queries whether the plugin requires data from a data source.

Return values:

<Data>	0	1
	0
	Data source is not required
	1
	Data source is required
	*RST: 0
Example:	See Example"Loading plugins in the repository" on page 685
Usage:	Query only

ANTenna:MODel:PLUGin:VARiable:CATalog?
SCENario:PDW:PLUGin:VARiable:CATalog?

IPM:PLUGin:VARiable:CATalog?
PULSe:MOP:PLUGin:VARiable:CATalog?

Queries the variables used in the plugin.

Return values:

<Catalog>   string

Example: See Using a plugin as a modulation source

Usage: Query only

Manual operation: See "Plug-in Variables" on page 183

SCENario:PDW:PLUGin:VARiable:SELect <Select>
IPM:PLUGin:VARiable:SELect <Select>
PULSe:MOP:PLUGin:VARiable:SELect <Select>

Selects a plugin variable.

Parameters:

<Select>   string

Example: See Using a plugin as a modulation source

Manual operation: See "Plug-in Variables" on page 183

SCENario:PDW:PLUGin:VARiable:VALue <Value>
IPM:PLUGin:VARiable:VALue <Value>
PULSe:MOP:PLUGin:VARiable:VALue <Value>

Sets the values of the selected variable.

Parameters:

<Value>   string

Example: See Using a plugin as a modulation source

Manual operation: See "Plug-in Variables" on page 183

PULSe:MOP:PLUGin:VARiable:SELect:ID <Id>
SCENario:PDW:PLUGin:VARiable:SELect:ID <Id>

Selects a plugin variable ID.

Parameters:

<Id>   float
                  *RST: 0

PULSe:MOP:PLUGin:VARiable:RESet
SCENario:PDW:PLUGin:VARiable:RESet

Resets the variable values to the defaults.

Usage: Event

SCENario:PDW:ENABle <Enable>

Enables generation of Pulse Descripter Word (PDW) reports.

Parameters:

<Enable>   ON | OFF | 1 | 0

Example: See Example"Generating reports" on page 687

Manual operation: See "Reporting Enable" on page 544

SCENario:PDW:PATH <Path>

Sets the target directory in that the generated report files are stored.

Parameters:

<Path>   string

Example: See Example"Generating reports" on page 687

Manual operation: See "Target Path, Set Path" on page 544

SCENario:PDW:TYPE <Type>

Sets the template used be the reporting function.

Parameters:

<Type>   DEFault | TEMPlate | PLUGin | AMMos

Manual operation: See "Type" on page 545

SCENario:PDW:HOST <Host>

Select a connected signal generator, with all options required for the scenario, to allow
reporting.

Parameters:

<Host>   string

Manual operation: See "Options Host" on page 545

SCENario:PDW:PLUGin:NAME <Name>

Selects and loads a reporting template. This template must exist in the "Plugin" library.

To query a list of available plugins, use the command `PLUGin:CATalog?`.

Parameters:

<Name>   string

Example: See Example"Generating reports" on page 687

Manual operation: See "Plugin and plugin variables" on page 547

SCENario:PDW:AMMos:PPDW <Ppdw>

If enabled, the format of the AMMOS file is set to PPDW. Otherwise PDW is assumed.

Parameters:

<Ppdw>	ON | OFF | 1 | 0

Example: `SCENario:PDW:TYPE AMMos`
`SCENario:PDW:AMMos:PPDW 1`
`SCENario:PDW:AMMos:FRAMe 200`
`SCENario:PDW:AMMos:UTIMe:ENABle 1`
`SCENario:PDW:AMMos:UTIMe:ISO`
`SCENario:PDW:AMMos:UTIMe:ISO "2017-05-24T23:46:00"`
`SCENario:PDW:AMMos:AZIMuth RX`

Manual operation: See "Format" on page 546

SCENario:PDW:AMMos:FRAMe <Frame>

Sets the frame length.

Parameters:

<Frame>	float
	Range: 50 to 500

Example: See `SCENario:PDW:AMMos:PPDW` on page 691.

Manual operation: See "Frame Length" on page 547

SCENario:PDW:AMMos:UTIMe:ENABle <Enable>

Defines how the report start time is set.

Parameters:

<Enable>	ON | OFF | 1 | 0
	0
	The reporting start time is time at that the scenario calculation starts.
	1
	The reporting starts at user-defined moment, set with the command `SCENario:PDW:AMMos:UTIMe:ISO`.

Example: See `SCENario:PDW:AMMos:PPDW` on page 691.

Manual operation: See "Start Time, User Set Value" on page 547

SCENario:PDW:AMMos:UTIMe:ISO <Iso>

Sets the reporting start time, if `SCENario:PDW:AMMos:UTIMe:ENABle1`.

Parameters:

<Iso>	"<YYYY>-<Month>-<DD>T<HH:MM:SS>"

Example: See `SCENario:PDW:AMMos:PPDW` on page 691.

Manual operation: See "Start Time, User Set Value" on page 547

SCENario:PDW:AMMos:AZIMuth <Azimuth>

For `SCENario:PDW:TYPE`AMMos, defines whether the angle of the Rx antenna or the bearing is reported.

Parameters:

<Azimuth>   RX | BEARing

Manual operation: See "Azimuth" on page 547

SCENario:PDW:TEMPlate <Template>

Edits the selected template.

Parameters:

<Template>   string

Example:

```
SCPI
SCENario:PDW:TEMPlate?
# Pulse Sequencer PDW Report
#
Date:       <ISODATE>
Repository: <REPOSITORY>
Scenario:   <SCENARIO>


          TOA |         RF |        PW |      PA | MF | MOP |      BW |
           ns |        GHz |        us |     dBm |    |     |     kHz |
-----------------------------------------------------------------------
.HDR
 <TOA> | <RF> | <PW> | <PA> |  <MF> | <MOP> | <BW> |
# END OF REPORT
.OPT <TOA:12.0f:-9>
.OPT <RF:9.6f:9>
.OPT <PW:9.3f:-6>
.OPT <PA:6.1f:0>
.OPT <BW:6.0f:3>
```

Manual operation: See "Edit Template" on page 545

25.18 Pulse commands

The following are examples on how to create and configure pulses in remote environment.

Example: Creating an unmodulated pulse

```
SCPI
PULSe:CREate "P_1"
PULSe:CATalog?
// "P_1"
PULSe:SELect "P_1"
PULSe:COMMent "PW=100us"
PULSe:TIME:REFerence FULL
PULSe:TIME:WIDTh 100 us
PULSe:TIME:RISE 10 us
PULSe:TIME:FALL 10 us
PULSe:TYPE:RISE LINear
PULSe:TYPE:FALL LINear
PULSe:MOP:ENABle 0
PULSe:MOP:EXCLude:ENABle 0
// remane the pulse
PULSe:NAME "P1"
```

Example: Creating a linear chirp pulse

```
SCPI
PULSe:CREate "LinearChirp"
PULSe:CUSTom 0
PULSe:TIME:REFerence POWer
PULSe:TIME:RISE 1e-05
PULSe:TYPE:RISE RCOSine
PULSe:TIME:WIDTh 0.0001
PULSe:TIME:FALL 1e-05
PULSe:TYPE:FALL RCOSine
PULSe:LEVel:ON 0
PULSe:LEVel:OFF 100
PULSe:LEVel:DROop 1
PULSe:RIPPle 0
PULSe:RIPPle:FREQuency 50000

PULSe:MOP:ENABle 1
PULSe:MOP:TYPE CHIRP
PULSe:MOP:CHIRp:TYPE UP
PULSe:MOP:CHIRp:DEViation 5e+06
PULSe:MOP:EXCLude:MODE LEVel
PULSe:MOP:EXCLude:LEVel:STARt 25
PULSe:MOP:EXCLude:LEVel:STOP 25

// PULSe:MOP:TYPE PWISechirp
// PULSe:MOP:PIECewise:ADD
// PULSe:MOP:PIECewise:SELect 1
// PULSe:MOP:PIECewise:DURation 25
// PULSe:MOP:PIECewise:FREQuency 200
// PULSe:MOP:PIECewise:OFFSet 0
// PULSe:MOP:PIECewise:ADD
// PULSe:MOP:PIECewise:SELect 2
// PULSe:MOP:PIECewise:DURation 25
// PULSe:MOP:PIECewise:FREQuency -400
// PULSe:MOP:PIECewise:OFFSet 10E6
```

Example: Creating an FM step pulse

```
SCPI
PULSe:CREate "FM_Step"
PULSe:TIME:REFerence FULL
PULSe:TIME:RISE 0
PULSe:TIME:WIDTh 0.000125
PULSe:TIME:FALL 0
PULSe:LEVel:ON 0
PULSe:LEVel:OFF 100
PULSe:MOP:EXCLude:MODE WIDTh

PULSe:MOP:ENABle 1
PULSe:MOP:COMMent "fstart = -50 MHz, fend = 50 MHz, Df = 25 MHz (N = 5)"
PULSe:MOP:TYPE FMSTep
PULSe:MOP:FMSTep:ADD
PULSe:MOP:FMSTep:SELect 1
PULSe:MOP:FMSTep:DURation 2.49999993684469e-05
PULSe:MOP:FMSTep:FREQuency -50000000
PULSe:MOP:FMSTep:ADD
PULSe:MOP:FMSTep:SELect 2
PULSe:MOP:FMSTep:DURation 2.49999993684469e-05
PULSe:MOP:FMSTep:FREQuency -25000000
...
```

Example: Creating a BPSK pulse

```
SCPI
PULSe:CREate "Test"
PULSe:MOP:ENABle 1
PULSe:MOP:TYPE BPSK
PULSe:MOP:BPSK:TYPE NORMal
PULSe:MOP:BPSK:SRATe:AUTO 0
PULSe:MOP:BPSK:SRATe 1e+06
PULSe:MOP:BPSK:PHASe 180
PULSe:MOP:BPSK:TTYPe COSine
PULSe:MOP:BPSK:TTIMe 5
```

Example: Creating pulses with custom envelope

```
SCPI
PULSe:SELect "Test"
PULSe:CUSTom 1
PULSe:ENVelope:MODE DATA
PULSe:ENVelope:DATA:LOAD "C:\_PS_files\mop shape custom.csv"
PULSe:ENVelope:DATA:UNIT VOLTAGE
PULSe:ENVelope:DATA:MULTiplier 1
PULSe:ENVelope:DATA:OFFSet 0
PULSe:ENVelope:DATA:ITEM:SELect 1
PULSe:ENVelope:DATA:ITEM:VALue?
// 0.2
PULSe:ENVelope:DATA:ITEM:SELect 7
PULSe:ENVelope:DATA:ITEM:VALue?
// 0.3
PULSe:ENVelope:DATA:ITEM:COUNt?
// 7

// PULSe:ENVelope:MODE EQUation
// PULSe:ENVelope:EQUation "# Sech Pulse
// 1 / ( cosh( ( 2 * (t-Tr-Ton/2) * ln( 1+sqrt(2) ) ) / Ton ) )^2"
```

PULSe:CUSTom <Custom>

Enables the use of a custom envelope function

Parameters:

<Custom>   ON | OFF | 1 | 0

Manual operation: See "Custom Envelope" on page 152

PULSe:ENVelope:DATA:ITEM:VALue <Value>

Sets the value of the selected item.

Parameters:

<Value>   float

                          Range: -1e+09 to 1e+09

Example: See Example"Creating pulses with custom envelope"
                          on page 696

Manual operation: See "Envelope definition as a function of imported data"
                          on page 160

PULSe:ENVelope:DATA:LOAD <Load>

Loads an envelope description form an ASCII file.

Parameters:

<Load>   string

                          File path, file name, and file extension

Example: See Example"Creating pulses with custom envelope"
                          on page 696

Manual operation: See "Envelope definition as a function of imported data"
                          on page 160

PULSe:ENVelope:DATA:UNIT <Unit>

Sets the data format.

Parameters:

<Unit>	VOLTage | WATTs | DB

Example:	See Example"Creating pulses with custom envelope" on page 696
Manual operation:	See "Envelope definition as a function of imported data" on page 160

PULSe:ENVelope:DATA:MULTiplier <Multiplier>

Sets a multiplier factor.

Parameters:

<Multiplier>	float
	Range: -100 to 100

Example:	See Example"Creating pulses with custom envelope" on page 696
Manual operation:	See "Envelope definition as a function of imported data" on page 160

PULSe:ENVelope:DATA:OFFSet <Offset>

Sets an offset for the envelope.

Parameters:

<Offset>	float
	Range: -100 to 100

Example:	See Example"Creating pulses with custom envelope" on page 696
Manual operation:	See "Envelope definition as a function of imported data" on page 160

PULSe:ENVelope:DATA:SAVE <Save>

Stores the custom envelope into file.

Parameters:

<Save>	string
	File path, file name, and file extension

Example:	`PULSe:ENVelope:DATA:LOAD "C:` `\_PS_files\mop_shape.csv"`

PULSe:ENVelope:EQUation <Equation>

Determines the envelope mathematically.

Parameters:

<Equation>   string

Manual operation: See "Envelope Definition as an Equation" on page 159

PULSe:ENVelope:MODE <Mode>

Selects the type of the custom envelope function.

Parameters:

<Mode>   DATA | EQUation

Example: See Example"Creating pulses with custom envelope"
on page 696

Manual operation: See "Custom Envelope" on page 152

PULSe:LEVel:DROop <Droop>

Sets the amplitude droop.

Parameters:

<Droop>   float

          Range: 0 to 50

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Attenuation" on page 156

PULSe:LEVel:OFF <Off>
PULSe:LEVel:ON <On>

Sets the power during the pulse on time or the pulse off time.

Parameters:

<On>   float

          Range: 0 to 100

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Attenuation" on page 156

PULSe:MOP:AM:FREQuency <Frequency>

Sets modulation frequency.

Parameters:

<Frequency>   float

          Range: 0.001 to 1e+09

Example:	See `PULSe:MOP:AM:TYPE` on page 701

Manual operation:	See "AM" on page 164

PULSe:MOP:AM:MDEPth <Mdepth>

Sets the modulation depth.

Parameters:

<Mdepth>	float
	Range: 0 to 100
	Default unit: percent

Example:	See `PULSe:MOP:AM:TYPE` on page 701

Manual operation:	See "AM" on page 164

PULSe:MOP:AM:TYPE <Type>

Selects the modulation type.

Parameters:

<Type>	STD	LSB	USB	SB

Example:	`PULSe:MOP:TYPE AM`
	`PULSe:MOP:AM:TYPE STD`
	`PULSe:MOP:AM:FREQuency 50000`
	`PULSe:MOP:AM:MDEPth 50`

Manual operation:	See "AM" on page 164

PULSe:MOP:AMSTep:DURation <Duration>
PULSe:MOP:FMSTep:DURation <Duration>

Sets the step time.

Parameters:

<Duration>	float
	Range: 0 to 3600
	Default unit: s

Example:	See Example"Creating an FM step pulse" on page 695

Manual operation:	See "FM Step" on page 167

PULSe:MOP:AMSTep:LEVel <Level>

Sets the step level.

Parameters:

<Level>	float
	Range: -100 to 0

Example: See Example"Creating an FM step pulse" on page 695

Manual operation: See "AM Step" on page 165

PULSe:MOP:FMSTep:FREQuency <Frequency>

Sets the step frequency.

Parameters:
<Frequency>   float

                       Range: -1e+09 to 1e+09

Example: See Example"Creating an FM step pulse" on page 695.

Manual operation: See "FM Step" on page 167

PULSe:MOP:MSK:INVert <Invert>
PULSe:MOP:FSK:INVert <Invert>
PULSe:MOP:ASK:INVert <Invert>

Inverts the modulation.

Parameters:
<Invert>   ON | OFF | 1 | 0

Example: `PULSe:MOP:TYPE ASK`
                       `PULSe:MOP:ASK:MDEPth 100`
                       `PULSe:MOP:ASK:INVert 1`
                       `PULSe:MOP:ASK:SRATe 1e+06`

Manual operation: See "ASK" on page 166

PULSe:MOP:ASK:MDEPth <Mdepth>

Sets the modulation depth.

Parameters:
<Mdepth>   float

                       Range: 0 to 100
                       Default unit: percent

Example: See `PULSe:MOP:ASK:INVert` on page 702

Manual operation: See "ASK" on page 166

PULSe:MOP:MSK:SRATe <Srate>
PULSe:MOP:ASK:SRATe <Srate>

Sets the symbol rate.

Parameters:
<Srate>   float

                       Range: 1 to 1e+09

Example: See `PULSe:MOP:ASK:INVert` on page 702

Manual operation: See "ASK" on page 166

PULSe:MOP:BARKer:CODE <Code>

Selects the code sequence.

Parameters:

<Code>   R2A | R2B | R3 | R4A | R4B | R5 | R7 | R11 | R13

Example: `PULSe:MOP:TYPE BARKer`
                     `PULSe:MOP:BARKer:CODE R13`
                     `PULSe:MOP:BARKer:TTIMe 10`
                     `PULSe:MOP:BARKer:BLANk 1`

Manual operation: See "Barker" on page 176

PULSe:MOP:BARKer:TTIMe <Ttime>

Sets the transition time.

Parameters:

<Ttime>   float

                       Range: 0 to 50
                       Default unit: percent

Example: See `PULSe:MOP:BARKer:CODE` on page 703

Manual operation: See "Barker" on page 176

PULSe:MOP:BARKer:BLANk <Blank>

Blanks out the signal during the transition time.

Parameters:

<Blank>   ON | OFF | 1 | 0

Example: See `PULSe:MOP:BARKer:CODE` on page 703

Manual operation: See "Barker" on page 176

PULSe:MOP:BPSK:PHASe <Phase>

Sets the phase.

Parameters:

<Phase>   float

                       Range: 0 to 180
                       Default unit: degree

Example: See Example"Creating a BPSK pulse" on page 695

Manual operation: See "BPSK" on page 179

PULSe:MOP:BPSK:SRATe <Srate>

Sets the symbol rate.

Parameters:

<Srate>   float

Range: 1 to 1e+09

Example: See Example"Creating a BPSK pulse" on page 695

Manual operation: See "BPSK" on page 179

PULSe:MOP:BPSK:SRATe:AUTO <Auto>

Enables automatic adjusting of the bits in the pulse width.

Parameters:

<Auto>   ON | OFF | 1 | 0

Example: See Example"Creating a BPSK pulse" on page 695

Manual operation: See "BPSK" on page 179

PULSe:MOP:BPSK:TTIMe <Ttime>

Sets the transition time.

Parameters:

<Ttime>   float

Range: 0 to 50
Default unit: percent

Example: See Example"Creating a BPSK pulse" on page 695

Manual operation: See "BPSK" on page 179

PULSe:MOP:BPSK:TTYPe <Ttype>

Selects the transition type.

Parameters:

<Ttype>   LINear | COSine

Example: See Example"Creating a BPSK pulse" on page 695

Manual operation: See "BPSK" on page 179

PULSe:MOP:BPSK:TYPE <Type>

Sets the modulation type.

Parameters:

<Type>   STANdard | CONStant

Example: See Example"Creating a BPSK pulse" on page 695

Manual operation: See "BPSK" on page 179

PULSe:MOP:CHIRp:TYPE <Type>

Selects the modulation type.

Parameters:

<Type>   UP | DOWN | SINE | TRIangular | PIECewise

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Linear Chirp" on page 170

PULSe:MOP:CHIRp:DEViation <Deviation>

Sets the modulation deviation.

Parameters:

<Deviation>   float

                    Range: 1 to 1e+09

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Linear Chirp" on page 170

PULSe:MOP:FSK:TYPE <Type>

Selects the FSK modulation type.

Parameters:

<Type>   FS2 | FS4 | FS8 | FS16 | FS32 | FS64

Example: `PULSe:MOP:TYPE FSK`
                    `PULSe:MOP:FSK:TYPE FS8`

Manual operation: See "FSK, 4FSK, 8FSK, 16FSK, 32FSK, 64FSK" on page 169

PULSe:MOP:PIECewise:DURation <Duration>

Set the length of the chirp interval as a percentage of the duration the MOP is applied on.

Parameters:

<Duration>   float

                    Range: 0 to 100

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Piecewise Linear Chirp" on page 171

PULSe:MOP:PIECewise:RATE <Rate>

Set the chirp rate.

Parameters:

<Rate>	float
	Range: -1e+15 to 1e+15
	Default unit: Hz/s

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Piecewise Linear Chirp" on page 171

PULSe:MOP:PIECewise:OFFSet <Offset>

Offsets the start frequency of the chirp.

Parameters:

<Offset>	float
	Range: -1e+09 to 1e+09

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Piecewise Linear Chirp" on page 171

PULSe:MOP:CCHirp:FREQuency <Frequency>

Set the frequency of the custom chirp.

Parameters:

<Frequency>	float
	Range: -1e+09 to 1e+09

Example:
```
// custom chirp with 4 frequencies
PULSe:MOP:TYPE CCHIPRP
PULSe:MOP:CCHirp:ADD
PULSe:MOP:CCHirp:COUNt 4
PULSe:MOP:CCHirp:SEL 1
PULSe:MOP:CCHirp:FREQuency 1e+07
PULSe:MOP:CCHirp:SEL 2
PULSe:MOP:CCHirp:FREQuency 5e+06
PULSe:MOP:CCHirp:SEL 3
PULSe:MOP:CCHirp:FREQuency -1e+07
PULSe:MOP:CCHirp:SEL 4
PULSe:MOP:CCHirp:FREQuency 0
```

Manual operation: See "Custom Chirp" on page 172

PULSe:MOP:QAM:TYPE <Type>

Selects the QAM type.

Parameters:

<Type>	Q16	Q32	Q64	Q128	Q256

Example:
```
PULSe:MOP:TYPE QAM
PULSe:MOP:QAM:TYPE Q256
```

Manual operation:　　See "16QAM, 32QAM, 64QAM, 128QAM, 256QAM"
　　　　　　　　　　　　　on page 181

PULSe:MOP:DATA:CODing <Coding>

Selects the data coding scheme.

Parameters:

<Coding>　　　　　　NONE | DIFFerential | GRAY | DGRay

Example:　　　　　See Using a plugin as a modulation source

Manual operation:　　See "Coding" on page 184

PULSe:MOP:DATA:DSRC <Dsrc>

Selects the data source for the modulation, see DSRC:CREate.

Parameters:

<Dsrc>　　　　　　　string

Example:　　　　　See Using a plugin as a modulation source

Manual operation:　　See "Data Source" on page 184

PULSe:MOP:DATA:DSRC:RESet <Reset>

Resets the data source at the end of the pulse.

Parameters:

<Reset>　　　　　　ON | OFF | 1 | 0

Example:　　　　　`PULSe:MOP:DATA:DSRC:RESet 1`

Manual operation:　　See "Data Source" on page 184

PULSe:MOP:ENABle <Enable>

Defines whether a MOP is applied.

Parameters:

<Enable>　　　　　　ON | OFF | 1 | 0

Example:　　　　　See Example"Creating a linear chirp pulse" on page 694

Manual operation:　　See "Enable" on page 164

PULSe:MOP:EXCLude:ENABle <Enable>

Activates the restriction of the modulation.

Parameters:

<Enable>　　　　　　ON | OFF | 1 | 0

Example: Example"Creating an unmodulated pulse" on page 693

Manual operation: See "Restrict MOP Range" on page 185

PULSe:MOP:EXCLude:LEVel:STARt <Start>
PULSe:MOP:EXCLude:LEVel:STOP <Stop>

Sets the threshold levels at the beginning and the end of a pulse for the modulation to be excluded.

Parameters:

<Stop>   float

Range: 0 to 100

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Restrict MOP Range" on page 185

PULSe:MOP:EXCLude:MODE <Mode>

Selects the parameter that determines the area on that the MOP is applied.

Parameters:

<Mode>   TIME | LEVel | WIDTh

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Restrict MOP Range" on page 185

PULSe:MOP:EXCLude:TIME:STARt <Start>
PULSe:MOP:EXCLude:TIME:STOP <Stop>

Sets a time span to be excluded at the beginning and at the end of the pulse.

Parameters:

<Stop>   float

Range: 0 to 5e-07

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Restrict MOP Range" on page 185

PULSe:MOP:FILTer:BT <Bt>

Sets the B x T filter parameter.

Parameters:

<Bt>   float

Range: 0.15 to 2.5

Example: See Example"Using a plugin as a modulation source" on page 686

Manual operation: See "Filter" on page 185

PULSe:MOP:FILTer:BWIDth <Bwidth>

Sets the transition bandwidth of the filter.

Parameters:

<Bwidth>   float

                  Range: 1 to 1e+09

Example: See Example"Using a plugin as a modulation source"
                  on page 686

Manual operation: See "Filter" on page 185

PULSe:MOP:FILTer:LENGth <Length>

Sets the filter length.

Parameters:

<Length>   integer

                  Range: 1 to 64

Example: See Example"Using a plugin as a modulation source"
                  on page 686

Manual operation: See "Filter" on page 185

PULSe:MOP:FILTer:ROLLoff <Rolloff>

Sets the roll off factor.

Parameters:

<Rolloff>   float

                  Range: 0.05 to 1

Example: See Example"Using a plugin as a modulation source"
                  on page 686

Manual operation: See "Filter" on page 185

PULSe:MOP:FILTer:TYPE <Type>

Selects the filter type.

Parameters:

<Type>   NONE | RECTangular | COS | RCOS | GAUSs | LPASs |
                  FSKGauss | SOQPsk | SMWRect

Example: See Example"Using a plugin as a modulation source"
                  on page 686

Manual operation: See "Filter" on page 185

PULSe:MOP:TYPE <Type>

Select the modulation scheme.

Parameters:

<Type>   AM | ASK | AMSTep | FM | FSK | FMSTep | CHIRp | PCHirp |
                          BARKer | POLYphase | PLISt | BPSK | QPSK | NOISe |
                          PWISechirp | CCHiprp | PSK8 | QAM | MSK | NLCHirp | PLUGin

Example: See Example"Creating a linear chirp pulse" on page 694.

Manual operation: See "MOP Type" on page 164

Manual operation: See Chapter 6.2.5, "Modulation on pulse (MOP) settings",
on page 162.

PULSe:MOP:FM:DEViation <Deviation>

Sets the modulation deviation.

Parameters:

<Deviation>   float

                          Range: 0.1 to 1e+09
                          Default unit: Hz

Manual operation: See "FM" on page 167

PULSe:MOP:FM:FREQuency <Frequency>

Sets the modulation frequency.

Parameters:

<Frequency>   float

                          Range: 0.002 to 1e+09

Manual operation: See "FM" on page 167

PULSe:MOP:8FSK:DEViation <Deviation>
PULSe:MOP:4FSK:DEViation <Deviation>
PULSe:MOP:FSK:DEViation <Deviation>

Sets the modulation deviation.

Parameters:

<Deviation>   float

                          Range: 0.001 to 1e+09
                          Default unit: Hz

Manual operation: See "FSK, 4FSK, 8FSK, 16FSK, 32FSK, 64FSK" on page 169

PULSe:MOP:8PSK:SRATe <Srate>
PULSe:MOP:8FSK:SRATe <Srate>

PULSe:MOP:4FSK:SRATe <Srate>
PULSe:MOP:QAM:SRATe <Srate>
PULSe:MOP:FSK:SRATe <Srate>

Sets the symbol rate of the modulated signal.

Parameters:

<Srate>   float

                    Range: 1 to 1e+09

Manual operation: See "FSK, 4FSK, 8FSK, 16FSK, 32FSK, 64FSK" on page 169

PULSe:MOP:NLCHirp:EQUation <Equation>

Determines the chirp mathematically.

Parameters:
<Equation>   string

Manual operation: See "Equation" on page 174

PULSe:MOP:NOISe:BWIDth <Bwidth>

Sets the bandwidth.

Parameters:
<Bwidth>   float

                    Range: 1 to 1e+09
                    Default unit: Hz

Manual operation: see Chapter 6.2.5.7, "Noise", on page 182

PULSe:MOP:PCHirp:COEFficient <Coefficient>

Sets the coefficient of the chirp polynomial.

Parameters:
<Coefficient>   float

                    Range: -1e+32 to 1e+32

Manual operation: See "Polynomial Chirp" on page 174

PULSe:MOP:PCHirp:TERM <Term>

Sets the term of the chirp polynomial.

Parameters:
<Term>   float

                    Range: 0 to 32

Manual operation: See "Polynomial Chirp" on page 174

PULSe:MOP:PLISt:VALue <Value>

Sets the phase.

Parameters:
<Value>   float
                          Range: -180 to 180
                          Default unit: degree
Manual operation: See "Custom Phase" on page 178

PULSe:MOP:POLY:LENGth <Length>

Sets the polyphase length (code order).

Parameters:
<Length>   integer
                          Range: 1 to 100
Manual operation: See "Poly Phase" on page 178

PULSe:MOP:POLY:TYPE <Type>

Selects the modulation type.

Parameters:
<Type>   FRANk | P1 | P2 | P3 | P4
Manual operation: See "Poly Phase" on page 178

PULSe:MOP:QPSK:SRATe <Srate>

Sets the symbol rate.

Parameters:
<Srate>   float
                          Range: 1 to 1e+09
Manual operation: See "QPSK" on page 180

PULSe:MOP:QPSK:TYPE <Type>

Selects the modulation type.

Parameters:
<Type>   NORMal | OQPSk | DQPSk | ASOQpsk | BSOQpsk | TGSoqpsk
Manual operation: See "QPSK" on page 180

PULSe:MOP:QPSK:SOQPsk:IRIG <Irig>

Enables differential encoding according to the telemetry standard IRIG 106-04.

Parameters:

<Irig>   ON | OFF | 1 | 0

Example: `PULSe:MOP:QPSK:TYPE TGSOQPSK`

                          `PULSe:MOP:QPSK:SRATe 1e+06`

                          `PULSe:MOP:QPSK:SOQPsk:IRIG 1`

Manual operation: See "QPSK" on page 180

PULSe:OVERshoot <Overshoot>

Sets the overshoot level value.

Parameters:

<Overshoot>   float

                          Range: 0 to 50

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Overshoot" on page 157

PULSe:OVERshoot:DECay <Decay>

Sets the number of peaks.

Parameters:

<Decay>   float

                          Range: 1 to 100

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Overshoot" on page 157

PULSe:RIPPle <Ripple>

Sets the ripple level.

Parameters:

<Ripple>   float

                          Range: 0 to 50

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Ripple" on page 158

PULSe:RIPPle:FREQuency <Frequency>

Sets the ripple frequency.

Parameters:

<Frequency>   float

                          Range: 0 to 3e+08

                          Default unit: Hz

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Ripple" on page 158

PULSe:TIME:FALL <Fall>
PULSe:TIME:RISE <Rise>

Sets the transition time of the rising and falling edges.

Parameters:

<Rise>   float

                            Range: 0 to 3600

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Pulse Shape Settings" on page 153

PULSe:TIME:REFerence <Reference>

Selects a predefined envelope profile.

Parameters:

<Reference>   VOLTage | POWer | FULL

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Standard Timing Profile" on page 152

PULSe:TIME:WIDTh <Width>

Sets the time during that the pulse is on top power.

Parameters:

<Width>   float

                            Range: 0 to 3600
                            Default unit: s

Example: See Example"Creating a linear chirp pulse" on page 694

Manual operation: See "Pulse Shape Settings" on page 153

PULSe:TYPE:FALL <Fall>
PULSe:TYPE:RISE <Rise>

Sets the slope type for the rising and falling edges.

Parameters:

<Rise>   LINear | COSine | RCOSine | SQRT

Example: Example"Creating an unmodulated pulse" on page 693

Manual operation: See "Pulse Shape Settings" on page 153

PULSe:SETTings <Settings>

Switches between the displayed settings.

Setting parameters:

<Settings>   TIMing | MOP | MKR | GENeral | | LEVel

Example: `PULSe:SETTings MOP`

`PULSe:PREView:MODE MOP`

`PULSe:PREView:MOP IQ`

Usage: Setting only

PULSe:PREView:MODE <Mode>

Switches between the envelope and modulation graphs.

Setting parameters:

<Mode>   ENVelope | MOP

Example: See `PULSe:SETTings` on page 715

Usage: Setting only

Manual operation: See "Envelope graph" on page 187

PULSe:PREView:MOP <Mop>

Sets the displayed modulation characteristics.

Setting parameters:

<Mop>   IQ | PHASe | FREQuency

Example: See `PULSe:SETTings` on page 715

Usage: Setting only

Manual operation: See "Time domain display" on page 188

25.19 Receiver commands

Example: Creating receivers

```
RECeiver:CREate "Receiver DF"
RECeiver:CATalog?
// "Receiver1", "Receiver DF"
RECeiver:SELect?
// Receiver DF
RECeiver:NAME?
// Receiver DF
RECeiver:MODel?
// INTerfero

RECeiver:ANTenna:ADD
RECeiver:ANTenna:SELect 1
RECeiver:ANTenna:ALIas "Antenna 1"
RECeiver:ANTenna:POSition:X 0.14
RECeiver:ANTenna:POSition:Y 0
RECeiver:ANTenna:POSition:RADius 0.14
RECeiver:ANTenna:POSition:ANGLe 0
RECeiver:ANTenna:POSition:HEIGht 0
ANTenna:CATalog?
// "Isotropic","My_PencilBeam","My_Cosecant","My_PlanarAntenna","CustomAntennPattern"
RECeiver:ANTenna:PATTern "My_PlanarAntenna"
RECeiver:ANTenna:SCAN "NONE"
RECeiver:ANTenna:GAIN 2
RECeiver:ANTenna:DIRection:AWAY 1
RECeiver:ANTenna:DIRection:ELEVation 0
RECeiver:ANTenna:DIRection:AZIMuth?
// 0

RECeiver:ANTenna:ADD
RECeiver:ANTenna:SELect 2

RECeiver:ANTenna:ALIas "Antenna 2"
RECeiver:ANTenna:POSition:X 0
RECeiver:ANTenna:POSition:Y 0.14
RECeiver:ANTenna:POSition:RADius 0.14
RECeiver:ANTenna:POSition:ANGLe 90
RECeiver:ANTenna:POSition:HEIGht 0
RECeiver:ANTenna:PATTern "My_PlanarAntenna"
RECeiver:ANTenna:SCAN "NONE"
RECeiver:ANTenna:GAIN 0
RECeiver:ANTenna:DIRection:AWAY 1
RECeiver:ANTenna:DIRection:ELEVation 0
RECeiver:ANTenna:DIRection:AZIMuth?
// 90
```

See also Example"Moving emitters and receivers" on page 768.

RECeiver:ANTenna:ALIas <Alias>

Sets an alias name for the selected antenna element.

Parameters:

<Alias>   string

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Antenna Configuration" on page 354

RECeiver:MODel <Model>

Sets the receiver model.

Parameters:

<Model>   INTerfero | TDOA | COMBined

For details, see "Model" on page 353.

INTerfero
Interferometer
Calculates the relative phase difference between the single antenna ports.

TDOA
Time difference of arrival
Calculates the absolute time of arrival (TOA) of the incoming signal for each antenna.

COMBined
Calculates the relative phases between the antenna ports and calculates the the individual TOAs for each antenna port.

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Model" on page 353

RECeiver:ANTenna:POSition:X \<X\>
RECeiver:ANTenna:POSition:Y \<Y\>

Sets the antenna element position as X and Y values, relative to the receiver origin.

Parameters:

\<Y\>   float

                              Range: -1e+06 to 1e+06

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Position" on page 356

RECeiver:ANTenna:POSition:RADius \<Radius\>

Sets the distance from the antenna element to the receiver origin.

Parameters:

\<Radius\>   float

                              Range: 0 to 1e+06

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Position" on page 356

RECeiver:ANTenna:POSition:ANGLe \<Angle\>

Sets the antenna element position as an angle offset from the X-axis.

Parameters:

\<Angle\>   float

                              Range: 0 to 360

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Position" on page 356

RECeiver:ANTenna:POSition:HEIGht \<Height\>

Sets the antenna element height, relative to the receiver origin.

Parameters:

\<Height\>   float

                              Range: -1e+06 to 1e+06

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Position" on page 356

RECeiver:ANTenna:PATTern \<Pattern\>

Assigns an existing antenna pattern, see `ANTenna:CATalog?`.

Parameters:

<Pattern>   string

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Properties" on page 355

RECeiver:ANTenna:SCAN <Scan>

Sets the antenna scan.

Parameters:

<Scan>   string

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Properties" on page 355

RECeiver:ANTenna:GAIN <Gain>

Sets the gain of the individual antenna element.

Parameters:

<Gain>   float

                           Range: -120 to 120

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Properties" on page 355

RECeiver:ANTenna:DIRection:AWAY <Away>

Sets the azimuth automatically, so that the beam axis is radial to the receiver origin.

Parameters:

<Away>   ON | OFF | 1 | 0

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Properties" on page 355

RECeiver:ANTenna:DIRection:AZIMuth <Azimuth>

Turns the antenna beam axis.

Parameters:

<Azimuth>   float

                           Range: 0 to 360

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Properties" on page 355

RECeiver:ANTenna:DIRection:ELEVation <Elevation>

Turns the antenna beam axis.

Parameters:

<Elevation>   float

                     Range: -90 to 90

Example: See Example"Creating receivers" on page 716.

Manual operation: See "Properties" on page 355

25.20 Repository commands

(i)   Do not change the default folder structure and the repository file names.

      Changing file paths and file names may lead to data loss and irreparable faults in the
      data sources.

Example: Working with repositories

```
SCPI
// create new empty repository
REPository:CREate "Repository for tests"
REPository:CATalog?
// line brakes added to improve the readability
//"Repository for tests",
//"C:\Users\Public\Documents\Rohde-Schwarz\Pulse Sequencer\Repositories",
//"K300 and K301 Tests",
//"C:\Users\Public\Documents\Rohde-Schwarz\Pulse Sequencer\Repositories",
//"Rep","C:\Users\Public\Documents\Rohde-Schwarz\Pulse Sequencer\Repositories"
REPository:SAVE
REPository:SELect "Repository for tests"
REPository:AUTHor "My Company Name"
REPository:DATE?
REPository:SECurity LEV1
REPository:VERSion "1.1"
REPository:COMPlexity EMITter
REPository:XPOL:ATTenuation 30
REPository:SAVE
REPository:PATH?
// C:\Users\Public\Documents\Rohde-Schwarz\PulseSequencer\Repositories\20140722_113521
REPository:FILename?
// C:\Users\Public\Documents\Rohde-Schwarz\PulseSequencer\Repositories
    \20140722_113521\Config.ps_rep
REPository:VOLatile:PATH?
// C:/_PS_files/Rohde-Schwarz/Pulse Sequencer/Storage/3012b039-89b5-45e1-82e2-d8390712bc7a/11
REPository:ACCess?
// RW,Login=no,Pass=no,Uname=testuser
REPository:REMove "Rep"

REPManager:PATH:LIST?
// "C:\Users\Public\Documents\Rohde-Schwarz\PulseSequencer\Repositories"
REPManager:PATH:ADD "C:\_ps_files"
REPManager:PATH:LIST?
// "C:\Users\Public\Documents\Rohde-Schwarz\PulseSequencer\Repositories","C:\_ps_files"
REPManager:CATalog?
// "Repository for tests",
    "C:\Users\Public\Documents\Rohde-Schwarz\PulseSequencer\Repositories",
// "K300 and K301 Tests",
    "C:\Users\Public\Documents\Rohde-Schwarz\PulseSequencer\Repositories",
// "My_Tests","C:\_ps_files"
REPManager:LOAD "K300 and K301 Tests"
REPManager:EXPort "K300 and K301 Tests","c:\_ps_files\my.psarch"
REPManager:DELete "My_Tests"
REPManager:CATalog?
// "Repository for tests",
    "C:\Users\Public\Documents\Rohde-Schwarz\PulseSequencer\Repositories",
```

```
// "K300 and K301 Tests",
   "C:\Users\Public\Documents\Rohde-Schwarz\PulseSequencer\Repositories"
```

REPository:ACCess?

Queries information on the access rights of the current user.

Return values:

<Access>   <permission>,<login>,<pass>,<Uname>

          <permission>
          Permission of the current user, for example `RW` (read-write)

          <login>,<pass>
          `Login/Pass=No`: Password not required
          `Login/Pass=Yes`: Password required

          <Uname>
          User name of the current user

Example: See Example"Working with repositories" on page 721

Usage: Query only

Manual operation: See "Users" on page 557

REPository:AUTHor <Author>

Enters information on the author.

Parameters:

<Author>   string

Example: See Example"Working with repositories" on page 721

Manual operation: See "Info" on page 107

REPository:COMPlexity <Complexity>

Sets the complexity level.

Parameters:
<Complexity>   PTRain | EMITter | DIRection

Example: See Example"Working with repositories" on page 721

Manual operation: See "Functional Range" on page 108

REPository:DATE <Date>

Queries the creation data.

Parameters:
<Date>   string

Example: See Example"Working with repositories" on page 721

Manual operation: See "Info" on page 107

REPository:FILename?

Queries the file name of the repository archive.

Return values:
<Filename>   string

                           File path, incl. file name, and extension

Example: See Example"Working with repositories" on page 721

Usage: Query only

Manual operation: See "Storage" on page 109

REPository:PATH?

Queries the directory in that the repository archive is stored.

Return values:
<Path>   string

Example: See Example"Working with repositories" on page 721

Usage: Query only

Manual operation: See "Storage" on page 109

REPository:VOLatile:PATH?

Queries the directory in that the volatile data is saved.

Return values:

<Path>   string

Example: See Example"Working with repositories" on page 721

Usage: Query only

REPository:SAVE

Stores the repository archive.

To query the storage location, use the command `REPository:PATH?`.

Example: See Example"Working with repositories" on page 721

Usage: Event

Manual operation: See "Storage" on page 109

REPository:SECurity <Security>

Sets the security level.

Parameters:

<Security>   LEV0 | LEV1 | LEV2 | LEV3 | LEV4

Example: See Example"Working with repositories" on page 721

Manual operation: See "Classification" on page 108

REPository:VERSion <Version>

Sets the repository version.

Parameters:

<Version>   string

Example: See Example"Working with repositories" on page 721

Manual operation: See "Info" on page 107

REPository:XPOL:ATTenuation <Attenuation>

Sets the attenuation used to calculate the cross-polarized antenna patterns.

Parameters:

<Attenuation>   float

                        *RST: 0

Example: See Example"Working with repositories" on page 721

Manual operation: See "Antenna Cross Polarization > Attenuation" on page 108

REPository:UUID?

Queries the repository's Universally Unique Identifier (UUID).

Return values:

<Uuid>	string
Example:	REPository:UUID? // 89d00568-e9d7-4470-8b8d-2179336da541
Usage:	Query only

REPManager:CATalog?

Queries available repository elements in the database.

Return values:

<Catalog>	"<RepositryName>","<path>" <RepositryName> is the name of the repository as defined with the command REPository:CREate <Path> is the compete file path
Example:	See Example"Working with repositories" on page 721
Usage:	Query only
Manual operation:	See "Discovered Repositories on the Mass Storage" on page 111

REPManager:LOAD <RepName>[,<Path>[,<Username>,<Passwd>]]

Loads the selected repository to the workspace.

If more than one repository with the same name exist, loaded is the first repository with a name match.

To query the available repository elements in the database, use the command REPository:CATalog?.

Setting parameters:

<RepName>	string Repository name, as configured in the workspace.
<Path>	string Compete file path, as queried with the command REPManager:PATH:LIST?. The <Path> must be specified, if <Username> and <Passwd> are used.
<Username>	string Required if the repository is password protected
<Passwd>	string Required if the repository is password protected

Example: See Example"Working with repositories" on page 721

Usage: Setting only

Manual operation: See "Load" on page 112

REPManager:DELete <RepName>[,<Path>[,<Username>,<Passwd>]]

Deletes the entire repository from the permanent mass storage.

Setting parameters:

<RepName>   string

Repository name, as configured in the workspace.
If more than one repository with the same name exists, the
<Path> must be specified.

<Path>   string

Compete file path, as queried with the command REPManager:
PATH:LIST?.
The <Path> must be specified, if the <RepName> is not unique
and if <Username> and <Passwd> are used.

<Username>   string

Required if the repository is password protected

<Passwd>   string

Required if the repository is password protected

Example: See Example"Working with repositories" on page 721

Usage: Setting only

Manual operation: See "Delete" on page 112

REPManager:EXPort <RepName>[,<Path>],<PSArchiveFile>

Exports the selected repository file to an archive file.

Setting parameters:

<RepName>   string

Repository name, as configured in the workspace.

<Path>   string

Compete file path, as queried with the command REPManager:
PATH:LIST?.

<PSArchiveFile>   Complete file path, incl. file name, and extension (*.psarch).

Example: See Example"Working with repositories" on page 721.

Usage: Setting only

Manual operation: See "Export" on page 112

REPManager:IMPort <Import>

Imports a repository from a directory.

Setting parameters:

<Import>   string

Usage: Setting only

Manual operation: See "Import" on page 112

REPManager:PATH:ADD <Add>

Add the selected directory.

Setting parameters:

<Add>   string

                        Complete file path

Example: See Example"Working with repositories" on page 721

Usage: Setting only

Manual operation: See "Add Path" on page 112

REPManager:PATH:DELete <Delete>

Removes the selected file path.

Setting parameters:

<Delete>   string

                        File path

Example: See Example"Working with repositories" on page 721

Usage: Setting only

Manual operation: See "Remove Path" on page 113

REPManager:PATH:LIST?

Queries the directory in that the repository files are stored.

Return values:

<List>   string

                        Compete file path

Example: See Example"Working with repositories" on page 721

Usage: Query only

25.21 Scenario commands

Example: Creating simple pulse train scenario

```
SCPI
SCENario:CREate "SimplePulseTrain"
SCENario:TYPE SEQuence
SCENario:ID?
// 4
SCENario:OUTPut:FREQuency 6e+09
SCENario:OUTPut:LEVel -10
SCENario:OUTPut:RESet:ENABle 1
SCENario:OUTPut:RUNMode CONTinious
SEQuence:CATalog?
// "My_S1_PT","My_S2_S1-F1_S1-F2","My_PT_PRI-Stagger","My_PT_F-Hops",
// "My_WV_Seq","My_TestSequence"
SCENario:SEQuence "My_S1_PT"
SCENario:OUTPut:TARGet INSTrument
GENerator:CATalog?
// "My_R&S SMW","My_R&S SMBV","R&S SMW"
SCENario:GENerator "R&S SMW"
SCENario:GENerator:PATH?
// 1
SCENario:CACH:REPository:ENABle 1

SCENario:OUTPut:CLOCk:MODE AUTO
SCENario:OUTPut:CLOCk:AUTO:OVERsampling 8
SCENario:OUTPut:CLOCk:AUTO:BORDer 1000000
// SCENario:OUTPut:CLOCk:MODE MAN
// SCENario:OUTPut:CLOCk:USER?
// 1e+07
SCENario:OUTPut:DURation:MODE AUTO
// SCENario:OUTPut:DURation:TIME 0.10
SCENario:OUTPut:FORMat WV
SCENario:OUTPut:THReshold -100
SCENario:OUTPut:RECall:ENABle 1

SCENario:CALCulate
SCENario:STATe?
// RUN
SYSTem:PROGress?
// 74
SYSTem:PROGress?
// 100
SCENario:CACHe:VOLatile:VALid?
// 1
SCENario:CACHe:VOLatile:RELease

// SCENario:CACHe:VOLatile:CLEar
```

```
// SCENario:STOP
// SCENario:SANitize REPository

SCENario:STARt
```

Example: Creating sequence collection scenario

```
SCPI
SCENario:CREate "My_PT_Collection"
SCENario:TYPE CSEQuence
SCENario:CSEQuence:ADD
SCENario:CSEQuence:SELect 1
SCENario:CSEQuence:ALIas "PT"
SCENario:CSEQuence "My_S2_S1-F1_S1-F2"
SCENario:CSEQuence:ADD
SCENario:CSEQuence:SELect 2
SCENario:CSEQuence:ALIas "F-Hops"
SCENario:CSEQuence "My_PT_F-Hops"
SCENario:CSEQuence:CURRent 1
...
SCENario:CALCulate
```

Example: Creating a simple emitter scenario

```
SCPI
SCENario:CREate "SimpleEmitterScenario"
SCENario:TYPE EMITter
EMITter:CATalog?
// "My_EmitterGuidance","TestEmitter"
SCENario:EMITter "My_EmitterGuidance"
SCENario:EMITter:MODE 2
SCENario:EMITter:MODE:BEAM 1
SCENario:EMITter:DIRection:YAW -10
SCENario:EMITter:DIRection:PITCh -5
...
SCENario:CALCulate
```

Example: Creating a scenario with multiple emitters/interferers and background emitters

```
SCPI
SCENario:CREate "LocalizedEmitters"
SCENario:TYPE LOCalized
EMITter:CATalog?
// "My_EmitterGuidance","TestEmitter"
WAVeform:CATalog?
// "My_Waveform","WV_Matlab","CW","AWGN"
ANTenna:CATalog?
// "Isotropic","My_PencilBeam","My_Cosecant","My_PlanarAntenna","Testantenna"
SCAN:CATalog?
// "My_RasterScan","My_Circular","Test Antenna Scan"

SCENario:LOCalized:RECeiver:ANTenna "My_Cosecant"
SCENario:LOCalized:RECeiver:SCAN "My_Circular"
SCENario:LOCalized:RECeiver:GAIN 5
SCENario:LOCalized:RECeiver:HEIGht 1
SCENario:LOCalized:RECeiver:DIRection:YAW 0
SCENario:LOCalized:RECeiver:DIRection:PITCh -5

SCENario:LOCalized:ADD
SCENario:LOCalized:SELect?
// 4
SCENario:LOCalized:SELect 5
SCENario:LOCalized:TYPE WAVeform
SCENario:LOCalized:ALIas "I3"
SCENario:LOCalized:WAVeform "My_Waveform"
SCENario:LOCalized:WAVeform:EIRP 50
SCENario:LOCalized:WAVeform:LEVel?
SCENario:LOCalized:WAVeform:FREQuency 3e+09
SCENario:LOCalized:WAVeform:ANTenna "Testantenna"
SCENario:LOCalized:WAVeform:SCAN "Test Antenna Scan"
SCENario:LOCalized:DIRection:TRACk 1
SCENario:LOCalized:DIRection:PITCh 0
SCENario:LOCalized:DIRection:YAW -53.3035
SCENario:LOCalized:LOCation:EAST 2860.34
SCENario:LOCalized:LOCation:NORTh -3837.93
SCENario:LOCalized:DISTance?
// 4786.57

SCENario:LOCalized:ADD
SCENario:LOCalized:SELect 6
SCENario:LOCalized:TYPE EMITter
SCENario:LOCalized:ALIas "E4"
SCENario:LOCalized:EMITter "My_EmitterGuidance"
SCENario:LOCalized:EMITter:MODE 1
SCENario:LOCalized:EMITter:MODE:BEAM 2
SCENario:LOCalized:DIRection:TRACk 1
SCENario:LOCalized:DIRection:PITCh 0
```

```
SCENario:LOCalized:DIRection:YAW -87.8901
SCENario:LOCalized:LOCation:EAST 253.448
SCENario:LOCalized:LOCation:NORTh -6879.31
SCENario:LOCalized:DISTance?
// 6883.98

SCENario:LOCalized:ADD
SCENario:LOCalized:SELect 7
SCENario:LOCalized:TYPE BKG
SCENario:LOCalized:ALIas "BKG"
WAVeform:CATalog?
// "My_Waveform","WV_Matlab","CW","Noise","MC","BKG"
SCENario:LOCalized:WAVefrom "BKG"

SCENario:LOCalized:ADD
SCENario:LOCalized:SELect 7
SCENario:LOCalized:TYPE BKG
SCENario:LOCalized:ALIas "PT"
SEQuence:CATalog?
// "My_S2_S1-F1_S1-F2","My_PT_PRI-Stagger","My_PT_F-Hops",
// "My_TestSequence","My_PulseTrain_Seq"
SCENario:LOCalized:SEQuence "My_PulseTrain_Seq"

...
SCENario:CALCulate
```

Example: Direction finding scenario

```
SCENario:NAME "DF"
SCENario:DF:RECeiver:MOVement:VEHicle STATIONARY
SCENario:DF:RECeiver "Receiver DF"
SCENario:DF:LOCation:REC:PMODe STATIC
SCENario:DF:RECeiver:LATitude -3.781666183E+01
SCENario:DF:RECeiver:LONGitude 1.4496665955E+02
SCENario:DF:RECeiver:HEIGht 1.E+02
SCENario:DF:RECeiver:DIRection:YAW 0.E+00
SCENario:DF:RECeiver:DIRection:PITCh 0.E+00
SCENario:DF:RECeiver:DIRection:ROLL 0.E+00

SCENario:DF:TYPE EMITTER
SCENario:DF:ALIas "TestEmitter"
SCENario:DF:EMITter "TestEmitter"
SCENario:DF:MOVement:VEHicle
SCENario:DF:MCHG:STATe 0
SCENario:DF:EMITter:MODE 1
SCENario:DF:EMITter:MODE:BEAM 1
SCENario:DF:EMITter:MODE:TRACkrec 1
SCENario:DF:LOCation:PMODe MOVING
SCENario:DF:LOCation:EAST 3.998600097656E+03
SCENario:DF:LOCation:NORTh 6.902350097656E+03
SCENario:DF:LOCation:HEIGht 0.E+00
SCENario:DF:DISTance 7.977545898438E+03
SCENario:DF:LOCation:AZIMuth 3.008413852012E+01
SCENario:DF:LOCation:ELEVation 0.E+00
SCENario:DF:MOVement:SPEed 1.E-01
SCENario:DF:MOVement:ACCeleration 0.E+00
SCENario:DF:MOVement:EAST 0.E+00
SCENario:DF:MOVement:NORTh 0.E+00
SCENario:DF:MOVement:HEIGht 0.E+00
SCENario:DF:MOVement:RMODe ONEWAY
...
```

Example: Enabling mode changing

The following is a simple example on how to enable 3 modes for the emitter with alias E4 in the example Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730.

```
SCENario:LOCalized:SELect 6
SCENario:LOCalized:ALIas?
// "E4"
SCENario:LOCalized:TYPE?
// EMITter
SCENario:LOCalized:EMITter?
// "My_EmitterGuidance"
// enable mode change and add 3 entries
SCENario:LOCalized:MCHG:STATe 1
SCENario:LOCalized:MCHG:ADD
SCENario:LOCalized:MCHG:ADD
SCENario:LOCalized:MCHG:COUNt?
// 3
// start configuration of the last (entry#3)
SCENario:LOCalized:MCHG:SELect 3
// configure the stop time first
SCENario:LOCalized:MCHG:STOP 10
SCENario:LOCalized:MCHG:STARt 5
SCENario:LOCalized:EMITter:MODE 2
SCENario:LOCalized:EMITter:MODE:BEAM 2
// configure entry#2
SCENario:LOCalized:MCHG:SELect 2
// set stop time of entry#2 = start time of entry#3
SCENario:LOCalized:MCHG:STOP 5
SCENario:LOCalized:MCHG:STARt 3
SCENario:LOCalized:EMITter:MODE 2
SCENario:LOCalized:EMITter:MODE:BEAM 1
// configure entry#1
SCENario:LOCalized:MCHG:SELect 1
// set stop time of entry#1 = start time of entry#2
SCENario:LOCalized:MCHG:STOP 3
SCENario:LOCalized:MCHG:STARt 0
SCENario:LOCalized:EMITter:MODE 1
SCENario:LOCalized:EMITter:MODE:BEAM 1

// SCENario:LOCalized:MCHG:CLEar
```

Example: Configuring position steps

The following is a simple example on how to enable 3 positions for the emitter with alias E4.

```
SCENario:LOCalized:SELect 6
SCENario:LOCalized:ALIas?
// "E4"
SCENario:LOCalized:TYPE?
// EMITter
SCENario:LOCalized:EMITter?
// "My_EmitterGuidance"
// enable position steps and add 2 entries
SCENario:LOCalized:PSTep:STATe 1
SCENario:LOCalized:PSTep:ADD
SCENario:LOCalized:PSTep:ADD
SCENario:LOCalized:PSTep:COUNt?
// 2
// position step#1 is already defiend

SCENario:LOCalized:PSTep:SELect 2
SCENario:LOCalized:DIRection:YAW -150
SCENario:LOCalized:LOCation:EAST 4.439169921875E+03
SCENario:LOCalized:LOCation:NORTh 7.702899902344E+03

SCENario:LOCalized:PSTep:SELect 3
SCENario:LOCalized:DIRection:TRACk 1
SCENario:LOCalized:LOCation:EAST 2.323706054688E+03
SCENario:LOCalized:LOCation:NORTh 4.177126464844E+03
```

Example: Configuring different modes

The following is a simple example on how to enable emitter with different modes.

```
SCENario:LOCalized:EMITter "TestEmitter"
SCENario:LOCalized:EMITter:STATe:ENABle 1
SCENario:LOCalized:EMITter:STATe:ADD
SCENario:LOCalized:EMITter:STATe:SELect 1
SCENario:LOCalized:EMITter:STATe:DURation 1
SCENario:LOCalized:EMITter:STATe:VALue 1
SCENario:LOCalized:EMITter:STATe:ADD
SCENario:LOCalized:EMITter:STATe:COUNt?
// 2
SCENario:LOCalized:EMITter:STATe:SELect 2
SCENario:LOCalized:EMITter:STATe:DURation 2
SCENario:LOCalized:EMITter:STATe:VALue 0

SCENario:LOCalized:EMITter:STATe:LOOP 1
```

Example: Creating and configuring PDW list scenario

The following is a simple example on how to create a PDW list scenario.

```
// create scenario
SCENario:TYPE PDW
SCENario:CREate
SCENario:NAME "PDW List"


// create a waveform element to import your PDW list
WAVeform:CREate "PDW"
WAVeform:SELect "PDW"
WAVeform:TYPE PDW
WAVeform:NAME "PDW"
// import your PDW file
```

See Example"PDW import (unmodulated pulses)" on page 669.

```
// repeat the same steps for all your PDW lists
// create, for example, a second waveform object or
// select an existing one
// WAVeform:CATalog?
// "My_Waveform","WV_Matlab","CW","Noise","MC","BKG","txt","PDW","PDW_2"


// configure the PDW list scenario
SCENario:CPDW:ADD
SCENario:CPDW:SELect 1
SCENario:CPDW:ALIas "PDW Custom"
SCENario:CPDW:NAME "PDW"
SCENario:CPDW:LVABs 0
SCENario:CPDW:FREQ 3e+9


SCENario:CPDW:ADD
SCENario:CPDW:SELect 2
SCENario:CPDW:ALIas "My PDW"
SCENario:CPDW:NAME "PDW_2"
SCENario:CPDW:LVABs -3
SCENario:CPDW:FREQ 3e+9


SCENario:CPDW:INTerleaving 0

// convert and calculate the signal from PDW list "PDW Custom"
SCENario:CPDW:CURRent 1
SCENario:OUTPut:TARGet FILE
SCENario:OUTPut:FORMat ESEQencing
SCENario:STARt
// convert and calculate the signal from the other PDW lists in the same manner
// SCENario:CPDW:CURRent 2

// convert and calculate the signal from PDW list "PDW Custom"
SCENario:CPDW:CURRent 1
SCENario:DESTination "Destination 1"
// convert and calculate the signal from the other PDW lists in the same manner
// SCENario:CPDW:CURRent 2
```

```
// calculate the multiplexed signal of the two PDW lists
SCENario:CPDW:INTerleaving 1
SCENario:OUTPut:THReshold -1.E+02
// set the priority of each PSW list
// the higher the number the higher the priority
SCENario:CPDW:SELect 1
SCENario:CPDW:ENABle 1
SCENario:CPDW:DELay 0
SCENario:CPDW:PRIority 10
SCENario:CPDW:LEVel 0
SCENario:CPDW:LDELay 10
SCENario:CPDW:GROup:CATalog?
// "Default","Group 2"
SCENario:CPDW:GROup "Default"

SCENario:CPDW:SELect 2
SCENario:CPDW:ENABle 1
SCENario:CPDW:DELay 0.01
SCENario:CPDW:PRIority 5
SCENario:CPDW:LEVel -5
SCENario:CPDW:LDELay 0
SCENario:CPDW:GROup "Default"

// convert and calculate the multiplexed signal
SCENario:CPDW:CURRent 1
SCENario:STARt
SCENario:INTerleave
SCENario:CACHe:REPository:ENABle:INTerleave

// remove the second file from the PDW list
SCENario:CPDW:SELect 2
SCENario:CPDW:DELete
// remove all entries in the PDW list scenario
// SCENario:CPDW:CLEar
```

Example: Creating and configuring PDW list scenario

The following is a simple example on how to configure interleaving groups. It uses as an example a PDW list scenario, for example as configured in Example"Creating and configuring PDW list scenario" on page 735.

```
// create scenario
SCENario:TYPE PDW
SCENario:CREate
SCENario:NAME "PDW List"

SCENario:INTerleaving:GROup:COUNt?
// 1
SCENario:CPDW:GROup:SELect 1
SCENario:CPDW:GROup:ALIas?
// "Default"
SCENario:CPDW:GROup:ADD
```

```
SCENario:CPDW:GROup:COUNt?
// 2
SCENario:CPDW:GROup:SELect 2
SCENario:CPDW:GROup:ALIas "Group 2"
SCENario:CPDW:GROup:CATalog?
// "Default","Group 2"

SCENario:CPDW:INTerleaving 1
SCENario:CPDW:SELect 1
SCENario:CPDW:ENABle 1
SCENario:CPDW:GROup "Default"
SCENario:CPDW:SELect 2
SCENario:CPDW:ENABle 1
SCENario:CPDW:GROup "Default"
SCENario:CPDW:SELect 3
SCENario:CPDW:ENABle 1
SCENario:CPDW:GROup "Group 2"

// the first and the second PDW lists are interleaved;
// the third one belongs to a different interleaving group
```

SCENario:TYPE <Type>

Sets the scenario type.

Parameters:

<Type>	SEQuence | CSEQuence | EMITter | CEMitter | LOCalized | DF | PDW | WAVeform
Example:	See Example"Creating simple pulse train scenario" on page 728
Manual operation:	See "Scenario Type" on page 124

SCENario:ID?

Queries the database identifier of the selected scenario.

Return values:

\<Id>	float
	*RST: 0

Example: See Example"Creating simple pulse train scenario" on page 728

Usage: Query only

SCENario:LOCalized:INTerleaving \<Interleaving>
SCENario:DF:INTerleaving \<Interleaving>
SCENario:CEMit:INTerleaving \<Interleaving>
SCENario:CPDW:INTerleaving \<Interleaving>

If enabled, multiple PDW lists are interleaved into a single output file using a priority-based dropping algorithm.

Set the priority with the command SCENario:CPDW:PRIority.

Parameters:

\<Interleaving>   ON | OFF | 1 | 0

Example: See Example"Creating and configuring PDW list scenario"
                     on page 735.

Manual operation: See "Interleaving" on page 124

SCENario:DF:INTerleaving:FREQagility \<Freqagility>
SCENario:LOCalized:INTerleaving:FREQagility \<Freqagility>
SCENario:CEMit:INTerleaving:FREQagility \<Freqagility>

Enables frequency agility in interleaving.

Requires R&S SMW with firmware version 5.xx.xxx and higher. To query the installed firmware version of the selected instrument, use the command INSTrument:FIRMware? on page 647.

Parameters:

\<Freqagility>   ON | OFF | 1 | 0

Example: SCENario:TYPE?
                     // CEMitter
                     SCENario:CEMit:INTerleaving ON
                     SCENario:CEMit:INTerleaving:FREQagility ON

Manual operation: See "Frequency Agility" on page 124

SCENario:CALCulate

Starts the signal calculation.

Example:	See Example"Creating simple pulse train scenario" on page 728.
Usage:	Event
Manual operation:	See "Calculate" on page 132

SCENario:STARt

Starts the signal generation.

Example:	See Example"Creating simple pulse train scenario" on page 728.
Usage:	Event
Manual operation:	See "Config" on page 130

SCENario:STOP

Stops the signal calculation.

Example:	See Example"Creating simple pulse train scenario" on page 728.
Usage:	Event
Manual operation:	See "Config" on page 130

SCENario:INTerleave

If `SCENario:CEMit:INTerleaving`|`SCENario:CPDW:INTerleaving`|
`SCENario:LOCalized:INTerleaving`|`SCENario:DF:INTerleaving` 1, triggers
the calculation of a single output file.

The output file comprises the individual PDWs or pulses, where overlapping PDWs or
pulses within an interleaving group are dropped, based on a defined priority.

Example:	See Example"Creating and configuring PDW list scenario" on page 735.
Usage:	Event
Manual operation:	See "Interleave" on page 135

SCENario:STATe?

Queries the current scenario status.

Return values:

<State>	IDLE	RUN
Example:	See Example"Creating simple pulse train scenario" on page 728
Usage:	Query only

Manual operation: See "Config" on page 130

SCENario:CACHe:REPository:ENABle \<Enable\>

Enables file storage in the repository.

Parameters:
\<Enable\>   ON | OFF | 1 | 0

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Signal Storage" on page 133

SCENario:CACHe:REPository:ENABle:INTerleave \<Interleave\>

Enables file storage in the repository when interleaving is active.

Parameters:
\<Interleave\>   ON | OFF | 1 | 0

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Results" on page 134

SCENario:CACHe:REPository:VALid?
SCENario:ILCache:VOLatile:VALid?
SCENario:CACHe:VOLatile:VALid?

Queries whether the volatile/repository memory contains a valid signal file.

Return values:
\<Valid\>   ON | OFF | 1 | 0

Example: See Example"Creating simple pulse train scenario" on page 728

Usage: Query only

SCENario:CACHe:VOLatile:RELease

Exports and stores the generated signal files in the repository.

Example: See Example"Creating simple pulse train scenario" on page 728

Usage: Event

Manual operation: See "Results" on page 131

SCENario:CACHe:VOLatile:RESTore

Loads signal files from the storage.

Example:	`SCENario:CACH:REPository:ENABle 1`
	`SCENario:CACHe:VOLatile:VALid?`
	`// 1`
	`SCENario:CACHe:VOLatile:RELease`
	`SCENario:CACHe:VOLatile:RESTore`
Usage:	Event
Manual operation:	See "Results" on page 131

SCENario:CACHe:REPository:CLEar
SCENario:ILCache:VOLatile:CLEar
SCENario:CACHe:VOLatile:CLEar

Deletes the files from the volatile/repository memory.

Example:	See Example"Creating simple pulse train scenario" on page 728
Usage:	Event
Manual operation:	See "Results" on page 131

SCENario:GENerator <Generator>

Sets the signal generator.

Parameters:

<Generator>	string
	Use the command `GENerator:CATalog?` to query a list of configured generator.
Example:	See Example"Creating simple pulse train scenario" on page 728
Manual operation:	See "Assign" on page 131

SCENario:GENerator:PATH <Path>

Selects the signal path that will play the generated signal.

Parameters:

<Path>	float
	Range: 1 to 32
Example:	See Example"Creating simple pulse train scenario" on page 728
Manual operation:	See "Output Port" on page 128

SCENario:SEQuence <Sequence>

Assigns a pulse sequence, see `SEQuence:CATalog?` on page 587.

Parameters:

<Sequence>	string

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Single sequence and Waveform scenario settings" on page 138

SCENario:SEQuence:CLEar

Usage: Event

Manual operation: See "Single sequence and Waveform scenario settings" on page 138

SCENario:DF:ALIas <Alias>
SCENario:CEMit:ALIas <Alias>
SCENario:CSEQuence:ALIas <Alias>
SCENario:LOCalized:ALIas <Alias>

Enters an alias name.

Parameters:
<Alias>   string

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "Alias Name" on page 438

SCENario:CPDW:CURRent <Current>
SCENario:DF:CURRent <Current>
SCENario:LOCalized:CURRent <Current>
SCENario:CSEQuence:CURRent <Current>

Sets the sequence/emitter that is used by the scenario.

Parameters:
<Current>   float

Number of the sequence/emitter in the list with multiple sequences

*RST: 1

Example: See Example"Creating sequence collection scenario" on page 729

Manual operation: See "Sequence (Collection) Scenario Settings" on page 139

SCENario:DF:DIRection:YAW <Yaw>
SCENario:EMITter:DIRection:YAW <Yaw>
SCENario:CEMit:DIRection:YAW <Yaw>
SCENario:LOCalized:DIRection:YAW <Yaw>

SCENario:DF:RECeiver:DIRection:YAW <Yaw>
SCENario:LOCalized:RECeiver:DIRection:YAW <Yaw>

Sets the yaw.

Parameters:

<Yaw>   float

                        Range: 0 to 360

Example: See Example"Creating a scenario with multiple emitters/interfer-
                        ers and background emitters" on page 730

Manual operation: See "Attitude > Pitch, Yaw, Roll" on page 361

SCENario:DF:DIRection:PITCh <Pitch>
SCENario:DF:RECeiver:DIRection:PITCh <Pitch>
SCENario:EMITter:DIRection:PITCh <Pitch>
SCENario:CEMit:DIRection:PITCh <Pitch>
SCENario:LOCalized:RECeiver:DIRection:PITCh <Pitch>
SCENario:LOCalized:DIRection:PITCh <Pitch>

Sets the pitch.

Parameters:

<Pitch>   float

                        Range: -90 to 90
                        Default unit: grad

Example: See Example"Creating a scenario with multiple emitters/interfer-
                        ers and background emitters" on page 730

Manual operation: See "Attitude" on page 445

SCENario:DF:DIRection:ROLL <Roll>
SCENario:DF:RECeiver:DIRection:ROLL <Roll>
SCENario:EMITter:DIRection:ROLL <Roll>
SCENario:CEMit:DIRection:ROLL <Roll>
SCENario:LOCalized:RECeiver:DIRection:ROLL <Roll>
SCENario:LOCalized:DIRection:ROLL <Roll>

Sets the roll.

Parameters:

<Roll>   float

                        Range: 0 to 360

Manual operation: See "Attitude" on page 445

SCENario:DF:EMITter <Emitter>
SCENario:EMITter <Emitter>

SCENario:CEMit:EMITter <Emitter>
SCENario:LOCalized:EMITter <Emitter>

Assigns an existing emitter or an existing waveform, see WAVeform:CATalog? and EMITter:CATalog?.

Parameters:

<Emitter>   string

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "Emitter/Interferer Name" on page 438

SCENario:DF:EMITter:ENABle <Enable>
SCENario:LOCalized:EMITter:ENABle <Enable>
SCENario:CEMit:EMITter:ENABle <Enable>

In a map-based sceanrio, enable selected item for calculation.

Parameters:

<Enable>   ON | OFF | 1 | 0

SCENario:DF:EMITter:MODE <Mode>
SCENario:EMITter:MODE <Mode>
SCENario:CEMit:EMITter:MODE <Mode>
SCENario:LOCalized:EMITter:MODE <Mode>

Set the emitter mode.

Parameters:

<Mode>   float

                          Range: 1 to 32

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "Mode" on page 441

SCENario:DF:EMITter:MODE:BEAM <Beam>
SCENario:EMITter:MODE:BEAM <Beam>
SCENario:CEMit:EMITter:MODE:BEAM <Beam>
SCENario:LOCalized:EMITter:MODE:BEAM <Beam>

Sets the used beam of the current mode.

Parameters:

<Beam>   float

                          Range: 1 to 32

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "Beam" on page 441

SCENario:DF:DIRection:TRACk <Track>
SCENario:LOCalized:DIRection:TRACk <Track>

Turns the antenna in the direction of the receiver.

Parameters:

<Track>   ON | OFF | 1 | 0

Example: See Example"Creating a scenario with multiple emitters/interfer-
                       ers and background emitters" on page 730

Manual operation: See "Attitude" on page 445

SCENario:DF:EMITter:MODE:TRACkrec <Trackrec>
SCENario:CEMit:EMITter:MODE:TRACkrec <Trackrec>
SCENario:LOCalized:EMITter:MODE:TRACkrec <Trackrec>

If enabled, the scan follows the receiver automatically.

Parameters:

<Trackrec>   ON | OFF | 1 | 0

Manual operation: See "Track Receiver" on page 441

SCENario:DF:LOCation:PSTep:STATe <State>
SCENario:LOCalized:LOCation:PSTep:STATe <State>

Enables the definition of multiple discrete postions for the same emitter/interferer.

Parameters:

<State>   ON | OFF | 1 | 0

Example: See Example"Configuring position steps" on page 734.

SCENario:DF:DISTance <Distance>
SCENario:LOCalized:DISTance <Distance>

Sets the distance to the receiver.

Parameters:

<Distance>   float

                       Range: 0 to 1e+09
                       Default unit: m

Example: See Example"Creating a scenario with multiple emitters/interfer-
                       ers and background emitters" on page 730

Manual operation: See "East, North, Height/Distance, Azimuth, Elevation"
                       on page 444

SCENario:DF:LOCation:EAST <East>
SCENario:DF:LOCation:NORTh <North>

SCENario:LOCalized:LOCation:EAST <East>
SCENario:LOCalized:LOCation:NORTh <North>

Sets the emitter coordinates.

Parameters:

<North>	float
	Range: -1e+09 to 1e+09 Default unit: m
Example:	See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730
Manual operation:	See "East, North, Height/Distance, Azimuth, Elevation" on page 444

SCENario:DF:LOCation:AZIMuth <Azimuth>
SCENario:LOCalized:LOCation:AZIMuth <Azimuth>

Sets the azimuth.

Parameters:

<Azimuth>	float
	Range: 0 to 360
| **Example:** | ```
SCENario:LOCalized:LOCation:NORTh 7000
SCENario:LOCalized:LOCation:EAST 0
SCENario:LOCalized:LOCation:ALTitude 0

SCENario:LOCalized:DISTance?
// 7000
SCENario:LOCalized:LOCation:AZIMuth?
// 0
SCENario:LOCalized:LOCation:ELEVation?
// 0

SCENario:LOCalized:LOCation:AZIMuth 90
SCENario:LOCalized:LOCation:NORTh?
// 4.28626e-13
SCENario:LOCalized:LOCation:EAST?
// 7000

SCENario:LOCalized:LOCation:ALTitude 10
SCENario:LOCalized:LOCation:ELEVation?
// 0.0818511
SCENario:LOCalized:DISTance?
// 7000.01
``` |
| **Manual operation:** | See "East, North, Height/Distance, Azimuth, Elevation" on page 444 |

SCENario:DF:LOCation:ELEVation <Elevation>
SCENario:LOCalized:LOCation:ELEVation <Elevation>

Sets the elevation.

Parameters:

<Elevation>   float

                     Range: -90 to 90

Example: See `SCENario:LOCalized:LOCation:AZIMuth`
                     on page 750

Manual operation: See "East, North, Height/Distance, Azimuth, Elevation"
                     on page 444

SCENario:DF:RECeiver:LATitude <Latitude>
SCENario:DF:RECeiver:LONGitude <Longitude>
SCENario:LOCalized:RECeiver:LATitude <Latitude>
SCENario:LOCalized:RECeiver:LONGitude <Longitude>

Sets the latitude/longitude coordinates of the static receiver.

Parameters:

<Longitude>   float

                     Range: -180 to 180

Example: See Example"Direction finding scenario" on page 732.

Manual operation: See "Position > Latitude, Longitude, Altitude, Yaw, Pitch, Roll"
                     on page 351

SCENario:DF:RECeiver:HEIGht <Height>
SCENario:DF:LOCation:HEIGht <Height>
SCENario:LOCalized:RECeiver:HEIGht <Height>
SCENario:LOCalized:LOCation:HEIGht <Height>

Sets the height of the antenna.

Parameters:

<Height>   float

                     Range: -1e+09 to 1e+09

Example: See Example"Creating a scenario with multiple emitters/interfer-
                     ers and background emitters" on page 730

Manual operation: See "East, North, Height/Distance, Azimuth, Elevation"
                     on page 444

SCENario:DF:WAVeform:ANTenna <Antenna>
SCENario:LOCalized:RECeiver:ANTenna <Antenna>
SCENario:LOCalized:WAVeform:ANTenna <Antenna>

Assigns an existing antenna pattern, see `ANTenna:CATalog?` on page 587.

Parameters:

<Antenna>   string

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

SCENario:DF:WAVeform:SCAN <Scan>
SCENario:LOCalized:RECeiver:SCAN <Scan>
SCENario:LOCalized:WAVeform:SCAN <Scan>

Assigns an existing antenna scan, see `SCAN:CATalog?` on page 587.

Parameters:

<Scan>   string

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

SCENario:DF:TYPE <Type>
SCENario:LOCalized:TYPE <Type>

Defines whether an emitter/interferer is configured.

Parameters:

<Type>   EMITter | | WAVeform

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "Type" on page 438

SCENario:LOCalized:RECeiver:GAIN <Gain>

Sets the antenna Gain.

Parameters:

<Gain>   float

                     Range: -120 to 120

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "Gain" on page 351

SCENario:DF:SEQuence <Sequence>
SCENario:LOCalized:SEQuence <Sequence>

Assigns a sequence to the background signal.

Parameters:

<Sequence>   string

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "Background Signals" on page 428

SCENario:DF:WAVeform <Waveform>
SCENario:LOCalized:WAVeform <Waveform>

Assigns an existing emitter or an existing waveform, see `WAVeform:CATalog?` and `EMITter:CATalog?`.

Parameters:
<Waveform>   string

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "Emitter/Interferer Name" on page 438

SCENario:DF:WAVeform:LEVel <Level>
SCENario:DF:WAVeform:EIRP <Eirp>
SCENario:LOCalized:WAVeform:LEVel <Level>
SCENario:LOCalized:WAVeform:EIRP <Eirp>

Sets the EIRP of the interferer.

Parameters:
<Eirp>   float

                          Range: -200 to 200

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "EIRP" on page 440

SCENario:DF:WAVeform:FREQuency <Frequency>
SCENario:LOCalized:WAVeform:FREQuency <Frequency>

Sets the frequency of the interferer.

Parameters:
<Frequency>   float

                          Range: 1000 to 1e+11

Example: See Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730.

Manual operation: See "Frequency" on page 441

SCENario:DF:FREQuency?
SCENario:LOCalized:FREQuency?
SCENario:CEMit:FREQuency?
SCENario:CPDW:FREQ <Freq>

Sets the frequency for the selected emitter.

Parameters:

<Freq>   float

                           Range: -1000 to 1e+11

Example: See Example"Creating and configuring PDW list scenario"
                           on page 735.

Manual operation: See "Frequency" on page 515

SCENario:CEMit:FQOFfset <Fqoffset>

Sets the frequency offset for the selected emitter.

Parameters:

<Fqoffset>   float

                           Range: -2e+09 to 2e+09

Manual operation: See "Frequency offset" on page 315

SCENario:CEMit:SCNDelay <Scndelay>

Sets the scan delay for the selected emitter.

Parameters:

<Scndelay>   float

                           Range: -3600 to 3600

Example: SCENario:CEMit:SCNDelay 100

Manual operation: See "Scan delay" on page 315

SCENario:CEMit:LVABs <Lvabs>
SCENario:CPDW:LVABs <Lvabs>

Sets the absolute level for the selected PDW list.

Parameters:

<Lvabs>   float

                           Range: -130 to 30

Example: See Example"Creating and configuring PDW list scenario"
                           on page 735.

Manual operation: See "Absolute Level" on page 515

SCENario:DF:SYNChronize:ENABle <Enable>
SCENario:LOCalized:SYNChronize:ENABle <Enable>

Enables synchronized setup.

Parameters:

<Enable>   ON | OFF | 1 | 0

SCENario:DF:RECeiver <Receiver>

Selects an existing receiver, see `RECeiver:CATalog?` on page 587.

Parameters:

<Receiver>   string

Example: `RECeiver:CATalog?`
                       `// "Receiver DF","DF"`
                       `SCENario:DF:RECeiver "Receiver DF"`

Manual operation: See "Receiver" on page 358

SCENario:CSEQuence <Csequence>

Select an existing sequence, see `SEQuence:CATalog?` on page 587.

Parameters:

<Csequence>   string

Example: See Example"Creating sequence collection scenario"
                       on page 729

Manual operation: See "Sequence (Collection) Scenario Settings" on page 139

SCENario:CSEQuence:VARiable <Variable>

Sets the collection variable.

Parameters:

<Variable>   string

Example: `SCENario:CSEQuence:VARiable?`

Manual operation: See "Variables" on page 210

SCENario:DF:MAPS:ENABle <Enable>
SCENario:LOCalized:MAPS:ENABle <Enable>

Enable maps for the selected scenario. This operation cannot be undone.

Parameters:

<Enable>   ON | OFF | 1 | 0

SCENario:DF:MAPS:LOAD <Load>
SCENario:LOCalized:MAPS:LOAD <Load>

This command loads a georeferenced map for the selected scenario.

Supported formats:

- `.tif`
- `.tiff`

Setting parameters:
<Load>

Usage:　　　　　　　　Setting only

SCENario:DF:MCHG:STATe <State>
SCENario:CEMit:MCHG:STATe <State>
SCENario:LOCalized:MCHG:STATe <State>

Enables mode changes.

Parameters:
<State>　　　　　　　　ON | OFF | 1 | 0

Example:　　　　　　　See Example"Enabling mode changing" on page 733.

Manual operation:　　See "Single Mode" on page 439

SCENario:DF:MCHG:STARt <Start>
SCENario:DF:MCHG:STOP <Stop>
SCENario:CEMit:MCHG:STARt <Start>
SCENario:CEMit:MCHG:STOP <Stop>
SCENario:LOCalized:MCHG:STARt <Start>
SCENario:LOCalized:MCHG:STOP <Stop>

Sets the start and end time per mode entry.

Parameters:
<Stop>　　　　　　　　float

　　　　　　　　　　　*RST:　　　0

Example:　　　　　　　See Example"Enabling mode changing" on page 733.

Manual operation:　　See "Timing" on page 440

SCENario:CEMit:MCHG:CLEar
SCENario:DF:MCHG:CLEar
SCENario:LOCalized:MCHG:CLEar

Removes all defined modes.

Example:　　　　　　　See Example"Enabling mode changing" on page 733.

Usage:　　　　　　　　Event

Manual operation:　　See " Append, Remove Current, Remove All" on page 440

SCENario:DF:EMITter:STATe:ENABle <Enable>
SCENario:LOCalized:EMITter:STATe:ENABle <Enable>

Enables that an emitter can use on and off states.

Parameters:
<Enable>　　　　　　　ON | OFF | 1 | 0

Example:	See Example"Configuring different modes" on page 734.
Manual operation:	See "Use on/off states with emitter mode" on page 449

SCENario:DF:EMITter:STATe:DURation <Duration>
SCENario:LOCalized:EMITter:STATe:DURation <Duration>

Sets the duration during that the emitter remains in the current state.

Parameters:

<Duration>	float
	Range: -1e+06 to 1e+06

Example:	See Example"Configuring different modes" on page 734.
Manual operation:	See "Duration" on page 450

SCENario:DF:EMITter:STATe:VALue <Value>
SCENario:LOCalized:EMITter:STATe:VALue <Value>

Sets the emitter state during the selected period.

Parameters:

<Value>	ON | OFF | 1 | 0

Example:	See Example"Configuring different modes" on page 734.
Manual operation:	See "State" on page 450

SCENario:DF:EMITter:STATe:LOOP <Loop>
SCENario:LOCalized:EMITter:STATe:LOOP <Loop>

Repeats the states definition cyclically.

Parameters:

<Loop>	ON | OFF | 1 | 0

Example:	See Example"Configuring different modes" on page 734.
Manual operation:	See "Execute States in Loop" on page 450

SCENario:CPDW:NAME <Name>

Selects the waveform element, used to import the PDW list.

Query the list of waveform elements with the command `WAVeform:CATalog?`.

Parameters:

<Name>	string

Example:	See Example"Creating and configuring PDW list scenario" on page 735.
Manual operation:	See "PDW List" on page 515

SCENario:CPDW:ALIas <Alias>

Enters an alias name.

Parameters:

<Alias>   string

Example: See Example"Creating and configuring PDW list scenario"
                     on page 735.

Manual operation: See "Alias Name" on page 515

SCENario:DF:ENABle <Enable>
SCENario:LOCalized:ENABle <Enable>
SCENario:CEMit:ENABle <Enable>
SCENario:CPDW:ENABle <Enable>

If enabled, the PDW list is included in the output file.

Parameters:

<Enable>   ON | OFF | 1 | 0

Example: See Example"Creating and configuring PDW list scenario"
                     on page 735.

Manual operation: See "Enable" on page 517

SCENario:DF:PRIority <Priority>
SCENario:LOCalized:PRIority <Priority>
SCENario:CEMit:PRIority <Priority>
SCENario:CPDW:PRIority <Priority>

Sets the priority of the selected PDW list , where the higher the value the higher the
priority.

Parameters:

<Priority>   float

                   Range: 1 to 100

Example: See Example"Creating and configuring PDW list scenario"
                     on page 735.

Manual operation: See "Priority [0 = Highest]" on page 517

SCENario:DF:LDELay <Ldelay>
SCENario:LOCalized:LDELay <Ldelay>
SCENario:CEMit:LDELay <Ldelay>
SCENario:CPDW:LDELay <Ldelay>

If interleaving is enabled, shifts the processing of the selected PDW list in time.

Parameters:

<Ldelay>   float

                   Range: -1e+09 to 1e+09

Example:	See Example"Creating and configuring PDW list scenario" on page 735.
Manual operation:	See "Time Offset" on page 517

SCENario:DF:LEVel <Level>
SCENario:LOCalized:LEVel <Level>
SCENario:CEMit:LEVel <Level>
SCENario:CPDW:LEVel <Level>

Adds a level offset.

Parameters:

<Level>	float
	Range: -200 to 0
Example:	See Example"Creating and configuring PDW list scenario" on page 735.
Manual operation:	See "Level Offset" on page 518

SCENario:DF:GROup <Group>
SCENario:LOCalized:GROup <Group>
SCENario:CEMit:GROup <Group>
SCENario:CPDW:GROup <Group>

Assigns the emitter to one of the available interleaving groups.

Parameters:

<Group>	string
	Query a list of the alias names of the existing interleaving groups with the command `SCENario:CPDW:GROup:CATalog?`.
Example:	See Example"Creating and configuring PDW list scenario" on page 735.
Manual operation:	See "Group" on page 518

SCENario:CEMit:GROup:CATalog?
SCENario:DF:GROup:CATalog?
SCENario:LOCalized:GROup:CATalog?
SCENario:CPDW:GROup:CATalog?

Queries the alias names of the configured interleaving groups.

Return values:

<Catalog>	string
	A list of coma-separated alias names.
Example:	See Example"Creating and configuring PDW list scenario" on page 736.
Usage:	Query only

Manual operation:	See "Select, Insert, Append/Prepend, Remove, Clear" on page 518

SCENario:CEMit:GROup:ALIas <Alias>
SCENario:DF:GROup:ALIas <Alias>
SCENario:LOCalized:GROup:ALIas <Alias>
SCENario:CPDW:GROup:ALIas <Alias>

Sets an alias name for the selected interleaving group.

See also `ASSignment:GROup:SELect` on page 640.

Parameters:

<Alias>	string
Example:	See Example"Creating and configuring PDW list scenario" on page 736.
Manual operation:	See "Alias Name" on page 519

SCENario:OUTPut:CLIPping <Clipping>

Sets a maximum level to limit the dynamic range of the signal. Pulses at levels above this threshold are reduced (clipped) to the configured level.

Parameters:

<Clipping>	float
	Range: -100 to 20 Default unit: dBm
Manual operation:	See "Clipping Level" on page 478

SCENario:OUTPut:CLOCk:MODE <Mode>

Sets the clock mode.

Parameters:

<Mode>	AUTO | MANual
	AUTO Clock rate is retrieved from the generated waveform.
	MANual Clock rate is user-defined
Example:	See Example"Creating simple pulse train scenario" on page 728
Manual operation:	See "Clock, Duration" on page 479

SCENario:OUTPut:CLOCk:AUTO:OVERsampling <Oversampling>

Sets the minimum oversampling factor.

Parameters:

<Oversampling>	float
	Range: 1 to 1000

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Clock, Duration" on page 479

SCENario:OUTPut:CLOCk:AUTO:BORDer <Border>

Sets the minimum clock rate.

Parameters:

<Border>	float
	Range: 1000 to 1e+08

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Clock, Duration" on page 479

SCENario:OUTPut:CLOCk:USER <User>

Sets a user defined clock rate.

Parameters:

<User>	float
	Range: 1 to 2.4e+09

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Clock, Duration" on page 479

SCENario:OUTPut:DURation:MODE <Mode>

Sets how the waveform duration is defined.

Parameters:

<Mode>	AUTO | MANual
	AUTO Sets the simulation time to maximum of sequence, scan or movement duration.
	MANual Sets the simulation time to a fixed value.

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Duration" on page 125

SCENario:OUTPut:DURation:TIME <Time>

Sets the duration of the generated waveform.

Parameters:

<Time>   float

Range: 1e-06 to 1.8432e+06
Simulation time longer than 7200s requires R&S SMW with firm-
ware version 5.xx.xxx and higher. To query the installed firmware
version of the selected instrument, use the command
`INSTrument:FIRMware?` on page 647.

Example: See Example"Creating simple pulse train scenario"
on page 728.

Manual operation: See "Duration" on page 125

SCENario:OUTPut:DURation:AUTO?

Requires `SCENario:OUTPut:DURation:MODE AUTO`.

Queries the value of the automatically determined signal duration.

Return values:

<Auto>   float

Range: 1e-06 to 1.8432e+06

Usage: Query only

Manual operation: See "Duration" on page 125

SCENario:OUTPut:FORMat <Format>

Sets the type of the generated waveform file.

Parameters:

<Format>   WV | MSW

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Output Format" on page 477

SCENario:OUTPut:RECall:ENABle <Enable>

Stores the current signal generator configuration as a save/recall file.

Parameters:

<Enable>   ON | OFF | 1 | 0

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Create save/recall configuration" on page 478

SCENario:TRIGger:TIME:STATe <State>

Activates a time-based trigger.

Parameters:

<State>   ON | OFF | 1 | 0

Manual operation: See "Start Time" on page 126

SCENario:TRIGger:TIME:DATE <Date>

Sets the date when the simulation is started.

Parameters:
<Date>

Manual operation: See "Start Time" on page 126

SCENario:TRIGger:TIME:TIME <Time>

Sets the time when the simulation is started.

Parameters:
<Time>

Manual operation: See "Start Time" on page 126

SCENario:TRIGger:TIME:REFerence <Reference>

Sets the time reference in synchronization to the common time reference of the signal generator in the test setup.

Parameters:

<Reference>   GENTime | EXTernal

GENTime
Use this setting if the signal generator works with its internal clock as timebase for triggering.

EXTernal
Use this setting if the signal generator works with an external time source as timebase for triggering.

Manual operation: See "Start Time" on page 126

SCENario:TRIGger:TIME:SOURce <Source>

Select a signal generator as time source for time-based triggering.

You can select a certain signal generator as time source in a multi instrument setup, where no primary and secondary instrument is defined.

Parameters:

<Source>   string

Manual operation: See "Start Time" on page 126

SCENario:OUTPut:FREQuency <Frequency>

Sets the carrier RF frequency of the generated signal.

Parameters:
<Frequency>   float

Range: 1000 to 1e+11

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Frequency" on page 127

SCENario:OUTPut:LEVel <Level>

Sets the reference level used by the calculation of the pulse envelope.

Parameters:
<Level>   float

Range: -130 to 30

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Level" on page 127

SCENario:OUTPut:PATH <Path>

Sets the directory the generated waveform is stored in.

Parameters:
<Path>   string

File path

Example: See Example"Creating a simple emitter scenario" on page 729

Manual operation: See "Config" on page 130

SCENario:OUTPut:SUPRess:ENABle <Enable>

Enable to prevent waveform recalculation if the RF frequency is changed.

Parameters:
<Enable>   ON | OFF | 1 | 0

Example: SCENario:OUTPut:SUPRess:ENABle 1

Manual operation: See "No Recalculation" on page 128

SCENario:OUTPut:RESet:ENABle <Enable>

Restarts the connected instrument on scenario start.

Parameters:
<Enable>   ON | OFF | 1 | 0

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Initial Reset" on page 128

SCENario:OUTPut:RUNMode <Runmode>

Defines the way the generated signal is processed.

Parameters:
<Runmode>   CONTinuous | SINGle

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Play Mode" on page 128

SCENario:OUTPut:TARGet <Target>

Defines whether the software creates an ARB file or transfers the generated waveform to a connected physical generator.

To assign a generator, use the command `SCENario:GENerator` on page 745.

To set the name and the directory the ARB file is stored in, use the command `SCENario:OUTPut:PATH`.

Parameters:
<Target>   INSTrument | FILE

Firmware/software: see Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Assign" on page 131

SCENario:CEMit:THReshold <Threshold>
SCENario:CPDW:THReshold <Threshold>
SCENario:LOCalized:THReshold <Threshold>
SCENario:DF:THReshold <Threshold>
SCENario:OUTPut:THReshold <Threshold>

Sets a threshold. Pulses at levels below this threshold are omitted.

Parameters:
<Threshold>   float

                         Range: -100 to 0

Example: See Example"Creating simple pulse train scenario" on page 728

Manual operation: See "Threshold" on page 328

SCENario:OUTPut:ARB:DETails:ALBS <Albs>

Enables you to calculate the antenna attenuation for each sample. Otherwise a lookup at center position is used.

Parameters:

<Albs>   ON | OFF | 1 | 0

Manual operation: See "ARB Details" on page 478

SCENario:OUTPut:ARB:DETails:TRUNcate <Truncate>

Enables signal truncation.

If enabled, the calculation process allows truncated signals at the end of the signal duration, for example, only a fraction of a pulse if the time ends within the last pulse.

Parameters:

<Truncate>   ON | OFF | 1 | 0

Manual operation: See "ARB Details" on page 478

SCENario:OUTPut:MULTithread <Multithread>

Enable to optmize the calculation speed.

Parameters:

<Multithread>   ON | OFF | 1 | 0

Example: `SCENario:OUTPut:MULTithread ON`

Manual operation: See "Optimize calculation speed" on page 479

SCENario:OUTPut:MTMode <Mtmode>

If multithreading is enabled with `SCENario:OUTPut:MULTithread` on page 766, sets the mode to use for multithreading.

Parameters:

<Mtmode>   AUTO | MANual

Example: `SCENario:OUTPut:MTMode?`
                         `// SCENario:OUTPut:MTMode AUTO`

Manual operation: See "Optimize calculation speed" on page 479

SCENario:OUTPut:MTTHreads <Mtthreads>

In manual mode, sets the required number of threads for the signal calculation.

Parameters:

<Mtthreads>   float

                        Range: 0 to 1000

Example: `SCENario:OUTPut:MTMode`
                         `MANUAL SCENario:OUTPut:MTTHreads?`
                         `// SCENario:OUTPut:MTTHreads 16`

Manual operation: See "Optimize calculation speed" on page 479

SCENario:VOLatile:SEL <Sel>

If several files are created, select the one to be visualized.

Parameters:

<Sel>	float
	Subsequent number, indicating the files in the volatile memory.

Manual operation: See "Results" on page 131

SCENario:VOLatile:VIEW

If a waveform exists in the volatile memory, opens the "Waveform Viewer" and displays this waveform.

Usage: Event

Manual operation: See "Results" on page 131

SCENario:SANitize <Sanitize>

Removes uploaded waveforms from the hard disk of the signal generator.

Setting parameters:

<Sanitize>	SCENario | REPository | ALL
	SCENario
	Removes the current scenario waveforms
	REPository
	Removes the waveforms of all scenarios from the current repository
	ALL
	Removes all waveforms created by the R&S Pulse Sequencer

Example: SCENario:SANitize SCENario

Usage: Setting only

Manual operation: See "Config" on page 130

SCENario:TRIGger

Triggers the scenario, if separate trigger for scenario start is enabled.

See PROGram:SCENario:XTRG:ENABle.

Usage: Event

Manual operation: See "Start preview after trigger" on page 127

SCENario:WAVeform <Waveform>

Specify the name of the "Signal Generation" output file.

Parameters:

<Waveform>   string

25.22   Moving emitters/interferers and receivers commands

Example: Moving emitters and receivers

The following is a simple example on how to enable a moving emitter. The same configuration steps apply for interferer and the receivers in the localized emitters and direction finding scenarios.

```
SCENario:NAME "Dynamic Scenario"
SCENario:LOCalized:TYPE EMITTER
SCENario:LOCalized:ALIas "Plane"
SCENario:LOCalized:MOVement:VEHicle AIRPLANE
SCENario:LOCalized:LOCation:PMODe MOVING
SCENario:LOCalized:MOVement:TYPE WAYPOINT
SCENario:LOCalized:MOVement:WAYPoint "C:/Users/Public/Documents/Rohde-Schwarz/
Pulse Sequencer/Example Files/Movements/Waypoints/Munich_Flight.xtd"
SCENario:LOCalized:MOVement:RFRame WGS
SCENario:LOCalized:MOVement:RMODe ONEWAY

SCENario:LOCalized:MOVement:VFIle "Big_Aircraft.xvd"
SCENario:LOCalized:MOVement:SMOothening 1


SCENario:LOCalized:MOVement:ATTitude MOTION
SCENario:LOCalized:MOVement:YAW?
// 0
SCENario:LOCalized:MOVement:PITCh?
// 0
SCENario:LOCalized:MOVement:ROLL 1.E-01
SCENario:LOCalized:MOVement:IMPort

SCENario:LOCalized:ADD
SCENario:LOCalized:TYPE EMITTER
SCENario:LOCalized:ALIas "Ship"
SCENario:LOCalized:MOVement:VEHicle SHIP
SCENario:LOCalized:LOCation:PMODe MOVING


SCENario:LOCalized:MOVement:TYPE ARC
// start position
SCENario:LOCalized:LOCation:EAST 2.386050048828E+03
SCENario:LOCalized:LOCation:NORTh 1.016520019531E+03
SCENario:LOCalized:LOCation:HEIGht 0.E+00
SCENario:LOCalized:DISTance 4.767236816406E+03
SCENario:LOCalized:LOCation:AZIMuth 6.692472065778E+01
SCENario:LOCalized:LOCation:ELEVation 0.E+00
// center position
```

```
SCENario:LOCalized:MOVement:SPEed 4.E+01
SCENario:LOCalized:MOVement:ANGLe 1.6988E+02
SCENario:LOCalized:MOVement:EAST 2.3103E+03
SCENario:LOCalized:MOVement:NORTh 1.59021E+03
SCENario:LOCalized:MOVement:RMODe ROUNDTRIP

SCENario:LOCalized:RECeiver:MOVement:VEHicle AIRPLANE
SCENario:LOCalized:LOCation:REC:PMODe MOVING
SCENario:LOCalized:RECeiver:MOVement:TYPE LINE
// start position
SCENario:LOCalized:RECeiver:LATitude 0.E+00
SCENario:LOCalized:RECeiver:LONGitude 0.E+00
SCENario:LOCalized:RECeiver:HEIGht 4.E+03


SCENario:LOCalized:RECeiver:MOVement:SPEed 1.8E+02
SCENario:LOCalized:RECeiver:MOVement:ACCeleration 0.E+00
// end position
SCENario:LOCalized:RECeiver:MOVement:EAST -1.360382727273E+03
SCENario:LOCalized:RECeiver:MOVement:NORTh 5.4037425E+03
SCENario:LOCalized:RECeiver:MOVement:HEIGht 4.E+03
SCENario:LOCalized:RECeiver:MOVement:RMODe CYCLic
```

SCENario:DF:LOCation:PMODe <Pmode>
SCENario:LOCalized:LOCation:PMODe <Pmode>

Sets if the emitter is static or moving.

Parameters:

<Pmode>	STATic | STEPs | MOVing

Example:	See Example"Moving emitters and receivers" on page 768.
Manual operation:	See "Mode" on page 443

SCENario:DF:LOCation:REC:PMODe <Pmode>
SCENario:LOCalized:LOCation:REC:PMODe <Pmode>

Sets if the receiver is static or moving.

Parameters:

<Pmode>	STATic | MOVing

Example: See Example"Moving emitters and receivers" on page 768.

Manual operation: See "Mode" on page 360

SCENario:DF:MOVement:TYPE <Type>
SCENario:DF:RECeiver:MOVement:TYPE <Type>
SCENario:LOCalized:RECeiver:MOVement:TYPE <Type>
SCENario:LOCalized:MOVement:TYPE <Type>

Defines the trajectory shape.

Setting the parameter to "NONE" sets the positioning mode to "static".

Parameters:
<Type>   LINE | ARC | WAYPoint | TRACe | NONE

Example: See Example"Moving emitters and receivers" on page 768.

Manual operation: See "Trajectory" on page 452

SCENario:DF:LOCation:ALTitude <Altitude>

Use for defining the altitude of a fixed emitter (no movement) on a georeferenced map.

Parameters:
<Altitude>   float

                    Range: -1e+09 to 1e+09

Example:
```
SCEN:DF:LOC:ALT 42
SCEN:DF:LOC:LAT 48.1351
SCEN:DF:LOC:LONG 11.5820
```

Manual operation: See "Longitude, Latitude, Altitude" on page 444

SCENario:LOCalized:LOCation:ALTitude <Altitude>

Sets the altitude of the antenna.

Parameters:
<Altitude>   float

                    Range: -1e+09 to 1e+09

Example: see Example"Creating a scenario with multiple emitters/interferers and background emitters" on page 730

Manual operation: See "Longitude, Latitude, Altitude" on page 444

SCENario:DF:LOCation:LATitude <Latitude>
SCENario:DF:LOCation:LONGitude <Longitude>
SCENario:LOCalized:LOCation:LATitude <Latitude>
SCENario:LOCalized:LOCation:LONGitude <Longitude>

Use for defining the position of a fixed emitter (no movement) on a georeferenced map.

Positive values represent °East.

Negative values represent °West.

Parameters:

<Longitude>   float

                            Range: -180 to 180

Example: SCEN:LOC:LOC:ALT 42

                            SCEN:LOC:LOC:LAT 48.1351

                            SCEN:LOC:LOC:LONG 11.5820

Manual operation: See "Longitude, Latitude, Altitude" on page 444

SCENario:DF:MOVement:ALTitude <Altitude>
SCENario:LOCalized:MOVement:ALTitude <Altitude>

Use for defining the altitude of a moving emitter (line trajectory) on a georeferenced map. Use to define the altitude of the end-points of the line.

Parameters:

<Altitude>   float

                            Range: -1e+09 to 1e+09

Example: SCEN:LOC:MOV:ALT 42

                            SCEN:LOC:MOV:LAT 48.1351

                            SCEN:LOC:MOV:LONG 11.5820

Manual operation: See "End Position" on page 453

SCENario:DF:MOVement:LATitude <Latitude>
SCENario:DF:MOVement:LONGitude <Longitude>
SCENario:LOCalized:MOVement:LATitude <Latitude>
SCENario:LOCalized:MOVement:LONGitude <Longitude>

Use for defining the movement of an emitter (line trajectory) on a georeferenced map. Use to define the end-points of the line.

Positive values represent °East.

Negative values represent °West.

Parameters:

<Longitude>   float

                            Range: -180 to 180

Example: SCEN:LOC:MOV:ALT 42

                            SCEN:LOC:MOV:LAT 48.1351

                            SCEN:LOC:MOV:LONG 11.5820

Manual operation: See "End Position" on page 453

SCENario:DF:MOVement:CLATitude <Clatitude>
SCENario:DF:MOVement:CLONgitude <Clongitude>

SCENario:LOCalized:MOVement:CLATitude <Clatitude>
SCENario:LOCalized:MOVement:CLONgitude <Clongitude>

Use for defining the movement of an emitter (arc trajectory) on a georeferenced map. Use to define the center-point of the arc.

Positive values represent °East.

Negative values represent °West.

Parameters:

<Clongitude>	float
	Range: -180 to 180

Example: `SCEN:LOC:MOV:CLAT 48.1351`
                      `SCEN:LOC:MOV:CLONG 11.5820`

Manual operation: See "Center Position" on page 454

SCENario:DF:RECeiver:MOVement:PSTep:SELect <Select>
SCENario:LOCalized:RECeiver:MOVement:PSTep:SELect <Select>

Selects the specified point on a trace trajectory.

Parameters:

<Select>	float
	*RST: 1

Example: `SCEN:LOC:REC:MOV:PST:SEL 7`

Manual operation: See "Trace Points" on page 367

SCENario:DF:MOVement:SPEed <Speed>
SCENario:DF:RECeiver:MOVement:SPEed <Speed>
SCENario:LOCalized:RECeiver:MOVement:SPEed <Speed>
SCENario:LOCalized:MOVement:SPEed <Speed>

Sets the speed of the moving emitter.

Parameters:

<Speed>	float
	Range: 0 to 5999

Example: See Example"Moving emitters and receivers" on page 768.

Manual operation: See "Speed" on page 453

SCENario:DF:MOVement:ACCeleration <Acceleration>
SCENario:DF:RECeiver:MOVement:ACCeleration <Acceleration>
SCENario:LOCalized:RECeiver:MOVement:ACCeleration <Acceleration>
SCENario:LOCalized:MOVement:ACCeleration <Acceleration>

Sets the acceleration of the moving emitter.

Parameters:

<Acceleration>	float
	Range: -100 to 100

Example: See Example"Moving emitters and receivers" on page 768.

Manual operation: See "Acceleration" on page 453

SCENario:DF:MOVement:ANGLe <Angle>
SCENario:DF:RECeiver:MOVement:ANGLe <Angle>
SCENario:LOCalized:RECeiver:MOVement:ANGLe <Angle>
SCENario:LOCalized:MOVement:ANGLe <Angle>

Sets the arc angle and thus defines the arc length.

Parameters:

<Angle>	float
	Range: -360 to 360

Example: See Example"Moving emitters and receivers" on page 768.

Manual operation: See "Angle" on page 454

SCENario:DF:MOVement:EAST <East>
SCENario:DF:RECeiver:MOVement:EAST <East>
SCENario:LOCalized:RECeiver:MOVement:EAST <East>
SCENario:LOCalized:MOVement:EAST <East>
SCENario:DF:MOVement:NORTh <North>
SCENario:DF:RECeiver:MOVement:NORTh <North>
SCENario:LOCalized:RECeiver:MOVement:NORTh <North>
SCENario:LOCalized:MOVement:NORTh <North>

Sets the East/North coordinates of the emitter at the end of the movement.

Parameters:

<North>	float
	Range: -1e+09 to 1e+09

Example: See Example"Moving emitters and receivers" on page 768.

Manual operation: See "End Position" on page 453

SCENario:DF:MOVement:HEIGht <Height>
SCENario:DF:RECeiver:MOVement:HEIGht <Height>
SCENario:LOCalized:RECeiver:MOVement:HEIGht <Height>
SCENario:LOCalized:MOVement:HEIGht <Height>

Sets the height of the emitter at the end of the movement.

Parameters:

<Height>	float
	Range: -1e+09 to 1e+09

Example:　　　　　　　See Example"Moving emitters and receivers" on page 768.

Manual operation:　　See "End Position" on page 453

SCENario:DF:MOVement:WAYPoint <Waypoint>
SCENario:DF:RECeiver:MOVement:WAYPoint <Waypoint>
SCENario:LOCalized:RECeiver:MOVement:WAYPoint <Waypoint>
SCENario:LOCalized:MOVement:WAYPoint <Waypoint>

Loads the selected waypoint file.

To import and apply the files, send the command `SCENario:LOCalized:MOVement:IMPort`.

Parameters:

<Waypoint>　　　　　　string

Filename or complete file path, incl. file extension.
Waypoint files must have the extension `*.txt`, `*.kml` or `*.xtd`.
Example files are provided with the software.
For description, see Chapter A.3, "Movement files",
on page 850.

Example:　　　　　　　See Example"Moving emitters and receivers" on page 768.

Manual operation:　　See "▤Load, Delete" on page 458

SCENario:DF:RECeiver:MOVement:WAYPoint:CLEar
SCENario:DF:LOCation:WAYPoint:CLEar
SCENario:LOCalized:LOCation:WAYPoint:CLEar
SCENario:LOCalized:RECeiver:MOVement:WAYPoint:CLEar

Discards the selected file.

Example:　　　　　　　See Example"Moving emitters and receivers" on page 768.

Usage:　　　　　　　　Event

Manual operation:　　See "▤Load, Delete" on page 363

SCENario:DF:MOVement:ATTitude <Attitude>
SCENario:DF:RECeiver:MOVement:ATTitude <Attitude>
SCENario:LOCalized:RECeiver:MOVement:ATTitude <Attitude>
SCENario:LOCalized:MOVement:ATTitude <Attitude>

Defines how the attitude information is defined.

Parameters:

<Attitude>　　　　　　WAYPoint | MOTion | CONStant

WAYPoint
The attitude parameters are extracted from the selected way-
point file.

MOTion
Enables a constant rate of change of the roll.
See `SCENario:LOCalized:MOVement:ROLL` on page 778

Constant
The attitude is constant values.

Example:	See Example"Moving emitters and receivers" on page 768.
Manual operation:	See "Attitude" on page 460

SCENario:DF:RECeiver:MOVement:VEHicle <Vehicle>
SCENario:LOCalized:RECeiver:MOVement:VEHicle <Vehicle>

Assigns the selected icon.

Parameters:

<Vehicle>	LVEHicle | SHIP | AIRPlane | STATionary | RECeiver
Example:	See Example"Moving emitters and receivers" on page 768.
Manual operation:	See "Vehicle Icon" on page 351

SCENario:DF:MOVement:VEHicle <Vehicle>
SCENario:LOCalized:MOVement:VEHicle <Vehicle>

Assigns the selected icon.

Parameters:

<Vehicle>	LVEHicle | SHIP | AIRPlane | STATionary | DEFault | CAR
Example:	See Example"Moving emitters and receivers" on page 768.
Manual operation:	See "Vehicle Icon" on page 439

SCENario:DF:MOVement:VFILe <Vfile>
SCENario:DF:RECeiver:MOVement:VFILe <Vfile>
SCENario:LOCalized:RECeiver:MOVement:VFILe <Vfile>
SCENario:LOCalized:MOVement:VFILe <Vfile>

Loads the selected vehicle description file (`*.xvd`).

To import and apply the files, send the command `SCENario:LOCalized:MOVement:IMPort`.

Parameters:

<Vfile>	string
	Filename or complete file path, incl. file extension. Example files are provided with the software. For description, see Chapter A.4, "Vehicle description files (Used for smoothening)", on page 858.
Example:	See Example"Moving emitters and receivers" on page 768.
Manual operation:	See "▤Load, Delete" on page 459

SCENario:DF:MOVement:VFILe:CLEar
SCENario:DF:RECeiver:MOVement:VFILe:CLEar
SCENario:LOCalized:RECeiver:MOVement:VFILe:CLEar
SCENario:LOCalized:MOVement:VFILe:CLEar

Discards the selected vehicle description file.

Example:	See Example"Moving emitters and receivers" on page 768.
Usage:	Event
Manual operation:	See "▤Load, Delete" on page 459

SCENario:DF:MOVement:YAW <Yaw>
SCENario:DF:RECeiver:MOVement:YAW <Yaw>
SCENario:LOCalized:RECeiver:MOVement:YAW <Yaw>
SCENario:LOCalized:MOVement:YAW <Yaw>
SCENario:DF:MOVement:PITCh <Pitch>
SCENario:DF:RECeiver:MOVement:PITCh <Pitch>
SCENario:LOCalized:RECeiver:MOVement:PITCh <Pitch>
SCENario:LOCalized:MOVement:PITCh <Pitch>
SCENario:DF:MOVement:ROLL <Roll>
SCENario:DF:RECeiver:MOVement:ROLL <Roll>
SCENario:LOCalized:RECeiver:MOVement:ROLL <Roll>
SCENario:LOCalized:MOVement:ROLL <Roll>

Sets the angles of rotation in the corresponding direction.

Parameters:

<Roll>	float	
	Range:	-180 to 180

Example:	See Example"Moving emitters and receivers" on page 768.
Manual operation:	See "Yaw, Pitch, Roll" on page 460

SCENario:DF:MOVement:RFRame <Rframe>
SCENario:DF:RECeiver:MOVement:RFRame <Rframe>
SCENario:LOCalized:RECeiver:MOVement:RFRame <Rframe>
SCENario:LOCalized:MOVement:RFRame <Rframe>

Select the reference frame used to define the emitters coordinates.

Parameters:

<Rframe>	WGS	PZ

Example:	See Example"Moving emitters and receivers" on page 768.
Manual operation:	See "Reference Frame" on page 459

SCENario:DF:MOVement:RMODe <Rmode>
SCENario:DF:RECeiver:MOVement:RMODe <Rmode>

SCENario:LOCalized:RECeiver:MOVement:RMODe <Rmode>
SCENario:LOCalized:MOVement:RMODe <Rmode>

Defines the behavior of the moving object when the end of the trajectory is reached.

Parameters:

<Rmode>   CYCLic | ROUNdtrip | ONEWay

Example: See Example"Moving emitters and receivers" on page 768.

Manual operation: See "Mode" on page 454

SCENario:DF:MOVement:SMOothening <Smoothening>
SCENario:DF:RECeiver:MOVement:SMOothening <Smoothening>
SCENario:LOCalized:RECeiver:MOVement:SMOothening <Smoothening>
SCENario:LOCalized:MOVement:SMOothening <Smoothening>

If a vehicle description file is loaded, activates smoothening.

See `SCENario:LOCalized:MOVement:VFILe` on page 777.

Parameters:

<Smoothening>   ON | OFF | 1 | 0

Example: See Example"Moving emitters and receivers" on page 768.

Manual operation: See "Smoothing for waypoints > State" on page 459

SCENario:DF:MOVement:IMPort
SCENario:DF:RECeiver:MOVement:IMPort
SCENario:LOCalized:RECeiver:MOVement:IMPort
SCENario:LOCalized:MOVement:IMPort

Imports the selected waypoint and vehicle description files into the repository and **applies** them.

Example: See Example"Moving emitters and receivers" on page 768.

Usage: Event

Manual operation: See "Calculate" on page 460

SCENario:DF:MOVement:CLEar
SCENario:DF:RECeiver:MOVement:CLEar
SCENario:LOCalized:RECeiver:MOVement:CLEar
SCENario:LOCalized:MOVement:CLEar

Discards the waypoint and vehicle description file.

Example: See Example"Moving emitters and receivers" on page 768.

Usage: Event

Manual operation: See " Clear" on page 461

SCENario:DF:SUBitem:CURRent <Current>
SCENario:LOCalized:SUBitem:CURRent <Current>
Parameters:

<Current>	float
	Range: 1 to 4096

SCENario:DF:SUBitem:SELect <Select>
SCENario:LOCalized:SUBitem:SELect <Select>
Parameters:

<Select>	float
	Range: 1 to 4096

PREView:POSition?

If movement is enabled, queries the current positions of the TX items and RX items.

Return values:

<Position>   string

Semicolon-separated string with the format:
```
TIME=<time_from_simulation_start>;
ID=<Item ID>;NAME=<Item alias
name>;DIST=<distance>m;LEVATT=<Level at Rx
origin>dBm;
AZI=<Azimuth>deg;ELEV=<Elevation>deg;N=<North>m;
E=<East>m;H=<Height>m;LON=<Longitude>deg;
LAT=<Latitude>deg;ALT=<Altitude>m;VEL=<Velocity>m/s
```

Example:
```
PREView:POSition?
// line breaks added for better readability
// TIME=00:00:24.012;
// ID=1;NAME=Ground;DIST=2323.635km;LEVATT=8dBm;AZI=15.2deg;
ELEV=0.0deg;N=2241.90m;E=610.87m;H=0.00km;LON=0.00549deg;
LAT=0.02028deg;ALT=0.43m;VEL=0.00m/s
// ID=2;NAME=Plane;DIST=3812.188m;LEVATT=-4dBm;AZI=291.9deg;
ELEV=13.9deg;N=1383.20m;E=-3432.21m;H=916.23m;LON=-0.03083deg;
LAT=0.01251deg;ALT=917.30m;VEL=100.00m/s
// ID=3;NAME=Ship;DIST=1055.920m;LEVATT=12dBm;AZI=324.0deg;
ELEV=-11.9deg;N=836.01m;E=-606.84m;H=-218.61m;LON=-0.00545deg;
LAT=0.00756deg;ALT=-218.57m;VEL=100.00m/s
// ID=4;NAME=Receiver;DIST=---;LEVATT=---;AZI=---;ELEV=---;
N=0.00m;E=0.00m;H=0.00m;LON=0.00000deg;LAT=0.00000deg;ALT=0.00m;
VEL=0.00m/s
```

Usage: Query only

Manual operation: See "2D Live Preview Stats" on page 425

25.23 Platform command

PLATform:NAME <Name>

Queries the name of the platform.

Parameters:

<Name>   string

Manual operation: See "Name" on page 337

PLATform:EMITter <Emitter>

The string must be unique within the repository.

Letters, numbers, spaces and some special characters can be used.

Examples of special characters:

- Supported: !$%()=?-+_.
- Not supported: §&/:äöü

Parameters:

<Emitter>   string

Manual operation: See "Emitter" on page 342

PLATform:EMITter:ID <Id>

Queries the ID number of the emitter.

Parameters:

<Id>   float

                           *RST: 1

PLATform:EMITter:ALIas <Alias>

Sets an alias name for the selected platform emitter element.

Parameters:

<Alias>   string

Manual operation: See "List of emitters" on page 339

PLATform:EMITter:ANGLe <Angle>

You can set the position of the selected emitter relative to the platform's origin, using this command combined with `PLATform:EMITter:RADius`.

- `PLATform:EMITter:ANGLe` sets the angle of the emitter element on the azimuth plane, relative to the platform's heading.
- `PLATform:EMITter:RADius` sets the distance of the emitter element on the azimuth plane, relative to the platform's origin.

Parameters:

<Angle>   float

                           Range: 0 to 360
                           Increment: 0.01

Manual operation: See "Angle" on page 344

PLATform:EMITter:AZIMuth <Azimuth>

Angle of the emitter element's pointing direction relative to the platform's heading.

Parameters:

<Azimuth>   float

                           Range: 0 to 360
                           Increment: 0.01

Manual operation: See "Azimuth" on page 342

PLATform:EMITter:BLANkranges <Blankranges>

Creates a set of blank ranges for the currently selected platform emitter.

This approach is more efficient than using several blank range start/stop commands.

The ranges are defined as a set of comma-separated values.

Parameters:
<Blankranges>

Manual operation: See "Set of ranges" on page 345

PLATform:EMITter:BLANkranges:SELect <Select>

Selects a particular blank range for the currently selected platform emitter.

Use with the following commands to configure a blank range:

- `PLATform:EMITter:BLANkranges:STARt` on page 783
- `PLATform:EMITter:BLANkranges:STOP` on page 783

To configure several blank ranges with a single command, you can use `PLATform:EMITter:BLANkranges` on page 782 . This approach is more efficient than using several blank range start/stop commands.

Parameters:

<Select>	float
	*RST: 1

Manual operation: See "No" on page 344

PLATform:EMITter:BLANkranges:STARt <Start>

Sets the start angle for the selected blank range.

The reference value (i.e. 0°) is the configured "Azimuth" value for the selected emitter.

Use together with `PLATform:EMITter:BLANkranges:SELect` on page 592.

To configure several blank ranges with a single command, you can use `PLATform:EMITter:BLANkranges` on page 782 . This approach is more efficient than using several blank range start/stop commands.

Parameters:

<Start>	float
	Range: 0 to 360

Example: `PLATform:EMITter:BLANkranges:SELect 1 ; PLATform:EMITter:BLANkranges:STARt 160`

Manual operation: See "Start" on page 345

PLATform:EMITter:BLANkranges:STOP <Stop>

Sets the stop angle for the selected "Blank Range".

The reference value (i.e. 0°) is the configured "Azimuth" value for the selected emitter.

Use together with `PLATform:EMITter:BLANkranges:SELect` on page 592.

To configure several blank ranges with a single command, you can use `PLATform:EMITter:BLANkranges` on page 782 . This approach is more efficient than using several blank range start/stop commands.

Parameters:

<Stop>	float
	Range: 0 to 360

Example: `PLATform:EMITter:BLANkranges:`
`SELect 1 ; PLATform:EMITter:BLANkranges:STOP`
`200`

Manual operation: See "Stop" on page 345

PLATform:EMITter:DIRection:AWAY <Away>

This command automatically configures the transmission direction of the selected emit-
ter.

No effect if emitter is at origin.

Affects emitters whose position (relative to the origin) has been defined by one of the
following methods:

- Set X and Y values
- Set Angle and Radius values

The transmission direction is configured so that it is directly away from the origin.

Parameters:

<Away>   ON | OFF | 1 | 0

Manual operation: See "Attitude" on page 342

PLATform:EMITter:ELEVation <Elevation>

Elevation of the emitter item's pointing direction, relative to the azimuth plane.

Parameters:

<Elevation>   float

                  Range: -90 to 90

Manual operation: See "Elevation" on page 342

PLATform:EMITter:HEIGht <Height>

Height of the selected emitter element relative to the platform's origin.

Can be used, for example, to differentiate between:

- Radars mounted on different parts of a ship or aircraft.
- Various radars situated across a land-based radar installation.

Parameters:

<Height>   float

                  Range: -500 to 500

Manual operation: See "Height" on page 343

PLATform:EMITter:RADius <Radius>

You can set the position of the selected emitter relative to the platform's origin, using this command combined with `PLATform:EMITter:ANGLe`.

- `PLATform:EMITter:ANGLe` sets the angle of the emitter element on the azimuth plane, relative to the platform's heading.
- `PLATform:EMITter:RADius` sets the distance of the emitter element on the azimuth plane, relative to the platform's origin.

Parameters:

<Radius>   float

                        Range: 0 to 2000

Manual operation: See "Radius" on page 343

PLATform:EMITter:ROLL <Roll>

Roll of the emitter item's pointing direction relative to the platform's up direction.

Can be used, for example, to simulate the emissions from a mast-mounted radar on a marine platform affected by wind.

Parameters:

<Roll>   float

                        Range: -180 to 180

Manual operation: See "Roll" on page 342

PLATform:EMITter:X <X>

Set the position of the selected emitter relative to the platform's origin, using this command combined with `PLATform:EMITter:Y`.

X and Y represent the two principle axis of the platform.
- The Y-axis represents the axis along the center-line of the platform.
  This axis:
  – Corresponds to its heading.
  – Passes through the origin.
- The X-axis:
  – Is at right-angles to the Y-axis.
  – Passes through the origin.

- `PLATform:EMITter:Y` sets the distance of the emitter element from the origin, along the Y-axis.
  Positive values are towards the heading.
- `PLATform:EMITter:X` sets the distance of the emitter element from the origin, along the X-axis.

Parameters:

<X>	float
	Range: -2000 to 2000

Manual operation: See "X" on page 343

PLATform:EMITter:Y <Y>

Set the position of the selected emitter relative to the platform's origin, using this command combined with `PLATform:EMITter:X`.

X and Y represent the two principle axis of the platform.

- The Y-axis represents the axis along the center-line of the platform.
  This axis:
  - Corresponds to its heading.
  - Passes through the origin.

- The X-axis:
  - Is at right-angles to the Y-axis.
  - Passes through the origin.

- `PLATform:EMITter:Y` sets the distance of the emitter element from the origin, along the Y-axis.
  Positive values are towards the heading.
  Step = 0.01 m

- `PLATform:EMITter:X` sets the distance of the emitter element from the origin, along the X-axis.
  Step = 0.01 m

Parameters:

<Y>	float
	Range: -2000 to 2000

Manual operation: See "Y (Heading)" on page 343

25.24 Sequence commands

In the following examples, we assume that a pulse-base sequence "Test Sequence" and the required Pulses have been created.

See also:

- Example"Handling items" on page 581
- Example"Creating an unmodulated pulse" on page 693

Example: Creating a filler segment

```
SCPI
SEQuence:CREate "Test Sequence"
SEQuence:ITEM:ADD
SEQuence:ITEM COUNt?
// 2
SEQuence:ITEM:SELect 2
SEQuence:ITEM:TYPE FILLer
SEQuence:ITEM:FILLer:SIGNal BLANk
SEQuence:ITEM:FILLer:MODE DURation
SEQuence:ITEM:FILLer:TIME FIXed
SEQuence:ITEM:FILLer:TIME:FIXed 5e-3
```

Example: Creating a simple sequence with two pulses repeated different number of times

```
SCPI
SEQuence:SELect "Test Sequence"
SEQuence:TYPE?
SEQuence:PHASe:MODE ABSolute
SEQuence:TIME:MODE PRI

SEQuence:ITEM:ADD
SEQuence:ITEM:SELect 1; SEQeunce:ITEM:TYPE PULSe
SEQunece:ITEM:PULSe "P1"
SEQuence:ITEM:PRI 2.5E-03
// SEQuence:TIME:MODE PRF
// SEQuence:ITEM:PRF?
// 4.E+02
SEQuence:ITEM:PDELay 2E-03
SEQuence:ITEM:REP:TYPE FIXed
SEQuence:ITEM:REP:COUNt:FIXed 2

SEQuence:ITEM:ADD
SEQuence:ITEM:SELect 2; SEQeunce:ITEM:TYPE LOOP
SEQuence:ITEM:LOOP:TYPE VARiable
SEQuence:ITEM:LOOP:COUNt:MINimum 2
SEQuence:ITEM:LOOP:COUNt:MAXimun 10
SEQuence:ITEM:LOOP:COUNt:STEP 2

SEQuence:ITEM:ADD
SEQuence:ITEM:SELect 3; SEQeunce:ITEM:TYPE PULSe
SEQunece:ITEM:PULSe "P1"
SEQuence:ITEM:PRI 5 ms
SEQuence:ITEM:FREQuency:OFFSet 1000000
SEQuence:ITEM:LEVel:OFFSet -10
SEQuence:ITEM:REP:TYPE DURation
SEQuence:ITEM:REP:COUNt:DURation 0.001
SEQuence:ITEM:REP:COUNt:ROUNding UP
SEQuence:ITEM:INDent 1
```

Example: Creating an overlay

```
SCPI
SEQuence:SELect "Test Sequence"
SEQuence:ITEM COUNt?
// 3
SEQuence:ITEM:ADD
SEQuence:ITEM:SELect 4; SEQeunce:ITEM:TYPE PULSe
SEQunece:ITEM:PULSe "P1"
SEQuence:ITEM:PRI 2.5 ms

SEQuence:ITEM:SELect 2
SEQeunce:ITEM:TYPE OVL
SEQuence:ITEM:OVL:WTIMe 0.001

SEQuence:ITEM:SELect 3; SEQuence:ITEM:INDent 1
SEQuence:ITEM:SELect 4; SEQuence:ITEM:INDent 1
```

SEQuence:TYPE <Type>

Sets the sequence type.

Parameters:

<Type>   PULSe | | WAVeform

Example: See Example"Handling items" on page 581

Manual operation: See "Sequence Name, Comment, Type, Sequence Description
                          Table, Block Diagram" on page 201

SEQuence:PHASe:MODE <Mode>

Defines how the phase is set at each pulse start.

Parameters:

<Mode>   ABSolute | CONTinuous | MEMory

Example: See Example"Creating a filler segment" on page 787

Manual operation: See "Phase Mode" on page 201

SEQuence:TIME:MODE <Mode>

Switches between time-based (PRI) and frequency-based (PRF) pulse repetition defi-
nition.

Parameters:

<Mode>   PRI | PRF

Example: See Example"Creating a simple sequence with two pulses
                          repeated different number of times" on page 787

Manual operation: See "Timing Mode" on page 203

SEQuence:ITEM:FILLer:MODE <Mode>

Sets how the filler duration is determined.

Parameters:

<Mode>   DURation | TSYNc

Example: See Example"Creating a filler segment" on page 787

Manual operation: See "Mode" on page 209

SEQuence:ITEM:FILLer:SIGNal <Signal>

Sets the signal type.

Parameters:

<Signal>   BLANk | | CW | HOLD

Example: See Example"Creating a filler segment" on page 787

Manual operation: See "Signal" on page 209

SEQuence:ITEM:FILLer:TIME <Time>

Defines the way the duration is defined.

Parameters:

<Time>   FIXed | EQUation

Example: See Example"Creating a filler segment" on page 787

Manual operation: See "Time" on page 210

SEQuence:ITEM:FILLer:TIME:EQUation <Equation>

Sets the filler duration as an equation.

Parameters:

<Equation>   string

Example: `SEQuence:ITEM:FILLer:TIME:EQUation?`

Manual operation: See "Time" on page 210

SEQuence:ITEM:FILLer:TIME:FIXed <Fixed>

Sets the duration of the filler.

Parameters:

<Fixed>   float

                           Range: 0 to 1e+09
                           Default unit: sec

Example: See Example"Creating a filler segment" on page 787

Manual operation: See "Time" on page 210

SEQuence:ITEM:FREQuency:OFFSet <Offset>

Enables a frequency offset.

Parameters:

<Offset>   float

                           Range: -1e+09 to 1e+09
                           Default unit: Hz

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Δ Freq." on page 205

SEQuence:ITEM:INDent \<Indent\>

Indents the selected item rows to include it, for example, in a loop.

Parameters:

\<Indent\>   float

Range: 0 to 5

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Nesting" on page 203

SEQuence:ITEM:LEVel:OFFSet \<Offset\>

Sets a level offset.

Parameters:

\<Offset\>   float

Range: -100 to 0
Default unit: dB

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Δ Level" on page 205

SEQuence:ITEM:LOOP:COUNt:FIXed \<Fixed\>

Sets the repetition number as a numeric value.

Parameters:

\<Fixed\>   float

Range: 1 to 65535

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Repetition" on page 204

SEQuence:ITEM:LOOP:COUNt:MINimum \<Minimum\>
SEQuence:ITEM:LOOP:COUNt:MAXimum \<Maximum\>

Sets the value range of the loop count.

Parameters:

\<Maximum\>   float

Range: 1 to 65535

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Loop Repetition" on page 207

SEQuence:ITEM:LOOP:COUNt:STEP <Step>

Sets the loop count granularity.

Parameters:

<Step>   float

                     Range: 1 to 65535

Example: See Example"Creating a simple sequence with two pulses
                     repeated different number of times" on page 787

Manual operation: See "Loop Repetition" on page 207

SEQuence:ITEM:LOOP:TYPE <Type>

Sets how the loop repetition is defined.

Parameters:

<Type>   FIXed | VARiable

Example: See Example"Creating a simple sequence with two pulses
                     repeated different number of times" on page 787

Manual operation: See "Loop Repetition" on page 207

SEQuence:ITEM:LOOP:VARiable <Variable>

Sets a loop variable.

Parameters:

<Variable>   string

Example: See Example"Creating a simple sequence with two pulses
                     repeated different number of times" on page 787

Manual operation: See "Loop Variables" on page 207

SEQuence:ITEM:OVL:VARiable <Variable>

Sets a variable.

Parameters:

<Variable>   string

Manual operation: See "Overlay Variables" on page 208

SEQuence:ITEM:OVL:WTIMe <Wtime>

Sets the duration of the overlay.

Parameters:

<Wtime>   float

Range: 0 to 3600
Default unit: sec

Example: See Example"Creating an overlay" on page 788

Manual operation: See "Duration" on page 208

SEQuence:ITEM:PDELay <Pdelay>

Enables a start delay.

Parameters:

<Pdelay>   float

Range: 0 to 1e+09
Default unit: sec

Example: See Example"Creating a simple sequence with two pulses
repeated different number of times" on page 787

Manual operation: See "Delay" on page 205

SEQuence:ITEM:PHASe:OFFSet <Offset>

Sets a phase offset.

Parameters:

<Offset>   float

Range: -180 to 180

Example: SEQuence:ITEM:PHASe:OFFSet 30

Manual operation: See "Phase" on page 205

SEQuence:ITEM:PRF <Prf>
SEQuence:ITEM:PRI <Pri>

Sets the pulse repetition interval (PRI) or the pulse repetition frequency (PRF).

Parameters:

<Pri>   float

Range: 0 to 1e+09

Example: See Example"Creating a simple sequence with two pulses
repeated different number of times" on page 787

Manual operation: See "PRI/PRF" on page 205

SEQuence:ITEM:WAVeform <Waveform>
SEQuence:ITEM:PULSe <Pulse>

Assigns a pulse or a waveform to the selected item.

Use the commands `PULSe:CATalog?` and `WAVeform:CATalog?` to querry the available pulses and waveforms.

Parameters:

<Pulse>	string
	Pulse name

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Pulse/Waveform" on page 204

SEQuence:ITEM:REP:COUNt:DURation <Duration>

Sets a time duration.

Parameters:

<Duration>	float
	Range: 0 to 1e+09
	Default unit: sec

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Repetition Number" on page 206

SEQuence:ITEM:REP:COUNt:FIXed <Fixed>

Sets the repetition number as a numeric value.

Parameters:

<Fixed>	float
	Range: 1 to 65535

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Repetition" on page 204

SEQuence:ITEM:REP:COUNt:MINimum <Minimum>
SEQuence:ITEM:REP:COUNt:MAXimum <Maximum>

Sets the value range of the repetition count.

Parameters:

<Maximum>	float
	Range: 1 to 65535

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Repetition Number" on page 206

SEQuence:ITEM:REP:COUNt:ROUNding <Rounding>

Sets how the repetition number is rounded.

Parameters:

<Rounding>   DN | UP

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Repetition Number" on page 206

SEQuence:ITEM:REP:COUNt:STEP <Step>

Sets the repetition count granularity.

Parameters:

<Step>   float

Range: 1 to 65535

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Repetition Number" on page 206

SEQuence:ITEM:REP:TYPE <Type>

Sets how the repetition number is defined.

Parameters:

<Type>   FIXed | VARiable | DURation

Example: See Example"Creating a simple sequence with two pulses repeated different number of times" on page 787

Manual operation: See "Repetition Number" on page 206

SEQuence:ITEM:REP:VARiable <Variable>

Seta a repetition variable.

Parameters:

<Variable>   string

Manual operation: See "Repetition Variables" on page 206

SEQuence:ITEM:TYPE <Type>

Sets the content type of the selected item.

Parameters:

<Type>   PULSe | FILLer | LOOP | | OVL | SUBSequence | WAVeform

Example: See Example"Creating a filler segment" on page 787

Manual operation: See "Element Type" on page 204

25.25 Status commands

This system contains the commands for the status reporting system.

Refer to the R&S SMW/SMBV/SGT user manual for detailed information.

Value ranges

- **Return parameters**
  Queries return the current value of the respective register, which permits a check of the device status.
  Range: A decimal value in the range 0 to 32767 (=2^{15}-1)
- **Setting parameters**
  The configuration commands set the respective register thus determining which status changes of the R&S Pulse Sequencer cause the status registers to be changed.
  Range: A decimal value in the range 0 to 32767 (=2^{15}-1)

STATus:OPERation:CONDition?

Queries the content of the CONDition part of the STATus:OPERation register. This part contains information on the action currently being performed in the instrument. The content is not deleted after being read out because it indicates the current status.

Return values:

<Condition>	float

Example:	:STATus:OPERation:CONDition? Queries the Status:Operation:Condition register.
Usage:	Query only

25.26 System, message log and program commands

The SYSTem, MSGLog and PROGram subsystems contain a series of commands for general functions which do not directly affect signal generation.

SYSTem:ERRor?

Queries the error/event queue for the oldest item and removes it from the queue. The response consists of an error number and a short description of the error.

Positive error numbers are instrument-dependent. Negative error numbers are reserved by the SCPI standard.

Return values:

<Error>	string

	`Error/event_number,`
	`"Error/event_description>[;Device-dependent`
	`info]"`
	If the queue is empty, the response is `0,"No error"`

Example:	See `SYSTem:ERRor:ALL?` on page 797
Usage:	Query only

SYSTem:ERRor:ALL?

Queries the error/event queue for all unread items and removes them from the queue. The response is a comma-separated list of error number and a short description of the error in FIFO order.

Positive error numbers are instrument-dependent. Negative error numbers are reserved by the SCPI standard.

Return values:

<All>	string

	List of: `Error/event_number,`
	`"Error/event_description>[;Device-dependent`
	`info]"`
	If the queue is empty, the response is `0,"No error"`

Example:	`SYSTem:ERRor:ALL?`
	Queries all entries in the error queue
	`Response: 0, 'no error'`
	No errors have occurred since the error queue was last read out

Usage:	Query only

SYSTem:ERRor:COUNt?

Queries the number of entries in the error queue. If the error queue is empty, '0' is returned.

Return values:

<Count>	string

Example:	`SYSTem:ERRor:COUNt?`
	Queries the number of entries in the error queue
	`Response: 1`
	One error has occurred since the error queue was last read out

Usage:	Query only

SYSTem:PROGress?

Queries the signal generation progress status.

Return values:

\<Progress\>　　　　　　　float

Example:　　　　　　　See Example"Creating simple pulse train scenario" on page 728

Usage:　　　　　　　　Query only

Manual operation:　　See "Config" on page 130

MSGLog:ERRor?

Queries the last error listed in the "Message Log" dialog.

Return values:

\<Error\>　　　　　　　　string

Example:　　　　　　　`MSGLog:POPup 1`
　　　　　　　　　　　　`MSGLog:ERRor?`
　　　　　　　　　　　　`// 09:38:41:837    Cannot open input file.`

Usage:　　　　　　　　Query only

Manual operation: See Chapter 26, "Troubleshooting", on page 831.

MSGLog:POPup \<Popup\>

Opens/closes the "Message Log" dialog.

Parameters:

\<Popup\>　　　　　　　　ON | OFF | 1 | 0

Manual operation: See Chapter 26, "Troubleshooting", on page 831.

PROGram:HIDE
PROGram:SHOW

Minimizes/maximizes the R&S Pulse Sequencer workspace.

Usage:　　　　　　　　Event

25.27 Waveform commands

Example: Creating waveforms

```
SCPI
WAVeform:CREate "NOISe"
WAVeform:SELect "NOISe"
WAVeform:TYPE NOISe
WAVeform:NOISe:BWIDth 2e+6

WAVeform:CREate "WV"
WAVeform:SELect "WV"
WAVeform:TYPE WAVeform
WAVeform:SIGCont COMM
WAVeform:WAVeform:LOAD "C:\_PS_files\dl_IQ_10MBW.wv"
WAVeform:LEVel:REFerence?
// 14.4695
WAVeform:WAVeform:CLEar

WAVeform:CREate "BKG"
WAVeform:SELect "BKG"
WAVeform:TYPE BEMitter
WAVeform:BEMitter:COUNt 64
WAVeform:BEMitter:BWIDth 2e+07
WAVeform:BEMitter:DURation 0.05
WAVeform:BEMitter:PW:MIN 5e-05
WAVeform:BEMitter:PW:MAX 0.00025
WAVeform:BEMitter:PRI:RATio:MIN 10
WAVeform:BEMitter:PRI:RATio:MAX 20
WAVeform:BEMitter:LEVel:RANGe 20
```

WAVeform:TYPE <Type>

Sets the type of the waveform.

Parameters:

<Type>   CW | NOISe | WAVeform | USER | BEMitter | MT | PDW | AIF |
                        APDW | IQDW

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Type" on page 374

WAVeform:NOISe:BWIDth <Bwidth>

Sets the bandwidth of the generated AWGN waveform.

Parameters:

<Bwidth>   float

                        Range: 0 to 2e+09
                        Default unit: Hz

Example: See Example"Creating waveforms" on page 799

Manual operation: See "White noise settings" on page 376

WAVeform:SIGCont <Sigcont>

Defines the waveform signal type.

Parameters:

<Sigcont>   PULSe | COMM

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Settings for importing externally created Rohde & Schwarz
                        waveforms or files with custom I/Q data" on page 377

WAVeform:WAVeform:LOAD <Load>

Load the selected waveform file (`*.wv`), see Table A-1.

Setting parameters:

<Load>   string

                        Complete file path with file name and extension

Example: See Example"Creating waveforms" on page 799

Usage: Setting only

Manual operation: See "Settings for importing externally created Rohde & Schwarz
                        waveforms or files with custom I/Q data" on page 377

WAVeform:LEVel:REFerence <Reference>

Queries the reference level.

Parameters:

<Reference>   float

                   Range: 0 to 100

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Settings for importing externally created Rohde & Schwarz waveforms or files with custom I/Q data" on page 377

WAVeform:IQ:CLEar
WAVeform:WAVeform:CLEar

Removes the imported waveform or file with I/Q data.

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Settings for importing externally created Rohde & Schwarz waveforms or files with custom I/Q data" on page 377

WAVeform:MT:COUNt <Count>

Sets the number of tones.

Parameters:

<Count>   integer

                   Range: 2 to 1024

Example: ```
WAVeform:CREate "MT"
WAVeform:SELect "MT"
WAVeform:TYPE MT
WAVeform:MT:COUNt 64
WAVeform:MT:SPACing 100000
```

Manual operation: See "CW and Multitone waveform settings" on page 374

WAVeform:MT:SPACing <Spacing>

Sets the tone spacing.

Parameters:

<Spacing>   float

                   Range: 100 to 1e+07

Example: See WAVeform:MT:COUNt on page 801

Manual operation: See "CW and Multitone waveform settings" on page 374

WAVeform:BEMitter:COUNt <Count>

Sets the number of background emitters.

Parameters:

<Count>   integer

                           Range: 1 to 255

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Background emitters settings" on page 375

WAVeform:BEMitter:BWIDth <Bwidth>

Sets the maximum permissible bandwidth of the resulting signal.

Parameters:

<Bwidth>   float

                           Range: 1000 to 2e+09

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Background emitters settings" on page 375

WAVeform:BEMitter:DURation <Duration>

Sets the signal duration.

Parameters:

<Duration>   float

                           Range: 0.0001 to 1

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Background emitters settings" on page 375

WAVeform:BEMitter:PRI:RATio:MINimum <Minimum>
WAVeform:BEMitter:PRI:RATio:MAXimum <Maximum>

Sets the value range for the PRI/PW ratio.

Parameters:

<Maximum>   float

                           Range: 10 to 1000

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Background emitters settings" on page 375

WAVeform:BEMitter:PW:MINimum <Minimum>
WAVeform:BEMitter:PW:MAXimum <Maximum>

Sets the value range for the pulse width values.

Parameters:

<Maximum>	float
	Range: 5e-05 to 1

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Background emitters settings" on page 375

WAVeform:BEMitter:LEVel:RANGe <Range>

Sets the maximum level difference between the strongest and the weakest emitter.

Parameters:

<Range>	float
	Range: 0 to 90

Example: See Example"Creating waveforms" on page 799

Manual operation: See "Background emitters settings" on page 375

WAVeform:PDW:CENTer <Center>

During the import of PDW list files, the software evaluates the frequency information in the file and calculates the center frequency of all pulses. The center frequency is calculated as the middle frequency between the min and the max frequency values included in the PDW file. Chirp frequency deviations are also considered. The pulses are calculated relatively to this center frequency.

If the actual frequency of the generator differs from the calculated one, use this command to set the center frequency of the generator.

Parameters:

<Center>	float
	Range: 0 to 1e+11

Example: `WAVeform:PDW:CENTer 1.E+09`

Manual operation: See "AMMOS IF and AMMOS PDW Import" on page 382

25.28 Waveform viewer and signal preview commands

WAVeform:VIEW:XMODe <Xmode>
SCENario:VOLatile:VIEW:XMODe <Xmode>

Sets the units (time or samples) used on the x axis.

Setting parameters:

<Xmode>	SAMPles | TIME

Usage: Setting only

Manual operation: See "Units" on page 403

WAVeform:VIEW:YMODe <Ymode>
SCENario:VOLatile:VIEW:YMODe <Ymode>

Sets the view mode.

Setting parameters:

<Ymode>	IQ | MAGDb | MAGW | MAGV | PHASe | FREQuency | PAV

Usage: Setting only

Manual operation: See "View mode" on page 402

WAVeform:VIEW:ZOOM:POINt <Point>
SCENario:VOLatile:VIEW:ZOOM:POINt <Point>

Sets center point of the displayed area.

Setting parameters:

<Point>	float
	Always related to time
	Default unit: s

Example:
```
SCENario:VOLatile:VIEW:ZOOM:POINt 300 us
SCENario:VOLatile:VIEW:ZOOM:RANGe 100u
// 100 us around the 300 us point, i.e.
// displayed is the time span of 200 us to 400 us

SCENario:VOLatile:VIEW:ZOOM:POINt 0.5 ms
SCENario:VOLatile:VIEW:ZOOM:RANGe 2 m
// displayed is the time span of 0 s to 2 ms
```

Usage: Setting only

Manual operation: See "X/Y zoom" on page 402

WAVeform:VIEW:ZOOM:RANGe <Range>
SCENario:VOLatile:VIEW:ZOOM:RANGe <Range>

Sets the displayed waveform part as a range around the selected center point, set with the command SCENario:VOLatile:VIEW:ZOOM:POINt.

Setting parameters:

<Range>	float
	Expressed as a time span (units can be omitted) or as number of samples
Example:	See `SCENario:VOLatile:VIEW:ZOOM:POINt`
Usage:	Setting only
Manual operation:	See "X/Y zoom" on page 402

25.29 List of remote commands

26 Troubleshooting

The R&S Pulse Sequencer monitors the functions performed and automatically detects errors and irregularities. The software logs status and error messages, warnings, SCPI commands send to the software, or additional information in a log file. Some of the messages are also entered in the error/event queue of the status reporting system.

26.1 Querying error messages

The following information sources help you determine the cause for an error or an unexpected program behavior:

- **Message log**
  The "Message Log" dialog lists information about all messages in a history list. If the software detects an error, the "Message Log" dialog pops up automatically and displays the error message.

- **Debug information**
  Additionally to the log information, the software provides a dialog with debug information. It lists information like the version of the used VISA library, the list of all loaded modules, or the report and data paths.

- The **psstartup.log** file
  If the software stops functioning properly, a log file is created automatically and stored in the %HOMEPATH% directory when the software is started again, see Table 2-2.

See:

- "To access the "Message Log" dialog" on page 831
- "To open the "Message Log" dialog in debug mode" on page 832
- "To access the "Debug Information" dialog" on page 833
- "To remove write lock that results from a previously crashed session" on page 834

To access the "Message Log" dialog

► Perform one of the following:

   a) In the toolbar, select the "Log" icon.
   b) In the menu bar, select "Window > Message Log".

1 = Info message
2 = Warning

The displayed information is read-only but you can mark and copy it.
Remote command:
MSGLog:POPup on page 798
MSGLog:ERRor? on page 798

To open the "Message Log" dialog in debug mode

1. In the toolbar, select the "Console Window" icon.

2. In the "Command Console" window, type *help*.

   The dialog lists all available commands.

3. Enter *set debug = 1* to enable the debug messages to be output in the "Message Log" view.

1 = Debug message
2 = Info message
3 = Error message

To access the "Debug Information" dialog

► In the menu bar, select "Help > Debug Information".

A dialog opens and displays information that helps the debugging.

To remove write lock that results from a previously crashed session

A red lock symbol 🔒 on the database icon in the "Repository Tree" indicates that a session was terminated improperly. A locked repository can be unlocked only from the same PC and by the same user with write permission that had opened the repository before.

1. In the project tree, double-click the repository name.

2. In the "Repository" dialog, select "Storage > Remove Write Lock".

   The repository is unlocked; you can change or store repository settings.
   See also Chapter 4.1, "Repository settings", on page 106.

26.2 Collecting information for technical support

If you encounter problems that you cannot solve yourself, contact your Rohde & Schwarz support center as listed at http://www.customersupport.rohde-schwarz.com. Our support center staff is optimally trained to assist you in solving problems.

The support center finds solutions more quickly and efficiently if you provide them with information on the software and an error description.

To collect error information in a debug file

1. Select "Help > Debug Information".

2. You can include or not the selected repository in the debug information file.

   a) Select "Include current repository" and in the "Export Debug Information" dialog confirm the export with "Yes" .

   b) If your repository contains sensitive data, deselect "Include current repository".

3. Select "Export Debug Information".

4. Navigate to the directory you want to save the file in.
   The filename is assigned automatically.

   The debug information and further required data is collected automatically.
   The debug file `DebugInfo.psdbarch` is created and saved in the selected directory.

The file contains:

- `*.ini` files, collected from the
  `%HOMEPATH%\AppData\Roaming\Rohde-Schwarz\Pulse Sequencer`
  directory.
- Text as it is displayed in the "Debug Information" dialog (`debug_info.txt`).
- If enabled, the repository (`*.psarch`).

Name	Size	Packed	Type	Modified
..			File folder	
colors.ini	1.319	1.319	Configuration setti…	17.09.2021 09:44
config.ini	2.745	2.745	Configuration setti…	17.09.2021 10:12
crash.ini	20	20	Configuration setti…	17.09.2021 09:54
debug_info.txt	7.571	7.571	Text Document	17.09.2021 10:34
help.psarch	107.520	107.520	PSARCH File	17.09.2021 10:34
mapping.ini	4.973	4.973	Configuration setti…	28.07.2021 08:54
settings_2.ini	44.204	44.204	Configuration setti…	17.09.2021 09:44

Attach the `DebugInfo.psdbarch` file to an email in which you describe the problem. Send the email to the customer support address for your region as listed on the Internet (http://www.customersupport.rohde-schwarz.com).

26.3 Contacting customer support

Technical support – where and when you need it

For quick, expert help with any Rohde & Schwarz product, contact our customer support center. A team of highly qualified engineers provides support and works with you to find a solution to your query on any aspect of the operation, programming or applications of Rohde & Schwarz products.

Contact information

Contact our customer support center at www.rohde-schwarz.com/support, or follow this QR code:

Figure 26-1: QR code to the Rohde & Schwarz support page

Annex

A Supported file types and file formats

This section summarizes the file types supported by the application and provides infor-
mation about the default file locations.

List of supported file types and file extensions

Table A-1: Supported file types

Extension	Description
`*.ps_hwsetup`	Hardware setup file Contains information on instruments in the "Signal Generators" dialog. See Chapter 3.6, "Signal Generators settings", on page 75.

`*.ant_pat`
`*.ffe`
`*.tsv`
`*.ffd`
`*.csv`
	Antenna pattern
	See Chapter A.2, "Antenna pattern file formats", on page 843.

`*.dll`	Plug-in, for example with user modulation, IPM profile
`*.ini`	Initialization file Contains information on default location for temporary files
`psstartup.log`	Log file created automatically if the software stops functioning properly
`*.wv`	Waveform file in a Rohde & Schwarz proprietary file format. Refer to the user manual of the signal generator for description of this file for-mat
`*.riq`	Rohde & Schwarz proprietary I/Q data file format used by the R&S®PR100 port-able receiver.
`*.wav`	Rohde & Schwarz proprietary audio-like file format that contains I/Q samples instead of the usual FM stereo signal. Although the file extension suggests that this file is a standard auto file, stereo players do not recognize it.
`*.iq.tar`	Rohde & Schwarz proprietary I/Q data file format used by some signal and spectrum analyzers, e.g. R&S®FSW. The `*.iq.tar` file contains I/Q data in binary format together with meta infor-mation that describes the nature and the source of data, e.g. the sample rate.
`*.mat`	MATLAB file that contains complex 1xN or Mx1 vectors.
`*.txt`	Standard ASCII files with comma-separated values. The `*.txt` files can be used to: • Describe settings that are based on lists, like for example the custom pulse envelope shapes, the list for IPM profiles, or data lists. • Describe custom I/Q data, with I and Q vectors in one file or in separated `*_i.txt` and `*_q.txt` files; If file pairs are stored in the same directory, they are recognized automati-cally by the "Import Wizard"

Extension	Description
`*.bin`	Files with custom I/Q data in binary format

`*.aif`
`*.dat`
`*.pdw` and `*.ppdw`
	AMMOS IF files
	AMREC recording files
	AMMOS PDW
	Rohde & Schwarz proprietary data formats used with Rohde & Schwarz monitoring equipment. These files typically contain multiple data streams.
	The R&S Pulse Sequencer extracts IF (I/Q) or PDW data from these files.

`*.ps_adr` `*.ps_def`	Automatically created Rohde & Schwarz proprietary files. File processing requires option Extended Sequencer (R&S SMW-K501).

A.1 File format of the reporting template

The template uses an ASCII text. Text that is not a token, a format (`.FMT`), or an option (`.OPT`) description is directly transferred to the final report.

Example: Example of a template that generates report in ASCII format

```
# R&S Pulse Sequencer, PDW Report              ← Text
#
Date:         <DATE>
Repository: <REPOSITORY>                        ← <Token>
Scenario:   <SCENARIO>


TOA      RF       PW       PA       MOP
         GHz      us       dBm
---------------------------------------
.HDR
<TOA> ; <RF> ; <PW> ; <PA> ; <MOP>              ← .HDR
# END OF REPORT                                   <Col#1> ; <Col#2> ; ...
.FMT TOA:12.0f:-9
.FMT RF:9.6f:9
.FMT PW:7.3f:-6                                 ← .FMT Token:Format:Exponent
.FMT PA:4.0f:0
```

Figure A-1: PDW template explanation

General template text and available tokens

The general template text is copied to the final report. All tokens are replaced with values or text provided by the software.

See Table A-2 for an overview of the available tokens.

Table A-2: List of available tokens

Token name	Description
`<USER>`	Current user name
`<TPATH>`	Target file path of the directory in that the generated report file is stored

Token name	Description
`<ISODATE>`	Date & time in ISO format
`<DATE>`	Date & time in local format
`<REPOSITORY>`	Repository name
`<RCOMM>`	Repository comment
`<VERSION>`	Repository version
`<AUTHOR>`	Repository author
`<RCDATE>`	Repository creation data
`<RPATH>`	Root path of repository
`<SCENARIO>`	Scenario name
`<SCOMM>`	Scenario comment

Adding and formatting data with the .HDR keyword

Each report contains one or more data lines. The position of these data lines in the report is indicated with the keyword `.HDR`. Usually, the reported data is formatted in table form, where the column names are defined with the information right after the `.HDR` line.

Each report template must have a `.HDR` line and subsequent second line defining the individual columns information and following the syntax:

`.HDR`

`<Col#1>;<Col#2>;...`

See Table A-3 for an overview of the available tokens.

Table A-3: List of available .HDR tokens

Token name	Description
Values related to the generated pulse data	
`<TOA>`	Time of arrival
`<PRI>`	PRI of current pulse (0% - 0%)
`<PRF>`	PRF of current pulse (1/PRI or 0)
`<PW>`	Pulse width (0% - 0%)
`<RT>`	Rise time
`<FT>`	Fall time
`<RF>`	Center frequency of pulse (absolute)
`<BW>`	RF bandwidth (FMOP) incl. BB filter
`<PA>`	Pulse amplitude (at pulse start)
`<MF>`	MOP flag, where 1 indicates that MOP is active
`<MOP>`	MOP type, AMOP, PMOD, FMOP

Token name	Description
`<DF>`	Frequency offset from carrier
`<RFC>`	Center frequency of emitter / sequence
`<DP>`	Phase offset (deg)
`<PHS>`	Absolute start phase at the beginning of pulse (deg)
`<RX_AZI>`	Azimuth angle of receive antenna at the beginning of pulse (deg)
`<RX_ELE>`	Elevation angle of receive antenna at the beginning of pulse (deg)
`<BEAR_AZI>`	Emitter bearing (e.g. simulate direction finder AoA output in PDW)
`<BEAR_ELE>`	Emitter elevation (e.g. simulate direction finder output in PDW)
`<TX_NORTH>`	Emitter distance in north direction from origin
`<TX_EAST>`	Emitter distance in east direction from origin
`<TX_UP>`	Emitter height above origin
`<TX_PITCH>`	Emitter attitude pitch angle
`<TX_YAW>`	Emitter attitude yaw angle
`<TX_ROLL>`	Emitter attitude roll angle
`<RX_NORTH>`	Receiver distance in north direction from origin
`<RX_EAST>`	Receiver in east direction from origin
`<RX_UP>`	Receiver height above origin
`<RX_PITCH>`	Receiver attitude pitch angle
`<RX_YAW>`	Receiver attitude yaw angle
`<RX_ROLL>`	Receiver attitude roll angle
Global variables from IPM profiles or loops, etc.	
`<VarName>`	Variable from the internal variable pool
Automatically generated count values related to the sequencing	
`<@ITEM>`	Sequence line item, 1 to N
`<@REP>`	Repetition, 1 to M
`<@COLL>`	Collection line item, 1 to L

Defining format with the .FMT keyword

Each format line must begin with `.FMT` and follow the syntax:

`.FMT Token:Format:Exponent`

The `.FMT` keyword formats the numeric data output, see Table A-4.

Table A-4: Format keyword syntax description

Parameter	Description
Token	Name of the token
Format	The format of the numeric data is compared to a regular expression: `[0-9]*.?[0-9]*[-+diufFeExX]` To prevent e.g. application crash, data that does not match this expression is ignored.
Exponent	Sets an exponent to convert the numeric value; e.g. -9 converts the reported data into nano.

See Figure A-1.

Defining options with the .OPT keyword

This keyword defines one or multiple options, that further control the output of the PDW report generator. Each option line must begin with `.OPT` and follow the syntax:

`.OPT TAG=<CURLY>,EXT=<html>`

Table A-5: Option keyword syntax description

Parameter	Description
TAG	`TAG=CURLY|ANGLE` Sets the brackets type
EXT	Sets the extension of the reporting file

Defining conditions with the .COND keyword

This keyword defines conditions, so that PDWs are generated only, if the condition is fulfilled. The condition line must begin with `.COND` keyword, can contain one or multiple comma-separated conditions and follow the syntax:

`.COND <token name> <condition> <value>,[<token name> <condition> <value>],`

Where the following <condition> are supported:

- Equal to: `==` or `=`
- Different that: `!=` or `<>`
- Greater/smaller than: `>` and `<`
- Greater/smaller or equal: `>=` and `<=`

Example:

The condition `.COND PA > -100.0` reports only the PDWs with pulse amplitude > -100 dBm.

Template examples

Template examples are provided in the repositories supplied with the software.

You can load the example repository or copy&paste the templates content in your own repository, try out them, and further develop them.

Access:

1. Select "File > Load and Manager Repositories".

2. In the "Repository Manager", double click the "Examples > K300 > Reports".

Created is a new repository with several scenarios using different kind of reporting templates.

3. Open a scenario.
   In the "Scenario" dialog, select "Waveform Generation > Config > Reporting > Template > Edit".

Example: Example of a template that generates report in HTML format

The following is the content of the template that creates a report in HTML format. The report is the same as in the ASCII file example shown on Figure 22-3, but the output is HTML formatted.

```
# Pulse Sequencer Report Template
# The following section defines the HTML report. All values
# in curly brackets are replaced by the actual values.
.OPT TAG=CURLY,EXT=html

<HTML> <HEAD> <TITLE>PDW Report</TITLE> <HEAD>
<BODY> <H2>PDW Report Generated with PS Example</H2>
  <P><TABLE style="width:100%;text-align:left;background-color:gold;">
    <colgroup> <col style="width:30%"> <col style="width:70%"> </colgroup>
    <TR style="background-color:blue;color:white;" >
    <TD>User</TD> <TD>{USER}</TD> </TR>
    <TR> <TD>Repository Path</TD> <TD>{TPATH}</TD> </TR>
    <TR> <TD>Date</TD> <TD>{DATE}</TD> </TR>
    <TR> <TD>ISO Date</TD> <TD>{ISODATE}</TD> </TR>
  </TABLE></P>
  <P><TABLE style="width:100%;text-align:left;background-color:gold;">
    <colgroup> <col style="width:30%"> <col style="width:70%"> </colgroup>
    <TR style="background-color:blue;color:white;" >
    <TD>Repository</TD> <TD>{REPOSITORY}</TD> </TR>
```

```
    <TR> <TD>Comment</TD> <TD>{RCOMM}</TD> </TR>
    <TR> <TD>Version</TD> <TD>{VERSION}</TD> </TR>
    <TR> <TD>Author</TD> <TD>{AUTHOR}</TD> </TR>
    <TR> <TD>Created</TD> <TD>{RCDATE}</TD> </TR>
    <TR> <TD>Path</TD> <TD>{RPATH}</TD> </TR>
  </TABLE></P>
<H3>{SCENARIO}</H3>
<P>{SCOMM}</P>


<P><TABLE style="width:100%;text-align:left;background-color:lightgrey;">
 <TR style="background-color:blue;color:white;" >
 <TD>TOA</TD> <TD>RF</TD> <TD>PW</TD> <TD>PA</TD> <TD>MF</TD> <TD>MOP</TD>
 <TD>BW</TD> <TD>Rep</TD> </TR>
 <TR style="background-color:blue;color:white;" >
 <TD>ns</TD> <TD>GHz</TD> <TD>us</TD> <TD>dBm</TD> <TD> </TD> <TD> </TD>
 <TD>kHz</TD> <TD> </TD> </TR>
.HDR
 <TR> <TD>{TOA}</TD> <TD>{RF}</TD> <TD>{PW}</TD> <TD>{PA}</TD> <TD>{MF}</TD>
 <TD>{MOP}</TD> <TD>{BW}</TD> <TD>{@REP}</TD> </TR>
 </TABLE>
</BODY>
# The following .FMT lines define the data format
#
.FMT TOA:12.0f:-9
.FMT RF:9.1f:9
.FMT PW:9.3f:-6
.FMT PA:6.1f:0
.FMT BW:6.0f:3
.FMT @REP:3i:0
```

A.2 Antenna pattern file formats

R&S Pulse Sequencer supports:
- "FEKO far field file format *.ffe" on page 844
- "Antenna Magus file format *.tsv" on page 844
- "ANSYS HFSS file format *.ffd" on page 844
- "Antenna patterns in *.csv file format" on page 845
- "Rohde&Schwarz proprietary antenna pattern *.ant_pat file format" on page 846

Examples of antenna pattern files

A subset of antenna pattern examples is included in the software.

Access:

► On your PC, open "Start > All Programs > R&S Pulse Sequencer > Example Files > Antenna Pattern".

You can load and import the patterns, try out them, and further develop them. See "To import a user-defined antenna pattern from file in one of the predefined file formats" on page 299.

FEKO far field file format *.ffe

FEKO is an electromagnetic simulation software tool distributed by the EM Software & Systems-S.A. (Pty) Ltd.

The R&S Pulse Sequencer supports the `*.ffe` files with version V1, V2, and V3.

For information on the file format, see the official product page http://www.feko.info/.

Antenna Magus file format *.tsv

Antenna Magus is an antenna design software tool.

The `*.tsv` files are text files that describe the antenna pattern in a tabular structure. These files contain a header section and data part with tab-separated values.

For information, see http://www.antennamagus.com/.

Example: Antenna Magus *.tsv file (extract)

```
# Far-field exported from Antenna Magus 5.3.0.1686
# On Friday, March 06, 2015 at 1:18:13 PM
# Frequency: 3.0E+9 Hz
# Power: 0.171146799151093 W
# Theta Samples: 181
# Phi Samples: 361
#Theta      Phi     Re(E_Theta)     Im(E_Theta)      Re(E_Phi)       Im(E_Phi)
0.000000000000E+000     0.000000000000E+000     4.303687774512E-016     0.000000000000E+000     -7.148424634567E+000     -1.436268526906E+000
1.000000000000E+000     0.000000000000E+000     4.425352450194E-016     0.000000000000E+000     -7.363209895869E+000     -1.542584165100E+000
...
```

ANSYS HFSS file format *.ffd

ANSYS HFSS software is the industry standard for simulating 3-D full-wave electro-magnetic fields.

The R&S Pulse Sequencer supports the `*.ffd` multi-frequency far field pattern files but imports only the first frequency.

For information, see http://www.ansys.com/.

Example: ANSYS HFSS *.ffd file (extract)

The values in the columns are the complex values of the electromagnetic field in theta and phi direction.

```
0 180 19
0 360 37
Frequencies 1
Frequency 8.680000000000000e+008
-3.029304169961667e+000 -3.592562203897908e+000 6.108290737961166e-001 1.376043194687679e+000
-2.877212877279787e+000 -3.299035718428788e+000 1.127582356352358e+000 1.978979886470077e+000
```

Antenna patterns in *.csv file format

Several of the popular antenna design software tools export antenna patterns also as `*.csv` files.

The `*.csv` files are text files that describe the antenna pattern in a tabular structure. These files contain a header section and data part with up to six columns. The values can be separated by commas, semicolons, or withe spaces.

When a file is selected, the "Import Wizard" tries to determine the used column separator, decimal delimiter and to data content of the columns. You can change the values afterwards.

Following parameters describe an antenna (see Figure A-2):

- Theta
  Elevation angle with value range 0° to 180°, where Theta = 0° means negative Z-axis direction
- Phi
  Azimuth angle with value range 0° to 360°, where Phi = 0° means positive Y-axis direction
- Use constant resolution step for both antenna parameters.

Figure A-2: Coordinate system

1 = Elevation angle with value range 0° to 180°
2 = Phi = 0°
3 = Theta = 0°
4 = Azimuth angle with value range 0° to 360°

Example: Antenna Magus file in *.csv format (extract)

```
FRQ=3.0E+9
PIN=0.171146799151093
THETA_SYM_0=0
THETA_SYM_90=0
PHI_SYM_0=0
PHI_SYM_90=0
INTERP_DBI=0
Theta(,deg) Phi(,deg) E_Theta(Re,) E_Theta(Im,) E_Phi(Re,) E_Phi(Im,)

# Far-field exported from Antenna Magus 5.3.0.1686
# On Friday, March 06, 2015 at 1:17:55 PM
# Theta Samples: 181
# Phi Samples: 361
0.000000000000E+000,0.000000000000E+000,4.303687774512E-016,0.000000000000E+000,-7.148424634567E+000,-1.436268526906E+000
1.000000000000E+000,0.000000000000E+000,4.425352450194E-016,0.000000000000E+000,-7.363209895869E+000,-1.542584165100E+000

...
```

Example: ANSYS HFSS files in *.csv format (extract)

The following are two examples of the ANSYS HFSS file in `*.csv` file format. The first file describes the electromagnetic fields, whereas the second one the gain in dBi.

```
Phi[deg],Theta[deg],re(rEPhi)[mV],im(rEPhi)[mV]
0,0,610.829,1376.04
10,0,1127.58,1978.98
...

Phi[deg],Theta[deg],dB(GainTotal)
0,0,-2.79687
10,0,-2.79687
...
```

Rohde&Schwarz proprietary antenna pattern *.ant_pat file format

The Table A-6 describes the used tags and parameters.

Table A-6: Format of *.ant_pat file

Element	Description
`<antenna_pattern>`	Root element of the antenna pattern file
`<az_res>`	Resolution of the columns in the `<data>` section
	Value in degrees integer divider of 360

Element	Description
<elev_res>	Resolution of the rows in the <data> section Value in degrees integer divider of 180
<data>	The file has to contain up to: • [1 + 360/<az_res>] columns • [1 + 180/<elev_res>] rows If the column resolution exceeds the resolution specified with the <az_res> element, an interpolation is applied. You can also define a subset of values, for example to define a beam antenna pattern. Missing values are internally set to zero, see Example"Beam antenna pattern" on page 847.

The following are two examples of the file format: the description of a beam-like antenna pattern and the description of an antenna pattern with back lobes and medium resolution.

Example: Beam antenna pattern

This example shows that partially defined antenna patterns are also allowed.

```
<?xml version="1.0" encoding="ISO-8859-1"?>
<antenna_pattern>
 <az_res> 5.00000000e+00 </az_res>
 <elev_res> 5.00000000e+00 </elev_res>
 <data>
        -30,-20,-10,0,  10, 20, 30
    -30,100,100,100,100,100,100,100,
    -20,100,100,100,100,100,100,100,
    -10,100,100,10, 100,10, 100,100,
      0,100,100,100,0,  100,100,100,
     10,100,100,10, 100,10,1 00,100,
     20,100,100,100,100,100,100,100,
     30,100,100,100,100,100,100,100,
 </data>
</antenna_pattern>
```

In this example, defined are only the subset of values that describe the beam pattern; all other values are assumed to be zero, see Figure A-3.

Figure A-3: 3D view of the imported partially defined antenna pattern

Example: Antenna pattern with back lobes

```
<?xml version="1.0" encoding="ISO-8859-1"?>
<antenna_pattern>
<az_res> 5.00000000e+00 </az_res>
<elev_res> 5.00000000e+00 </elev_res>
<data>
 -180,-160,-140,-120,-100,-80,-60,-40,-20,0,20, 40, 60, 80, 100,120,140,160,180
 -90,100,100,100,100,100,100,100,100,100,30,100,100,100,100,100,100,100,100,100
 -80,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 -70,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 -60,100,100,100,100,100,100,100,100,100,20, 100,100,100,100,100,100,100,100,100
 -50,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 -40,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 -30,100,100,100,100,100,100,100,100,100,10, 100,100,100,100,100,100,100,100,100
 -20,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 -10,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 0, 30, 100,100,20 ,100,100,10, 100,100,0,  100,100,10, 100,100,20, 100,100,30
 10,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 20,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 30,100,100,100,100,100,100,100,100,100,10, 100,100,100,100,100,100,100,100,100
 40,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 50,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 60,100,100,100,100,100,100,100,100,100,20, 100,100,100,100,100,100,100,100,100
 70,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 80,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100,100
 90,100,100,100,100,100,100,100,100,100,30, 100,100,100,100,100,100,100,100,100
</data>
</antenna_pattern>
```

Figure A-4: 3D view of the imported antenna pattern with back lobes

A.3   Movement files

A.3.1   Waypoint file format

A waypoint file is a simple text file (`*.txt`) that describes a trajectory as a sequence of positions called waypoints. The file contains a resolution [ms] that give the time between two consecutive waypoints and a list of waypoint coordinates, one row per waypoint.

Each waypoint is specified with its longitude [° (decimal format)], latitude [° (decimal format)], altitude [m].

Example:

The resolution command at the beginning of the file specifies the time (in ms) between two consecutive waypoints. In this example, it is 50 ms. The value is different than 10 ms; the waypoint file is resampled.

```
RESOLUTION: 50
144.966666334601,-37.8166633061788,100.000000009313
144.966662392613,-37.8166632247233,100.000000039116
144.966658453002,-37.8166630889914,100.00000008475
144.966654516955,-37.8166628990241,100.000000149943
144.966650585658,-37.8166626548785,100.000000235625
144.966646660296,-37.8166623566284,100.000000339001
...
etc.
```

A.3.2   Vector trajectory file format

In the vector trajectory file format, motion is defined in an East North plane which is tangent to earth at the specified reference point. The curvature of the Earth is not considered in the interpretation of the `Up (U)` coordinate elements when moving east or north.

The vector trajectory file format is suitable for description of movements where the curvature of the earth is negligible. This file format is not intended for long-distance simulations where the curvature of the earth is significant.

This file format uses a script containing the commands defined in the following table:

Command	Description
`REFERENCE: lon, Lat, alt`	Specifies the Cartesian Reference of the ENU coordinates system. Given as a geodetic WGS84 point (longitude, Latitude, Altitude).
`START: E ,N, U, Velocity`	Start location is the initial "current location" in the ENU Cartesian coordinate of center `REFERENCE`. East, North and UP (ENU) coordinates are then provided (m). The last argument is the start velocity in (m/s).
`ARC: E, N, Angle`	Specifies a 2-Dimensional ARC (East, North) with the first two arguments representing the center of the ARC (m) in the Cartesian basis. The last argument specifies the angle in degrees (°) of the Arc Starting the "current location". Angle sign is significant because it indicates positive direction (counterclockwise) or negative direction (clockwise). The end edge of the arc represents the new "current location". Velocity does not change when using an ARC command.
`LINE: ΔE, ΔN, Acceleration`	Specifies a 2-dimensional line form the current location or first edge (`Current loc E, Current loc N`) to the next location or second edge (`Current loc E + ΔE, Current loc N + ΔN`) (m). The argument `Acceleration` specifies a constant acceleration/deceleration, where `Acceleration = 0` indicates constant velocity. Unit for acceleration is (m/s2). The second edge is used as the current location for the next command. The speed at this second edge is also used as the start speed for the next command.

Command	Description
`LINE3D: ΔE, ΔN, ΔU, Acceleration`	Describes a 3-dimensional straight line in ENU coordinates, where the first edge is described as `(E, N, U)` and the second edge as `(E + ΔE, N + ΔN, U + ΔU)` (m).
	The argument `Acceleration` specifies a constant acceleration/deceleration, where `Acceleration = 0` indicates constant velocity. Unit for acceleration is (m/s2).
	`*********** MOVEMENT FILE *****************`
	`RESOLUTION: 10`
	`REFERENCE: 0,0,0`
	`START: 0, 0, 10000, 20`
	`LINE3D: 0, 0, -10000, 0`
`STAY: Time`	Stay at the current location for `Time` period (ms).

Example: Example of a waypoint file

This example explains a waypoint file, describing a moving emitter on a rectangular trajectory of 940m by 1440m with rounded corners defined in Table A-7 and Figure A-5.

The initial reference is first defined followed by acceleration to final speed of 100 km/h in 250 m. The emitter then maintains the speed for 400 m. The speed then decreases to 25 km/h in 250 m. The UE then turn 90 degrees with turning radius of 20 m at 25 km/h. The speed increases to 100 km/h in 250 m.

The sequence is repeated to complete the rectangle.

Table A-7: Trajectory parameters

Parameter	Distance (m)	Speed (km/h)
l_{11}, l_{15}, l_{21}, l_{25}	20	25
l_{12}, l_{14}, l_{22}, l_{24}	250	25 to 100 and 100 to 25
l_{13}	400	100
l_{23}	900	100

Figure A-5: Trajectory

The following is the content of the waypoint file.

> The start tag `MOVEMENT FILE` is a mandatory element for each movement description. If this tag is omitted, the software interprets the coordinates described in the waypoint file as static one.

```
****************************** MOVEMENT FILE ******************************
%%!! All empty lines or statements starting with %% will be ignored
%% This file format is used to load a continuous movement File into the signal generator
%% Enter the WGS-84 Reference Point's Coordinates S
REFERENCE: 144.966670277777777777777777777777777,-7.81666333333333333333333333333,100
%%From here on, we work in an ENU (EAST-NORTH-UPPER)basis centered around the ENU reference po
%%And more specifically the movement description will start from "START"
%%"START", "EAST COORDINATE E0 relative to S", "NORTH COORDINATE N0 relative to S",
%%"UPPER COORDINATE N0 relative to S", "SPEED in (m/s)"
START: 0, 0, 0, 6.94444444444444444
%%Each movement segment is described in one line command in the file
%%The segments are to be read sequentially.
%%It can be either a line description or an arc description
%%For a line, we start by the syntax "LINE"
%%For an arc, we start with the syntax "ARC"
%%In the current version, we provide only EAST-NORTH Movement segments
%%"ARC", "Center EAST Coordinate", "Center NORTH Coordinate",
%% "ARC in degrees (pos or neg is necessary for orientation)"
ARC: 0, 20, -90
%%"LINE", "Second Edge EAST Coordinate, first edge is the current point
%% or last point of previous segment","Second Edge NORTH Coordinate","Acceleration/Deceleratio
%%Acceleration is to be taken as the projection following the movement segment
LINE: 0, 250, 1.44675925925925925926
LINE: 0, 400, 0
LINE: 0, 250, -1.44675925925925925926
ARC: 0, 920, -90
```

```
LINE: 250, 0, 1.44675925925925925926
LINE: 900, 0, 0
LINE: 250, 0, -1.44675925925925925926
ARC: 1400, 920, -90
LINE: 0, -250, 1.44675925925925925926
LINE: 0, -400, 0
LINE: 0, -250, -1.44675925925925925926
ARC: 1400, 20, -90
LINE: -250, 0, 1.44675925925925925926
LINE: -900, 0, 0
LINE: -250, 0, -1.44675925925925925926
%%End of Trajectory description

****************************** MOVEMENT FILE ******************************
%%!! All empty lines or statements starting with %% will be ignored
%% Enter the WGS-84 Reference Point's Coordinates S
REFERENCE: 144.96667027777777777777777777777777,-7.81666333333333333333333333333,100
%%From here on, we work in an ENU (EAST-NORTH-UPPER)basis centered around the ENU reference po
%%And more specifically the movement description will start from "START"
%%"START", "EAST COORDINATE E0 relative to S", "NORTH COORDINATE N0 relative to S",
%%"UPPER COORDINATE N0 relative to S", "SPEED in (m/s)"
START: 0, 0, 0, 6.94444444444444444
%%Each movement segment is described in one line command in the file
%%The segments are to be read sequentially.
%%It can be either a line description or an arc description
%%For a line, we start by the syntax "LINE"
%%For an arc, we start with the syntax "ARC"
%%In the current version, we provide only EAST-NORTH Movement segments
%%"ARC", "Center EAST Coordinate", "Center NORTH Coordinate",
%% "ARC in degrees (pos or neg is necessary for orientation)"
ARC: 0, 20, -90
%%"LINE", "Second Edge EAST Coordinate, first edge is the current point
%% or last point of previous segment","Second Edge NORTH Coordinate","Acceleration/Deceleratio
%%Acceleration is to be taken as the projection following the movement segment
LINE: 0, 250, 1.44675925925925925926
LINE: 0, 400, 0
LINE: 0, 250, -1.44675925925925925926
ARC: 0, 920, -90
LINE: 250, 0, 1.44675925925925925926
LINE: 900, 0, 0
LINE: 250, 0, -1.44675925925925925926
ARC: 1400, 920, -90
LINE: 0, -250, 1.44675925925925925926
LINE: 0, -400, 0
LINE: 0, -250, -1.44675925925925925926
ARC: 1400, 20, -90
LINE: -250, 0, 1.44675925925925925926
LINE: -900, 0, 0
LINE: -250, 0, -1.44675925925925925926
%%End of Trajectory description
```

A.3.3 Trajectory description files

The trajectory description files use the file extension `*.xtd`. See Example"Trajectory description files in <positiononly> waypoint format" on page 856 for a simple example of the file format.

The Table A-8 describes the used tags and parameters.

*Table A-8: Format of *.xtd file*

Container	Tag name	Parameter	Description
<general>			

	<property>	<waypointfor-mat>	Defines the format of the waypoint data.
			The possible values are:
			• "positiononly"
			  Only waypoint data
			• "position_attitude"
			  Waypoint and attitude data
			• "position_velocitymagnitude"
			  Waypoints and velocity magnitude information per point
			• "position_velocitymagnitude_attitude"
			  Waypoints, velocity magnitude and attitude data
			• "position_velocityvector"
			  Waypoints and velocity vector per location
			• "position_velocityvector_attitude"
			  Waypoints, velocity vector and attitude per point
			
			Smoothening is only possible with `<property waypointformat="positiononly">` and waypoints and `*.kml`.
			
			The description of the waypoints has to follow the selected format.
			
			For example, `<waypointvector data="0.05,144.966662392613,-37.8166632247233, 100.000000039116"/>` with `<property waypointformat="positiononly"/>` and `<datavectorhastimestamp="yes">`.

		<datavectorhas-timestamp>	Determines the way the time stamp (elapsed time starting simulation time) is defined.
			• With `<datavectorhastimestamp="yes">`
			  The time stamp of a waypoint is retrieved from the `<waypointvector data>`, as the first data vector parameter.
			• With `<datavectorhastimestamp="no">`
			  the time stamp is internally calculated based on the following rule:
			  – The time stamp of first waypoint is $T_1 = 0$
			    The time stamp of the waypoint$_N$ is $T_N = (N-1)*$`<timeresolution>`

		<coordsystem>	Defines the coordinate system used by the definition of the `<waypointvector data>`. Possible values: "enu", "cart ecef", "geod wgs84" or "geod pz90"
		<timeresolution>	If `<datavectorhastimestamp="no">`, applies implicitly time stamps to waypoints. Otherwise, the time stamp is retrieved automatically, see `<datavectorhastimestamp>`.
		<enurefpoint>	Coordinates of the reference waypoint in ENU format
		<endbehaviour>	Determines the behavior of the moving object at the end of the trajectory. Possible values: "jump", "return", "stop"
		<attitudeunit>	Defines the unit ("rad" or "deg") the attitude is expressed in

Container	Tag name	Parameter	Description
		`<noofway-points>`	Number of the used waypoints vectors. Use the parameter to minimize the size of the waypoint without actually deleting the waypoints
		`<attitudecontinu-ous>`	If `<attitudecontinuous="no">`, the attitude angles to be read are bounded by 2*PI. The `<attitudecontinuous="yes">` means that they are not bounded.
`<waypoints>`			Description of the trajectory as a list of waypoint vectors.
	`<waypointvec-tor>`	`<data>`	Coordinates of the waypoint in the format selected by `<coordsystem>` • [time,]longitude,latitude,altitude (geod wgs84) • [time,]X,Y,Z (cart ecef) • [time,]EAST,NORTH,UPPER (enu) Depending on the `<datavectorhastimestamp>`, 3 or 4 values are evaluated with `<property waypointformat="positiononly"/>` Depending on the `<property waypointformat>`, more information can be provided. The following is a list of the information appended to the coordinates of the waypoint mentioned above. • `<property waypointformat="positiononly">` No additional information • `<property waypointformat="position_attitude">` **yaw/heading,pitch/elevation,roll/bank**, e.g. `[time,]longitude,latitude,altitude,yaw,pitch,roll` The vehicle attitude (yaw, pitch and roll) parameters are configured relative to the local horizon • `<property waypointformat="position_velocitymagnitude">` **vel**, e.g. `[time,]X,Y,Z,vel` • `<property waypointformat="position_velocitymagnitude_attitude">` **vel,yaw/heading,pitch/elevation,roll/bank**, e.g. `[time,]EAST,NORTH,UPPER,vel,yaw,pitch,roll` • `<property waypointformat="position_velocityvector">` Velocity vector **Vx,Vy,Vz**, e.g. `[time,]X,Y,Z,Vx,Vy,Vz` • `<property waypointformat="position_velocityvector_attitude">` **yaw/heading,pitch/elevation,roll/bank,Vx,Vy,Vz**, e.g. `[time,]longitude,latitude,altitude,yaw,pitch,roll,Vx,Vy,Vz` **Note:** If the waypoints are in geod wgs84, velocity vector is assumed in Cartesian ECEF coordinate system.

Example: Trajectory description files in <positiononly> waypoint format

The following are two examples in `<property waypointformat="positiononly">` format. The examples emphasize on the different formats and explain the used time stamp principle.

- The `<waypointvector data>` parameters in the following example are vectors with size of 3, because of the tag `<property datavectorhastimestamp="no">`. Even if a fourth value has been defined, it would have been ignored.
  The time stamp is defined with the tag `<property timeresolution="0.05">`: $T_1 = 0$, $T_2 = (2-1)*0.05 = 0.05$, and so on, $T_{12} = (12-1)*0.05 = 0.55$

```
<trajectory>
   <general>
      <property waypointformat="positiononly"/>
      <property datavectorhastimestamp="no"/>
      <property coordsystem="enu"/>
      <property timeresolution="0.05"/>
      <property enurefpoint="54.0,10.0,12"/>
      <property endbehaviour="return"/>
      <property noofwaypoints="12"/>
   </general>
   <waypoints>
      <waypointvector data="0,0,0"/>
      <waypointvector data="0,1,0"/>
      <waypointvector data="0,2,0"/>
      <waypointvector data="0,3,0"/>
      <waypointvector data="0,4,0"/>
      <waypointvector data="0,5,0"/>
      <waypointvector data="0,6,0"/>
      <waypointvector data="0,7,0"/>
      <waypointvector data="0,8,0"/>
      <waypointvector data="0,9,0"/>
      <waypointvector data="0,10,0"/>
      <waypointvector data="0,11,0"/>
   </waypoints>
</trajectory>
```

- In the following example, the tag `<property datavectorhastimestamp="yes">` and all four values in the `<waypointvector data>` parameters are evaluated. The time stamp is retrieved automatically; the time stamp of a waypoint is the first value in the corresponding `<waypointvector data>` parameter:
  $T_1 = 0$, $T_2 = 0.05$, and so on, $T_{12} = 0.55$.

```
<trajectory>
   <general>
      <property waypointformat="positiononly"/>
      <property datavectorhastimestamp="yes"/>
      <property coordsystem="geod wgs84" />
      <property endbehaviour="return"/>
      <property duration="2.0"/>
   </general>
   <waypoints>
      <waypointvector data="0,144.966666334601,-37.8166633061788,100.000000009313"/>
      <waypointvector data="0.05,144.966662392613,-37.8166632247233,100.000000039116"/>
      <waypointvector data="0.10,144.966658453002,-37.8166630889914,100.00000008475"/>
      <waypointvector data="0.15,144.966654516955,-37.8166628990241,100.000000149943"/>
      <waypointvector data="0.20,144.966650585658,-37.8166626548785,100.000000235625"/>
      <waypointvector data="0.25,144.966646660296,-37.8166623566284,100.000000339001"/>
      <waypointvector data="0.30,144.966642742053,-37.8166620043635,100.000000461936"/>
      <waypointvector data="0.35,144.966638832109,-37.81666159819,100.000000603497"/>
```

```
    <waypointvector data="0.40,144.966634931642,-37.8166611382304,100.000000762753"/>
    <waypointvector data="0.45,144.96663104183,-37.8166606246233,100.000000941567"/>
    <waypointvector data="0.50,144.966627163843,-37.8166600575235,100.000001138076"/>
    <waypointvector data="0.55,144.96662329885,-37.8166594371019,100.000001354143"/>
</waypoints>
</trajectory>
```

A.4 Vehicle description files (Used for smoothening)

The vehicle description files use the file extension `*.xvd`. The following is a simple example of the file format.

```
<vehicle>
    <info name="Car"/>
    <limits>
        <property maxspeed="100.0"/>
        <property maxg="0.6"/>
        <property maxg_lateral="0.6"/>
        <property maxjerk="15"/>
        <property maxjerk_lateral="15"/>
        <property maxyawrate="0"/>
        <property maxyawacceleration="0"/>
        <property maxyawjerk="0"/>
        <property maxpitchangle="0"/>
        <property maxpitchrate="0"/>
        <property maxpitchacceleration="0"/>
        <property maxpitchjerk="0"/>
        <property maxrollangle="0"/>
        <property maxrollrate="0"/>
        <property maxrollacceleration="0"/>
        <property maxrolljerk="0"/>
        <property proximity="0"/>
    </limits>
</vehicle>
```

The Table A-9 describes the used tags and parameters.

*Table A-9: Format of *.xvd file*

Container	Tag name	Parameter	Description
<info>			
	<name>		Vehicle name
<limits>			
	<property>	<maxspeed>	Maximal Speed (m/s)
		<maxg>	Maximum tangential acceleration in g unit
		<maxg_lateral>	Maximum radial acceleration (due to centrifugal force) in g unit

Container	Tag name	Parameter	Description
		<maxjerk>	Maximum tangential Jerk in m/s2
		<maxjerk_lateral>	Maximum radial Jerk in m/s2
		<maxyawrate>	Maximum rate of change of the yaw/heading (rad/s)
		<maxyawacceleration>	Maximum second rate of change (acceleration) of the yaw/heading (rad/s2)
		<maxyawjerk>	Maximum third rate of change (jerk) of the yaw/heading (rad/s3)
		<maxpitchangle>	Maximum pitch/elevation angle (rad)
		<maxpitchrate>	Maximum rate of change of the pitch/elevation (rad/s)
		<maxpitchacceleration>	Maximum second rate of change (acceleration) of the pitch/elevation (rad/s2)
		<maxpitchjerk>	Maximum third rate of change (jerk) of the pitch/elevation (rad/s3)
		<maxrollangle>	Maximum roll/bank angle (rad)
		<maxrollrate>	Maximum rate of change of the roll/bank (rad/s)
		<maxrollacceleration>	Maximum second rate of change (acceleration) of the roll/bank (rad/s2)
		<maxrolljerk>	Maximum third rate of change (jerk) of the roll/bank (rad/s3)
		<proximity>	Specifies the maximum allowed deviation from the original waypoints (m)

B Formula syntax

You can use mathematical expression to define custom modulation types or envelope shapes.

The R&S Pulse Sequencer uses the fast math parser library muParser, that is an extensible high performance math expression parser library written in C++.

This section list some of the default features supported by the parser. For detailed information, see the muParser product home page http://beltoforion.de/article.php?a=muparser&p=features.

Basic syntax elements

Table B-1: Mathematical operations

Element	Description
+	Addition
-	Subtraction
*	Multiplication
/	Division
^	By the power of

Table B-2: Built-in functions (Extract)

Element	Description
sin	sine function
cos	cosine function
exp	raised to the power of x
etc.	

Table B-3: Other operators

Element	Description	Syntax
?:	if then else operator	`<if_condition>?<then_expression>:<else_expression>`

Table B-4: Constant and variables

Element		Description
Constants:	PI	π
	T_r	Pulse rise time
	T_w	Pulse width time
	T_f	Pulse fall time

Element		Description
	T $T = T_r + T_w + T_f$	Total pulse time
Variables:	ip\<n>	Inter-pulse modulation value \<n>
	t $t = 0 .. T$	Time

Equations and parameters used to define custom pulse envelopes

Envelope name	Equation	Envelope shape
"Rectangular Pulse"	1	
"Triangular Pulse"	(t<T/2) ? t/(T/2) :1-(t-T/2)/(T/2)	
"Raised Cosine Pulse"	0.5 * (1 - cos(2 * PI * t/T))	
"Gaussian Pulse"	exp(-4 * ln(2) * ((t-Tr-Tw/2) / Tw)^2)	

Envelope name	Equation	Envelope shape
"Lorentzian Pulse"	$1 / ((1 + 4 * (sqrt(2)-1) * ((t-Tr-Tw/2) / Tw)^2))^2$	
"Sech Pulse"	$1 / (cosh((2 * (t-Tr-Tw/2) * ln(1+sqrt(2))) / Tw))^2$	

Where Amplitude value range is 0 V to 1 V.

C Plug-in programming API

The functionality of the R&S Pulse Sequencer can be extended by plug-ins. A plug-in is a Microsoft Windows DLL. This DLL can be loaded into a repository and then remains there.

You can load plug-ins to:

- Specify your custom modulation scheme (MOP)
- Define a custom inter-pulse modulation profile (IPM)
- Create a report file
- Define the file format of the custom antenna pattern files.

Access:

A subset of example plug-ins is included in the software.

► On your PC, open "Start > All Programs > R&S Pulse Sequencer > SDK".

You can load the plug-ins in your own repository, try out them, and further develop them.
See "To import a plugin" on page 530.

The following is a list of all functions that the plug-in has to provide:

C.1 Common functions

getType (char szModType[1024])

(mandatory)

Queries the plug-in type.

The type defines the purpose of the plug-in. It is the most important parameter returned to the main application.

Do not change the type of the plug-in after it is used within a repository. The type also defines which functions have to be provided by the plug-in (API). It also affects in which R&S Pulse Sequencer dialogs the plug-in becomes visible.

Parameters:

szModType   string

(return parameter)
A keyword that denominates the plug-in type.

"modulation" | "mop"
MOP plug-in

"ipm"
IPM plug-in

"report"
Reports plug-in for reports created during the waveform generation

getVersion (char szModVer[1024])

(mandatory)

Queries the version string of the plug-in.

The version string is not further evaluated in the software. It only serves as an additional piece of information. The returned version string is visible on the plug-in dialog panel.

Parameters:

szModVer   "<Major>.<Minor>.<Revision>"

(return parameter)
`<Major>`, `<Minor>`, and `<Revision>` are numerical values.

getComment (char szModComment[4096])

(mandatory)

Queries the comment string of the plug-in.

The comment string is not further evaluated in the software. It only serves as an additional piece of information. The returned comment string is visible on the plug-in dialog panel.

Parameters:

szModComment   string

(return parameter)
Line breaks are permissible. To generate new line, insert `'\n'` into the string.

---

getAuthor (char szModAuthor[1024])

(mandatory)

Queries the author string of the plug-in.

The author string is not further evaluated in the software. It only serves as an additional piece of information. The returned author string is visible on the plug-in dialog panel.

Parameters:

szModAuthor   string

(return parameter)
Empty string is allowed

---

getName (char szModName[1024])

(mandatory)

Queries the plug-in name.

The name string is not further evaluated in the software. It only serves as an additional piece of information. However, be careful changing names once the plug-in is in use. The returned name string is visible on the plug-in dialog panel.

Return values:

szModName   string

(return parameter)

---

getError (char szModError[1024])

(mandatory)

Queries the plug-ins for errors.

This function is called if a plug-in function returns an error. In addition, the function is called repeatedly to poll the plug-in for errors that are not related to a specific function call. This function must return an empty string if there are no pending error messages. Returning an error stops the R&S Pulse Sequencer calculations.

Error messages should only be returned once.

Parameters:

szModError   string

(return parameter)
Error message as zero-terminated ASCII string.

int getNextMsg (char szInfoMsg[4096])

(optional)

Sends messages to the application. The messages are passed to the "Message Log" of the application, merely for user information.

Parameters:

szInfoMsg

"<Prefix>:<Text>"

(return parameter)
The message can contain a prefix that denominates the type of information sent.
`D` = Debug
`I` = Information
`W` = Warning
`E` = Error
If no prefix is present, the information type (`I`) is assumed.

Return values:

boolean

The function is called repeatedly until it returns false (0).

int initPlugin (void)

(mandatory)

This function initializes the plug-in. It is called once after the plug-in is loaded into memory and can set internal variables.

Return values:

boolean

true (1)
The initialization completed successfully

false (0)
Error during the initialization. The plug-in is unloaded from memory

shutdownPlugin (void)

(optional)

This function is called before the plug-in gets unloaded from memory. This function can be used to release allocated resources. A plug-in is unloaded from memory if it gets removed from the repository or the main application closes.

int getPrivateMemSize (void)

(optional)

This function sets the required amount of user data.

Avoid storing parameter settings as global variables because global variables are shared among multiple instances of the plugin. Register a user data block in the application and pass the pointer `pcPrivateMem` as argument instead.

Return values:

PrivateMemSize   Required size of user memory in bytes

---

int getVariable (int iIndex, char szConfig[4096])

(mandatory)

Plug-ins can register variables in the R&S Pulse Sequencer user interface. These variables are initialized with a default value but you can change the value later. The modified variable value is stored as part of the repository. This mechanism allows the reuse of a plug-in in different configurations.

This function is called repeatedly by the main application and reads information about the variables to be registered. On each function call, the zero-based variable index value `iIndex` increases by one. The index uniquely references a variable.

Parameters:

iIndex
> **(input parameter)**
> Zero-based variable index

szConfig
> **(return parameter)**
> The string of variables to be registered is composed as follows:
>
> **DBL,<name>,<unit>,<default>,<min>,<max>,<dec. digits>**
> Real Number
>
> **INT,<name>,<unit>,<default>,<min>,<max>**
> Integer
>
> **BOOL,<name>,<default>**
> Boolean
>
> **STR,<name>,<default>[,<validator-regexp>]**
> String
>
> **SEL,<name>,<opt>|<opt>|...,<default index>**
> Selection
> Value range of `<default index>`: 0 to (number of <opt>-1)

Return values:

boolean
> **true**
> A variable is registered under this index number
>
> **false**
> No variable was registered; the function is not called anymore

Example:
> `SEL,Type,16-QAM|32-QAM|64-QAM|256-QAM,2`
> This field is named "Type" and lists the options:
> 16-QAM, 32-QAM, 64-QAM, and 128-QAM
> The default value is 64-QAM

---

int setVariable (int iIndex, const char szValue[4096])
int setVariableEx (void *pcPrivateMem, int iIndex, const char szValue[4096])

(optional)

Sets the value of a variable. The variable is referenced by the same zero-based index `iIndex` used in `getVariable()`. The variable value is transferred as string. This string must be interpreted according to the variable data type.

Parameters:

pcPrivateMem
: **(input parameter)**
User data memory pointer, see `int getPrivateMemSize`.

iIndex
: **(input parameter)**
The zero-based index value references the variable

szValue
: Zero-terminated ASCII string

: **(input parameter)**
Variable value as formatted ASCII string. Use functions such as `atof()` or `atoi()` for conversion.

C.2 MOP functions

int isDataNeeded (void)

(optional)

Queries whether this plug-in requires data from a data source or not.

Return values:

boolean
: **true (1)**
The plug-in requires a data source

: **false (0)**
The plug-in does not need data from a data source

double getModulationClockHint (void)
double getModulationClockHintEx (void *pcPrivateMem)

(optional)

Queries the minimum clock rate required by the plug-in to calculate the I and Q data.

The function is automatically called after the parameterization. This function evaluates all plugin-internal settings and return the minimum required clock rate to fulfil the Nyquist criteria.

Parameters:
pcPrivateMem   **(input parameter)**
                    User data memory pointer, see `int getPrivateMemSize`.

Return values:
ModClockHint   If this function is not implemented or the supplied value is outside the limits, the default value is used automatically.

                    *RST: 100 MHz

double getModulationSymbolRate (void)
double getModulationSymbolRateEx (void *pcPrivateMem)

(optional)

Queries the symbol rate.

This function informs the main application about the symbol rate. This figure is only required if a baseband filter is applied to the plug-in I/Q output. If the plug-in does not generate signals that require baseband filtering the function can be omitted.

Parameters:
pcPrivateMem   **(input parameter)**
                    User data memory pointer, see `int getPrivateMemSize`.

Return values:
ModSymbolRate   The symbol rate in symbols per second.

                    Range: 0 GHz to 10 GHz

setModulationParameter (const char *szType, const char szValue[1024])
setModulationParameterEx (void *pcPrivateMem, const char *szType, const char szValue[1024])

(mandatory)

Sets various parameters before a MOP calculation starts.

This function is repeatedly called from the main application. On each call, the parameter type is defined by a string ID. The value is transferred as string value and has to be interpreted accordingly.

Parameters:

pcPrivateMem   **(input parameter)**
                          User data memory pointer, see `int getPrivateMemSize`.

szType   Zero-terminated ASCII string

                          (input parameter)
                          The string ID identifies the MOP parameter

                          "mtime"
                          MOP duration

                          "srat"
                          Sample rate, used to calculate the MOP

                          "done"
                          This string ID is sent when the function is called for the last time.
                          Evaluating this ID allows you to check parameters or calcula-
                          tions.

szValue   Zero-terminated ASCII string

                          (input parameter)
                          The numeric value formatted as string. Use functions such as
                          `atoi()` or `atof()` to interpret this data.

---

int getBitsNeeded (double dTime, long *pcStart, int *pcCount)
int getBitsNeededEx (void *pcPrivateMem, double dTime, long *pcStart, int
    *pcCount)

(optional)

Requests a defined number of bits from the data source.

The function is called by the main application each time before a sample is calculated.
For performance reasons, new bits should be requested in blocks and only if they are
required for the calculation of the next symbol.

Parameters:

pcPrivateMem   **(input parameter)**
                          User data memory pointer, see `int getPrivateMemSize`.

dTime   **(input parameter)**
                          Timestamp of the following sample

pcStart   **(return parameter)**
                          Index of the first bit from the data source

pcCount   **(return parameter)**
                          Number of required bits. Return zero if no bits are required.

Return values:

boolean   **true (1)**
                          Successful

setBits (long lStart, int iCount, const char *cBuffer)
setBitsEx (void *pcPrivateMem, long lStart, int iCount, const char *cBuffer)

(optional)

This function is the response to the function `getBitsNeeded()`. It transmits the requested number bits to the plug-in.

Parameters:

pcPrivateMem   **(input parameter)**
     User data memory pointer, see `int getPrivateMemSize`.

lStart   **(input parameter)**
     Start index of the first bit; This value matches the `pcStart` value from `int getBitsNeededExgetBitsNeeded()`

iCount   **(input parameter)**
     Number of transmitted bits

cBuffer   **(input parameter)**
     The buffer holds the requested number of bits. The bits are transferred in binary format. Each character represents one single bit and has either the value zero or one.

int modulationCore (double dTime, double *pdAmplitude, double *pdPhase, short *puiMkr)
int modulationCoreEx (void *pcPrivateMem, double dTime, double *pdAmplitude, double *pdPhase, short *puiMkr)

(mandatory)

Transforms samples into I/Q data.

Parameters:

pcPrivateMem   **(input parameter)**
     User data memory pointer, see `int getPrivateMemSize`.

dTime   **(input parameter)**
     Timestamp of the sample to be calculated

pdPhase   **(input parameter)**
     The phase of the last sample in radian. This parameter is required for modulation types that rotate the phase from sample to sample (e.g. chirps).

pdAmplitude   **(return parameter)**
     The voltage-based sample amplitude. This value ranges from zero to one.

pdPhase   **(return parameter)**
     The sample phase in radian.

puiMkr   **(return parameter)**
     32-bit unsigned integer bit field defines the marker flags:
     Bit 0= Marker 1
     Bit 1= Marker 2

Bit 3= Marker 3

Bit 4= Marker 4

Return values:

boolean **true (1)**

Calculation was successful

---

generateDirac (int *pcGenI, int *pcGenQ)

generateDiracEx (void *pcPrivateMem, int *pcGenI, int *pcGenQ)

(optional)

Generates a Dirac impulse in the main application for the last sample.

If the function is implemented, it gets called after every call to `modulationCore ()`.

The functionality provided by this function is used if the sample returned by `modulationCore ()` is to be folded with a baseband filter. This, for example, is required if the `modulationCore ()` provides samples used with a digital I/Q modulation such as QPSK or QAM. Typically the output signal is created with oversampling factors of two or larger, thus multiple I/Q samples are generated for each symbol. The folding operation, however, only requires one single sample per symbol as a Dirac impulse. This function indicates which sample is used as the Dirac impulse.

The I and Q channel can be controlled separately. This is required if the constellation points rotate around the origin. In this case, the I and Q waveform are time delayed by a fraction of the symbol duration.

If the function is not implemented, all generated samples are assumed to be Dirac pulses. This is equal to a sample and hold operation.

Parameters:

pcPrivateMem **(input parameter)**

User data memory pointer, see `int getPrivateMemSize`.

*pcGenI, *pcGenQ **(return parameter)**

The return value for `*pcGenI` and `*pcGenQ` is true (1) if the last sample shall be used as the Dirac pulse.

In all other cases, the main application replaces the sample value by zero.

---

setGlobalVariable (const char szVarName[256], const char szValue[256])

(optional)

Sends a global variable value to the plug-in.

User defined global variables can be set in the IPM profiles or in the sequences (e.g. loops). In addition, the software provides predefined variables.

Parameters:

szVarName Zero-terminated ASCII string

(input parameter)

The following variables are predefined in R&S Pulse Sequencer:

RFC
RF center frequency, in Hz

LEV
RF level, in dBm

REP
Repetition count of the current sequence line item

ITEM
Item number in a sequence

szValue   Zero-terminated ASCII string

(input parameter)
Value of the variable formatted as string. Use `atoi()` or
`atof()` to interpret this parameter.

---

int getNextReportVariable (int iIndex, const char szName[256], const char
        szValue[256])

(optional)

MOP plug-ins may provide variables to the reporting core. This functionality is useful if
values calculated by the MOP plug-in shall be reported.

The function queries the report variables after a successful calculation and sends them
to the report generator. The function is called repeatedly until false (0) is returned.

Parameters:
iIndex   Zero-based variable index

(input parameter)

szName   Zero-terminated ASCII string

(return parameter)
Variable names

szValue   Zero-terminated ASCII string

(return parameter)
Variable value as formatted string.

Return values:
boolean   **true (1)**
Variable data returned for current index

false (0)
No variable data available. The function is not called anymore.

C.3 IPM functions

setRndFnPtr (void *pRndFn, void *pThis)

(optional)

Sets a pointer to a random function `pRndFn` and the associated `pThis` pointer. Both pointers are needed for calling the built-in random number function.

Plug-ins must use the random generator provided by the application. This generator produces a random sequence with a defined start seed. If necessary, the application can reproduce a particular calculation with the same values. This mechanism is required because the main application performs an initial dry run to determine parameters such as sequence duration and clock rate. This random generator ensures that the same values are used during dry run and the final calculation.

The prototype of the callback function is defined as follows:

```
typedef int (*RndFnCallback)( void*, int );
```

The random generator is called using the provided pointers:

```
int iRnd = pRndFn( pThis, iMaxVal );
```

The random generator returns a 32-bit random unsigned integer value in the range 0 to 65535.

---

restart ( )
restartEx (void *pcPrivateMem)

(optional)

Restarts the IPM pattern generation.

This function is called if the IPM pattern generation is requested to restart. The sequence "IPM Configuration" dialog contains the option "Restart IPM for this line item". Setting this option calls this function at the beginning of the line item calculation.

This option is useful if IPM patterns follow a certain rule or have memory, e.g. lists where items shall not be reused.

Parameters:

pcPrivateMem   **(input parameter)**
                    User data memory pointer, see `int getPrivateMemSize`

---

int calculateNextIpmValue (double dTime, uint64 iCount, double *pdValue)
int calculateNextIpmValueEx (void *pcPrivateMem, double dTime, uint64 iCount, double *pdValue)

(mandatory)

Requests the next IPM value from the IPM plug-in.

IPM values can either be calculated based on a timestamp or an index. The time is the start time of the current sequence line item, e.g. the pulse start time. The index is an integer number that increases by one each time an IPM value is requested from the plug-in.

Parameters:

pcPrivateMem   **(input parameter)**
                     User data memory pointer, see `int getPrivateMemSize`

dTime   **(input parameter)**
                     Timestamp of the next IPM value

iCount   **(input parameter)**
                     64-bit unsigned integer value
                     Zero-based index of the next IPM value

pdValue   **(return parameter)**
                     Calculated IPM value.

Return values:

boolean   **true (1)**
                     Calculation successful

C.4 Reporting functions

For information on the report generation function, see:

- Chapter A.1, "File format of the reporting template", on page 838
- Chapter 22, "Creating reports and documenting measurement results", on page 542

The following is a list of plug-in functions for creating reports:

setParam (const char *szType, const char szValue[1024])

(mandatory)

Sets various report parameters.

This function is repeatedly called. It sets different parameters on each call.

Because pulses can have different parameters, some pulse parameters have to be set before the calculation of the particular pulse starts. User variables that are defined and used within a sequence are set with their names (`<user_var>`) and values.

Parameters:

szType   Zero-terminated ASCII string

(input parameter)

Sets the type of data that is transmitted.

See also Chapter A.1, "File format of the reporting template", on page 838.

"REPOSITORY"

String

Repository name

„SCENARIO"

String

Scenario name

„ISODATE"

String

Date & time in ISO format

„VERSION"

String

Repository version

„AUTHOR"

String

Repository author

„TPATH"

String

Target file path of the directory in that the generated report file is stored

„TOA"

Float

Time of arrival (timestamp of pulse)

„PW"

Float

Pulse width (0% - 0%)

„PRI"

Float

PRI of current pulse (0% - 0%)

„BW"

Float

RF bandwidth (FMOP) incl. BB filter

„RFC"

Float

Center frequency of emitter / sequence

„@ITEM"

Integer

Sequence line item, 1 to N

„@COLL"

Integer

Collection line item, 1 to L

„@CNT"
Integer
Repetition count

„@REP"
Integer
Current repetition, 1...M

„<user_var>"
Variable type, depending on the variable
User variable name

szValue Zero-terminated ASCII string

(input parameter)

---

reportBegin ( )

(mandatory)

A new report begins.

This function is called once at the beginning of a calculation run. In case of collection scenarios, this function is called once before the first collection item gets processed.

The function can be used to open the report file and to set the header information.

See Chapter A.1, "File format of the reporting template", on page 838.

---

reportEnd ( )

(mandatory)

A report ends.

This function is called once when all calculations are completed. In case of collection scenarios, this function is called once when the last collection item is calculated.

---

reportEntryComplete ( )

(mandatory)

A pulse entry is generated.

This function is called after each pulse. It creates the report entry for each pulse.

C.5 Custom antenna pattern import functions

The following is a list of functions for importing antenna patterns in custom file format.

These functions are not necessary when importing antenna patterns in one of the files formats listed in Chapter A.2, "Antenna pattern file formats", on page 843.

bool importPattern (const char *szFile)

(mandatory)

Run the pattern import procedure.

This function imports the antenna pattern into the internal pattern memory. Since this import is done in the foreground thread, it should not take too much time.

Return values:

boolean	**true**
	Success

getConversionInfo (char szInfo[4096])

(optional)

Get the result from the antenna pattern import.

This function is called after the `importPattern()` call. It returns general information and error messages from the import procedure.

Parameters:

szInfo	"<Prefix>:<Text>"
	(return parameter)
	The text can contain multiple lines. Each line can start with a prefix that denominates the information type.
	`D` = Debug
	`I` = Information
	`W` = Warning
	`E` = Error

int getPatternCount ( )

(mandatory)

Read the number of imported patterns.

This function is the first one that gets called by the main application after a successful import.

bool getFrequency (int iPatIndex, double *pdFMin, double *pdFMax)

(mandatory)

Get the frequency for a specific pattern.

This function is called once for each imported pattern. It returns the *frequency range* assigned to an antenna pattern.

Parameters:
iPatIndex   Zero-based antenna index

Return values:
boolean   **true**
                             Success

bool getThetaRange (int iPatIndex, double *pdTMin, double *pdTMax, int *piValueCnt)

(mandatory)

Get the theta range of a specific pattern.

This function is called once for each imported pattern. It returns the *theta range* assigned to an antenna pattern.

Parameters:
iPatIndex   Zero-based antenna index

pdTMin   Minimum theta angle

pdTMax   Maximum theta angle

piValueCnt   Number of theta values

Return values:
boolean   **true**
                             Success

bool getPhiRange (int iPatIndex, double *pdPMin, double *pdPMax, int *piValueCnt)

(mandatory)

Get the phi range of a specific pattern.

This function is called once for each imported pattern. It returns the *phi range* assigned to an antenna pattern.

Parameters:
iPatIndex   Zero-based antenna index

pdPMin   Minimum phi angle

pdPMax   Maximum phi angle

piValueCnt   Number of phi values

Return values:

boolean **true**

Success

---

bool getPatValue (int iPatIndex, int iPhiIndex, int iThetaIndex, float *pfEHorizontal,
float *pfEVertical, float *pfPhaseVertical)

(mandatory)

Get one data point from a pattern.

This function is called once for each imported pattern and once for each data point. It returns the *pattern data* to the main application.

Parameters:

iPatIndex	Zero-based antenna index
iPhiIndex	Zero-based phi index
iThetaIndex	Zero-based theta index
pfEHorizontal	Voltage-based gain, horizontal
pfEVertical	Voltage-based gain, vertical
pfPhaseVertical	Vertical phase with respect to horizontal phase

Return values:

boolean **true**

Success

---

cleanUp (void)

Clean up all data.

This function is called after all data has been read from the plug-in. All memory can now be freed.

C.6 List of Plugin Functions

D Scripting API

The detailed description of the script syntax is out of scope of this user manual. Refer to the script standard, that can be found under the following link:

http://www.ecma-international.org/publications/standards/Ecma-262.htm

Alternatively, any tutorial or documentation on JavaScript could be used.

In addition to the script standard, the R&S Pulse Sequencer introduces 2 proprietary objects: the global object `ps` and the newable object `QVisa`.

The objects `QDir`, `QFile`, `QMessageBox`, `QFileDialog` are `Qt` classes, where R&S Pulse Sequencer supports only a subset of the methods.

In this section, you find a list of the supported methods per class and the description of the proprietary methods and objects.

The standard `Qt` classes and methods are not described here. For information, refer to the `Qt` documentation:

http://doc.qt.io/qt-5/index.html

D.1 Global script extensions

The global object `ps` extents the script functionality. It provides the following methods within the script context:

```
QString        scpi            ( const QString &sOut )

bool           sync            ( int iTimeout )

void           traceError      ( const QString &sMsg )
void           traceDebug      ( const QString &sMsg )
void           traceInfo       ( const QString &sMsg )

void           print           ( const QString &sMsg )

void           sleep           ( unsigned int iMs )

void           setGlobalVar    ( const QString &sToken, const QString &sValue )
QString        getGlobalVar    ( const QString &sToken )
QString        getScriptPath   ( )
```

ps.scpi (const QString &sOut)

Remote control of the application itself.

This method can be used as command or query. In the latter case, the returned data is a string value.

Example:
```
ps.scpi( '*RST' )
var sIdent = ps.scpi( '*IDN?' )
```

ps.sync (int iTimeout)

Suspends script execution until the R&S Pulse Sequencer has finished a calculation task.

This is done by polling the operation condition register of the SCPI parser and wait for the command `STATus:OPERation:CONDition?` to return zero.

The method returns true if the register value becomes zero within the timeout period `iTimeout`. If there is a timeout, the method returns false.

Example:
```
var iTimoMs = 5000
ps.sync( iTimoMs )
```

ps.traceError (const QString &sMsg)
ps.traceDebug (const QString &sMsg)
ps.traceInfo (const QString &sMsg)

Traces log messages and prints messages to the "Message Log" dialog.

Three methods exist to output text messages to the application message log. By default, the message log shows only information and error messages.

Perform one of the following to enable debug output:

- Start the application with the command-line argument `--debug`
- In the command console window, enter *Set Debug = 1*.
  See also "To open the "Message Log" dialog in debug mode" on page 832.

Example:
```
ps.info( 'This is an info text.' )
ps.debug( 'This is a debug message.' )
ps.error( 'This is an error message.' )
```

ps.print (const QString &sMsg)

Printing any text or string to the "Script Manager" dialog.

Convert non-string variables into strings before using them with this method.

Example:
```
ps.print( 'My Text' )
var dVal = 1e-3
ps.print( 'Value = ' + dVal.toString() )
```

---

ps.sleep (unsigned int iMs)

Sets a waiting period specified in milliseconds and suspends script execution for at least this amount of time.

Example:
```
var iTimeMs = 500
ps.sleep( iTimeMs )
```

---

ps.setGlobalVar (const QString &sToken, const QString &sValue)
ps.getGlobalVar (const QString &sToken)

Exchanging Data Between Scripts.

The `ps` script extension provides two methods that access a global variable pool. These variables are of the data type string and remain valid for as long as the application runs. The variables are not saved when the application terminates. Each variable is identified by a name token.

The method `setGlobalVar` registers a new variable or updates an existing variable. The method `getGlobalVar` reads data from a variable. If the variable does not exist, an empty string is returned.

Example:
```
ps.setGlobalVar( 'Var1', 'This is some text.' )
ps.setGlobalVar( 'Var2', dValue.toString() )

dValue = parseFloat( ps.getGlobalVar( 'Var2' ) )
```

---

ps.getScriptPath ( )

Returns the absolute file path of the directory where the script file is stored.

D.2   Newable script extensions

The term newable describes objects that can be created dynamically using the `new` operator. A script can create multiple instances of these objects. The script engine automatically destroys all dynamically created objects at the end of the script.

D.2.1   QVisa

`QVisa` is a Rohde & Schwarz proprietary class that uses `Qt` data types. The `QVisa` object is a newable object that enables remote control access to instruments.

Instrument connections must be defined before data can be exchanged between the script and the instruments. The following structure is mandatory.

```
<Object> = new QVisa();
<Object>.define( '<VISA_String>');
<Object>.open ();

...

<Object>.close ();
```

The following `QVisa` methods are available within the script context:

```
void        define        ( const QString &sRsc )
bool        setTimeout     ( int iValueMs )
bool        open           ( )
bool        close          ( )

bool        write          ( const QString &sOut )
QString     read           ( const QString &sOut )
bool        checkErr        ( )
bool        uploadFile     ( const QString &sLocal, const QString &sRemote )
bool        downloadFile   ( const QString &sLocal, const QString &sRemote )
```

visa.define (const QString &sRsc)

Instrument connections must be defined before data can be exchanged between the script and the instruments.

The device definition is done with the `define` method.

Parameters:

sRsc　　　　　　　　　　VISA resource string of the instrument

visa.setTimeout (int iValueMs)

Sets the VISA timeout.

Parameters:

iValueMs　　　　　　　　Timeout value in milliseconds

visa.open ( )

Opens the VISA connection.

If the device connection was opened successfully, the method returns true.

---

visa.close ( )

Closes a device connection.

The application automatically closes all open connection at the end of the script.

---

visa.write (const QString &sOut)

Sends a command to the instrument.

Parameters:

sOut   Remote command

---

visa.read (const QString &sOut)

Sends a query to the instrument and then reads the result.

The method returns the data returned from the instrument as string.

Parameters:

sOut   Outbound query

---

visa.checkErr ( )

Repeatedly sends the query `SYSTem:ERRor?` on page 796 to the instrument until no more errors are returned.

Return values:

boolean   **false**
                             If errors were returned

---

visa.uploadFile (const QString &sLocal, const QString &sRemote)

Sends a file to the instrument.

The method returns true if the transfer was successful. This method uses the SCPI mass memory commands for the file transfer.

The functionality is only tested for Rohde & Schwarz vector signal generators.

Parameters:

sLocal   Local file name

sRemote   File name on the instrument file system

---

visa.downloadFile (const QString &sLocal, const QString &sRemote)

Reads a file from the instrument.

The method returns true if the transfer was successful. This method uses the SCPI mass memory commands for the file transfer.

The functionality is only tested for Rohde & Schwarz vector signal generators.

Parameters:

sLocal   Local file name

sRemote   File name on the instrument file system

D.2.2 QDir

For description of the `QDir` class, see http://doc.qt.io/qt-5/qdir.html.

The following `QDir` methods are available within script context:

```
QString        absoluteFilePath    ( const QString & fileName ) const
QString        absolutePath        ( ) const
bool           cd                  ( const QString & dirName )
bool           cdUp                ( )
QString        dirName             ( ) const
bool           exists              ( ) const
QString        filePath            ( const QString & fileName ) const
bool           isReadable          ( ) const
bool           isRoot              ( ) const
bool           mkdir               ( const QString & dirName ) const
bool           mkpath              ( const QString & dirPath ) const
QString        path                ( ) const
bool           remove              ( const QString & fileName )
bool           rmdir               ( const QString & dirName ) const
void           setPath             ( const QString & path )
QStringList    entryList           ( quint32 filters,
                                      quint32 sort ) const
// static
QString        currentPath         ( )
QString        homePath            ( )
QString        rootPath            ( )
QString        tempPath            ( )
```

D.2.3 QFile

For description of the `QFile` class, see http://doc.qt.io/qt-5/qfile.html.

The two methods `writeString` and `readLine` are Rohde & Schwarz proprietary methods. They work on the `QTextStream` method.

The following `QFile` methods are available within script context:

```
bool              exists     ( ) const
```

```
bool              open        ( quint32 mode )
bool              remove      ( )
void              setFileName ( const QString & name )
bool              copy        ( const QString & newName )
qint64            size        ( ) const
void              close       ( )
quint32           error       ( ) const

// reimplemented on QTextStream
bool              seek        ( qint64 pos )
void              flush       ( )
bool              atEnd       ( ) const

// custom methods (not part of QFile, work on QTextStream)
bool              writeString ( const QString &sStr )
QString           readLine    ( int iMaxLen )
```

file.writeString (const QString &sStr)

Writes the string to the selected file. Strings are appended as separate lines.

file.readLine (int iMaxLen)

Reads the next line of text from the file.

Setting parameters:
iMaxLen   Maximum number of characters to be read.

D.2.4 QFileDialog

For description of the `QFileDialogFile` class, see http://doc.qt.io/qt-5/qfiledialog.html.

The following `QFileDialogFile` methods are available within script context:

```
void         setAcceptMode      ( quint32 mode )
void         setDefaultSuffix   ( const QString & suffix )
void         setDirectory       ( const QString & directory )
void         setFileMode        ( quint32 mode )
void         setNameFilter      ( const QString & filter )
void         setOption          ( quint32 option, bool on )
void         setViewMode        ( quint32 mode )
QStringList  selectedFiles      ( ) const
int          exec               ( )

// static public method
QString      getOpenFileName     ( const QString & caption,
                                   const QString & dir,
                                   const QString & filter,
```

```
                                        quint32 options )


    QString        getExistingDirectory( const QString & caption,
                                         const QString & dir,
                                         quint32 options )


    QString        getSaveFileName    ( const QString & caption,
                                         const QString & dir,
                                         const QString & filter,
                                         quint32 options )
```

D.2.5 QMessageBox

For description of the QMessageBox class, see http://doc.qt.io/qt-5/qmessagebox.html.

The following QMessageBox methods are available within script context:

```
    void           setDetailedText    ( const QString & text )
    void           setIcon            ( quint32 Icon )
    void           setInformativeText ( const QString & text )
    void           setStandardButtons ( quint32 buttons )
    void           setText            ( const QString & text )
    void           setTextFormat      ( quint32 format )
    void           setWindowTitle     ( const QString & title )


    // static methods
    int            critical           ( const QString & title,
                                         const QString & text,
                                         quint32 buttons = QMessageBox::Ok,
                                         quint32 defaultButton = QMessageBox::NoButton )


    quint32        information         ( const QString & title,
                                         const QString & text,
                                         quint32 buttons = QMessageBox::Ok,
                                         quint32 defaultButton = QMessageBox::NoButton )


    int            question            ( const QString & title,
                                         const QString & text,
                                         quint32 buttons = QMessageBox::StandardButtons
                                         ( QMessageBox::Yes | QMessageBox::No ),
                                         quint32 defaultButton = QMessageBox::NoButton )


    int            warning             ( const QString & title,
                                         const QString & text,
                                         quint32 buttons = QMessageBox::Ok,
                                         quint32 defaultButton = QMessageBox::NoButton )
```

D.3 Script examples

A subset of script examples is provided in the repositories supplied with the software.

Access:

1. Select "File > Load and Manager Repositories".

2. In the "Repository Manager", double click on the "Examples > K300 > Script Example".

3. Select "Script > Manage".

   The script examples are loaded in the "Script Manager".
   You can try out script examples and further develop them.

The following is the content of one of the scripts.

IPM Diagnostics

```
// This script reads 100 IPM values and writes
// them to a text file. the data can be used for an analysis
// of the IPM data.

// get the temporary path
dir = new QDir();
temppath = dir.tempPath();
fd = new QFileDialog();

// select file
filename = fd.getSaveFileName( 'Save File As', temppath, '', 0 );
if( filename.length > 0 ) {

  // create new IPM profile
  ps.scpi( 'IPM:CRE "ipm_test"' );
  ps.scpi( 'IPM:SEL "ipm_test"' );

  // switch to normal distribution
  ps.scpi( 'IPM:TYPE RAND' );
  ps.scpi( 'IPM:RAND:DIST NORM' );
```

```
// configure distribution
ps.scpi( 'IPM:RAND:NORM:MEAN 0' );
ps.scpi( 'IPM:RAND:NORM:STD 1' );
ps.scpi( 'IPM:RAND:NORM:LIM 100' );

// start internal diagnostic system
ps.scpi( 'IPM:DIAG:INIT' );

file = new QFile();
file.setFileName( filename );
file.open( 0x0002 );

ps.print( 'Writing file...' );

for( var i=1; i<100; i++ ) {
  value = ps.scpi( 'IPM:DIAG:READ?' );
  file.writeString( value + '\n' );
}

file.close();
ps.print( 'Done' );

// remove IPM profile
ps.scpi( 'IPM:REM "ipm_test"' );
}
```

D.4 List of Script Objects and Methods

E Common coding algorithms

In the notation used below a_n denotes the nth input symbol and b_n denotes the correspondingly coded output symbol. Individual bits in the symbols from the LSB (least significant bit) to the MSB (most significant bit) are denoted by a_{0n}, a_{1n}, etc. The same applies to the output symbols.

Common coding types are listed in Table E-1.

Table E-1: Common coding algorithms

Coding	Coding algorithm	Applicable for K bit/symbol
"None"	$b_n = a_n$	k = 1...8
"Differential"	$b_n = (a_n + b_{-1})$ modulo 2^k	k = 1...7
"Differential + Gray"	Gray coding with differential coding	k = 1...7

The following example illustrates how one of common coding schemes in combination with a modulation method influence the signal.

Example: Differential coding for QPSK modulation with K = 2 bit/symbol

Decimal display; the value range for modulation symbols is $a_n = \{0, 1, 2, 3\}$

According to Table E-1 and for $k = 2$, the recursive coding is defined as $b_n = (a_n + b_{-1})$ *modulo 4*

Depending on the state of a preceding modulation symbol b_{-1}, the coded modulation symbol b_n is obtained for example from modulation symbol $\mathbf{a_n = 2}$ as follows:

b_{-1}	0	1	2	3
b_n	2	3	0	1

With differential coding, the assignment of modulation symbols a_n (binary indication: MSB, LSB) to the phase differences shown in the following table is generated:

Table E-2: Phase difference for QPSK

Modulation symbol a_n	00	01	10	11
Phase difference	0°	90°	180°	270°

F Supported filter and impact of the filter parameters

The Table F-1 shows the filters that are available, together with their associated parameters. The filter characteristic is displayed in graphical form.

Table F-1: Overview of the supported filters

Cosine	Root Cosine
Roll Off Factor	Roll Off Factor
Bandwidth	
Gauss (FSK)	Rectangular
B x T	

The following is a simple description of the filter parameters and the way they affect the main filter characteristics. Changing filter parameters is an effective way to ensure that the entire bandwidth of the desired signal is allowed to pass and adjust the filter form to reach spectrum mask requirements.

Cut Off Frequency

The cutoff frequency or corner frequency is a filter characteristic that defines the frequency at the 3 dB down point. This frequency is the bound to the transition band; here the filter characteristic changes form the passband to the stopband, where the signal is suppressed.

Rolloff Factor

The roll off factor is a measure for the excess bandwidth compared to the ideal band-width of a "brick like" filter. The roll off factor affects the steepness of the filter flanks. A "Rolloff Factor" = 0 would result in the steepest flanks theoretical possible; values near to 1 make the flanks more flat.

Passband

The passband describes the frequency span that the filter passed unchanged. The total passband of a filter is given as follows:

```
Bandwidth = (1 + "Roll Off Factor") * "Symbol Rate"
```

G References

- R&S SMW200A Vector Signal Generator user manual, the latest version is available for download at www.rohde-schwarz.com/manual/smw200a.

- R&S SMBV100A Vector Signal Generator operating manual, the latest version is available for download at www.rohde-schwarz.com/manual/smbv100b.

- R&S SMBV100A Vector Signal Generator operating manual, the latest version is available for download at www.rohde-schwarz.com/manual/smbv100a.

- R&S SMM Vector Signal Generator operating manual, the latest version is available for download at www.rohde-schwarz.com/manual/smm100a.

- R&S SGT100A SGMA Vector RF Source operating manual, the latest version is available for download at www.rohde-schwarz.com/manual/sgt100a.

Glossary: Terms and abbreviations

A

AM: Amplitude modulation

Antenna: The antenna characterizes the radiation pattern of the Emitter

ARB: Arbitrary Waveform Generator
An I/Q modulation source forming a part of the supported signal generators. The ARB allows the playback and output of any externally calculated modulation signal in the form of waveform file. It also allows the generation of multi-carrier or multi-segment signals from waveform files.

ASK: Amplitude-shift keying modulation

ASR: Airport Surveillance Radar

AWGN: Additive white gaussian noise

B

Barker: Special sequences (codes) that assures phase modulated signals with low autocorrelation properties.

Boresight: Antenna boresight is the direction to which an antenna shows the maximum gain.

BPSK: Binary phase shift keying modulation

BW: Bandwidth

C

C-BPSK: BPSK modulation with a constant envelope

Chirp: Signal in which the frequency varies over the time.

CW: Continuous wave signal, that is an unmodulated signal.

D

DQPSK: Differential QPSK

DUT: Device under test

E

EIRP: Equivalent isotopically radiated power

ELINT: Electronic intelligence, meaning information that comes from signals that typically do not contain speech or text.

Emitter: In the context of this software, emitters emulate radar systems.

EW: Electronic warfare

F

FFT: Fast Fourier transform

FM: Frequency modulation

FSK: Frequency-shift keying modulation

G

Gain: Antenna gain is a measure of the antenna's ability to concentrate electromagnetic energy in a narrow beam.

GUI: Graphical User Interface

H

HPBW: Half-Power Beam Width

I

Inter-Pulse Modulation: IPM

Intra-Pulse Modulation: MOP

IPM: Inter-Pulse Modulation

M

MOP: Modulation on Pulse

MSW: Multi-segment waveform

O

OQPSK: Offset QPSK

P

PAPR: Peak to average power ratio

PDW: Pulse Descriptor Word
Format of the reported data.

Plugin: A loadable Microsoft Windows DDL module that describes custom modulation schemes or envelope shapes.

Poly Phase: polyphase

polyphase: Code used by pulse modulation and suitable for pulse compression.

PRF: Pulse repetition frequency

PRI: Pulse repetition interval
Defines the overall time of a pulse cycle.

PRT: Pulse repetition time

Pulse sequence: Pulse train

Pulse train: A sequence of repetitive pulses

Pulse-to-Pulse Modulation: IPM

PW: Pulse width

Q

QPSK: Quadrature phase shift keying modulation

R

RADAR: Radio Detecting and Ranging

RCS: Radar cross section, RCS, or σ
The RCS is a measure of the energy that an object intercepts and scatters back towards the radar.

Repository: In the context of this software, a repository is a file-based database located on the local hard drive or on a network storage drive.

S

Scenario: In the context of this software, a scenario is the top-level description of the signals to be generated.

Sequence: In the context of this software, a sequence describes how pulses are arranged to form a waveform.

Stagger PRI: Pulse train composed of two or more pulse subtrains that use the same PRI.

V

VSG: Vector Signal Generator

Index