PROPSIM Reference User Guide

Using PROPSIM hardware and software
PROPSIM F64 F8800A, F8800B
PROPSIM FS16 F8820A, F8820B

User Guide

KEYSIGHT TECHNOLOGIES

Notices

Copyright Notice

Manual Part Number

F8800-93111

Revision

Revision 10.2, 16th Sep 2024

Published by:

Keysight Technologies Finland Oy
Elektroniikkatie 10
90590 Oulu, Finland

Technology Licenses

The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license.

Third Party Software

This software uses MATLAB Runtime. MATLAB®. © 1984 – 2016. The MathWorks, Inc.

Open Source Licensing

Portions of this software are licensed by third parties including open source terms and conditions, copies of which may be found from "opensource" directory in PROPSIM software installation folder and PROPSIM External Unit Driver installation folder.

Declaration of Conformity

Declarations of Conformity for this product and for other Keysight products may be downloaded from the Web. Go to http://www.keysight.com/go/conformity and click on "Declarations of Conformity." You can then search by product number to find the latest Declaration of Conformity.

U.S.Government Rights

The Software is "commercial computer software," as defined by Federal Acquisition Regulation ("FAR") 2.101. Pursuant to FAR 12.212 and 27.405-3 and Department of Defense FAR Supplement ("DFARS") 227.7202, the U.S. government acquires commercial computer software under the same terms by which the software is customarily provided to the public. Accordingly, Keysight provides the Software to U.S. government customers under its standard commercial license, which is embodied in its End User License Agreement (EULA), a copy of which can be found at http://www.keysight.com/find/sweula. The license set forth in the EULA represents the exclusive authority by which the U.S. government may use, modify, distribute, or disclose the Software. The EULA and the license set forth therein, does not require or permit, among other things, that Keysight: (1) Furnish technical information related to commercial computer software or commercial computer software documentation that is not customarily provided to the public; or (2) Relinquish to, or otherwise provide, the government rights in excess of these rights customarily provided to the public to use, modify, reproduce, release, perform, display, or disclose commercial computer software or commercial computer software documentation. No additional government requirements beyond those set forth in the EULA shall apply, except to the extent that those terms, rights, or licenses are explicitly required from all providers of commercial computer software pursuant to the FAR and the DFARS and are set forth specifically in writing elsewhere in the EULA. Keysight shall be under no obligation to update, revise or otherwise modify the Software. With respect to any technical data as defined by FAR 2.101, pursuant to FAR 12.211 and 27.404.2 and DFARS 227.7102, the U.S. government acquires no greater than Limited Rights as defined in FAR 27.401 or DFAR 227.7103-5 (c), as applicable in any technical data.

Warranty

THE MATERIAL CONTAINED IN THIS DOCUMENT IS PROVIDED "AS IS," AND IS SUBJECT TO BEING CHANGED, WITHOUT NOTICE, IN FUTURE EDITIONS. FURTHER, TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, KEYSIGHT DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED WITH REGARD TO THIS MANUAL AND ANY INFORMATION CONTAINED HEREIN, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. KEYSIGHT SHALL NOT BE LIABLE FOR ERRORS OR FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH THE FURNISHING, USE, OR PERFORMANCE OF THIS DOCUMENT OR ANY INFORMATION CONTAINED HEREIN. SHOULD KEYSIGHT AND THE USER HAVE A SEPARATE WRITTEN AGREEMENT WITH WARRANTY TERMS COVERING THE MATERIAL IN THIS DOCUMENT THAT CONFLICT WITH THESE TERMS, THE WARRANTY TERMS IN THE SEPARATE AGREEMENT WILL CONTROL.

Customer support

It is our goal to provide you with excellent Customer Support. To request assistance with any aspect of your test system, please create a Help Desk Request (HDR) using the NES Wireless Solutions Help Desk.

To access the NES Wireless Solutions Help Desk, and to download the latest releases of software and documentation, please log in to Keysight Support. On the Keysight Support home page, in the Additional Support, click the link for the NES Wireless Solutions Help Desk, or in the Assets, click Keysight Software Manager for software downloads.

Product security

Keysight is committed to providing information to assist you in protecting your products and solutions from external cyber threats. Please contact us immediately if you find a Keysight product or solution is impacted by a cyber security issue. Go to https://www.keysight.com/us/en/about/quality-and-security/security/product-and-solution-cyber-security.html for more information.

SAFETY NOTICES

Safety symbols and terms on Keysight equipment have the following meanings:

	This is a general hazard sign, which is used to draw the user's attention to warnings and cautions
WARNING!	The WARNING! sign denotes a hazard. It identifies conditions or practices that could result in personal injury or death.
CAUTION	The CAUTION sign denotes a hazard. It identifies conditions or practices that could result in damage to the equipment or to other property.

The following safety notices can be found on Keysight equipment:

WARNING!

Before this instrument is switched on, make sure it has been properly grounded through the protective conductor of the ac power cable to a socket outlet provided with a protective earth contact.

Any interruption of the protective (grounding) conductor, inside or outside the instrument, or disconnection of the protective earth terminal can result in PERSONAL INJURY.

Install the device to a location allowing easy access to the device main power switch. Ensure the installation site has sufficient clearances around the device cabinet.

If this instrument is used in a manner not specified by Keysight Technologies, the protection provided by the instrument may be impaired.

There are many points in the instrument that can, if contacted, cause personal injury. Be extremely careful. Any adjustments or service procedures that require operation of the instrument with the protective covers removed may be performed only by trained service personnel.

Do not cover the device. Make sure that the emulator has unrestricted airflow for the fan and ventilation openings.

The emulator should be carried by at least two people due to its size and weight.

The main switch of the device is equipped with over current protection. The switch bounces back to OFF position in case of over current. If this happens and the switch refuses to stay in ON position, do not try to force the switch.

CAUTION

Before this instrument is switched on, make sure its primary power circuitry has been adapted to the voltage of the ac power source.

Failure to set the ac power input to the correct voltage could cause damage to the instrument when the ac power is plugged in. See product specification for more details.

Note: This equipment has been tested and found to comply with the limits for Class A digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference when the equipment is operated in a commercial environment. The equipment generates, uses and can radiate radio frequency energy and, if not installed and used in accordance with the instruction manual, may cause harmful interference to radio communications. Operation of this equipment in a residential area is likely to cause harmful interference in which case the user is required to correct the interference at his own expense.

Note: Changes or modifications not expressly approved by the party responsible for compliance could void the user's authority to operate the equipment.

Important information concerning product disposal and recycling

When this crossed-out garbage bin symbol is attached to a product, it may not be mixed with general waste in case of product disposal.

There is a separate collection system for used electronic products in accordance with legislation that requires proper treatment, recovery and recycling.

In case of product disposal, customers of this product (PROPSIM) in the member states of the EU, Switzerland and Norway may return their used products free of charge to a retailer to be disposed properly.

For countries not mentioned above, please contact your local authorities for the correct method of disposal.

By doing so you ensure that your disposed product undergoes the necessary treatment, recovery and recycling, and thus prevents potential negative effects on the environment and human health.

For more information, please contact Keysight.

ABBREVIATIONS AND TERMS

Name	Description
ACU	Auto Calibration Unit, Keysight equipment to be used with PROPSIM for automated setup gain/phase calibration
AD	Analog-to-digital
AGC	Automatic gain control
AILC	Automatic input level control
AILS	Automatic input level settings
Auto alignment	User-controlled operation where PROPSIM aligns phases and levels between multiple channels in test setup
Auto calibration	See auto alignment.
.ASC	Text file format for channel impulse response data
ASO	Aerospace and Satellite Option
ATE	Automatic Test Equipment
AWGN	Additive White Gaussian Noise
BS	Base Station
BW	Bandwidth
C/I	Carrier-to-Interference
CIR	Channel Impulse Response
.CIR	File format for impulse response data
CIU	Common Interface Unit
.COR	Correlation file
CSS	Channel Sequence Simulation
.CSV	Data file for local data logging
.CTAP	File containing channel parameters
CW	Carrier Wave = Continuous Wave = Sine Wave
DA	Digital-to-analog
DoA	Direction of Arrival
DoD	Direction of Departure
DUT	Device Under Test
GCM	Geometric Channel Modeling
GSM	Global System for Mobile communications
GUI	Graphical User Interface
HDD	Hard Disk Drive
.ICS	CSS generated model file format
IR	Impulse Response
.IR	File format for impulse response data
LOS	Line-of-sight

Name	Description
LTE	Long Term Evolution
MANET	Mobile Ad-hoc Network
.MAT	Matlab file
MIMO	Multiple Input Multiple Output
MISO	Multiple Input Single Output
MPC	Multipath component
MS	Mobile Station
OTA	Over The Air
PHN	Phase noise
RF	Radio Frequency
RFLO	RF Local Oscillator signal
RRH	Remote Radio Head
RSRP	Reference Signal Received Power (LTE, 5GNR)
RTC	Run-Time Control (emulation type)
.RTC	Run-time channel model file
RX	Receiver
SD	Sample density: number of samples per half-wave. Half of the wavelength of the carrier wave divided by the sample distance. When simulating a moving vehicle, channel impulse responses are updated every λ/SD, where λ is the wavelength of the carrier wave.
.SHD	Shadowing file
.SIM	Emulator hardware control file
SIMO	Single Input Multiple Output
SIR	Signal-to-interference Ratio
SISO	Single Input Single Output
.SMU	Emulation configuration file
SNMP	Simple Network Management Protocol
SNR	Signal-to-noise ratio
.TAP	File containing channel parameters
TDD	Time Division Duplex
TDL	Tapped Delay Line
TX	Transmitter
UDP	User Datagram Protocol
UE	User Equipment
VDT	Virtual Drive Test
WCDMA / W-CDMA	Wideband Code Division Multiple Access
WiMAX	Worldwide Interoperability for Microwave Access
.WIZ	Last part in the name of a folder that contains all the related files in an emulation created in the Scenario Wizard. Folder naming convention: “[name of emulation].wiz”.

CONTENTS

1 INTRODUCTION

PROPSIM Product Specification and User Guidance Documentation consists of several documents: PROPSIM Quick Start Guide, PROPSIM Reference User Guide (this document), PROPSIM Modelling Tool documents, Test Scenario Pack specifications, Technical Notes/Application Notes and Release Note documents.

PROPSIM Quick Start Guide is a use case-oriented guide of the PROPSIM radio channel emulator. It covers the initial setup of the product and provides instructions on use of the views and applications for most typical applications. Quick Start Guide is included in the PROPSIM delivery as printed copy.

User Reference documentation includes more detailed information about the use, maintenance and troubleshooting as well as technical specifications and product configuration options of the PROPSIM. The User Reference is divided into chapters listed in Table 1.

When mentioned in this user reference, term “PROPSIM” applies to PROPSIM F64 (F8800A, F8800B) and PROPSIM FS16 (F8820A, F8820B). The specific product name is used when the topic is valid for that product only.

Some of the instructions in this user reference use PROSIM F64 as an example. However, all instructions apply to both PROPSIM F64 and PROPSIM FS16 unless stated otherwise.

Table 1 Chapters in User Reference Documentation

Chapter	Name	Contents
1	Introduction	Introduction to the PROPSIM and user documentation (this section)
2	Navigation in PROPSIM User Interface	Description of the main components of the Graphical User Interface
3	Scenario Wizard	Description of creating or editing an emulation using the Scenario Wizard
4	Emulation Control View	Description of the Emulation Control View and instructions for running and editing emulations
5	Data Views	Description of the Data Views window and the different views
6	Lab Setup Feature	Description of the lab setup feature
7	Utilities	Description of the tools in the Utilities submenu
8	Data Logging	Description of Data Logging feature
9	System Configuration and Shutdown	Description of the device configuration options and instructions for shutting down and restarting the system
10	SNMP Interface	Description of Simple Network Management Protocol support
11	User Alignment	Description of user alignment feature
12	Shadowing (Optional Feature)	Description of Shadowing controls and fields in GUI
13	Internal Interference Generator Optional Feature)	Description of Internal interference generator controls and fields in GUI
14	Phase Noise Generator (Optional Feature)	Description of phase noise generator controls and fields in GUI
15	Signal Capture (Optional Feature)	Description of signal capture controls and fields in GUI
16	Signal Waveform Playback (Optional Feature)	Description of signal waveform playback controls and fields in GUI
17	Integrated Setup Alignment Feature (Optional Feature)	Description of auto alignment GUI
18	Extended Frequency Range (Optional Feature)	Description of extending frequencies with external devices (CIU and RRH)

Chapter	Name	Contents
19	Aerospace and satellite modelling tool - ASO (Optional Feature)	Aerospace modelling feature description
20	Standard Tools Remote Control Interface and Functions	Description of ATE commands supported for test automation through LAN
21	File Formats	Description of file formats supported by PROPSIM for exporting and importing channel impulse response data
22	Maintenance	Instructions related to product maintenance
23	Troubleshooting	Explanation of error messages and advice on troubleshooting actions

Separate documentation is available about PROPSIM Standard Channel Models and for Geometric Channel Modelling (GCM) Tool, Virtual Drive Test (VDT) Modelling Tool, WLAN Modelling Tool and Test Scenario Packs.

All the documentation is also available in Keysight Software Manager. It can be accessed by logging into Keysight Support home page (https://support.keysight.com).

1.1 Document history

The following table lists the main changes to issues of this document.

Issue	Date	Summary of Changes
10.2	Sep 2024	Release 10.2. Updates to related to product security. Figure 240 with incorrect frequency information updated.
10.0	Apr 2024	Release 10.0. Updates to Emulation Control View, various feature enhancements and other minor updates.
9.0	Jan 2024	Release 9.0. Updates to Emulation Control View, various feature enhancements and other minor updates.
8.0	Apr 2023	Release 8.0. Updates to Emulation Control View, various feature enhancements and other minor updates.
7.0	Oct 2022	Release 7.0. Updates to Emulation Control View, various feature enhancements and other minor updates.
6.0	Dec 2021	Release 6.0. Updates to Emulation Control View, various feature enhancements and other minor updates.
5.0	Jun 2021	Release 5.0. Updates to Emulation Control View, various feature enhancements and other minor updates.
4.0	Mar 2021	Release 4.0. Updates to Emulation Control View and Data View chapters. New chapters for phase noise and signal capture added. Other minor updates.
3.0	Sep 2020	Release 3.0. Updates to Emulation Control View and Data View chapters. New chapter 19 Aerospace and satellite modelling tool – ASO (Optional Feature). Other minor updates.
2.0	Feb 2020	Release 2.0. Major updates corresponding to the updated PROPSIM GUI.
1.0	Jun 2019	Release 1.0

1.2 PROPSIM

Note: Some of the instructions in this user reference use PROSIM F64 as an example. However, all instructions apply to both PROPSIM F64 and PROPSIM FS16 unless stated otherwise.

The wireless environment imposes many constraints and limitations on the performance of wireless telecommunication systems. Radio channel propagation characteristics, such as attenuation, shadowing, fast fading, variable delays, Doppler effect, noise, and interference cause severe degradation to all wireless transmission. Thus, the development of wireless communication systems requires rigorous testing to ensure that the products can operate even under the most demanding propagation environments.

The traditional field-testing of wireless systems is generally labor intensive, time-consuming, and expensive. Furthermore, even under the same test setups and test scenarios the test results are typically non-repeatable since the propagation environment uncertainties plus external noise and interference affect the results.

The PROPSIM is a radio channel emulator that enables recreating the wireless channel propagation effects in a controlled laboratory environment. It is a one box solution for performing a realistic and accurate emulation of all typical radio channel propagation effects such as multipath propagation, fast fading, dynamic delays, attenuation, noise, interference, and shadowing. The PROPSIM supports multiple channels, wide bandwidth, high dynamic range, and channel emulation with very high accuracy. The physical radio channel characteristics can all be emulated independently on PROPSIM supplementing, or even replacing traditional field-testing. PROPSIM product family is shown in Figure 1.

Figure 1 PROPSIM F64 F8800A (left) and PROPSIM FS16 F8820A (right)

PROPSIM products have been developed to fulfill the requirements of both present and future wireless communication systems. With extremely high RF performance they provide superior performance and feature set for radio interface testing of current communication systems, such as WCDMA, GSM, TD-SCDMA, EV-DO / CDMA2000, 3GPP LTE, LTE-A, WiMAX, Wi-Fi and 802.11n/ac/ax and 5G.

1.2.1 Channel emulation concept

The PROPSIM is a generic channel emulator. It emulates only the radio channel excluding transmitter and receiver and is thus independent of system technology or modulation. The PROPSIM supports all major wireless standards and signal types in a broad frequency range. It supports the development of most demanding wireless applications, such as beamforming, 8×8 MIMO, software defined radios and aerospace satellite communications.

In a typical test scenario, the transmitter and receiver to be tested are connected to the PROPSIM, which then emulates a wireless propagation environment, replacing the real radio channel, as illustrated in Figure 2. The PROPSIM uses real-world signals generated by external test equipment as an input.

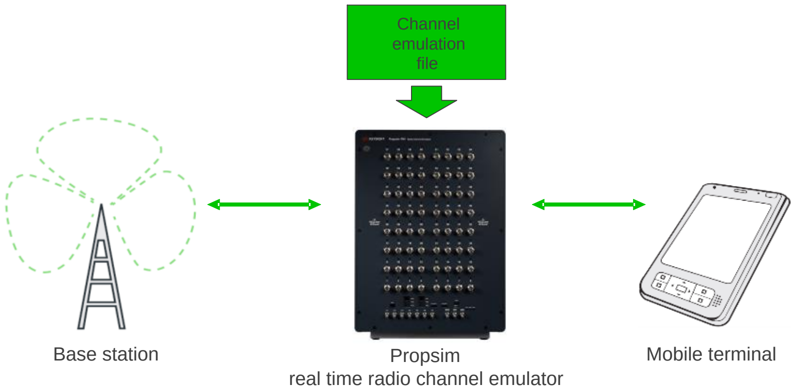


Figure 2 Replacing the radio channel with PROPSIM

The overall flexibility and extensive number of configuration options of the PROPSIM allow it to run several different types of tests and emulations. The emulations are based on preconfigured channel emulation files, stored in the PROPSIM. New emulation files can be added any time by the users. The file-based emulation approach ensures full repeatability and controllability of the emulations.

The PROPSIM includes pre-stored standard emulations (emulation connection diagram with channel models according to 3GPP). In basic use, the user only needs to load the emulation and adjust the power levels of the PROPSIM. For advanced use, all versatile channel modelling tools for creating user defined channel models and to modify pre-defined standard channel models are available.

The PROPSIM includes an easy to use GUI that guarantees quick emulation setup. The GUI can be used to control all aspects of the emulator functionality. It ensures that all emulation setup and configuration tasks require minimal amount of manual work.

The PROPSIM includes a “toolbox” of applications for creating channel models and emulations. Several channel models can also be combined for a single emulation which may use up to 128 fading channels, and thus up to 128 different channel models. The channel models are stored in the PROPSIM as pre-calculated files.

1.2.2 Physical connectors and LEDs

1.2.2.1 PROPSIM F64 (F8800A)

All external connectors and LEDs of the PROPSIM are found in the emulator front panel, as shown in Figure 3.

Figure 3 PROPSIM F64 (F8800A – ARF1)

The PROPSIM F64 is equipped with up to 64 duplex RF channels. Table 2 lists the physical connectors and LEDs found in the front panel of the PROPSIM F64.

Table 2 PROPSIM F64 connectors and LEDs

Name	Description
RF IN/OUT (1-64)	RF input/output connector (for each physical emulator channel).
SFP+	Digital data interface (for future use)
IOIOI	Digital communication interface (for future use)
LAN	Gigabit LAN, 10/100/1000 Base-T connector for LAN connectivity
USB	USB connectors (4 x USB 3.1, 2 x USB 2.0) for external keyboard, mouse, and memory. ***Note:*** *USB cable length should be less than 3m*
DisplayPort	DisplayPort connectors (2) for external displays
C1	Sync in
C2	Sync out
C3	Trigger in (Signal Waveform playback and Signal Capture)
C4	Trigger in (Signal Waveform playback and Signal Capture)
C5 – C7	Control connection to external units
200 MHz Ref Out	200 MHz sampling clock signal output
10 MHz Ref In	Input connector for 10 MHz reference signal
10 MHz Ref Out	Output connector for 10 MHz reference signal
Power indicator	LED inside the stand-by button
Power LED	Power status: Green — Power on
Status LED	System status: Green/off — Status ok Red — Warning (Indicated in case of overheating, over voltage or self-test failure. More detailed information about the reason of the warning is reported in the GUI.)
HDD LED	Hard disk status: Blinking green — HDD activity
CH STATUS LED	Channel status (for each RF connector): No indication — Connector not in use Green — Status ok Blinking — Emulation running Blue — Connector selected in GUI Red — Warning (Indicated in case of overheating, input cut-off, or self-test failure. More detailed information about the reason of the warning is reported in the GUI.)

1.2.2.2 PROPSIM FS16 (F8820A)

All external connectors and LEDs of the PROPSIM are found in the emulator front panel, as shown in Figure 4.

Figure 4 PROPSIM FS16

The PROPSIM FS16 is equipped with up to 16 Bidirectional TRX ports or 16 Unidirectional TX ports. Table 3 lists the physical connectors and LEDs found in the front panel of the PROPSIM FS16.

Table 3 PROPSIM FS16 connectors and LEDs

Name	Description
RF IN/OUT	RF In / Out connectors are even numbered
RF OUT	RF Out connectors are odd numbered
SFP+	Digital data interface (for future use)
IOIOI	Digital communication interface (for future use)
LAN	Gigabit LAN, 10/100/1000 Base-T connector for LAN connectivity
USB	USB connectors (4 x USB 3.1, 2 x USB 2.0) for external keyboard, mouse and memory ***Note:*** *USB cable length should be less than 3m*
DisplayPort	DisplayPort connectors (2) for external displays
C1	Sync in
C2	Sync out
C3	Trigger in (Signal Waveform playback and Signal Capture)
C4	Trigger in (Signal Waveform playback and Signal Capture)
C5 – C7	Control connection to external units
200MHz Ref Out	200 MHz sampling clock signal output
10 MHz Ref In	Input connector for 10 MHz reference signal
10 MHz Ref Out	Output connector for 10 MHz reference signal
Power indicator	LED inside the stand-by button
Power LED	Power status: Green Power on
Status LED	System status: Green/off Status ok Red Warning (Indicated in case of overheating, over voltage or self-test failure.

Name	Description
	More detailed information about the reason of the warning is reported in the GUI.)
HDD LED	Hard disk status: Blinking green HDD activity
CH STATUS LED	Channel status (for each RF connector): No indication Connector not in use Green Status ok Blinking Emulation running Blue Connector selected in GUI Red Warning (Indicated in case of overheating, input cut-off, or self-test failure. More detailed information about the reason of the warning is reported in the GUI.)

1.2.2.3 PROPSIM F64/FS16 (F8800B and F8820B)

All external connectors and LEDs of the PROPSIM are found in the emulator front panel, as shown in Figure 5.

Figure 5 PROPSIM F8800B and F8820B with BRF1 channel units

Figure 6 PROPSIM F8800B and F8820B with BRF2 channel units

Table 4 lists the physical connectors and LEDs found in the front panel of the PROPSIM F8800B and F8820B.

Table 4 PROPSIM F8800B and F8820B connectors and LEDs

Name	Description
RF IN/OUT	RF In / Out connectors, even numbered RF connectors in BRF1 channel units, all RF connectors in BRF2 channel units
RF OUT	RF Out connectors, BRF1 (odd numbered RF connectors)
SFP+	Digital data interface (for future use)
IOIOI	Digital communication interface (for future use)
LAN1, LAN2	Gigabit LAN, 10/100/1000 Base-T connector for LAN connectivity (2)
USB	USB A connectors (6) for external keyboard, mouse and memory. USB C connector (1) for Keysight smart accessories. Note, USB cable length should be < 3m.
DisplayPort	DisplayPort connectors (2) for external displays
C1	Sync in
C2	Sync out
C3	Trigger in (Signal Waveform playback and Signal Capture)
C4	Trigger in (Signal Waveform playback and Signal Capture)
C5 – C7	Control connection to external units
System Ref Out	System reference output for Keysight accessories
10 MHz Ref In	Input connector for 10 MHz reference signal
10 MHz Ref Out	Output connector for 10 MHz reference signal
Power indicator	LED inside the stand-by button
Power LED	Power status: Green Power on
Status LED	System status: Green/off Status ok Red Warning (Indicated in case of overheating, over voltage or self-test failure. More detailed information about the reason of the warning is reported in the GUI.)
HDD LED	N/A
CH STATUS LED	Channel status (for each RF connector): No indication Connector not in use Green Status ok Blinking Emulation running Blue Connector selected in GUI Red Warning (Indicated in case of overheating, input cut-off, or self-test failure. More detailed information about the reason of the warning is reported in the GUI.)

1.2.3 Removable hard disk

PROPSIM models F8800B and F8820B have a hard disk as a removable module. Removable hard disk for the PROPSIM models F8800A and F8820A is available as a separate option (F8800A-RHD / F8820A-RHD). Picture of removable hard disk in the front panel is show in Figure 7.

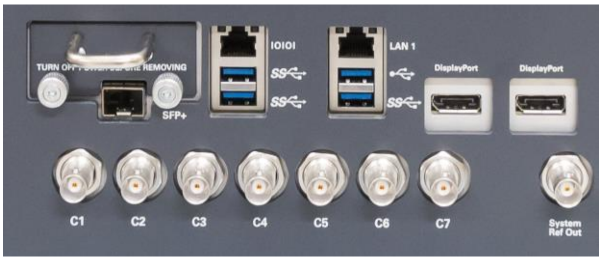


Figure 7 Removable hard disk in the PROPSIM front panel

Important: Always power off the device before removing the hard disk module. Do not power up the device without a hard disk module in place.

Important: Never install HDD modules of A-model devices (F8800A/F8820A) to B-model devices (F8800B/F8820B) or vice versa.

To remove the hard disk module, untighten two thumbscrews and gently pull the module from the handle and slide it out. To install the hard disk module, slide it to the slot and secure it in place by tightening the two thumbscrews.

After changing the hard disk module and powering up the device, always verify that the software starts without errors or warnings. If the software reports a warning about unsupported software/firmware configuration, as shown in Figure 8, install the latest PROPSIM firmware installer.

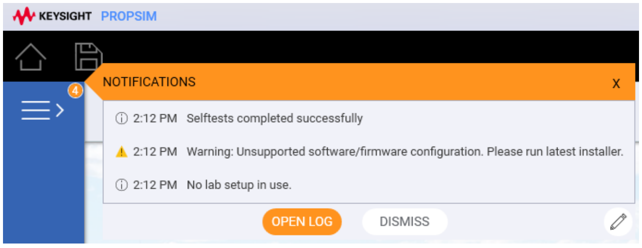


Figure 8 Warning about unsupported software/firmware configuration

Note: The hard disk module contains a PROPSIM software license file (C:\PROPSIM\lservrc) which is bind to a serial number of the PROPSIM device. When changing to a different hard disk module, license must be updated to match the device.

Typically changing the removable hard disk module will not modify the active boot device setting in BIOS/UEFI and operating system will boot normally. In case the operating system does not start, or the device shows black UEFI shell, power cycle the device and enter the BIOS/UEFI settings by pressing DEL key during startup. Change the “Boot option #1” to use “Windows Boot Manager”, as shown in Figure 9. Then select “Save & Exit” -> “Changes and exit”.

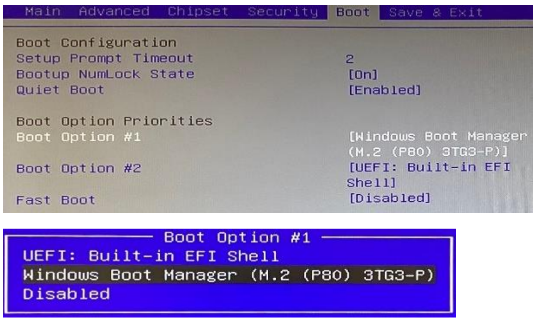


Figure 9. Change boot order in BIOS/UEFI

1.2.4 PROPSIM views and applications

The PROPSIM GUI has a navigation bar located on the left side. The navigation bar provides easy access to all views and applications in the PROPSIM. The GUI is shown in Figure 10 and Figure 11.

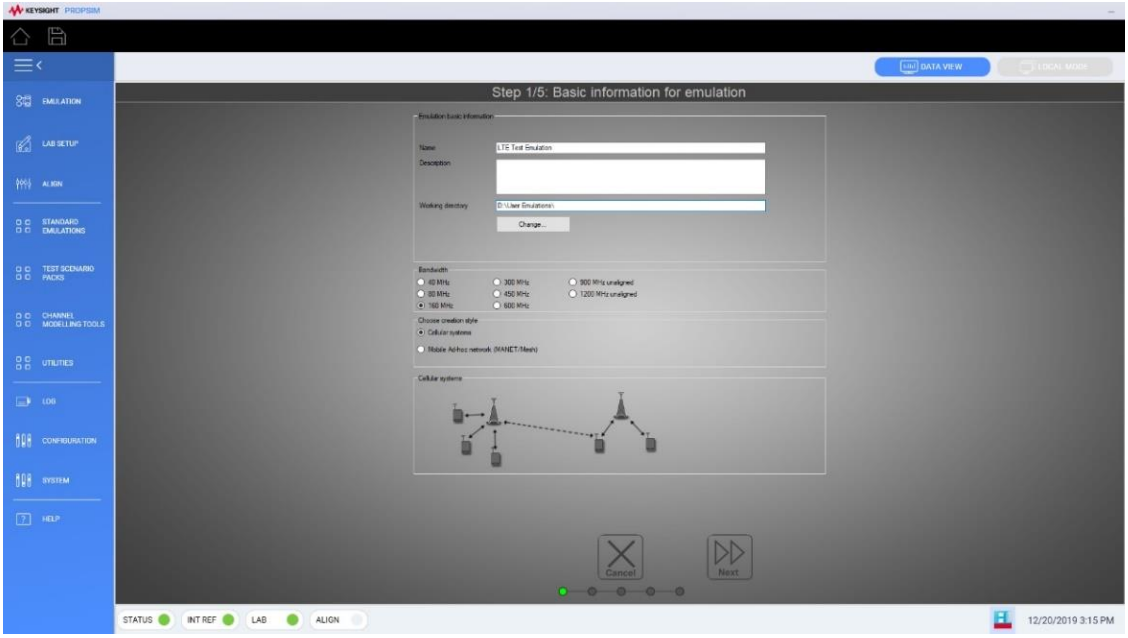


Figure 10 PROPSIM graphical user interface, Scenario Wizard

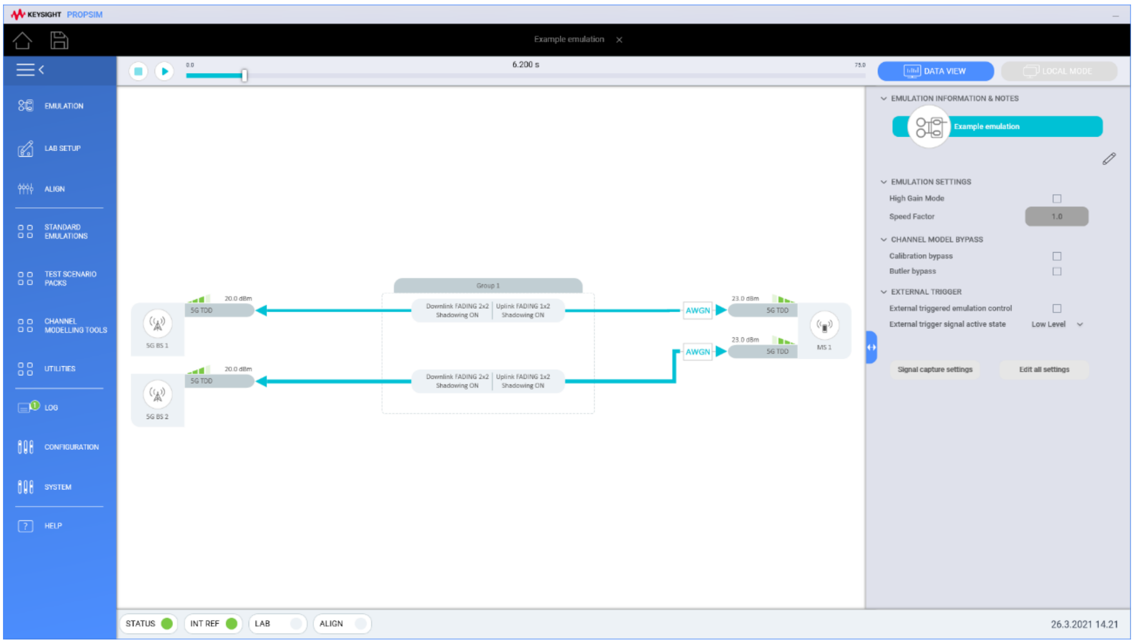


Figure 11 PROPSIM graphical user interface, Emulation Control View

The PROPSIM includes Standard Tools and optional Advanced Channel Modelling Tools for creating channel models and emulations - as well as tools to run the emulations and control the emulation playback.

- The Scenario Wizard is used to easily create emulations for different test scenarios, technologies, and lab environments.
- The Emulation Control View is used to run emulations pre-stored in the PROPSIM or created with the channel modelling tools. The Emulation Control View also allows you to control many of the hardware emulation operations of the PROPSIM.
- The Data Views offer different kinds of data about the emulation that is open or running.
- Lab Setup feature is used to store the common, typically fixed, laboratory and testing environment related PROPSIM parameters to separate ‘lab setups’. The most typical fixed parameters to store in lab setup are

frequencies, connector mapping and cable losses. The stored lab setups can be used with any suitable emulation, eliminating the need of defining the common parameters separately for each used emulation file.

- Auto Alignment (optional feature) is a tool for aligning phases and levels between multiple channels in the test setup.
- Standard Emulations are pre-installed emulations that enable quick emulator setup for standard test cases.
- Test scenario packs contain additional, separately installed, ready-made emulations and corresponding documentation for testing of technology or application specific standardized scenarios.
- Utilities:
  - Channel Model View is used to create statistical channel models.
  - Correlation Editor is used to define statistical correlation between channels of correlative or MIMO models.
  - ASO Editor is a tool for creating aerospace channel models.
  - Run-time Model Editor is used to create models for emulations using run-time control.
  - Multi Emulator Scaler is a tool for aligning emulation gains/levels in multi-emulator setups. This tool is internally used by channel modelling tools and used separately only in special cases.
  - Batch Builder is a tool for building all the selected emulations at once.
  - Running view is a legacy view that can be used to load and run emulations.
- Data logging can be used to store emulation run-time data, such as power levels, speeds, etc. to a local file or stream it out with LAN UDP protocol.
- Shadowing editor (optional feature) is used for defining shadowing profiles for channels. It can also be used to generate path loss triggered handovers / handoffs. The Shadowing editor can be launched in the Scenario Wizard.
- Interference generator (optional feature) is used to create interference sources for the emulations. Interferer can be added to an emulation in the Scenario Wizard.
- Phase noise generator can be added to any input/output to emulate receiver or transmitter phase noise (optional feature).
- Signal capture feature is used for capturing user signal from PROPSIM inputs (optional feature).
- Geometric Channel Modelling (GCM) Tool (optional feature) is a channel modelling tool for creating dynamic spatial channel models.
- VDT RF field to lab (optional feature) Automated, lab-based performance and interoperability test solution for mobile devices and network infrastructure.

  WLAN modelling tool (optional feature) is used to generate radio channel realizations according to IEEE 802.11n/ac/ax Channel Models specification.

The emulator tools are described in detail in other sections of the User Reference documentation. Separate documentation is available about PROPSIM Standard Channel Models and for Geometric Channel Modelling (GCM) Tool, Virtual Drive Test (VDT) Modelling Tool, WLAN Modelling Tool and Test Scenario Packs. All the documentation is also available in Keysight Software Manager (http://www.keysight.com/my). This requires myKeysight account.

1.2.5 Operating system, security, and 3rd party applications

1.2.5.1 Operating system and accounts

PROPSIM contains an integrated PC module with the following pre-installed software:

Models and version details	
Operating system	Windows™ 10 IoT. Normal installation level.
Adobe Acrobat Reader	Adobe Acrobat Reader version 19.12.20034 or newer.

Windows is a registered trademark of Microsoft Corporation in the United States and other countries.

Adobe Reader is a registered trademark of Adobe Systems Incorporated in the United States and/or other countries.

Table 5. Operating system default user account and password

Windows login user	Password	User rights
PROPSIM	propsim	Administrator rights, automatic log in

1.2.5.2 Security and hardening of operating system

Operating system of the device is applied with the latest applicable Keysight OS hardening standard available for the software and hardware configuration with which the device is shipped. List of settings which can be modified to improve product security:

- Remove automatic logon and/or change password or password expiration time (if changed, provide a password or restore the defaults before shipping the device to service/calibration).
- Preferably use the device in the isolated network (not open to public internet).
- If the device is connected to public internet, operating system automatic updates can be turned on to improve security. In that case Keysight cannot guarantee the operation with the latest updates.

Table 6 lists the TCP network ports that are actively listened from internal and external network. Preventing incoming connections to these ports with Windows Defender firewall can increase security but will have the listed impact on the device's features or functionality.

Table 6. List of listened TCP ports

TCP port	Functionality affected if blocked
80	Instrument web page is not available
23	ATE/SCPI not available at port 23
3334	ATE/SCPI not available at port 3334
5025	ATE/SCPI is not available at port 5025
5123	Some Keysight's additional tools and operation as a part of Keysight test systems will not operate correctly

1.2.5.3 Virus protection of the operating system

By default, the Windows Defender virus protection and firewall are enabled on PROPSIM. Product functionality is verified with these tools active. Product functionality with 3rd party virus protection tools cannot be fully guaranteed. If third party virus protection tools are used and the device is not functioning normally, please contact the product support.

1.2.5.4 User installations of 3rd party applications

Installation of any additional software application or external equipment to the emulator is fully at the user's risk and responsibility.

Keysight cannot guarantee that 3rd party applications, other than mentioned in section 1.2.5.1 on page 23, are fully functioning or that they do not cause any problems or errors to normal performance or use of the emulator.

2 NAVIGATION IN PROPSIM USER INTERFACE

The Graphical User Interface (GUI) of the PROPSIM opens when the emulator has been started up and the user has logged in, see Figure 12.

Note: By default, the user is logged in automatically.

Figure 12 PROPSIM Home view

The GUI is divided into a navigation bar and a view area.

- The navigation bar is organized as a column in the left side of the GUI and provides easy access to all views and applications of the PROPSIM.
- The main views and applications of PROPSIM open in the view area. The content of the view area (including the menus and the tools available in the toolbar) depend on the selected view or application.
- The status of the system, reference clock, lab setup, and auto alignment is shown in the status bar at the bottom of the GUI.

Note: The navigation bar is not always shown in GUI screen shot examples.

The Home view offers an access to pre-installed standard emulations, user defined emulations and Scenario Wizard to create emulations for different test scenarios and technologies.

You can return to the home view from other views (such as Lab Setup) by clicking the (Home) button in the top left corner of the software. The other views stay open in the background, and you can return to them by clicking the view buttons in the bottom right corner.

Note: When an emulation is open in the background, clicking the (Home) button opens the Emulation Control View instead of the Home view.

2.1 Navigation bar

The navigation bar provides access to the following views and applications:

Table 7 Navigation bar items

Menu Item	Submenu Item	Description
Home		Return to the home view from other views (such as Lab Setup). ***Note:*** *When an emulation is open in the background, clicking the (Home) button opens the Emulation Control View instead of the home view.*
Save		Save changes in the Emulation Control View.
Show/Hide Navigation Bar Menu		Show/hide navigation bar menu items.
Emulation	New	Create a new emulation in the Scenario Wizard. See chapter 3.1.
	Open	Open an existing emulation in the Emulation Control View. See chapter 4.
	Save	Save changes in the emulation that is open in the Emulation Control View. See chapter 4.3.
	Close	Close the current emulation.
	Close and edit	Closes the emulation from Emulation Control View and opens it to Scenario Wizard for editing.
Lab Setup	New	Create a new lab setup in the lab setup editor. The lab setup editor is used to configure operation parameters that can be repeated for several emulations. See chapter 6.
	Open	Open an existing lab setup for editing in the lab setup editor.
	Export	Export lab setup from the currently open emulation.
	Save	Save changes in the lab setup that is open in the lab setup editor.
	Deselect	Take the current lab setup out of use.
	Exit	Close the lab setup editor.
Align	New Auto Alignment	Create a new auto alignment.
	Open Auto Alignment	Select an existing auto alignment file for use.
	Deselect Auto Alignment	Take the current auto alignment out of use.
	New User Alignment	Create a new user alignment.
	Open User Alignment	Select an existing user alignment file for use.
	Deselect User Alignment	Take the current user alignment out of use.
Standard Emulations		Access to pre-installed standard emulations.

Menu Item	Submenu Item	Description
Test Scenario Packs		Access to test scenario packs. Test scenario packs contain additional, separately installed, ready-made emulations and corresponding documentation for testing of technology or application specific standardized scenarios.
Channel Modeling Tools		Access to channel modeling tools.
Utilities	Channel Model View	Open the channel modelling tool, see chapter 5.
	Correlation Editor	Open the tool for creating correlation matrices, see chapter 7.2.
	ASO Editor	Open the aerospace channel model editor.
	Runtime Model Editor	Open the runtime model editor.
	Multi Emulator Scaler	Open the multi emulator scaler, see chapter 7.3.
	Batch Builder	Open the emulation batch builder, see chapter 7.4.
	IR and ASC Converter	Converter tool between IR and ASC files.
	Running View (legacy)	Open the running view (legacy), see chapter **Error! Reference source not found.**.
Log		Open the system log. You can also set the log to show a notification callout for warnings and errors, see chapter 2.1.1.
Configuration	Device Information	Includes, for example, serial number and IP address.
	Device Configuration	Includes information about emulator hardware and option configuration
	Import License	Open a dialog for importing and/or updating licenses, see chapter 9.3.
	External RF unit configuration	Open a dialog for selecting configuration for external RF units (up/down -converters). Requires options F8800AUD1/ M1742A (RRH's), F8800ACI1(CIU)/S9165A or F8800AMP1.
	External unit manager	Opens external unit manager for creating configurations for external RF units. Requires options F8800AUD1/ M1742A (RRH's), F8800ACI1(CIU)/S9165A or F8800AMP1.
	System Backup / Restore	Opens the tool for taking backups of, for example, user data, lab setups, auto alignments, licenses, see chapter 7.6.
System	Windows Settings	Configuration of several emulator details, such as locale, time zone and network via Windows Settings dialog.
	Windows Control Panel	Configuration of several emulator details, such as locale, time zone and network via Windows Control Panel.
	Windows File Manager	Can be used for browsing, deleting or moving emulations.
	Windows Command Prompt	Text-based file- and control utility.
	Notepad	For small note keeping.
	Unplug or Eject Device	Utility to safely unplug removable devices, such as USB mass memories.
	Restart / Shutdown	Controlled restart and shutdown of emulator.

Menu Item	Submenu Item	Description
	Exit	Controlled exit of the PROPSIM software.
Help	Quick Guide	Quick Start Guide in PDF format.
	Technical support	Contact Technical support – web page link Keysight Community – web page link My Software Support – Product options and subscription list My Support ID – Product ID information Export System log files - Open a dialog for exporting the system logs to a zipped folder.
	Check for updates	Link to KeysightCare Service and Support web page.
	User reference	User Reference Document in PDF format.
	Application notes	Application Note documents in PDF format.
	About PROPSIM	Firmware release information and options.

2.1.1 Warning and error notifications on navigation bar

You can set a notification callout for warnings and errors that appears on the **Log** menu item of the navigation bar.

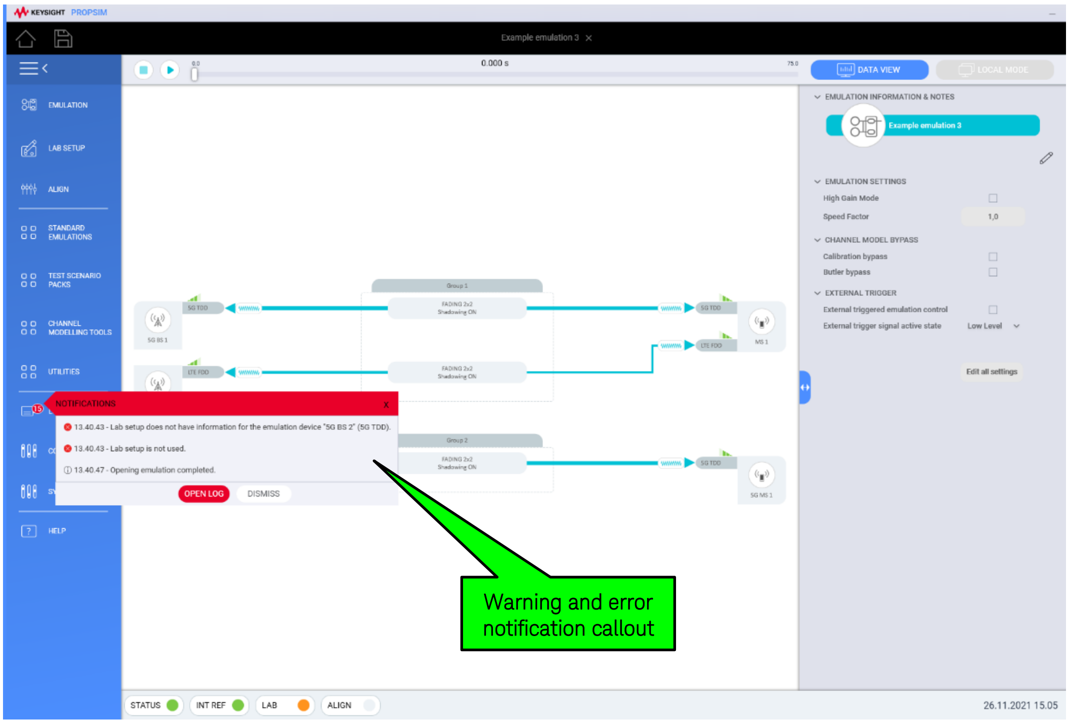


Figure 13 Warning and error notification callout on navigation bar

To set the callout and its duration, select **Configuration > Device Configuration** in the navigation bar. In the **Device configuration** dialog, select the option you want in the **Warning and error callout duration** list:

- “Permanent” – When a new warning or error appears, the notification callout is shown until you manually close it.
- “15 seconds” – When a new warning or error appears, the notification callout is shown for 15 seconds, after which it closes automatically.
- “Off” - When a new warning or error appears, the notification callout is not shown automatically. You can manually open the callout by clicking the small notification icon on the **Log** menu item in the navigation bar.

2.2 Status bar

The status of the system, reference clock, lab setup, and auto alignment is indicated with green, orange, red, and grey colors in the status bar as shown in Table 8.

Table 8 System status items

Category	Item	Color	Description
Status			Main level system status
	STATUS	Green	No errors or warnings
	STATUS	Orange	System errors.
	STATUS	Red	System errors.
Reference Clock			
	INT REF	Green	Internal reference clock selected.
	INT REF	Orange	Internal reference clock warming up.
	EXT REF	Green	External reference clock selected and OK.
	EXT REF	Orange	Locking error to external reference clock. ***Note:*** *Locking might take a few seconds.*
Lab Setup			
	LAB	Grey	No lab setup selected; lab setup is not in use.
	LAB	Green	Lab setup selected and in use.
	LAB	Orange	Lab setup selected but cannot be used with current emulation.
	LAB	Red	Error in selected lab setup file
Auto Alignment			(Optional feature)
	ALIGN	Grey	No user or auto alignment file selected; alignment is not in use.
	ALIGN	Green	Auto/User alignment file selected and in use. Tooltip shows detailed information.
	ALIGN	Orange	Auto/User alignment file selected, old alignment. Tooltip shows detailed information.
	ALIGN	Red	Auto/User alignment has expired or invalid alignment file. Tooltip shows detailed information.

2.2.1 STATUS button

Clicking the STATUS STATUS button opens the System log dialog shown in Figure 14. The system log collects system event and alarms, reference clock changes, etc.

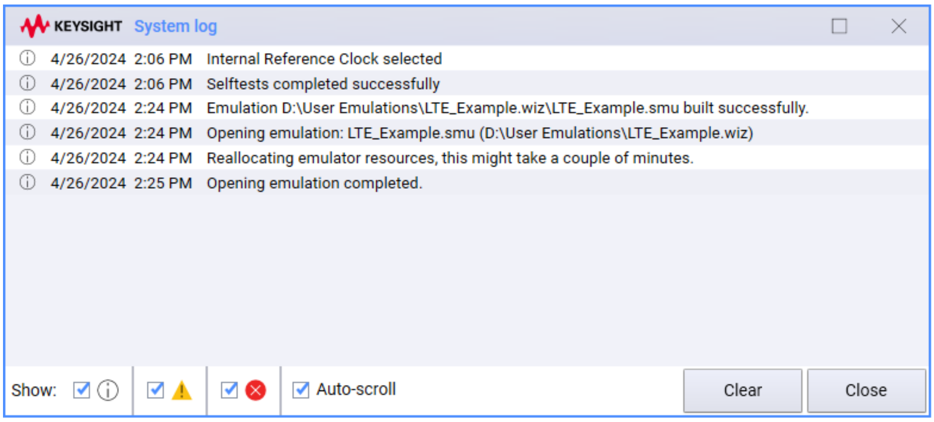


Figure 14 **System log** dialog

2.2.2 INT REF/EXT REF button

You can switch between the internal reference clock and the external reference clock by clicking the INT REF/EXT REF / button. When the external reference clock is selected, the emulator uses external 10 MHz reference clock instead of the internal signal. The external signal is also passed to Ref Out connector of the emulator front panel. This setting is system wide and not bound to current emulation.

Note: When disabling the external 10 MHz reference clock, warming up of the internal reference can take up to 10 minutes.

2.2.3 LAB button

Clicking the LAB button opens the Open Lab Setup dialog where you can select a lab setup to be edited or to be used with emulations. For more information, see chapter 6.

2.2.4 ALIGN button

Clicking the ALIGN button shows two options: “Auto Alignment” and “User alignment”.

Auto Alignment opens a dialog for selecting auto alignment to be used with emulations or create a new auto alignment file. For more information, see chapter 15.

User Alignment opens a dialog for selecting or creating a user alignment. For more information, see chapter 11.

3 SCENARIO WIZARD

With Scenario Wizard, you can create new emulations and edit existing emulations for different test scenarios and technologies.

- To create a new emulation with Scenario Wizard, select the New emulation button in the Home view. The Scenario Wizard is described in chapter 3.1.
- To open an existing emulation for editing or running, select the Edit or Run emulation button in the Home view. Opening an emulation for editing is described in chapter 3.2. Running an emulation in the Emulation Control View is described in chapter 4.

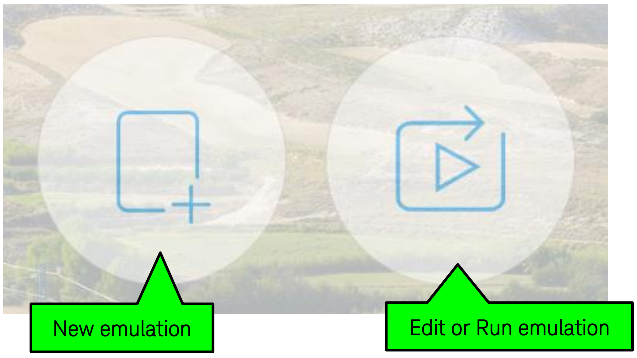


Figure 15 Home view buttons

3.1 Create emulation with Scenario Wizard

Selecting New emulation button in the Home view launches the Scenario Wizard (see Figure 16).

With Scenario Wizard, you can create new emulations for different test scenarios and technologies. The emulation is created in steps from basic settings to finalizing the test setup. There is a separate page for each step in the creation process.

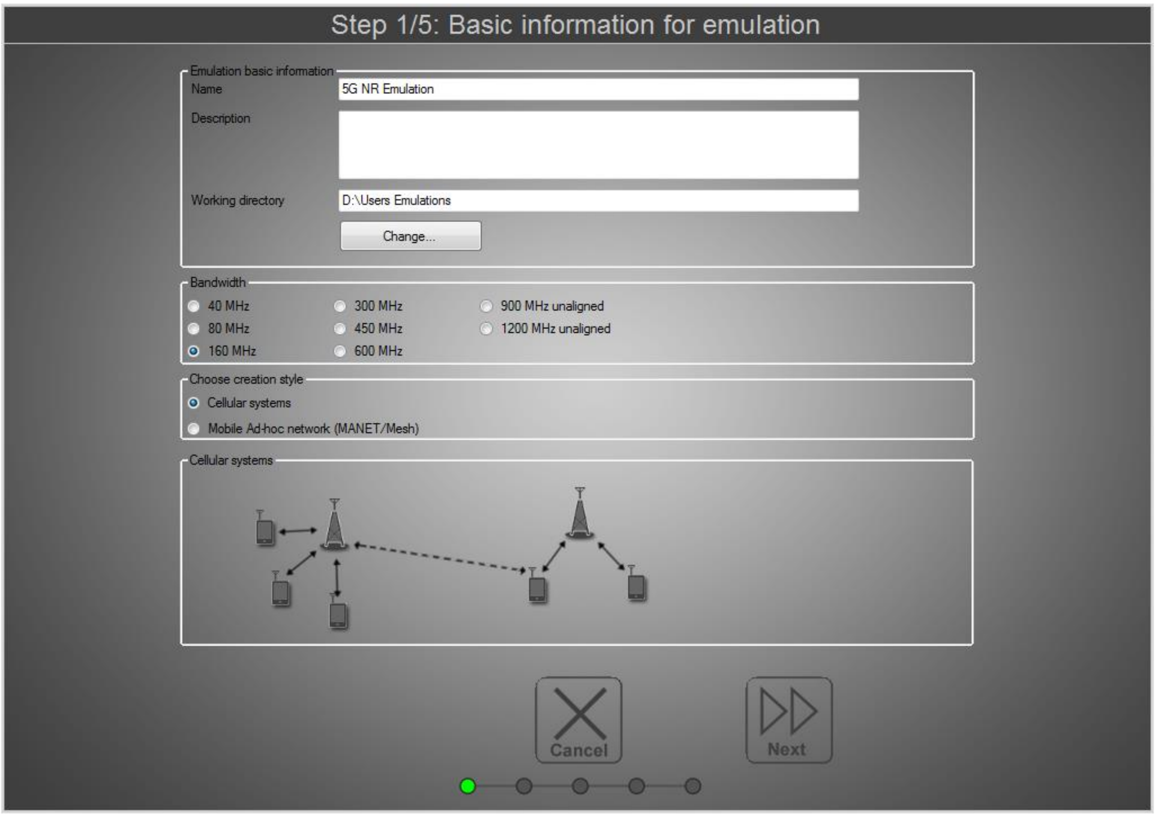


Figure 16 Scenario Wizard - First step of creating new emulation.

3.1.1 Scenario Wizard navigation

Scenario wizard contains the following navigation buttons (presented in Figure 17) for navigation between wizard steps:

- Back for returning to previous step.
- Cancel for cancelling emulation creation and discarding all changes.
- Next for accepting changes and proceeding to next step.

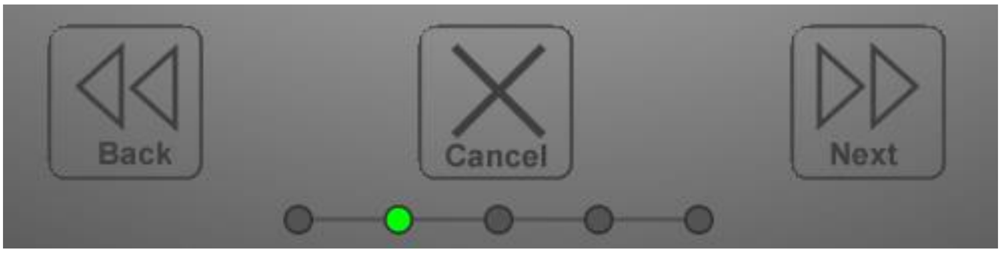


Figure 17 Scenario Wizard navigation

The navigation buttons are located at the bottom of wizard page. The current step is indicated with green color under the navigation buttons.

If an error has occurred on the page and proceeding to next step is not possible, the Next button is shown in red color. In that case, the invalid fields in the current step are also marked in red color and the corresponding error information is shown.

3.1.2 Step 1: Basic information for emulation

The following basic settings are defined on the first page of the wizard.

Emulation basic information (presented in Figure 18):

- Name: Emulation name.

  If you have opened an existing emulation, defining a new name makes an identical copy of the existing emulation leaving the original emulation untouched.
- Description: Emulation description
- Working directory: Emulation working directory. Scenario wizard creates emulations to user emulations folder. It is possible to browse the default working folder or make new working folders in it by clicking Change... button or typing folder name to empty working directory field.

Figure 18 Emulation basic information

3.1.2.1 Bandwidth:

In PROPSIM, emulation bandwidth is selected (see Figure 19).

Figure 19 Bandwidth selection in PROPSIM (F8800A and F8820A)

Available bandwidth options depend on the PROPSIM device type and bandwidth configuration.

Note: This bandwidth selection defines the bandwidth for all the devices. If needed, you can define device-specific bandwidths later in the Scenario Wizard, see section 3.1.3.1.1.

3.1.2.2 Creation style:

Creation style is selected from two options (see Figure 20): **Cellular systems** to enable various handover and user-defined emulation creation, or **Mobile Ad-hoc network** to enable MANET/Mesh emulation creation.

Figure 20 Creation style selection

3.1.2.3 Cellular systems

Cellular systems creation style is used for creating emulations based on pre-defined scenario templates. These templates contain configurations from a simple single link setup to various handover, multi-user and multi-RAT setups. Settings for the emulation are done on next page (see chapter 3.1.3.1).

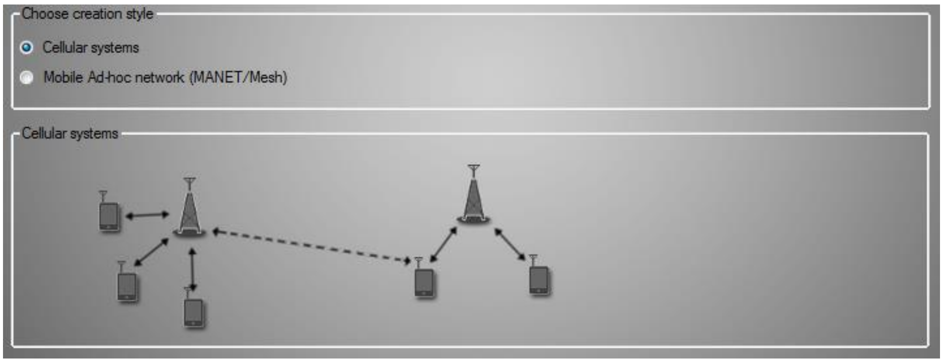


Figure 21 Cellular systems configuration

3.1.2.4 Mobile Ad-hoc network (MANET/Mesh)

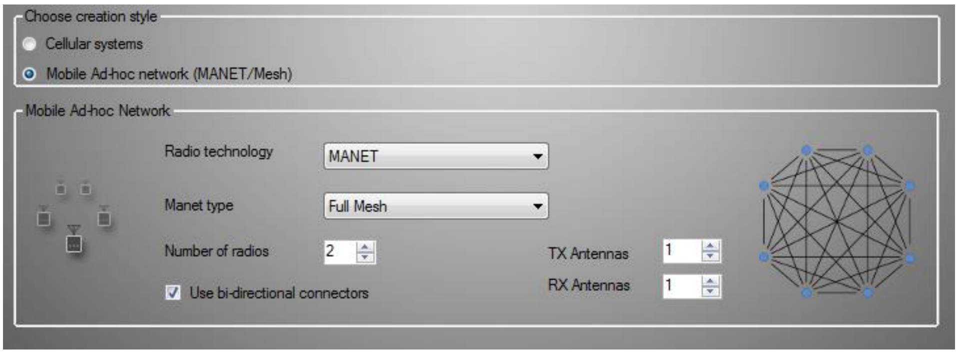


Figure 22 Mobile Ad-hoc Network options

Selecting **Mobile Ad-hoc network** creation style enables following options for creating MANET/Mesh emulation (see Figure 20):

Radio technology:

- Defines common environment variables for MANET/Mesh emulation

Manet type:

- Defines radio connection topology: full mesh, chain, loop, pyramid, diamond or star

Number of TX/RX antennas:

- Number of RX/TX antennas in each radio.

Number of radios:

- Number of radios in MANET/Mesh network. Maximum number of radios depends on hardware configuration.

Use bi-directional connectors:

- Selection to use uni- or bi-directional connectors

Settings for the emulation are done on next page (see chapter 3.1.3.2).

3.1.3 Step 2: Channel models, shadowing and interferences

3.1.3.1 Cellular systems creation style

When **Cellular Systems** creation style is selected, page in step 2 appears as shown in Figure 23.

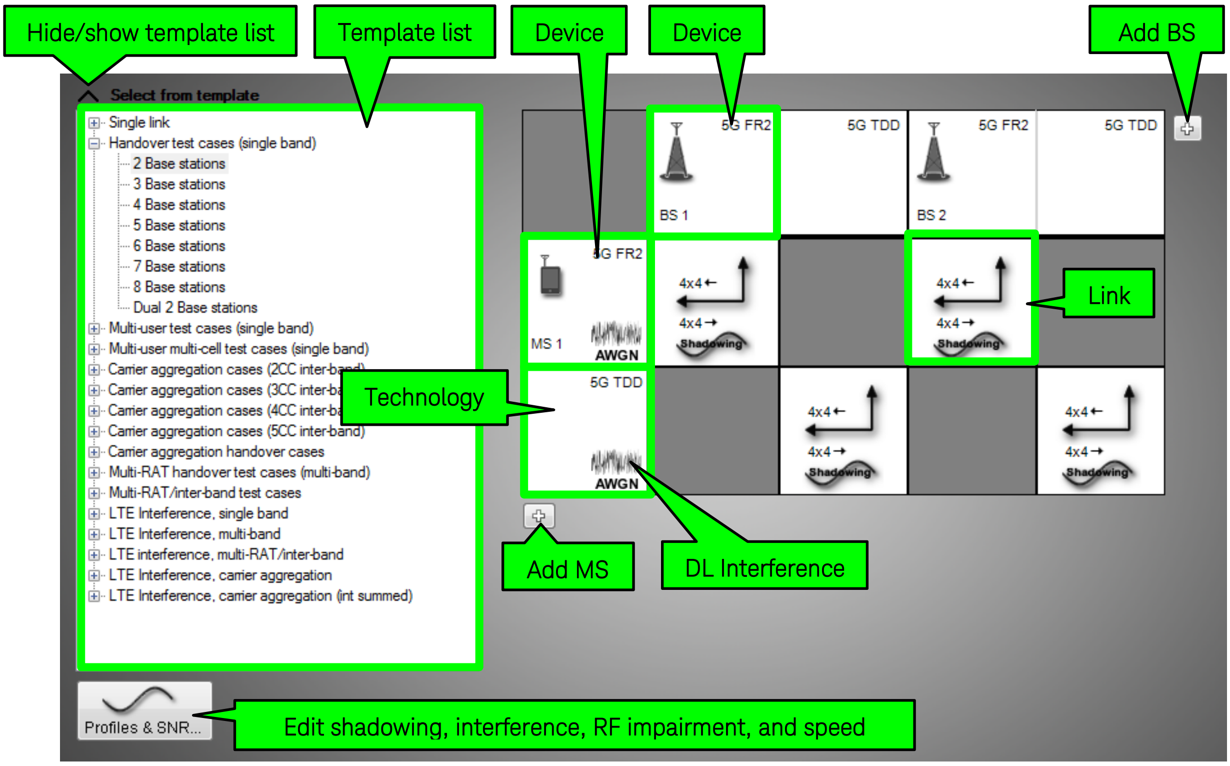


Figure 23 Device and link setup page

Template list

Template list on the left side of the figure contains available emulation templates grouped under specific categories:

- Single link (one mobile station and one base station)
- Handover test cases (one mobile station and two or more base stations)
- Multi-user test cases (one base station and two or more mobile stations)
- Multi-user Multi-cell test cases (several base stations and several mobiles)
- Carrier aggregation test cases
- Carrier aggregation handover
- Multi-RAT handover and multi-user test cases, where different technologies or frequency bands are combined to mobile station antennas.

If suitable template is not found from the list, then any template can be easily modified. All the templates have plus buttons for adding BS and MS devices (or interference source) for the emulation. For each device multiple radio technologies can be added by choosing Add technology from the device's right-click menu. This adds an empty row for MS and an empty column for BS (and interference) where the link needs to be added. Only one link between each MS/technology and each BS/technology can be added. Any existing device, technology and interference can be deleted from its right-click menu. All the changes only affect the emulation being created; the original template is never modified.

Devices with multiple radio technologies are automatically selected to active connectors supporting the device's technology count.

For instructions on creating Aerospace (ASO) emulations, see chapter 19.

Device and link information

Each template can contain several devices, technologies, links, and interferences as shown in Figure 23. To change properties of these, click the corresponding element in the page.

Right clicking any device or link shows options to edit the interferences, technologies or shadowing profiles related to the selected element (see Figure 24).

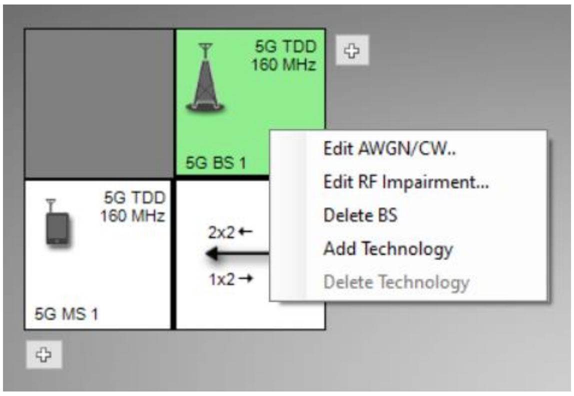
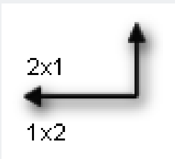
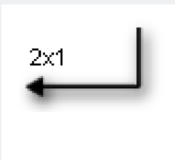
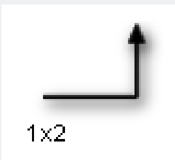
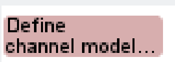


Figure 24 Additional options related to device/link with mouse right click

Symbols used in Link are explained in Table 9.

Table 9 Link items

Item	Description
	Bidirectional link. The number of downlink antennas is shown above the arrow, and the number of uplink antennas is shown below the arrow. Format: [number of TX antennas] x [number of RX antennas].
	Downlink. Here, the base station has two TX antennas and mobile station has one RX antenna.
	Uplink. Here, the mobile station has one TX antenna and base station has two RX antennas.
	Downlink and/or uplink channel model missing.
	Downlink and/or uplink has shadowing enabled.

3.1.3.1.1 Device Properties (MS and BS)

When device is clicked, Device properties dialog is opened as in Figure 25.

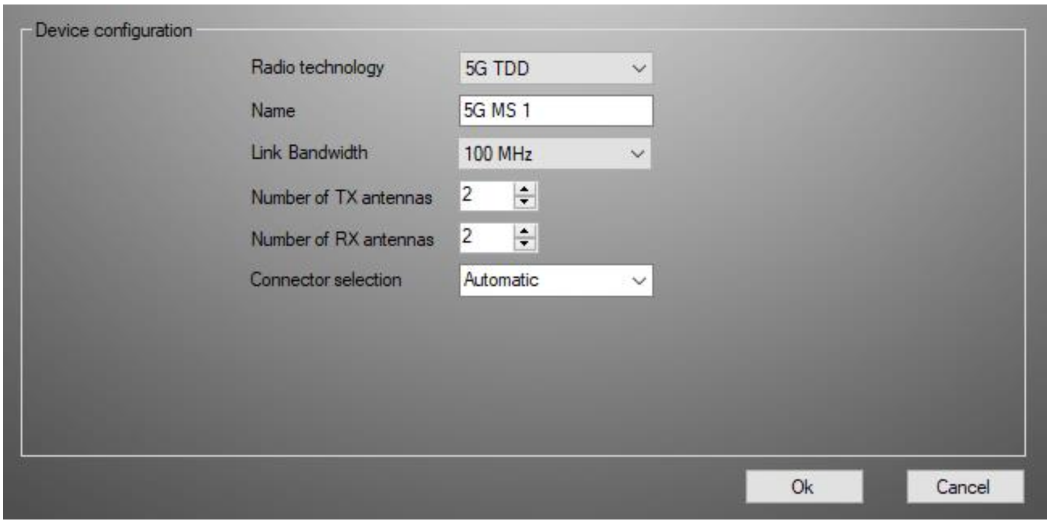


Figure 25 Device properties dialog

Radio technology

- This is the radio technology of the link to which the device connects to. When technology is changed, it is changed also to the device on the opposite end of the link.

Name

- Name of the device.

Link bandwidth

- Link bandwidth of the device. Changing the value here changes only the link bandwidth of the selected device.

Number of TX and RX antennas

- Number of device TX and RX antennas. If set to zero, corresponding link is removed. However, it is not possible to remove both links.

Connector selection

- Defines the connector selection for the ports for the device: Automatic, Duplex or Non-Duplex

Cell information for RSRP measurement and synchronization (LTE FDD, TD-LTE)

- Bandwidth and Physical Cell ID

3.1.3.1.2 Link properties

When link is clicked, Link properties dialog shown in Figure 26 opens.

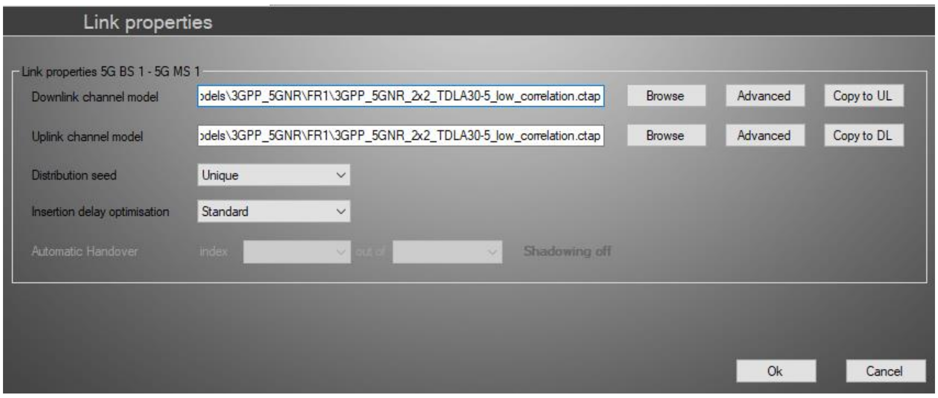


Figure 26 Link properties dialog

Link properties dialog is used to set channel models, antenna names (**Advanced**) and distribution seed for the generation of statistical channel models and automatic handover parameters (when shadowing is defined).

Downlink channel model, Uplink channel model

- Select pre-defined channel model for the downlink and uplink (**Browse** -button). Channel model setup is described in chapter 3.1.3.1.2.1.
- For the advanced channel model setup ("Advanced" -button), see chapter 3.1.3.1.2.2.
- For instructions on using Aerospace channel models, see chapter 19.
- **Copy to UL** and **Copy to DL** -buttons can be used to copy the selected model to another direction of the link.

Distribution seed (available when using .tap file as channel model)

- Defines the distribution seed for the random process of statistical channel model generation. Downlink and uplink always use the same distribution seed option. Options are:
  - "From File", distribution seed from inside the .tap -file is used
  - "Unique", each link in the emulation has unique distribution seed value.

    Note: "Unique" takes into account the "User defined" seeds from the other links of the emulation, but it does not take into account the links with "From File" setting.
  - "User defined", user defined distribution seed

Insertion delay optimisation

- Defines the insertion delay optimization setting for the link.
  - "Standard", all normal features are available, no specific delay optimization
  - "Short", optimized insertion delay is used with reduced fading resources and features. When this option is selected, internal signal routing is not available between channel units (connector rows). Make sure that all the connectors of the short link are placed in the same connector row. The feature requires an option F8800B-SD1/F8820B-SD1, device type F8800B or F88020B with a specific hardware revision.

3.1.3.1.2.1 Channel model setup

Clicking **Browse...** button under the channel model field opens a **Channel model setup** dialog (presented in Figure 27).

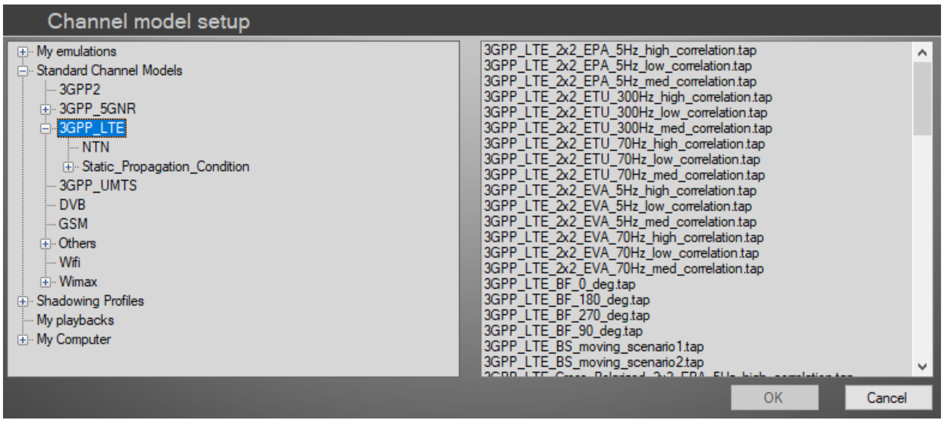


Figure 27 Channel model setup dialog

In **Channel model setup** dialog, folders are listed on the left and channel model files on the right.

File is selected by double clicking file in file list or clicking **OK** button. Channel model is set for all channels in the selected link.

Note: *Scenario Wizard supports following channel model files:*

- .tap
- .ctap
- .ir
- .cir
- .rtc
- .asc
- .mat
- .aso
- .caso

3.1.3.1.2.2 Advanced channel models setup

When using cellular systems creation style, advanced channel models setup page appears as shown in Figure 28. It can be used to select channel models individually for each MIMO channel or to enable concurrent channel models feature. If the model already contains a correlative tap (which internally defines all the MIMO channels), individual channels cannot be edited, as shown in Figure 28. To edit individual channels in this case, return to previous wizard page, remove the correlative channel model and enter the advanced channel model selections again.

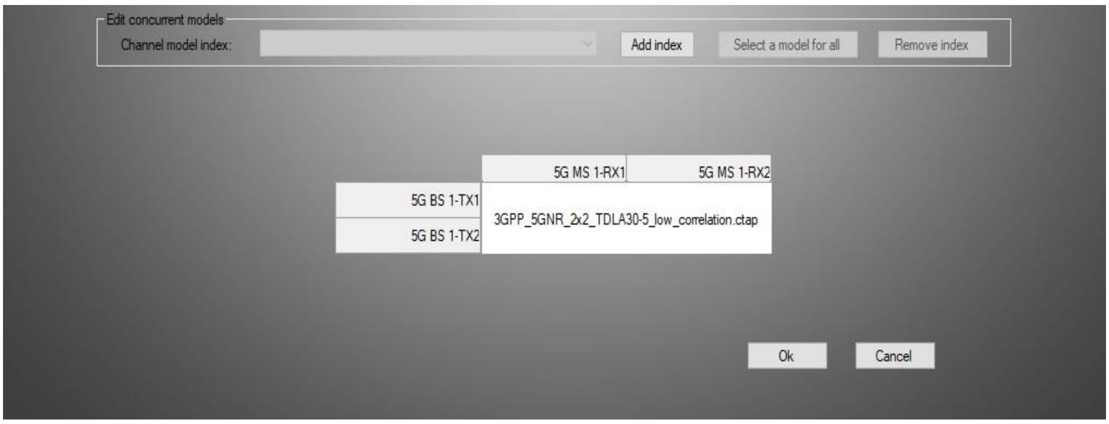


Figure 28 Channel models setup page in cellular systems creation style

Edit concurrent models

Concurrent channel models -feature allows adding multiple channel models into individual emulation links. All the selected channel models are bundled and built into the emulation. Selecting which of the models is active can be done while the emulation is loaded and in stop state. Channel model gains of the concurrent models are scaled to match the lowest gain of selected models.

Concurrent models have the following buttons / controls:

- Channel model index
    - Shows the selected concurrent channel model index and the name of the channel model in it. Dropdown list is used to select the index to show or edit.
- Add index
    - Pressing "Add index" creates a new concurrent index where the channel model can be then selected.

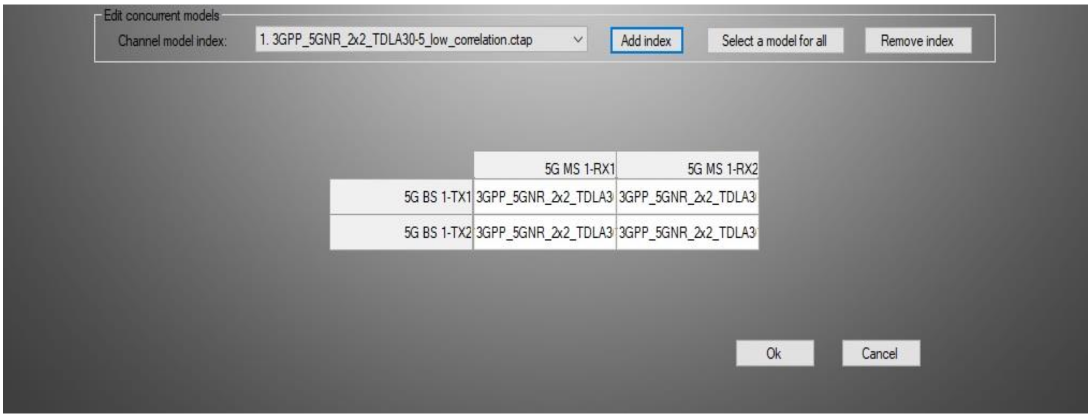


Figure 29 Creating a concurrent channel model by pressing the index button

- Select a model for all
    - Pressing "Select a model for all" opens a channel model browsing window as shown in chapter 3.1.3.1.2.1. After the selection is done, all the MIMO channels of the link get the same channel model. Concurrent channel model feature supports following channel model file types: .tap, .ctap, .ir, .cir and .asc.
    - Note: The same link cannot contain both correlated and non-correlated models simultaneously.
- Remove index
    - Press "Remove index" to delete the currently selected index.

Channel model fields

Own field for each channel. A channel model can be redefined by selecting wanted channel field between TX and RX antennas and pressing "..." button. It opens channel model setup dialog like presented in Figure 27. Selected channel model file name appears to channel model field.

Figure 30 illustrates a concurrent channel modeling example where multiple 3GPP 5GNR standard channel models are selected into a single link.

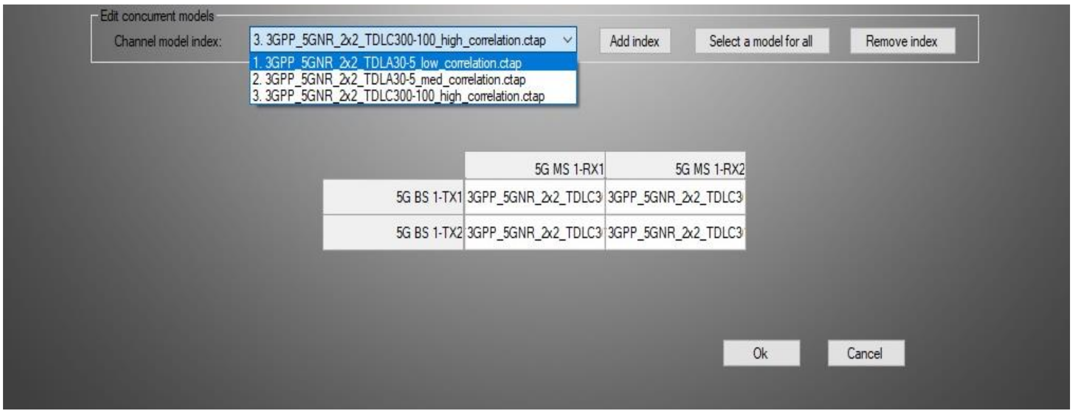


Figure 30 Concurrent channel models example

3.1.3.1.3 Adding shadowing profiles (optional)

When 

button is clicked, dialog opens for editing shadowing profiles, speed profiles, interferences and RF impairments. Dialog is shown in Figure 31.

Note: Scenarios using internal summing require channel specific shadowing license. Multi-RAT/Carrier aggregation scenarios with external summing require RF output shadowing license.

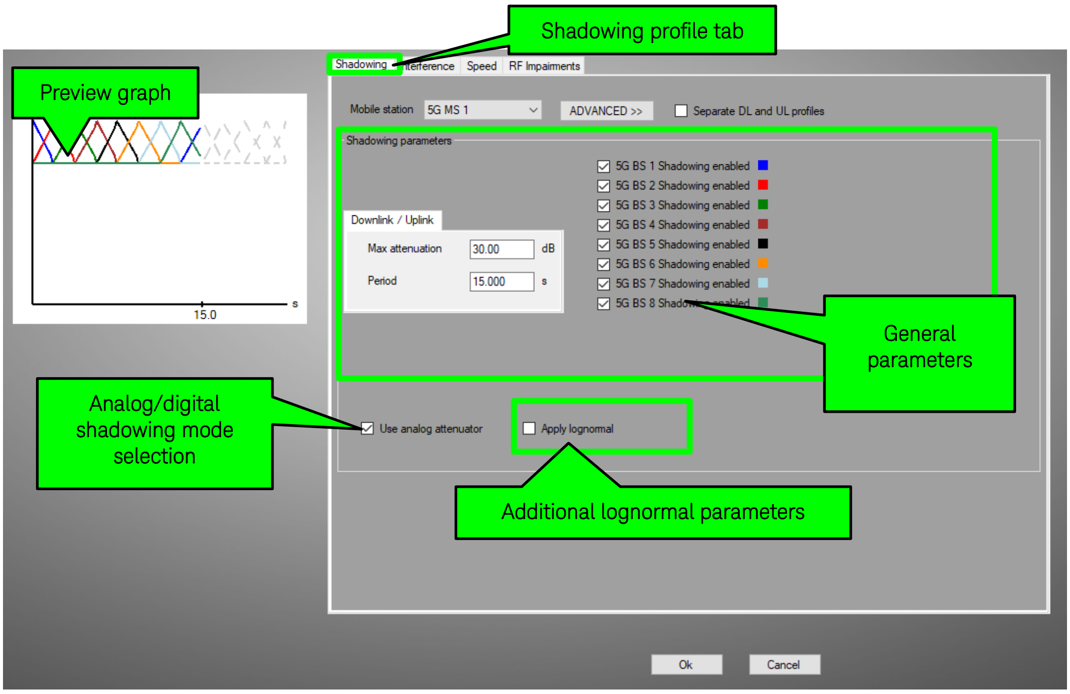


Figure 31 Edit link shadowing profiles

Edit link profiles and SNR dialog has separate tabs for shadowing, interference and RF Impairments. Select **Shadowing** tab to modify the shadowing profiles and parameters.

By default, a handover-style shadowing profile is created, where each base station in turn has zero attenuation while the others are attenuated more (default 30 dB attenuation). Attenuation transition is linear. Handover scenario with three base stations is illustrated in Figure 32.

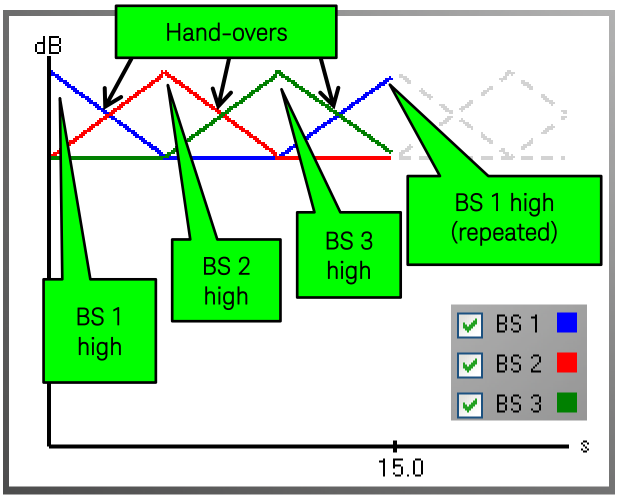


Figure 32 Default handover scenario with 3 base stations

General shadowing parameters and how they affect are illustrated in Figure 33.

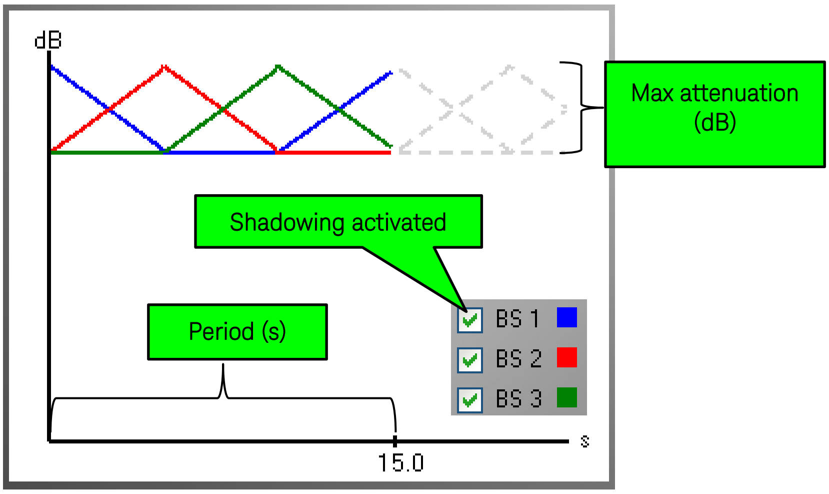


Figure 33 Shadowing parameters

When Apply lognormal is selected, all the shadowing curves have an additional lognormal variation according to specified standard deviation and correlation length.

Note: preview picture shows a coarse illustration of lognormal effect, but the details are not identical to final compiled curve.

Use Analog Attenuator enables the usage of RF output attenuator for the use of shadowing profiles. Attenuator is used to optimize signal dynamics during the shadowing playback - only the gain difference between the combined channels is done by scaling digitally. Analog attenuator is used to create the common part of the attenuation.

Note: AWGN and other interference generators should be used only when "Use Analog Attenuator" -setting is OFF. When "Use Analog Attenuator" setting is ON, behavior of interference level or SNR is undefined and depends on the selected shadowing profiles.

Pressing ADVANCED >> opens a table view of shadowing time points and attenuations as shown in Figure 34. Editing the values in the table allows creation of arbitrary curves for various testing purposes.

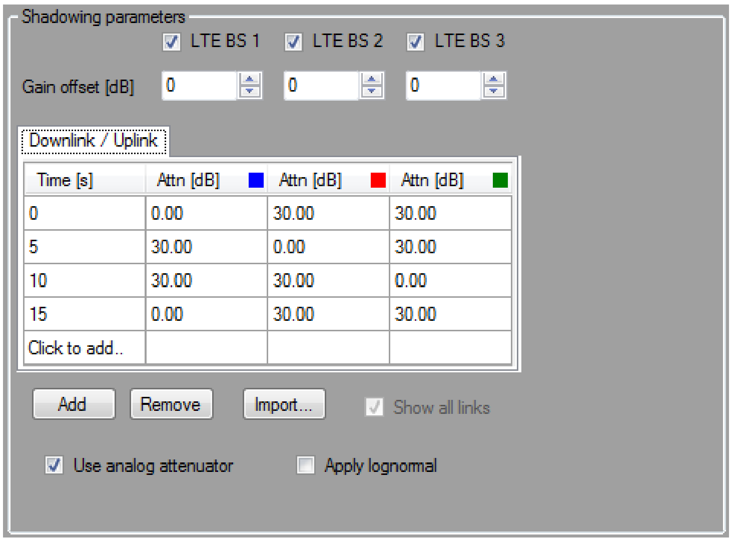


Figure 34	Advanced shadowing edit

Shadowing offset fields can be used to adjust curves to both positive and negative direction without altering the values in the table. Data will automatically saturate to highest and lowest possible levels. See chapter 12.3.2 for more details about shadowing offset.

Pressing 

in advanced view opens a dialog for browsing existing shadowing file (.shd) which will be imported. If a sequentially numbered series of shadowing files is available, for example from previously created emulation, user has a choice to import them all at once. Single shadowing file is imported to currently selected column. Also copy-pasting the whole table from Excel-tool (or tab-separated text file) is supported as shown in Figure 35. Right clicking the table opens a context menu for copy-pasting individual columns.

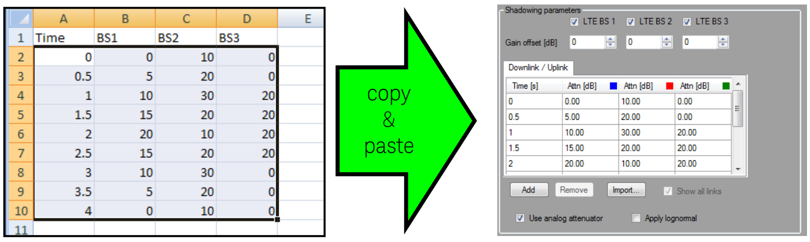


Figure 35	Copy-paste from Excel-tool

### 3.1.3.1.4	Adding interferences and interference power profiles (optional)

Right clicking any device and selecting **Edit AWGN/CW** (see Figure 24) or pressing 

button and selecting **Interference** tab shows the dialog for configuring interferences to the selected device. Dialog is shown in Figure 36.

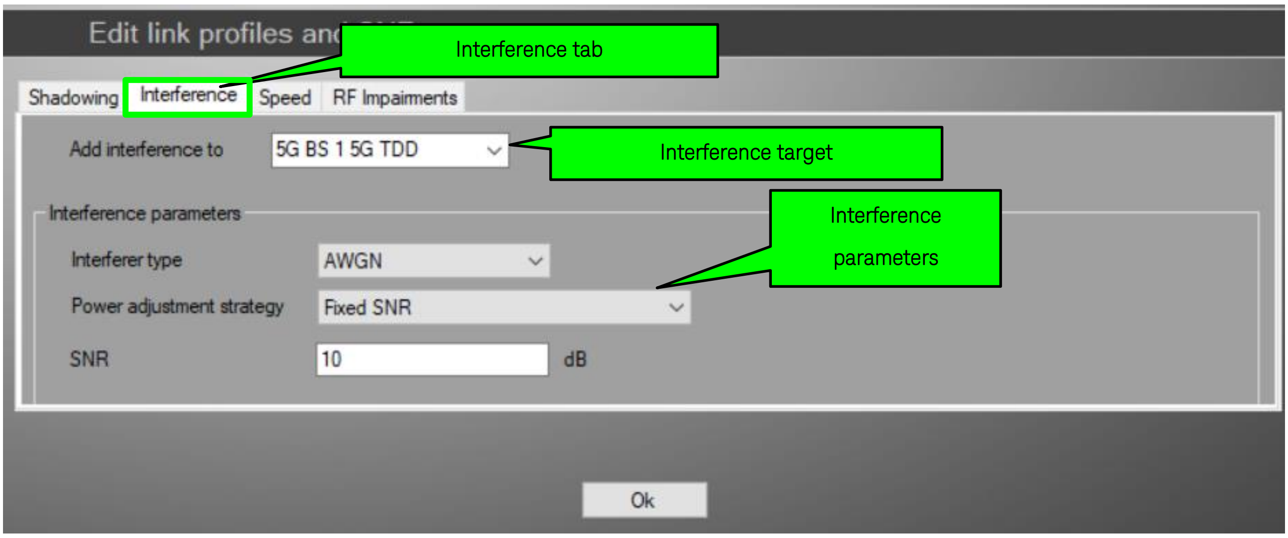


Figure 36 Editing interference parameters

Interference parameters are identical to Emulation Control View. More details can be found from Chapter 13 Internal Interference Generator (Optional Feature).

Scenario Wizard can be used to define a time variant power profile for the selected interferences. Select Dynamic Power Profile under Power Adjustment Strategy as shown in Figure 37.

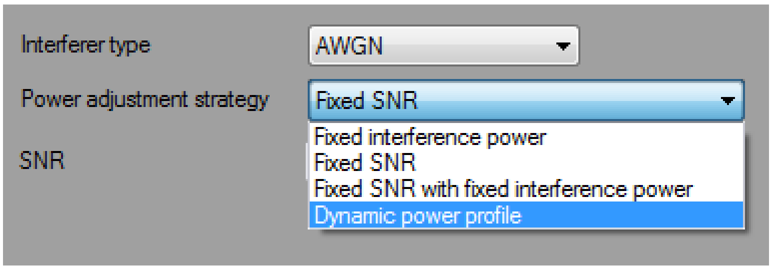


Figure 37 Interference dynamic power profile

When Dynamic power profile is used, total interference power consists of two parts combined together: “fixed interference power” part, defined in the interference settings and a “time variant attenuation” part, defined by the interference profile. “Fixed power” part can be adjusted also later in Emulation Control View when running the emulation.

Interference profile can be defined the same way as the shadowing profile, as discussed in the previous chapter 3.1.3.1.3. Figure 38 shows an example of user defined interference profile. Copy-paste from Excel-tool can be used for more convenient data input. Existing shadowing profile files (.shd) can also be used as interference profile data with “Import” button.

With scenarios using external summing, i.e. multi-RAT handover scenarios, interference profiles from different radio technologies can be edited individually when Same power profile from all transmitter devices is un-checked.

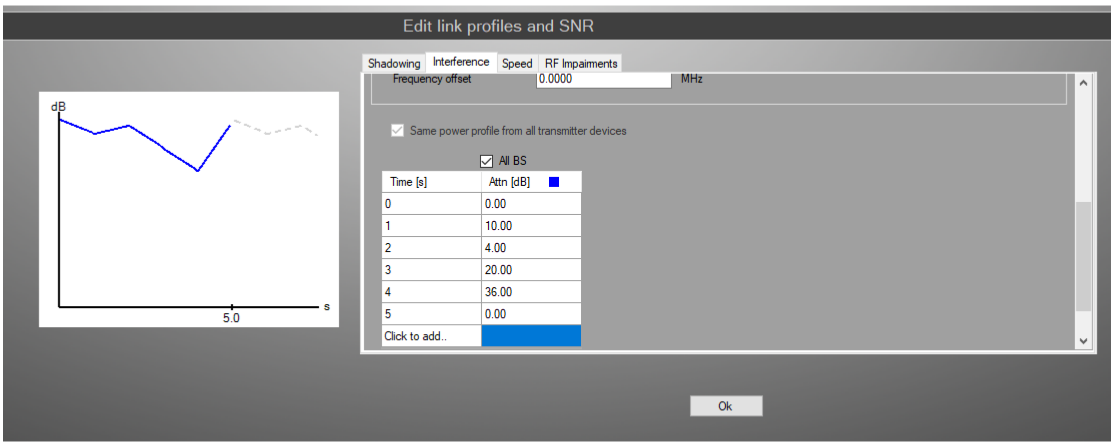


Figure 38 Editing interference profile

3.1.3.2 Mobile Ad-hoc Network creation style

Channel models setup page shown in Figure 39 is used to select channel models and shadowing profiles for Mobile Ad-hoc network setup.

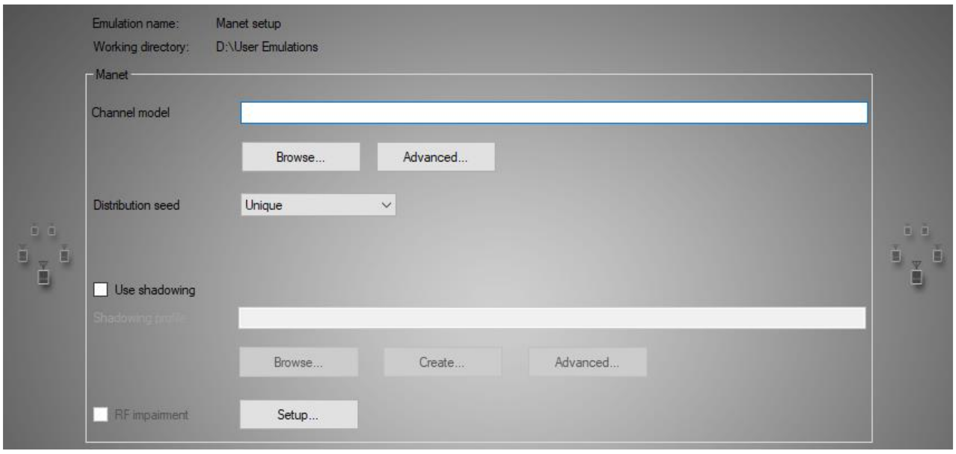


Figure 39 Channel model setup page for MANET configuration

Page contains following information:

Channel model

Channel models for links can be set in two ways:

- By browsing pre-defined channel model for all channels in the link (**Browse...** button).
    - Channel model setup is described in chapter 3.1.3.1.2.1.
- By browsing pre-defined channel model for each channel between radios separately (**Advanced...** button). In advanced channel model setup, it is also possible to give names for the radios and their RX and TX connectors.
    - Advanced channel model setup is described in chapter 3.1.3.2.1.

Distribution seed

Distribution seed can be defined, when the selected channel model is .tap. More information about distribution seed options can be found from the chapter 3.1.3.1.2.

Shadowing profile (optional)

Shadowing for the link is enabled by checking **Use shadowing** option under the channel model selection (see Figure 40).

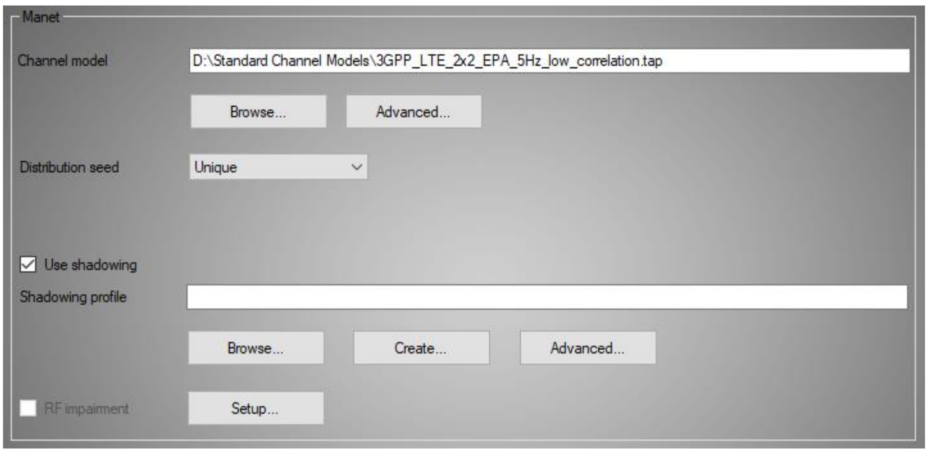


Figure 40 Shadowing enabled

Shadowing profile for link can be added in two ways:

- By browsing pre-defined shadowing profile for the link (**Browse...** button).
  - Shadowing profile setup is described in chapter 3.1.3.2.2.
- By creating new shadowing profile for the link (**Create...** button).
  - Shadowing profile creation is described in chapter 12.4.

RF impairment (optional)

RF impairments (phase noise) can be added by clicking “Setup...”. The dialog for adding RF impairments is similar as shown in the Figure 219. Checkbox indicates is any RF impairments have been activated.

3.1.3.2.1 Advanced channel model setup

Clicking **Advanced...** button in channel model setup page opens the **Advanced channel model setup** dialog (see Figure 41).

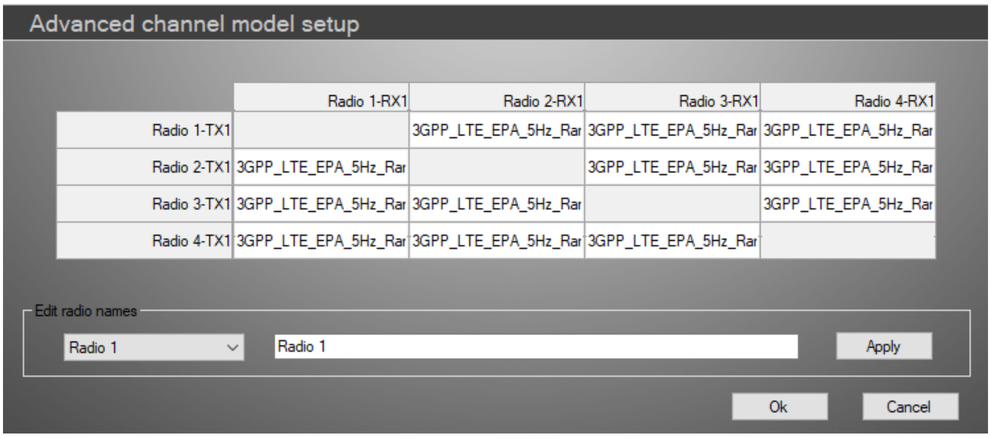


Figure 41 Advanced channel model setup

In **Advanced channel model setup,** it is possible to set pre-defined channel model for each link between radios (or device antennas) and rename TX and RX antennas.

Dialog contains following functions:

Antenna names

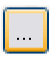
- Antennas default names are based on TX and RX device names.

Radio names

- Radio names can be changed to help identifying the devices in running graph.

Channel model fields

- Own field for each channel
- Selected channel field is indicated with browse button and TX and RX antenna names of channel are marked in bold.



Browse

- Opens channel model setup dialog.

Renaming antennas:

Renaming antennas can be done by clicking wanted antenna field and typing a new name for antenna.

Setting channel models:

Channel model is set by selecting wanted channel field between TX and RX antennas and clicking browse button. It opens channel model setup dialog like presented in Figure 27. Selected channel model file name appears to channel model field.

Channel model path information can be checked from the tooltip (see Figure 42).

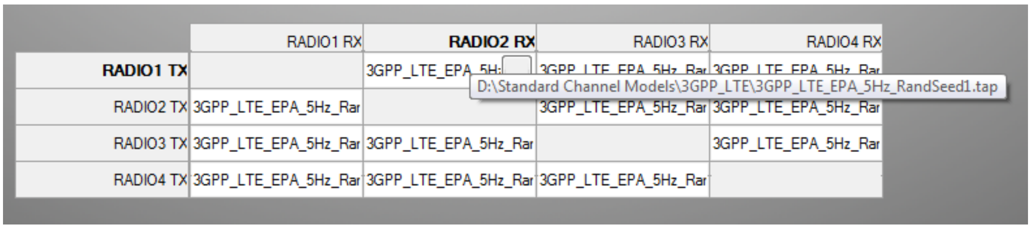


Figure 42 Selected channel model path information in tooltip

Changes are accepted by clicking **OK** button.

3.1.3.2.2 Select shadowing profile (optional)

Clicking **Browse...** under Shadowing profile field opens a **Shadowing profile setup** dialog (see Figure 43).

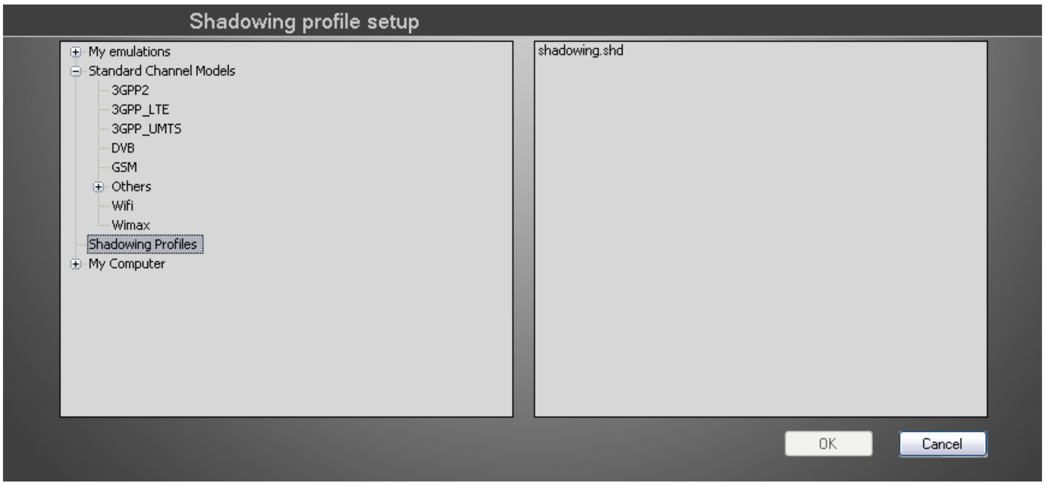


Figure 43 Shadowing profile selection

Selected shadowing profile is added to **Shadowing profile** field (see Figure 44).

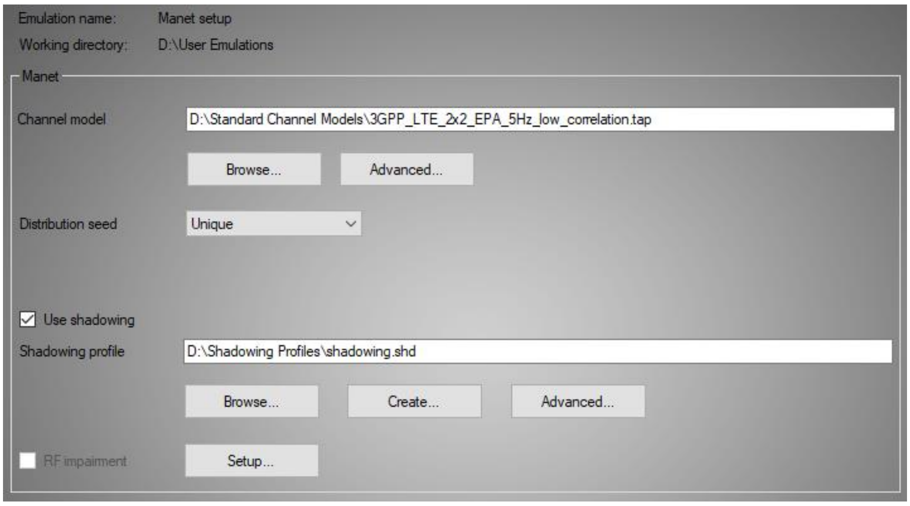


Figure 44 Selected shadowing profile

To create a new shadowing profile, under the **Shadowing profile field,** click **Create...** . For more information on the shadowing profile types, see section 12.4.

To select channel specific shadowing profiles for each link between the radios, clicking **Advanced.**. under the **Shadowing profile field**..

3.1.4 Step 3: Environment variables

Environment variables page presents links with their predefined environment variables. Predefined values are based on the technology selected in the first page (see chapter 3.1.2). Variables of the selected link are also illustrated in example picture as presented in Figure 45.

Explanation and possible limit values for selected variable are shown above the variable table.

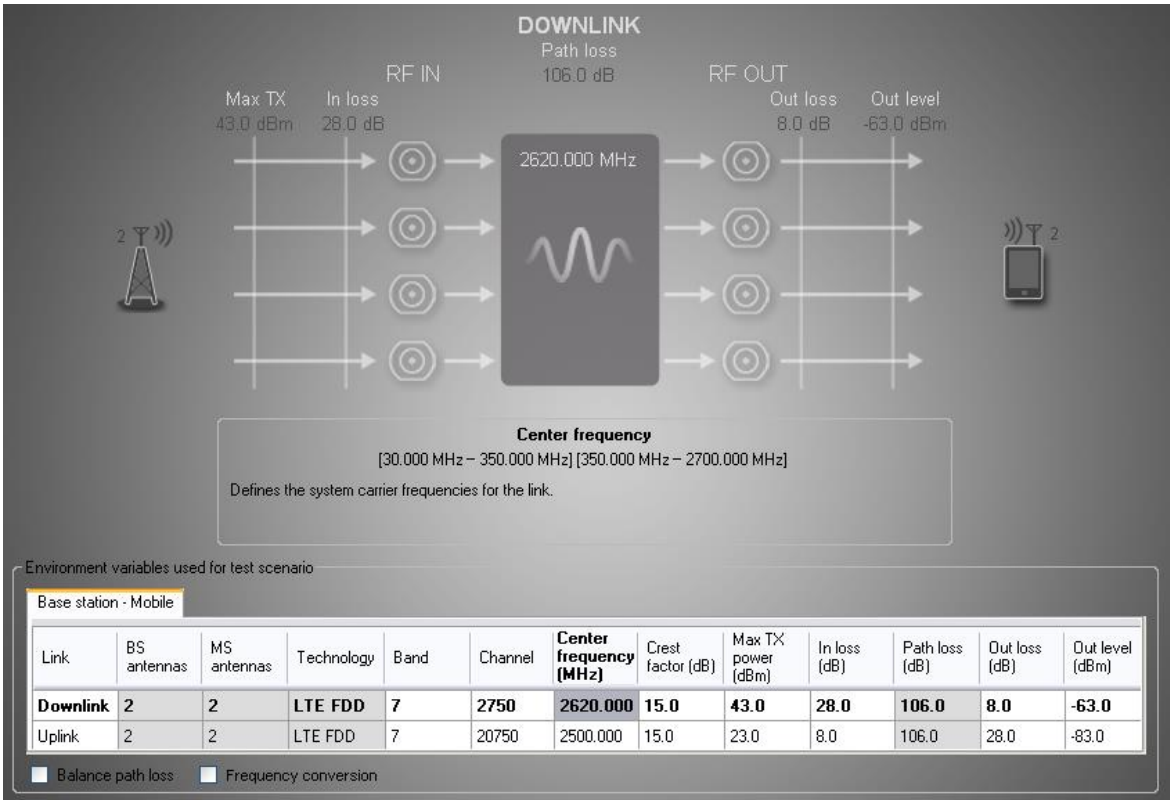


Figure 45 Emulation environment variables for bi-directional FDD 2x2 MIMO emulation.

Environment variables used for emulation:

Link

- Defines the name or identification of the link.

BS Antennas (read only)

- Displays the number of base station antennas defined on first wizard page.

MS Antennas (read only)

- Displays the number of mobile station antennas defined on first wizard page.

Technology (read only)

- Defines the name of the selected technology.

Band / Channel

- Defines the system carrier frequency as a band and channel number. Available for LTE and WCDMA technologies.

Center Frequency

- Defines the system carrier frequencies for the link.

Crest Factor

- Defines the crest factor (peak/average transmit power) of used modulation.

Max TX Power

- Defines the maximum RMS transmit power of the base station or mobile without cables or external losses.

In loss

- Defines the loss of cables and external components connected between device (BS or MS) and PROPSIM.

Path loss

- Defines the total loss between devices.

Out loss

- Defines the loss of cables and external components connected between PROPSIM and device (BS or MS).

Out level

- Defines the signal level seen by the device (BS or MS), after fading, cable losses and external components.
- This level is achieved when transmitter sends with its maximum TX power.

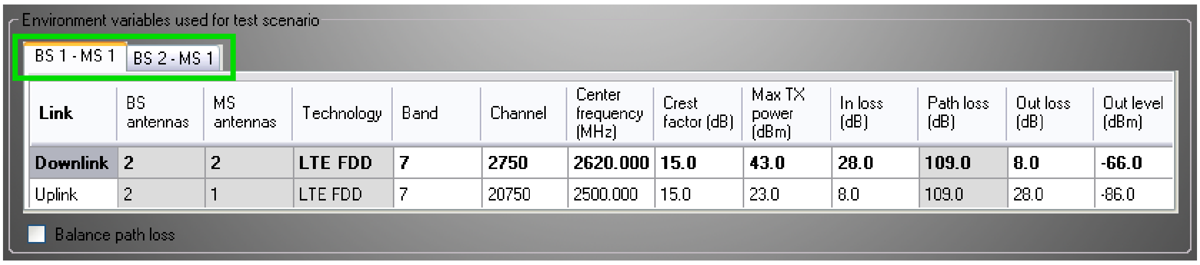
Balance Pathloss

- When selected, output levels (Out level) are adjusted automatically to achieve identical path loss for uplink and downlink.
- Setting must be defined for each link separately.

Frequency conversion

- This check box is shown if frequency conversion is enabled in the **Device Configuration** dialog (**Configuration > Device configuration** in the Navigation bar).
- If you select the **Frequency conversion** check box, you can define the **Input frequency** and **Output frequency** separately for uplink and downlink.
- When frequency conversion is used, **Virtual frequency** of the link is used to calculate the relation between Doppler and mobile speed.

When creating emulation with **Cellular systems** creation style (see chapter 3.1.3.1), environment variables of the links are on separate sheets as shown in Figure 46 and can therefore be configured independently. Parameter setup is identical to single link configuration.



Link	BS antennas	MS antennas	Technology	Band	Channel	Center frequency (MHz)	Crest factor (dB)	Max TX power (dBm)	In loss (dB)	Path loss (dB)	Out loss (dB)	Out level (dBm)
Downlink	**2**	**2**	**LTE FDD**	**7**	**2750**	**2620.000**	**15.0**	**43.0**	**28.0**	**109.0**	**8.0**	**-66.0**
Uplink	2	1	LTE FDD	7	20750	2500.000	15.0	23.0	8.0	109.0	28.0	-86.0

Figure 46 Emulation environment variables for two links.

For MANET/Mesh emulations, environment variables are configured automatically. If needed, settings can be modified later in Emulation Control View, see section 4.3.

3.1.5 Step 4: Active connectors selection

Scenario Wizard automatically allocates physical RF connectors that are used when running the emulation. Allocation is based on selections in previous steps. In this page it is possible to verify these default connector settings or modify connector locations to match test setup cable connections.

Figure 47 illustrates the layout of the active connector selection page:

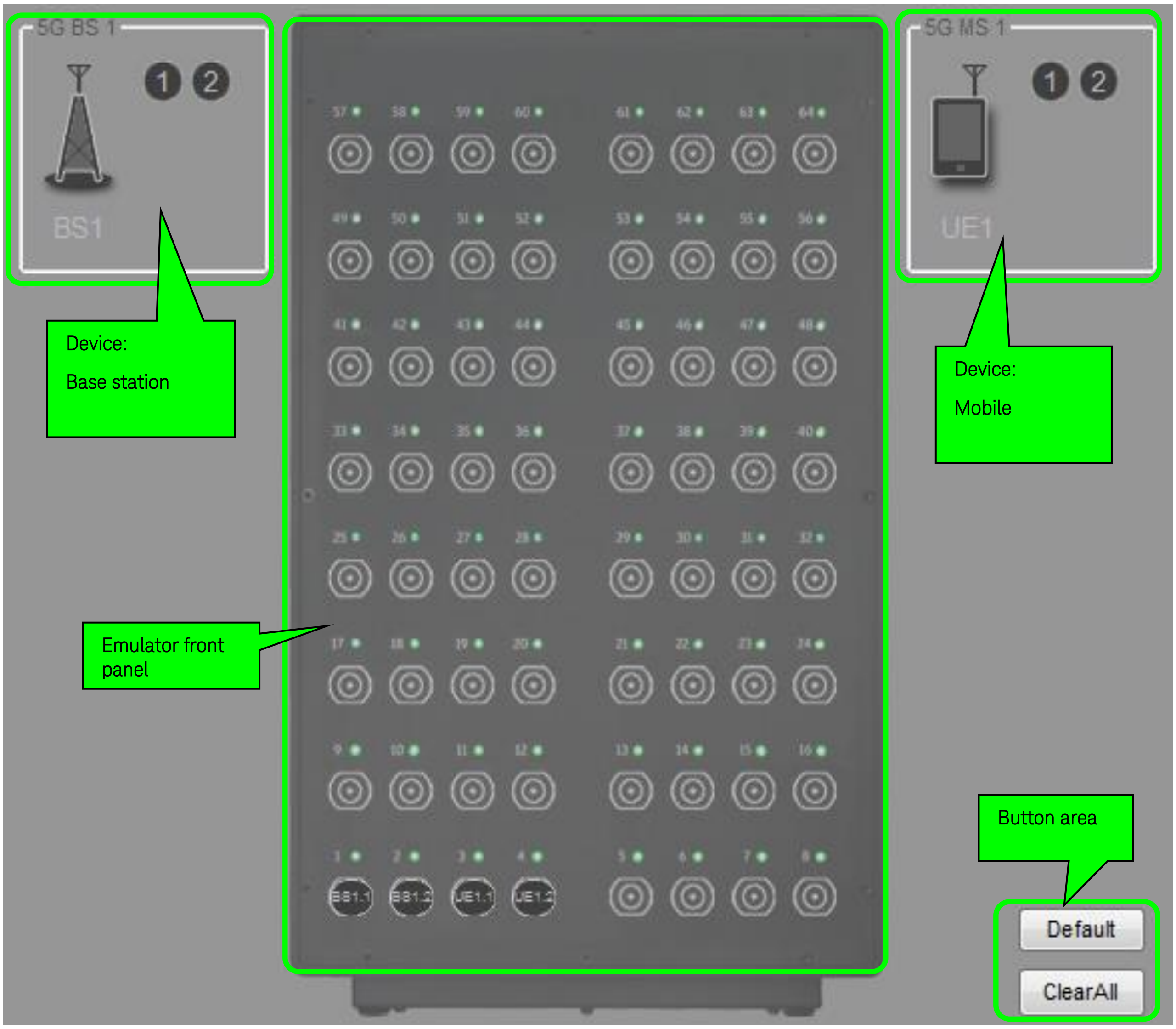


Figure 47 Active connector selection page layout – PROPSIM F64

Emulator front panel

- Presents available RF connectors and allocated RF connectors in current hardware configuration
  - Available RF connectors are shown in grey color: 
  - Allocated RF connectors are shown with identification information: 

Identification information is based on device identification (BS1 and UE1 in Figure 47) and connected device antenna number.

Devices

- Illustrated with representative symbols and antenna amounts on the both sides of the emulator front panel
- If Mobile ad-hoc network (MANET/Mesh) option selected on the first page of wizard (see chapter 3.1.2), MANET radios are shown on the left of the front panel

Button area

- Button(s) for restoring/resetting all allocated connectors.

- **Default** button restores the default connector setup
- **Clear All** clears all active connectors

3.1.5.1 Active connectors selection

In this page, it is possible to verify default connector settings, or modify connector locations to match test setup cable connections. Connectors can be relocated one by one to suitable RF connectors.

Figure 48 presents the PROPSIM F64 default active connectors for bi-directional 2x2 MIMO emulation, as located in the emulator front panel. All PROPSIM F64 connectors are duplex, for example, in Figure 48, Base station TX 1 and RX 1 antennas (BS1.1) are allocated to RF1 (IN/OUT).

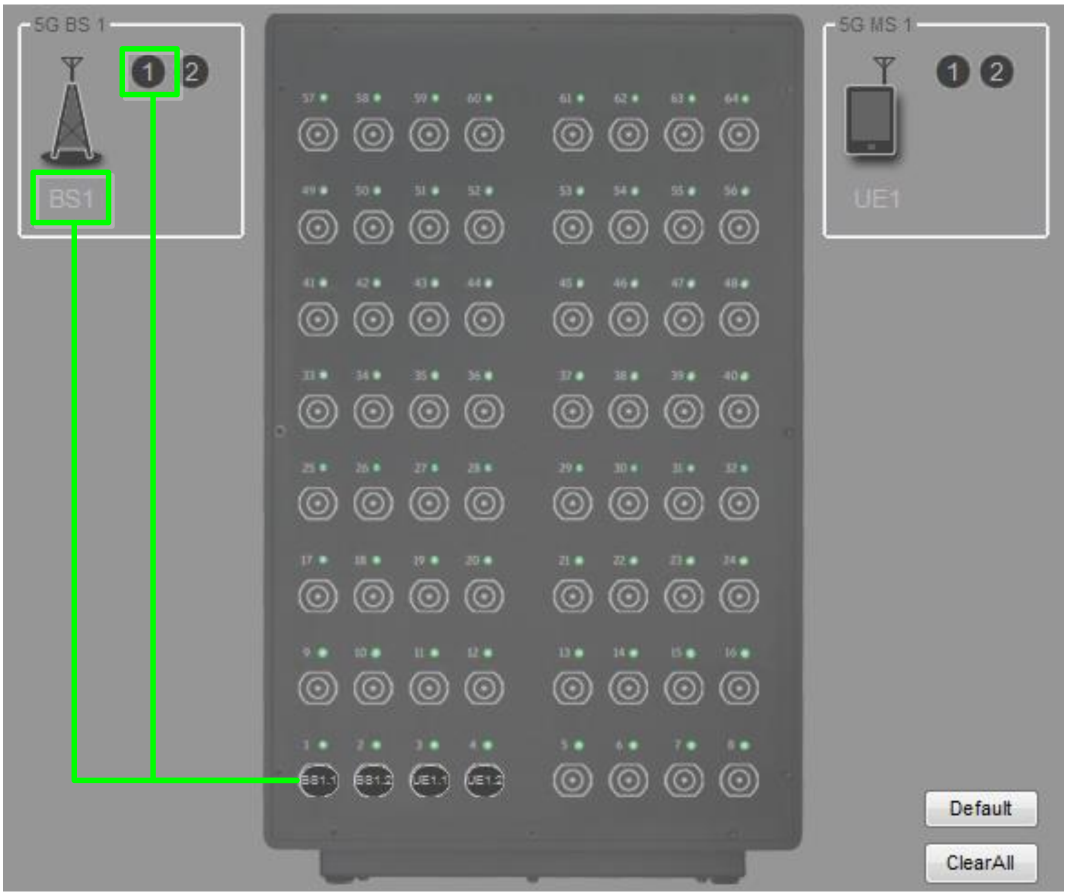


Figure 48 Active connectors selection – F64

Relocating connectors

It is possible to change the default connector setup. It can be done in two ways:

- All active connectors are cleared by clicking **Clear All** button and then all antennas are dragged one by one to wanted RF connector or
- Antenna is dragged from the front panel to the wanted RF connector and dropped there. When dragging antenna to new connector, suitable connector is shown as green in the front panel: 

Default button restores the default connector setup.

3.1.6 Step 5: Summary view

Last page of the wizard presents a summary of selected emulation configuration (see Figure 49).

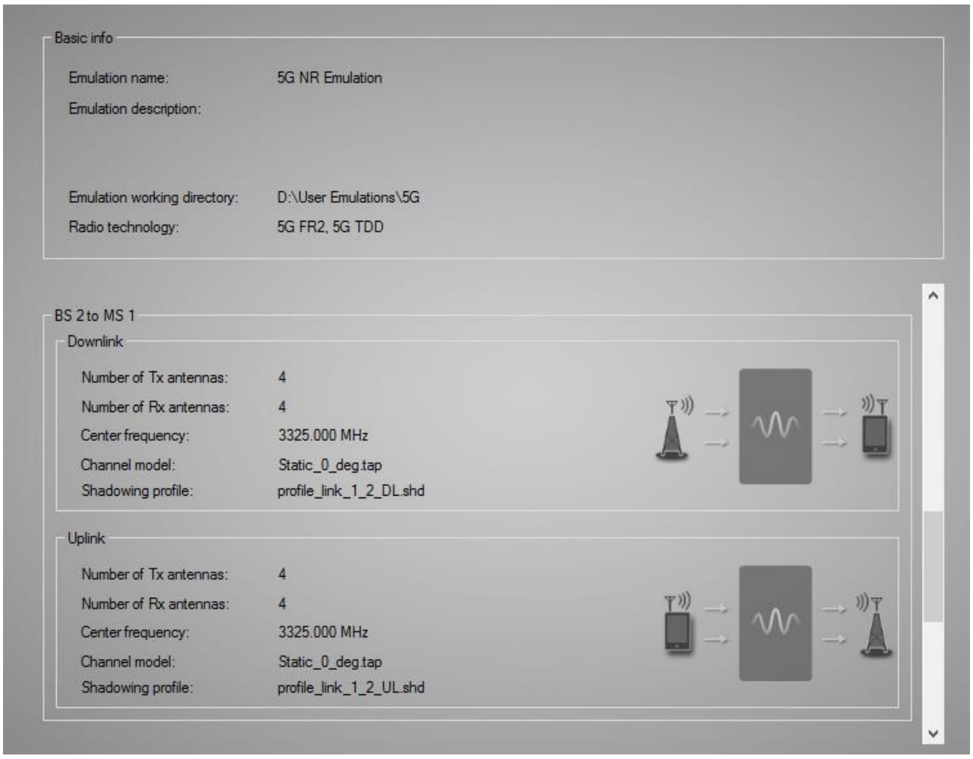


Figure 49 Emulation summary

On summary page, **Finish** button replaces the **Next** button. When **Finish** button is clicked, following options are opened:

- **Build & Finish emulation**: Save emulation and build it.
- **Build & Run emulation**: Save emulation, build it, and open to the Emulation Control View.
- **Finish emulation**: Save emulation.
- **Cancel**: Cancel and return to summary page.

Emulation is saved to individual folder in emulation working directory, containing all related files. Folder name consists of emulation name and .wiz extension in format <emulation name>.wiz (e.g. MyEmulation.wiz).

Emulation building is described in chapter 3.1.6.1.

3.1.6.1 Emulation building

Emulation build is started when **Build and Finish emulation** or **Build and Run emulation** option has been selected on summary page. Build progress bar (Figure 50) appears at the bottom of the window.

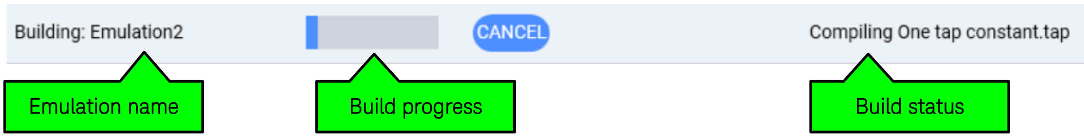


Figure 50 Build progress bar

Build progress bar has the following information and functionality:

Emulation name

- Name of the emulation

Build progress

- Indicates the status of the current build

Build status

- Status description of the build
- To get more specific build messages, hover the mouse over the build progress bar.

Cancel

- Cancel the ongoing build

3.2 Open emulation for editing in Scenario Wizard

Selecting the Edit or Run emulation in the Home view, or selecting Emulation > Open in the navigation bar, opens a dialog where user can browse and select an emulation file to be opened for editing.

Opening emulations for editing can be also used for creating new emulations based on existing ones. This can be done by opening an existing emulation for editing and giving a new name for it in the basic information page of the Scenario wizard. This will create a copy of the emulation with a new name and leave the original emulation untouched.

Figure 51 Open emulation

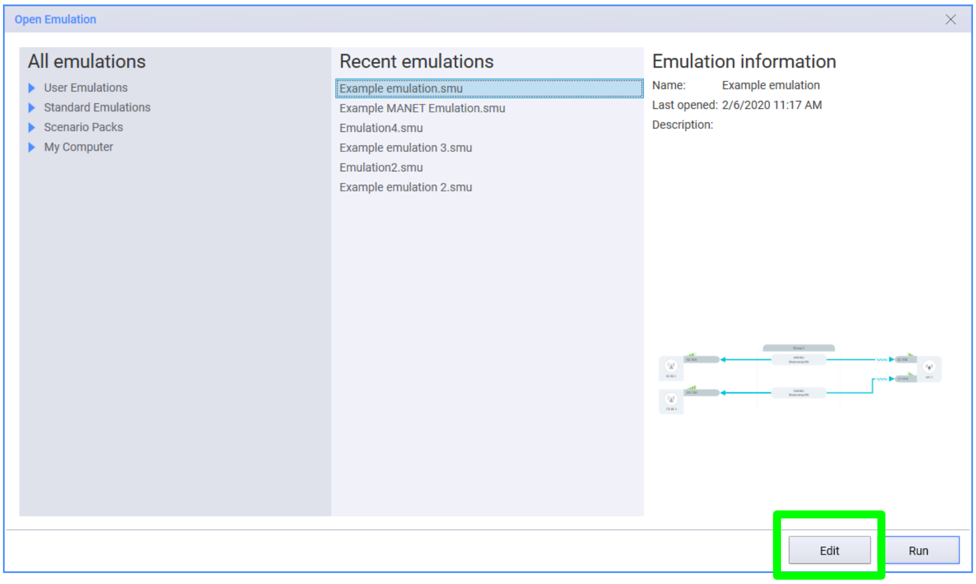


Figure 52 Select emulation to edit

The dialog includes the following lists for browsing the emulation file:

- **All emulations**: Pre-installed standard emulations (**Standard Emulations**), user defined emulations (**User Emulations**), and emulation files on the computer in hierarchical list.
- **Recent emulations**: List of recently used emulation files of current user.

Emulation information and graph preview are shown on the right.

Emulation is opened to Scenario Wizard. In the Scenario Wizard, it is possible to edit emulation description, technology, channel models, antenna configuration, and connector configuration by following steps described in chapter 3.1. Changing the emulation name creates a new copy of the selected emulation, leaving the original emulation untouched.

3.2.1.1 Modified environment variables

Environment variables page presents links with their environment variables. When new emulation is created, defined environment variables are set to all link antennas in emulation.

When emulation is opened for editing in Scenario wizard, it is possible that variables were modified in Emulation Control View and might differ between link antennas. If the value is not the same in all link antennas, the text **Values Differ** is shown in environment variables table (see Figure 53). Range of current values for selected variable is shown above the variable table along with possible limit values.

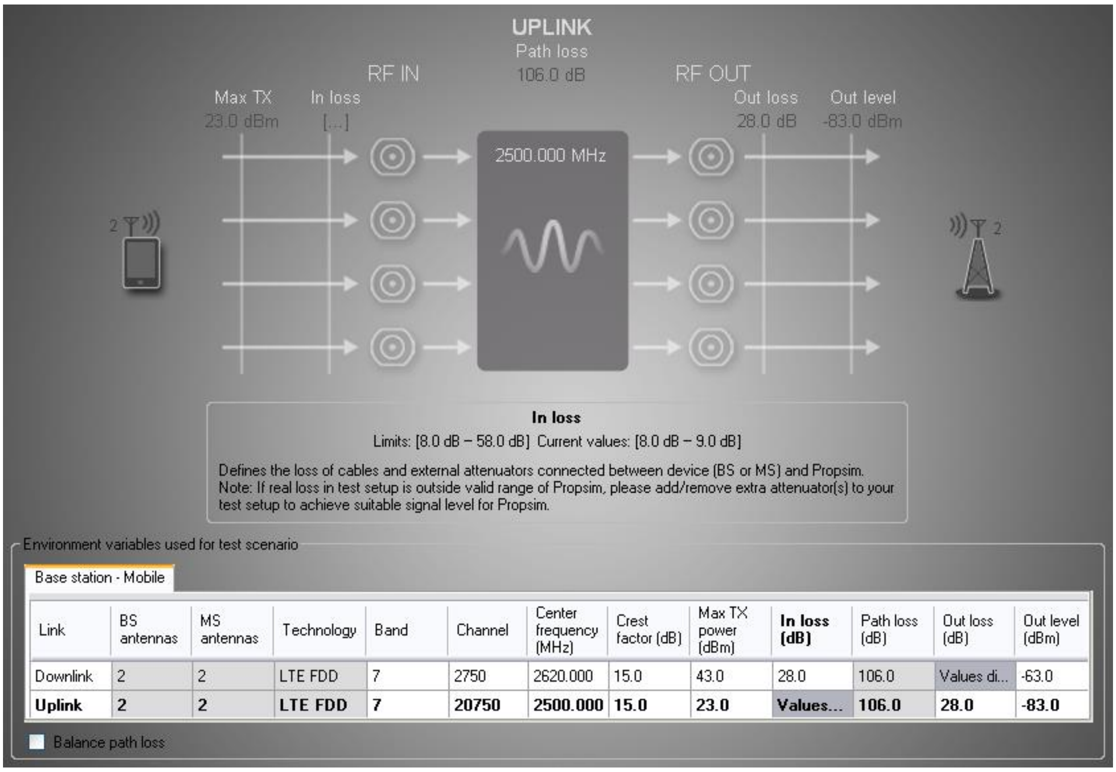


Figure 53 In loss and Out loss values differ

If new value is entered to edit field, new value is set for all antennas in link.

Note: *Changing the value might also affect to limits of other variables (depending on edited variable) and this way change also related variables.*

3.3 Start emulation

For instructions to start the selected emulation in Emulation control view, see chapter 4.2.

4 EMULATION CONTROL VIEW

The Emulation Control View is used to load and run standard emulations or emulations that have been created with the Scenario wizard or channel modelling tools. In this view, the user can view settings of the emulation and the hardware connectors, change parameters of the emulation, and control how the emulation is run. User can also save the changes to the emulation for future use.

To open an emulation in the Emulation Control View, click the **Edit or Run emulation** button in the Home view, or select **Emulation > Open** in the navigation bar. In the **Open Emulation** dialog, select the emulation you want to run and click **Run**.

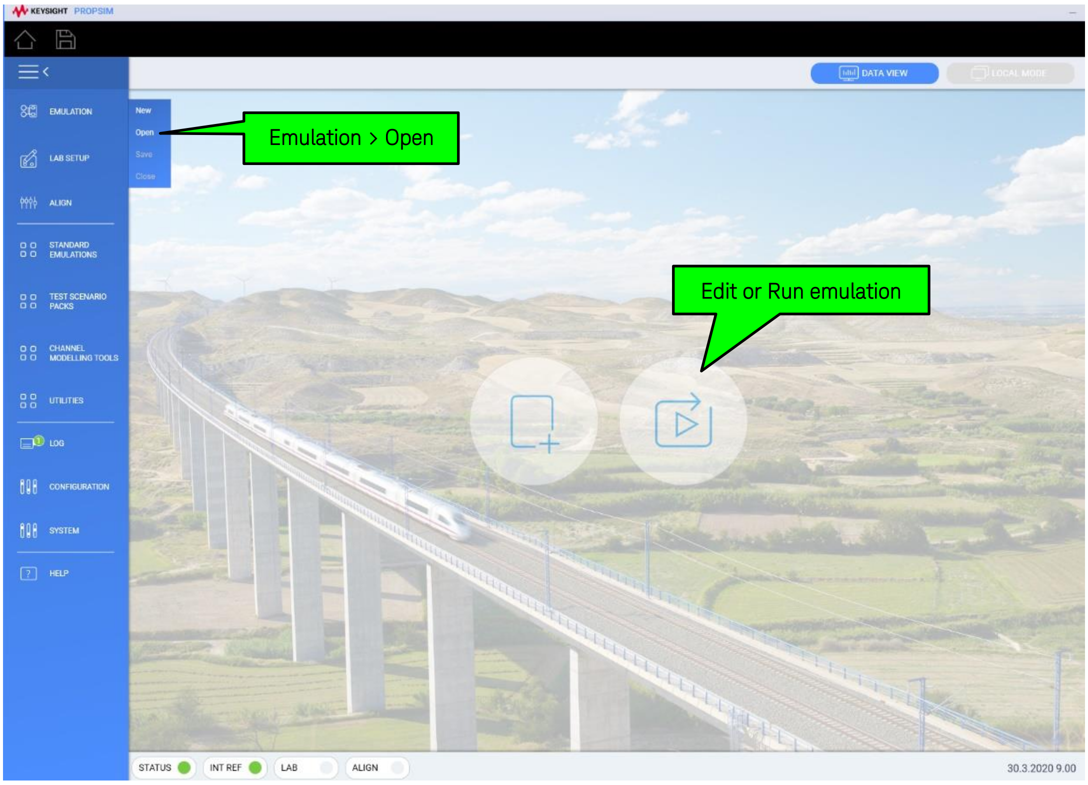


Figure 54 Opening emulation

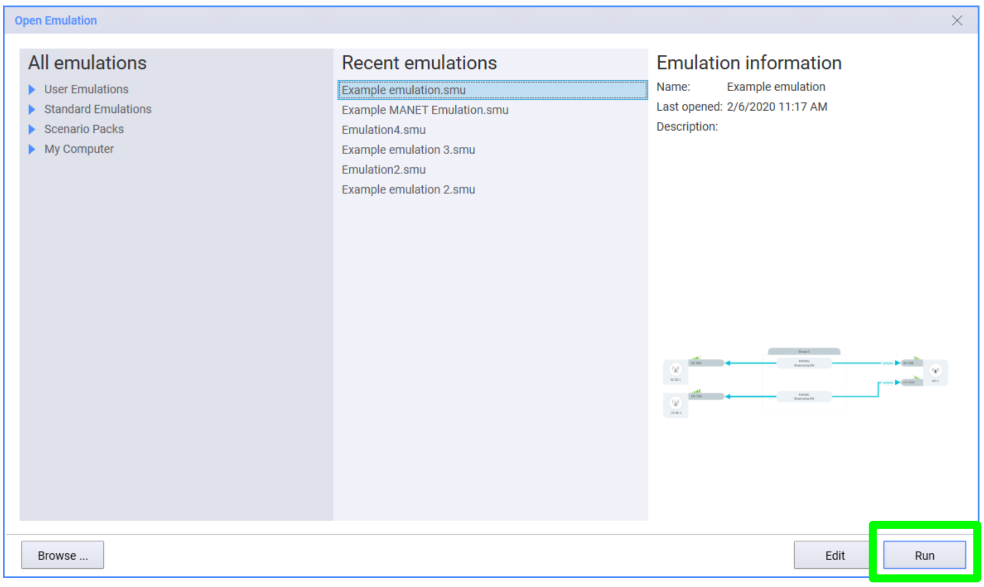


Figure 55 Select emulation to run

Pre-generated emulation file (.SMU) is first loaded to the Emulation control view. Modified emulation parameters can be saved to the emulation file if desired. The actual emulation data for each channel is stored in the emulation hardware control files (.SIM) not visible to the user.

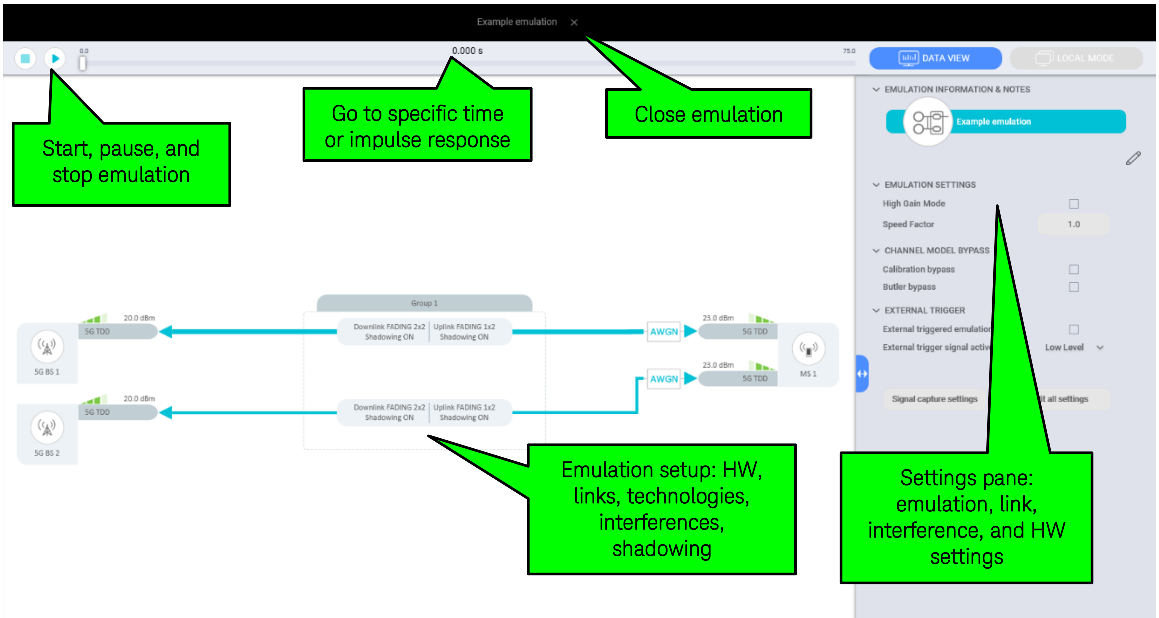


Figure 56 Main parts of Emulation Control View

- For more information on the emulation diagram, see section 4.1.
- For more information on starting emulations, see section 4.2.
- For more information on the different settings in the settings pane, see section 4.3.

4.1 Emulation setup diagram

The emulation setup diagram is a high-level presentation of the emulation setup. Each link may contain multiple antenna connections.

You can zoom in and out of the diagram with the ctrl-button and mouse wheel.

For instructions on viewing and editing the emulation and element settings, see chapter 4.3.

4.1.1 Cellular systems emulation

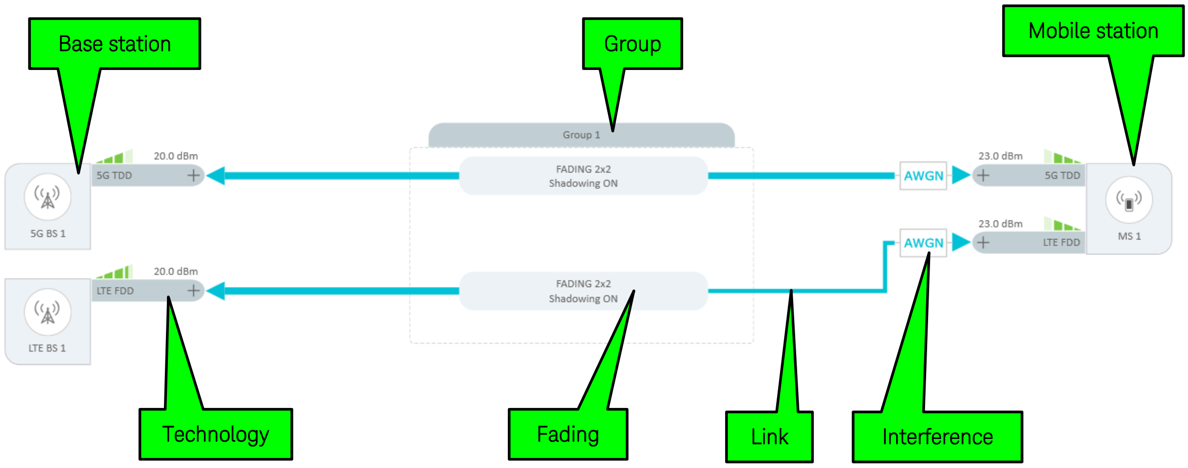


Figure 57 Setup diagram for a hand-over cellular systems emulation

The diagram of a cellular systems emulation consists of the following elements:

- Base station
- Technology (radio technology used by BS or MS)
- Mobile station
- Group (consists of base stations and mobile stations that have links to each other)
- Fading (fading in the radio signal between BS and MS). This element shows the following data:
  - Whether FADING or BYPASS is enabled for the uplink, downlink, or both
  - Antenna topology. The format depends on other link settings and on the number of antennas in the BS and MS.
    - *[number of transmitters] x [number of receivers]*
      This format is used when the link settings are the same for uplink and downlink, and BS and MS have the same number of antennas.
    - *Downlink [number of BS transmitters] x [number of MS receivers]*
      This format is used for the downlink topology when the link settings are different for uplink and downlink. The number of BS and MS antennas may be different.
    - *Uplink [number of MS transmitters] x [number of BS receivers]*
      This format is used for the uplink topology when the link settings are different for uplink and downlink. The number of BS and MS antennas may be different.
  - Whether shadowing is ON or OFF for the uplink, downlink, or both
- Link (radio link between BS and MS, containing one or more channels). The ✕ and ⓘ icons indicate possible disabled links, channels, and outputs as well as modifications in adjustment gain:
  - If the entire uplink or downlink is disabled, the ✕ (Link disabled) icon is shown on the link next to the Fading element.
  - If one or more channels are disabled, or if adjustment gain has been modified, the ⓘ (Balance adjusted) icon is shown on the link next to the Fading element.
  - If all the outputs of a BS or MS are disabled, the ✕ (Outputs disabled) icon is shown on the link next to the BS or MS.
  - If one or more outputs of a BS or MS are disabled (but not all outputs), the ⓘ (Some of the outputs are disabled) icon is shown on the link next to the BS or MS.
  - For more information on disabling and enabling links, channels, and outputs, see chapter 4.3.3.1.
- Interference (AWGN or CW interference to BS or MS)

4.1.2 MANET emulation

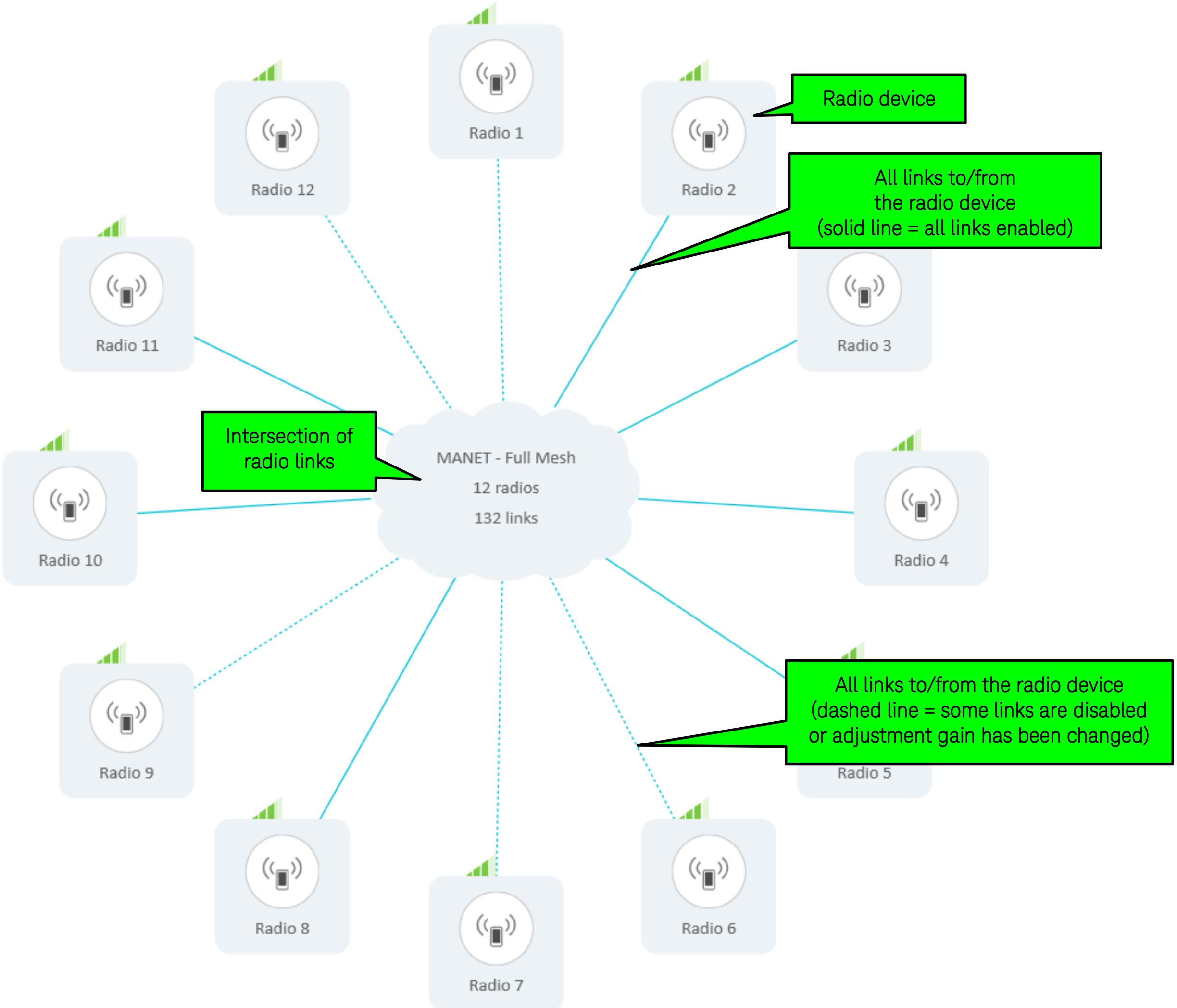


Figure 58 Setup diagram for MANET emulation with 12 radio devices (no radio device selected)

The diagram of a MANET emulation consists of the following elements:

- Radio devices.
- All the links between the radio device and all other radio devices in the emulation. To see the direct links of one radio device, select the radio device.
  - Solid line = All the links to/from the radio device are enabled and adjustment gain has not been changed.
  - Dashed line = One or more links to/from the radio device are disabled or adjustment gain has been changed.
- Intersection of radio links (shown when the emulation contains more than 10 radio devices). This is a visual presentation of the two-way radio links. Each individual radio device is linked to all radio devices in the setup. To see the direct links of one radio device, select the radio device.

When you click a radio device, the diagram shows all the links to/from that radio device. The diagram also shows whether the links between radio devices are enabled or disabled and whether their adjustment gain has been changed. Clicking a link opens the settings of that link in the settings pane.

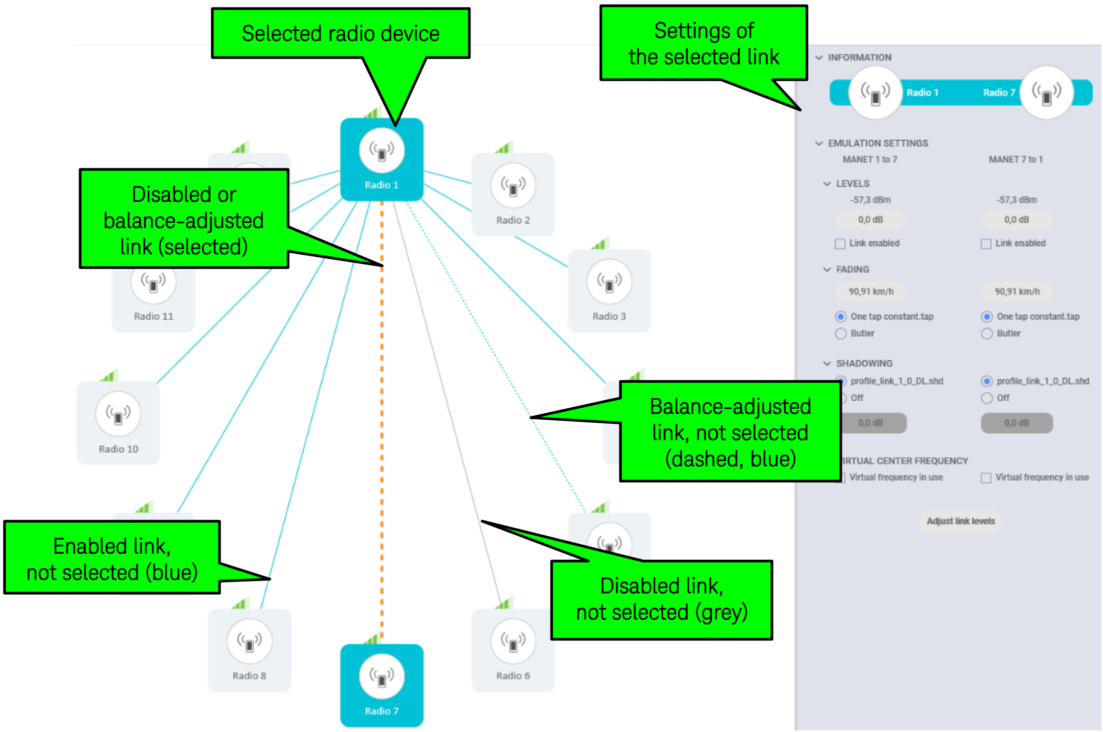


Figure 59 MANET emulation with a radio device and link selected

You can enable and disable all the links to/from a radio device, as well as links between two radio devices (in both directions or one direction):

- To enable/disable all the links to/from a single radio device, right-click the radio device and select **Enable all links** or **Disable all links**.
- To enable/disable a link between two radio devices in both directions, select a radio device, right-click the link you want to enable/disable, and select **Enable link** or **Disable link**.
- To enable/disable a link between two radio devices in one direction only, select the radio device and, in the link settings pane, select/unselect the **Link enabled** checkbox under the link direction you want.

4.2 Controlling the emulation run

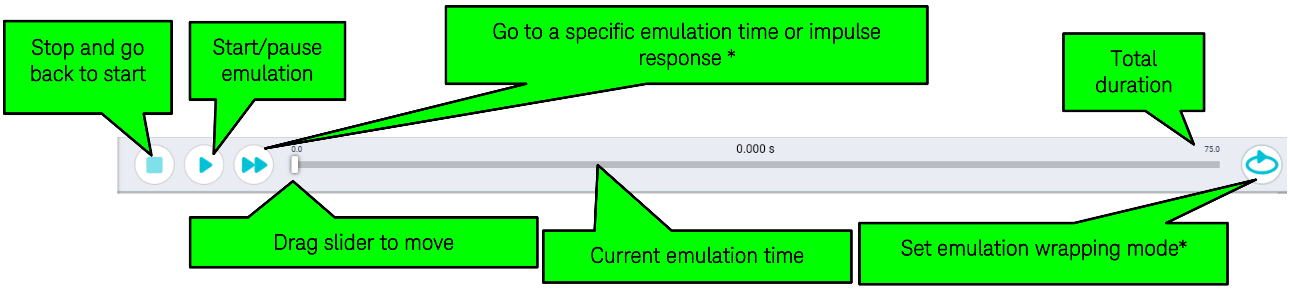


Figure 60 Emulation timeline (* *You cannot move to another time on the timeline when the emulation is running.)*

You can control the emulation run using the tools in the emulation timeline:

- To start the emulation that is open, click the (**Start**) button.
  Note: You cannot drag the emulation slider when the emulation in running.
- To pause the emulation, click the (**Pause**) button. The emulation pauses in the current time on the timeline.
- To stop the emulation and go back to start, click the (**Stop**) button. The emulation jumps back to the beginning of the timeline.
- To move to another time on the emulation timeline, drag the slider or click and enter a specific emulation time or impulse response that you want to jump to.
  Note: You cannot move to another time on the timeline when the emulation is running.

- To set the emulation wrapping mode, click the (**Wrapping mode)** button and select continuous or single run. In the single run mode you can choose the specific time range you want to run and by selecting "*Stop and re-run segment repeatedly*" it will automatically jump back and play again.

The total duration of the emulation (in seconds) is shown on the right of the emulation timeline.

The Data view window shows live data of the running emulation. For more information, see chapter 5 Data Views.

4.3 Settings

The settings pane in the Emulation control view shows the settings for the emulation, links, HW (base stations, mobile stations, radio devices), interferences, and technologies. With user-defined emulations, you can edit these settings in the Emulation control view and save them for future use.

4.3.1 Saving changes to emulation settings

If you make changes to an emulation and you want to keep those changes, you can save the emulation by clicking the (Save) button on the top left corner of the PROPSIM window.

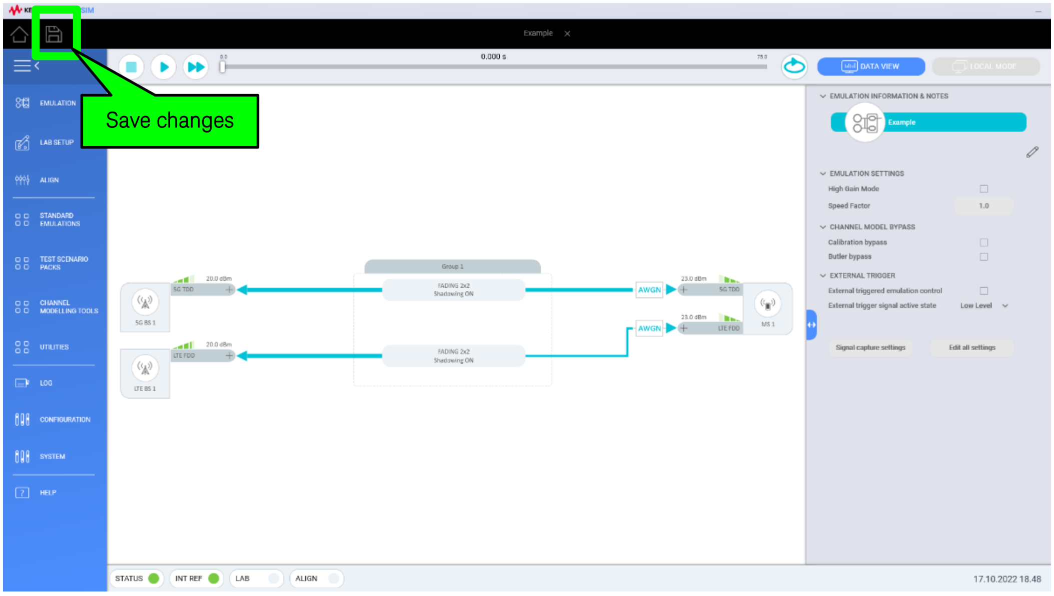


Figure 61 Saving changes to emulation

Note: It is possible to edit the settings and run the emulation without saving the changes. When you close the emulation with unsaved changes, a dialog appears where you can choose to save the changes.

4.3.2 Emulation settings

Emulation settings are shown when none of the emulation diagram elements are selected.

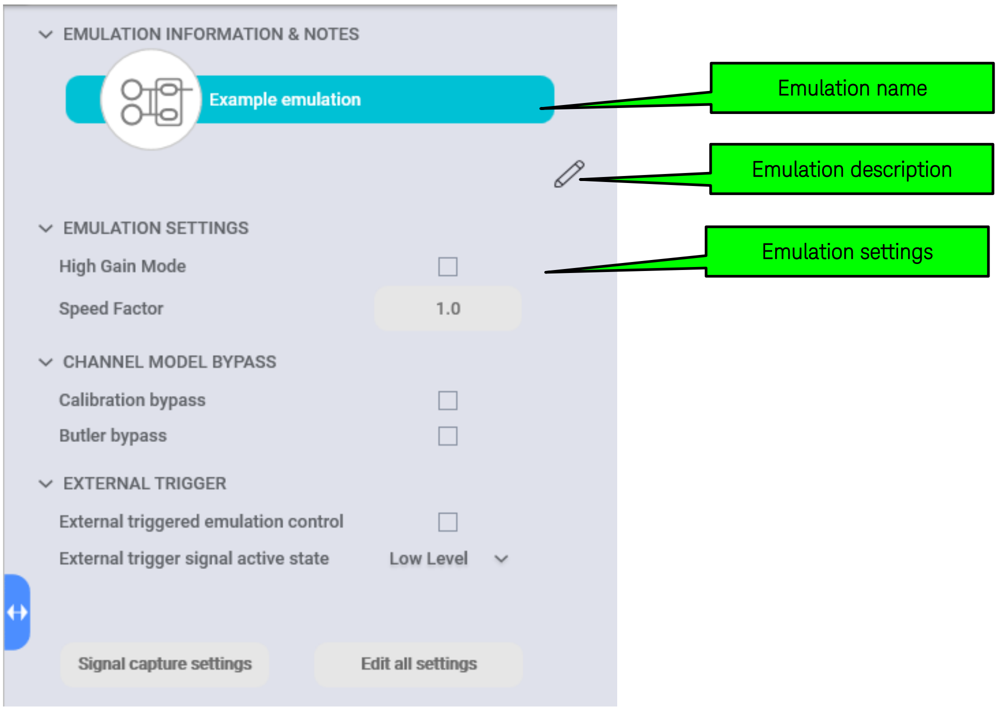


Figure 62 Emulation settings pane

Emulation name

Name of the emulation

Emulation description

Description of the emulation (optional)

High Gain Mode

Enabling this feature scales signal up +5 dB digitally.

Scaling signal up digitally can cause clipping depending on used fading, input signals and their phases. Please pay attention on clipping notifications (see section 4.3.3.3).

RF linearity and spurious specification is not guaranteed when applying High gain mode or positive channel gain.

Note: *Using high output gain in TDD emulation increases the risk of uncontrolled signal circulation (echo) which may have undesired impact on measurement results. Extra care must be taken to verify proper operation when using high gain mode in TDD case. Possible signal circulation depends on multiple factors as input and output settings, channel model, user signal and test setup outside PROPSIM.*

Speed factor

Multiplication factor for increasing the emulation running speed from the original. Default value is 1.0. This parameter applies to all time-variant parameters of the emulation: Doppler, shadowing profiles, speed profiles, and interference profiles.

Example: If the speed factor is set to 2.0, effective Doppler in the channels will be double compared to the original Doppler. Emulation duration, including fast fading, shadowing, speed and interference profiles, will be half compared to the original duration.

Note: *CIR graph in the Data view (chapter 5.5) will display the final mobile speed with speed factor applied.*

Calibration bypass

Calibration bypass replaces all the fading channels (uplink and downlink) in the emulation with static 1-path model with equal attenuation (-10 dB channel gain), delay and phase. Calibration bypass makes it possible to perform the phase and amplitude calibration for the test setup using emulations with different delays and amplitudes between fading channels. When calibration bypass is enabled, delay through the emulator is shown in the tooltip in the emulation settings pane as shown in Figure 63. Delay value depends on the emulation topology (SISO, MIMO, etc.) and the hardware configuration.

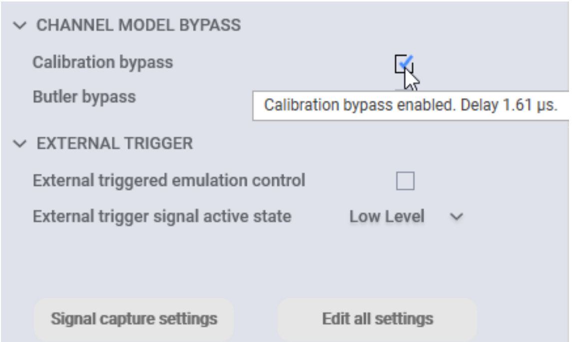


Figure 63 Calibration bypass delay

When calibration bypass is selected in the emulation settings pane, the Fading element indicates bypass is in use:

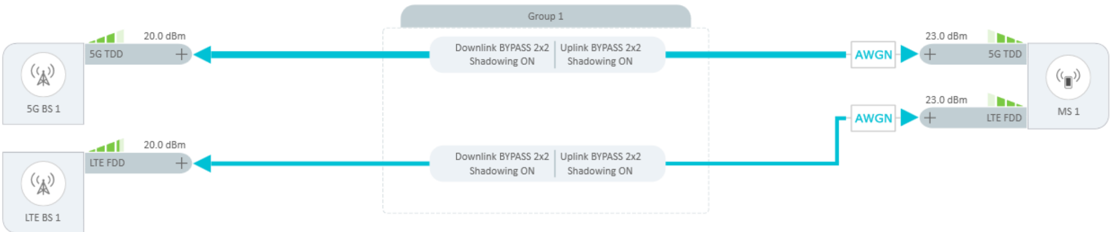


Figure 64 Bypass indicated in Fading element

Butler bypass

Butler bypass replaces all the fading channels (uplink and downlink) in the emulation with static 1-path model with attenuation at the average level of the corresponding fading channel model. Delay of the static channels is defined by the shortest path delay of the current position in the original channel model, as shown in Figure 65.

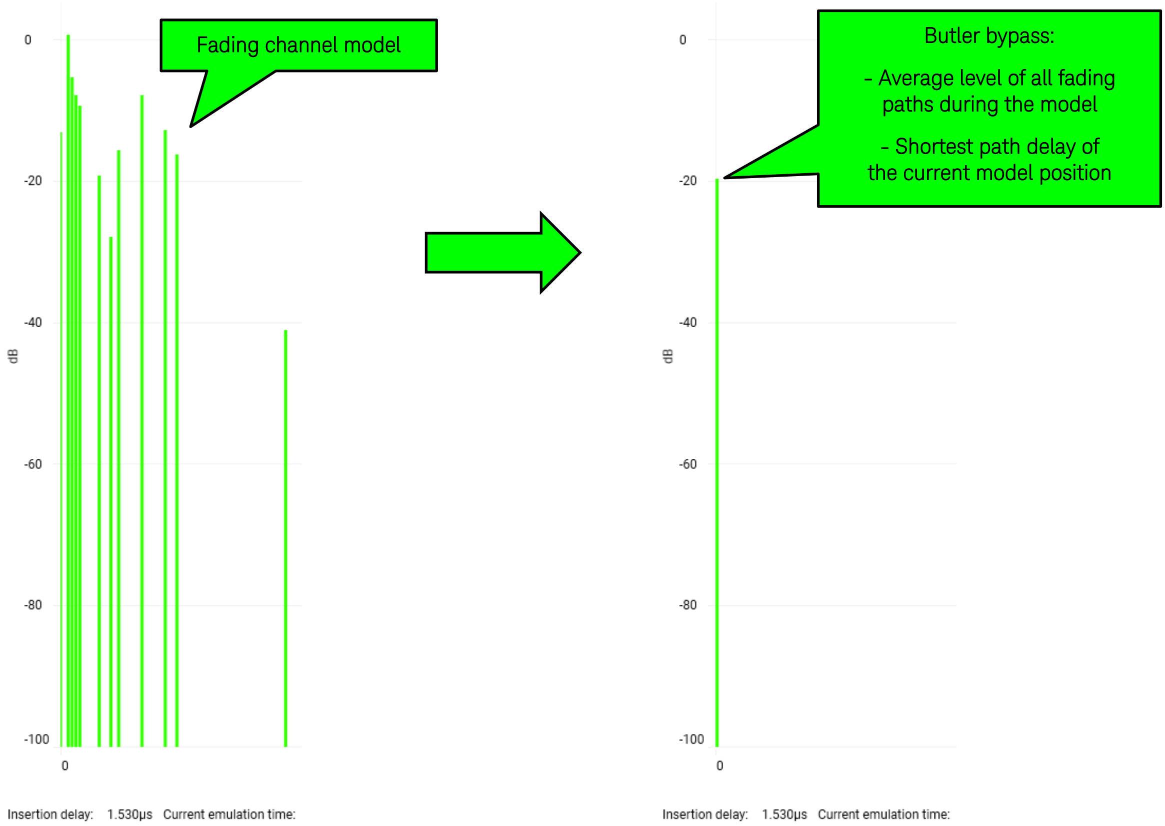


Figure 65 Butler bypass amplitude and delay behavior principle

Note: Bypass is not indicated in CIR View as shown as a principle in Figure 65.

Butler Bypass also modifies the phase components of the individual channels according to Butler matrix. This emulates the beaming effect, where receiver will see all the MIMO input streams arriving from different angles. This will create enough diversity for the receiver to separate different input streams and MIMO link can be established.

Butler phases are calculated with following formula (Radio wave Propagation and Antennas for Personal Communications, Second edition, Kazimierz Siwiak, 1998):

$$phase(input, output) = angle\left(\exp\left[j\left[\left[input - \frac{N+1}{2}\right]\left[output - \frac{N+1}{2}\right]\frac{2\pi}{N}\right]\right]\right) - angle\left(\exp\left[j\left[\left[input - \frac{N+1}{2}\right]\left[1 - \frac{N+1}{2}\right]\frac{2\pi}{N}\right]\right]\right)$$

where,

phase	is phase shift of the channel between input and output in radians
input	is input index in MIMO channel group (1...N)
output	is output index in MIMO channel group (1...N)
N	is maximum MIMO dimension of inputs and outputs

A beaming with Butler matrix is based on assumption that antenna elements are on horizontal plane with a distance λ/2 between elements.

For 4x4 MIMO the equation gives following phase matrix in degrees:

$$\begin{bmatrix} 0 & -135 & 90 & -45 \\ 0 & -45 & -90 & -135 \\ 0 & 45 & 90 & 135 \\ 0 & 135 & -90 & 45 \end{bmatrix}$$

Matrix rows represent the MIMO inputs and matrix columns MIMO outputs. Figure 66 shows the MIMO topology with Butler angles.

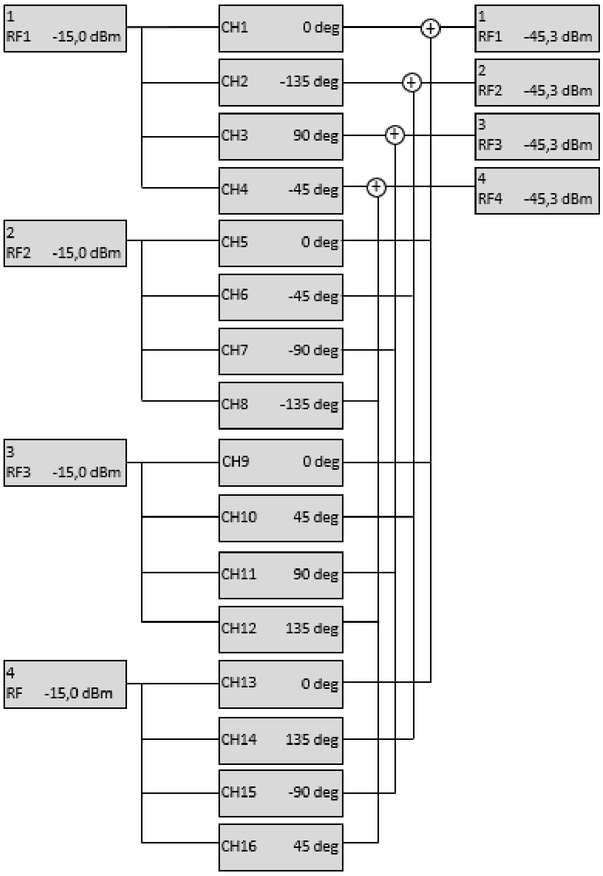


Figure 66 Butler phases of 4x4 MIMO

When Butler bypass is selected in the emulation settings pane, the Fading element indicates bypass is in use:

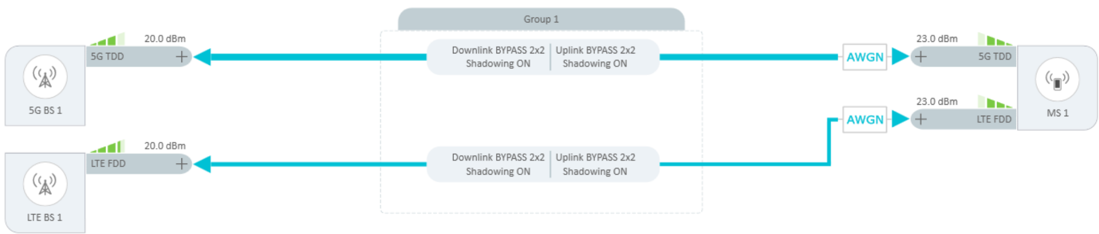


Figure 67 Bypass indicated in Fading element

External triggered emulation control

External trigger can be used for starting and stopping the emulation by using external connection. Trigger signal is connected to Sync In connector of the PROPSIM.

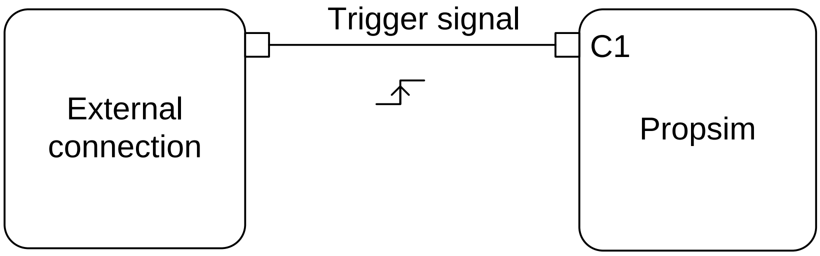


Figure 68 Example trigger connection

The current running state of the emulation can be acquired from the Sync Out connector of the emulator. When emulation is running, Sync Out is low, and when stopped, Sync Out is high.

Emulation start, stop and pause can be trigged in order to define the time of the actions precisely. Either level or edge active triggering can be used. However, emulation start/stop/pause commands from GUI or ATE are still needed.

Note: If triggering is disabled while the emulator is waiting for the external trigger, the emulation continues similarly as if trigger had occurred.

External trigger signal active state

Value: Low Level or High Level

- Starting emulation
  - When you click the (Start) button in the toolbar of Emulation control view, emulator starts to wait for trigger signal. The Start button begins to blink and the text “Waiting for external trigger” is shown in the button tooltip.
  - When changes to active level is detected on the signal connected to Sync In, the emulation starts to run.
- Running emulation
  - Emulation stops running when trigger signal goes to non-active state. The Start button begins to blink and the text “Waiting for external trigger” is shown in the button tooltip. When the trigger signal goes to active state, the emulation continues running.
  - Note: If triggering is disabled while the emulator is waiting for external trigger, the emulation continues.
- Pausing emulation
  - When you click the (Pause) button in the toolbar of Emulation control view, emulation pauses immediately. Clicking the (Start) button continues the emulation.
- Stopping emulation
  - When you click the (Stop) button in the toolbar of Emulation control view, emulation stops immediately and rewinds back to start (0 s).

Figure 140 shows an example of low-level triggering.

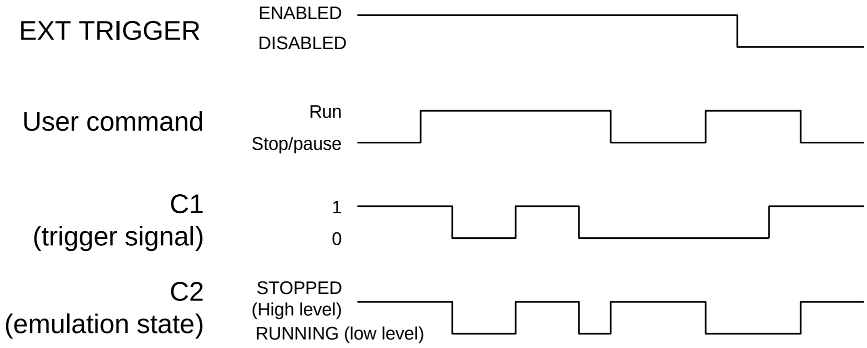


Figure 69 Low level triggering example

Value: Falling Edge or Rising Edge

- Starting emulation
  - When you click the (Start) button in the toolbar of Emulation control view, emulator starts to wait for trigger signal. The Start button begins to blink and the text “Waiting for external trigger” is shown in the button tooltip.
  - When active edge (rising or falling, depending on configuration) is detected on the signal connected to Sync In, the emulation starts to run.
- Running emulation
  - Emulation runs normally.
- Pausing emulation
  - When you click the (Pause) button in the toolbar of Emulation control view, emulation pauses when the signal connected to Sync In interface has an active edge. Clicking pause again starts the emulation on the next active edge in the input sync signal.
- Stopping emulation
  - When you click the (Stop) button in the toolbar of Emulation control view, emulation stops when the signal connected to Sync In interface has an active edge.

Figure 70 shows an example of falling edge triggering.

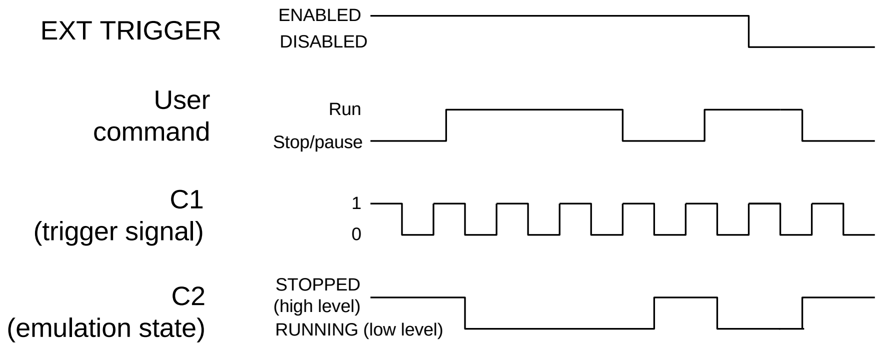


Figure 70 Falling edge triggering example

4.3.3 Link settings

Link settings are shown when the FADING element on the emulation diagram is selected.

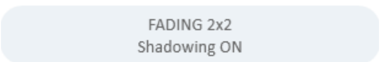


Figure 71 FADING element in emulation diagram

The FADING element shows the following data:

- Whether FADING or BYPASS is enabled for the uplink, downlink, or both
- Antenna topology
- Whether shadowing is ON or OFF for the uplink, downlink, or both

The link settings affect all the channels in the downlink and uplink between the base station and the mobile station. If needed, you can also define channel-specific values in the **All settings** dialog, see section ☐.

Some values are read-only and cannot be edited. These values have a dark grey background.

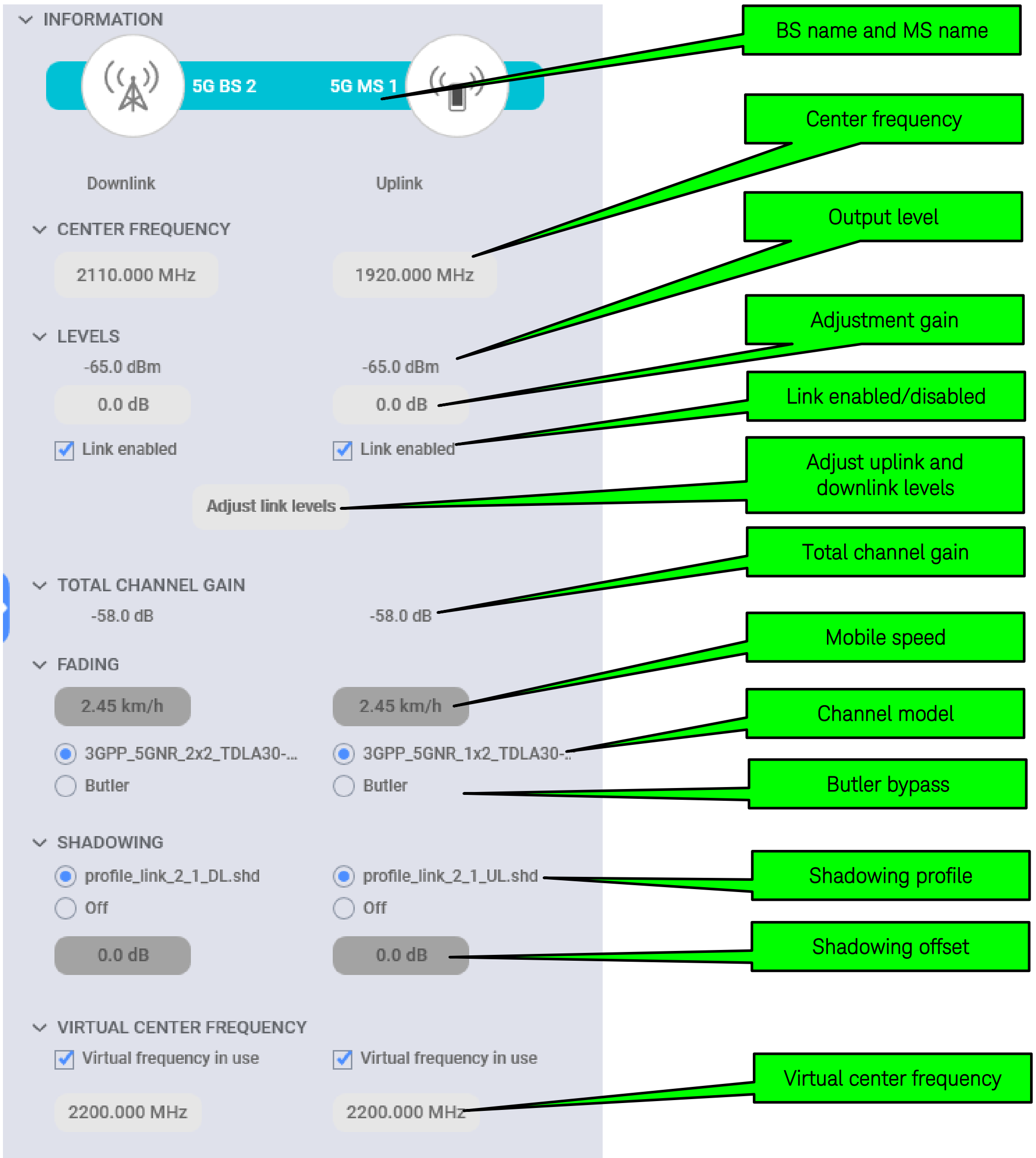


Figure 72 Link settings pane

BS name and MS name

- Names of the base station and mobile station (editable in Scenario wizard)

Output level

- Expected link output level, including current shadowing curve level.

Adjustment gain

- This field is used to adjust the link level digitally. Value goes to gain imbalance adjustment for each channel of the selected link.
- When using positive values (+5 – 0 dB), pay attention to digital clipping notifications (see section 4.3.3.1).

Link enabled/disabled

- Enable or disable the uplink or downlink

Mobile speed

- This field is used to change the mobile speed of each channel of the selected link.
- Mobile speed affects the CIR update rate according to the following equation:

$$f_{upd} = \frac{2 \cdot SD \cdot v \cdot f_c}{c} \text{ ,}$$

where

- SD sample density, samples per half-wave (in the channel model file),
- c speed of light,
- v mobile speed (with emulation speed factor),
- f_{upd} CIR update rate,
- f_c center frequency from channel group.

- Mobile speed is read-only if the CIR update rate has been locked in the channel model file (.tap, .ir, etc.).

Channel model

- Fading option that uses the selected fast fading channel models.

In case of concurrent channel models (multiple channel models in the same link, see chapter 3.1.3.1.2.2), the active channel model can be selected from the dropdown list, see Figure 73. Note that when swapping between channel model sets, the emulations must be stopped.

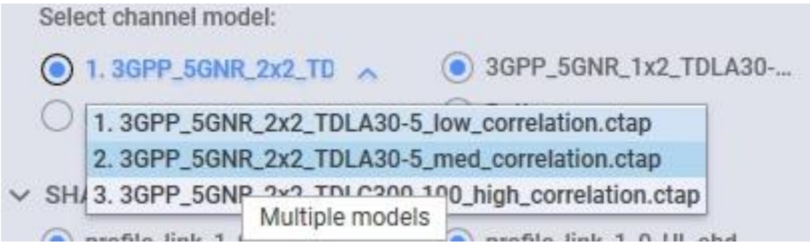


Figure 73 Concurrent channel model selection

Butler bypass

- Fading option that replaces the channel models with “Butler” models. For more information on butler bypass, see chapter 4.3.2, section Butler bypass.

Shadowing profile

- Shows the selected shadowing file name when channel specific shadowing is used.
- To disable shadowing, select **Off**.

Shadowing offset

- Shadowing offset for the link. Shadowing offset can be used to achieve the expected signal level for the DUT at specific shadowing curve point.

Virtual center frequency

- Virtual center frequency is used for calculating fading parameters (i.e. mobile speed, Doppler) in up/down conversion scenarios where RX and TX frequencies of the links are not equal or if application RF frequency is different from PROPSIM RF frequency (application IF frequency).

Adjust link levels

- Adjust uplink and downlink levels conveniently in one dialog. For more information, see section 4.3.3.1.

4.3.3.1 Adjusting link levels

You can adjust uplink and downlink levels conveniently in the **Link levels** dialog. To open the dialog, click **Adjust link levels** in the link settings pane, or right-click the FADING element and select **Adjust link levels**.

For descriptions of the settings, see Table 10 List of all settings on page 83.

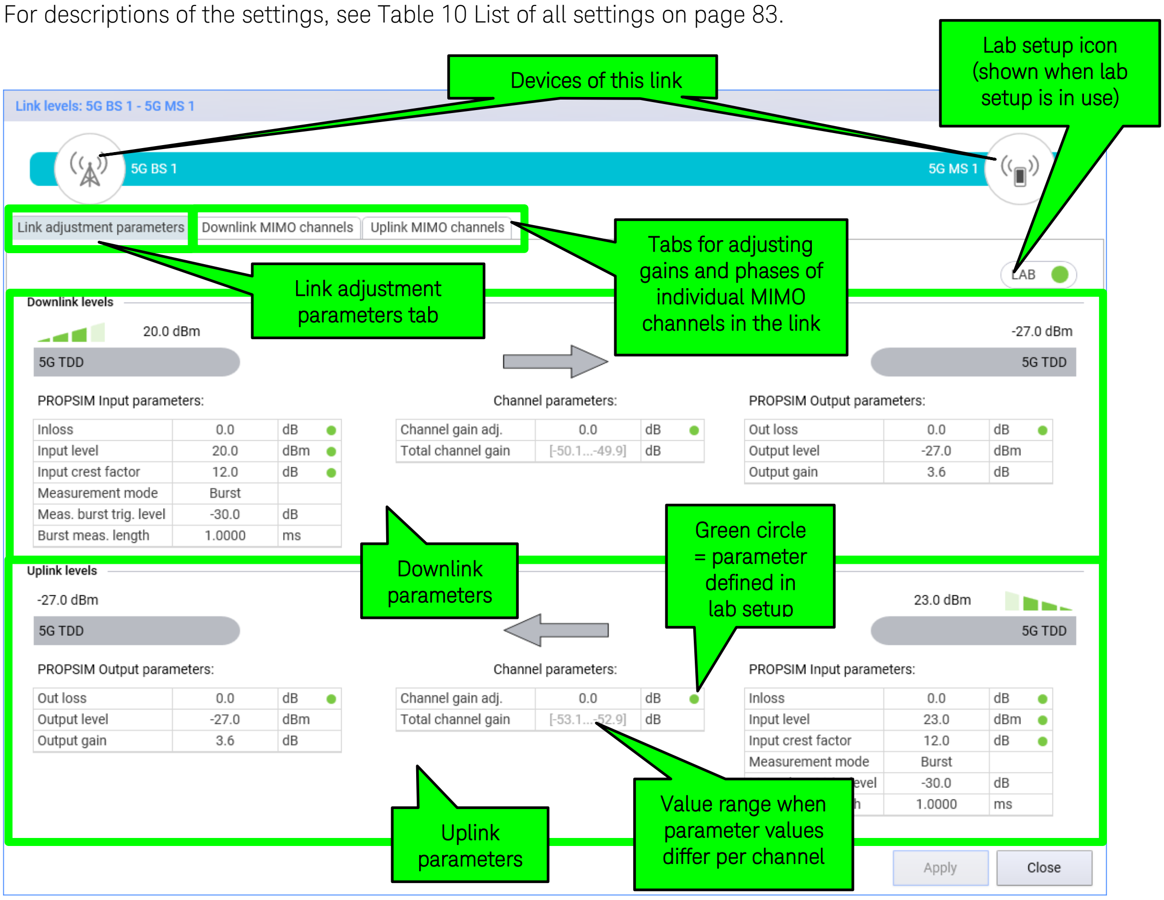


Figure 74. Link levels dialog

Note: You can also adjust the input/output levels and crest factors in the BS/MS settings pane, see section 4.3.4 on page 70.

The parameters listed under **Channel parameters** are common to all the channels in the link. If you change the value of these parameters, the change affects all the channels in the link. If the values of a parameter are currently not the same for all channels, the value field shows a value range, that is, the minimum and maximum value of the parameter. To adjust gains and phases of individual MIMO channels, use “Downlink MIMO channels” and “Uplink MIMO channels” tabs, as shown in Figure 75. To edit any parameters for each individual channel/input/output of the link, use **All settings** dialog, see section 4.3.7 on page 80.

Note: The parameter “Channel gain adj.” is the same as “Adjustment gain”.

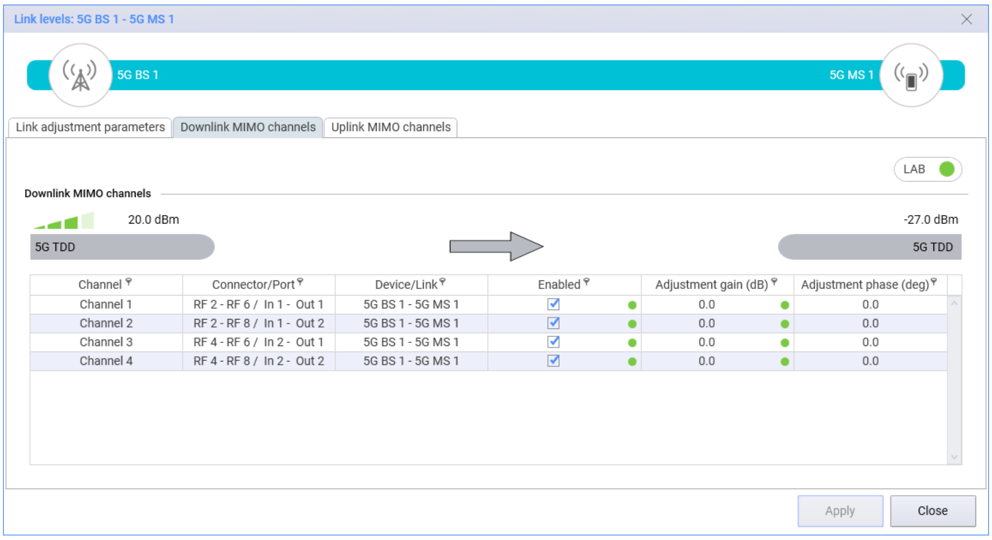


Figure 75. Downlink/Uplink MIMO channel of Link levels dialog

4.3.3.2 Disabling and enabling links, channels, and outputs

In the Emulation control view, you can disable and enable links, individual channels, and BS or MS outputs.

The and icons on the emulation setup diagram indicate possible disabled links, channels, and outputs as well as modifications in the adjustment gain of channels.

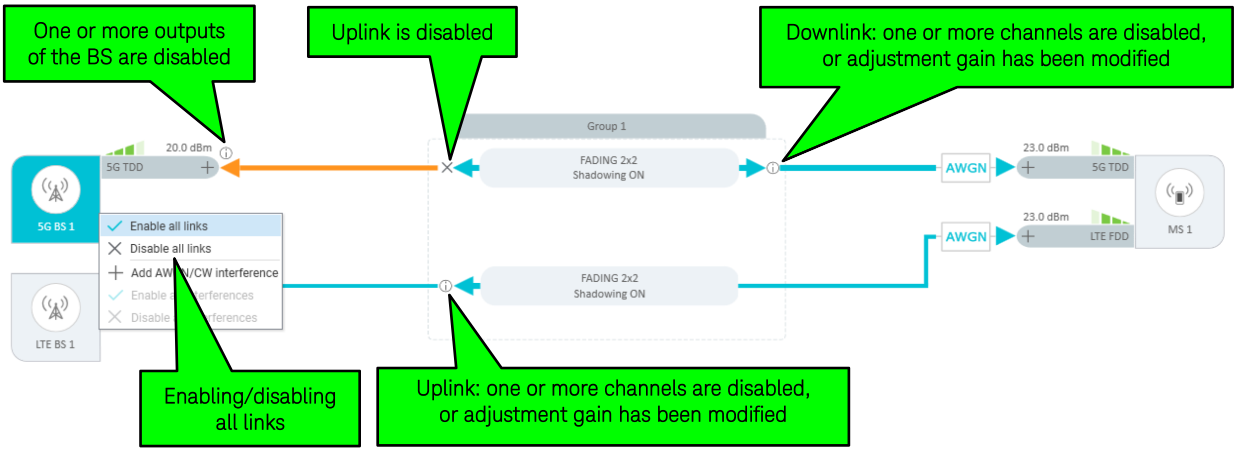


Figure 76 Indication of disabled links, channels, and outputs on emulation setup diagram

To disable/enable all the uplinks and downlinks of a BS or MS:

1. Right-click the device.
2. Select **Disable all links** or **Enable all links** from the list. When the link is disabled, the (Link disabled) icon is shown on the link next to the Fading element.

To disable/enable individual channels:

1. In the emulation settings pane, click the **Edit all settings** button to open the **All settings** dialog.
2. Filter the **All settings** list to show the “Channel enabled” parameter.
3. Disable/enable the channels you want. When one or more channels of a link are disabled (but not the entire link), the (Balance adjusted) icon is shown on the link next to the Fading element.

To disable/enable one or more outputs of a BS or MS:

1. In the emulation settings pane, click the **Edit all settings** button to open the **All settings** dialog.
2. Filter the **All settings** list to show the “Output enabled” parameter.

3. Disable/enable the outputs you want. When outputs are disabled, the ✕ (Outputs disabled) or ⓘ (Some of the outputs are disabled) icon is shown on the link next to the BS or MS.

4.3.3.3 Digital clipping

To avoid data overflow or signal clipping on digital domain, PROPSIM reserves necessary dynamics for digital calculation. However, if the option "High gain mode" is used or positive channel gain is added to models, it is possible that the dynamics reserved for calculation is not sufficient and clipping can occur.

Clipping is indicated by a warning icon and text “Digital clipping” in the FADING element in the emulation setup diagram, and the tooltip shows the per mill values (proportion of clipped samples in per mills), see Figure 77.

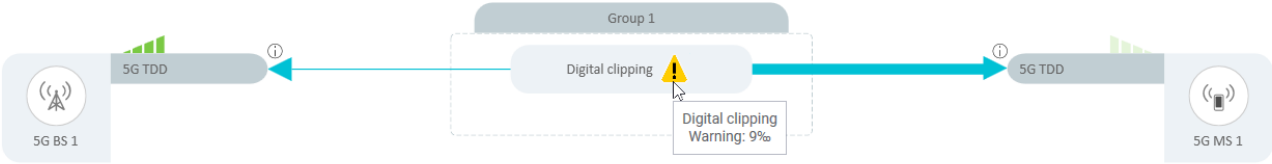


Figure 77 Digital clipping indication in emulation diagram

In addition, the beginning and ending of the clipping warning is shown in the **System log** dialog, as shown in Figure 78. To open the System log dialog, select **Log** in the navigation bar or click the **STATUS** button in the status bar.

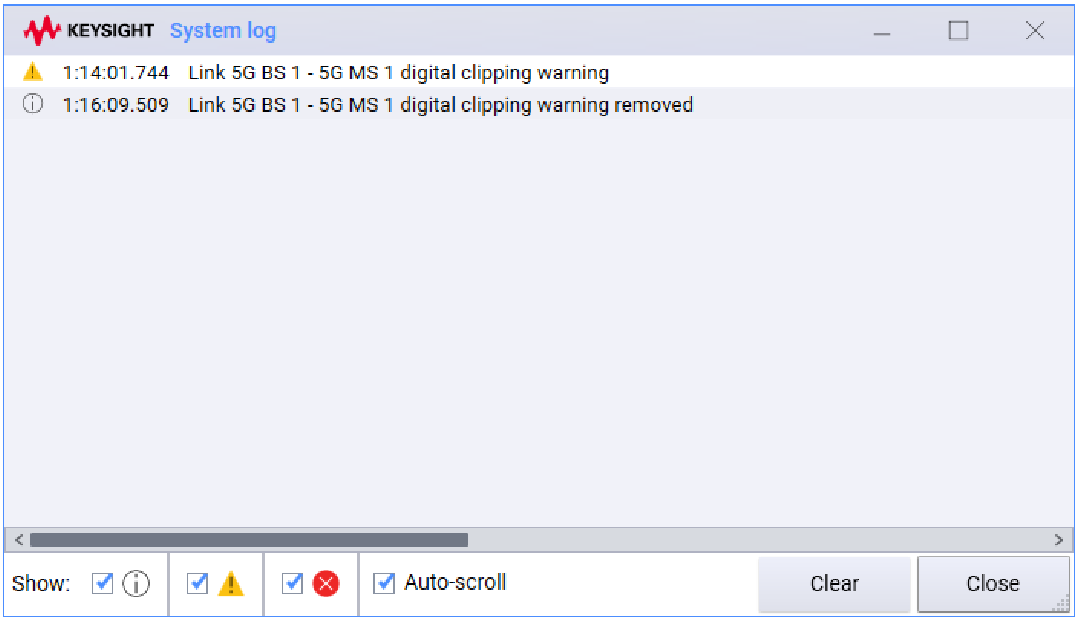


Figure 78 Digital clipping warning in system log

Note: Clipping indicator does not indicate the implications of the clipping for test system performance indicators (such as data throughput), just the relative share of the clipped samples in PROPSIM digital signal processing. Please monitor the test system performance for maximum tolerated digital clipping when applying the high gain mode in PROPSIM.

4.3.4 Base station/mobile station settings

To show the base station and mobile station settings, select the base station or mobile station on the emulation diagram.

The settings of each device affect all the antennas in that device (excluding the RF phase alignment setting). If needed, you can also define antenna-specific values in the **All settings** dialog, see section ☐.

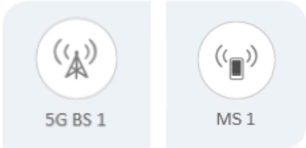


Figure 79 Base station and mobile station elements in emulation diagram

Figure 80 below shows an example of an LTE base station settings. The available settings for a mobile station are the same as for a base station.

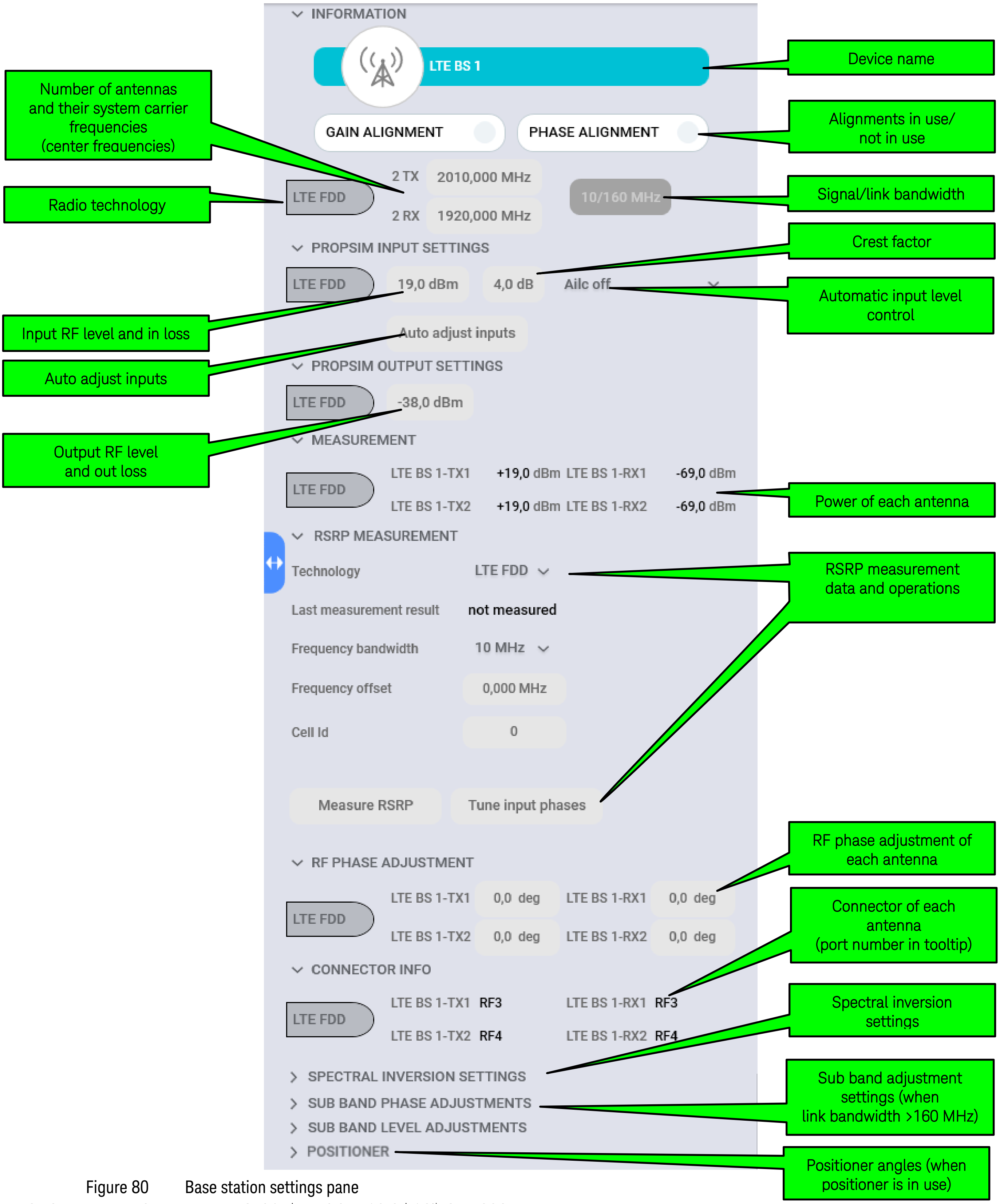


Figure 80 Base station settings pane

Device name

- Name of the base station or mobile station (editable in Scenario wizard)

Alignments in use/not in use

- Shows whether gain or phase alignment is in use for the BS or MS. For more information on auto alignment, see chapter 15.

TX frequency / RX frequency

- System carrier frequency for the link.
- For 5G, LTE, and WCDMA technologies, the system carrier frequency is also shown as a band and channel number in the tooltip.
- If frequency conversion is in use, this area shows the uplink/downlink input and output frequencies. See section 4.3.4.1

Bandwidth

- Link bandwidth
- With 5G/LTE technology: shows both 5G/LTE modulation bandwidth (e.g. 5G 100 MHz or LTE 10 MHz) and link bandwidth (e.g. 160 MHz)

Input RF level

- Defines the maximum RMS transmit power of the BS or MS without cables or external losses.

In loss

- Test setup loss between the transmitting device and PROPSIM input (external cables, attenuators and other equipment) in [dB]. When this value is measured and set correctly, input RF level defines the level in the beginning of the input cable – i.e. TX power of the transmitting device. Value in this field affects the limits of input RF level. Example: if attenuation in the cable between transmitting device and PROPSIM input is 3.6 dB, value 3.6 dB is used as In loss. Value can be also negative if external amplifiers are used.

Output RF level

- Defines the signal level seen by the device (BS or MS), after fading, cable losses and external components.
- This level is achieved when transmitter sends with its maximum TX power.

Out loss

- Test setup loss between the PROPSIM output and the receiving device (external cables, attenuators and other equipment) in [dB]. When this value is measured and set correctly, output level, interference level and “Level to DUT” define the level in the end of the output cable – i.e. the power level fed to the receiving device. Value in this field affects the limits of output level and interference level. Example: if attenuation in the cable between PROPSIM and the receiving device is 3.6 dB, value 3.6 dB is used as Out loss. Value can be also negative if external amplifiers are used.

Crest factor

- Crest factor indicates the difference between the average power level and the peak power in the input signal.

Automatic input level control (Ailc)

- When enabled, automatic input level control can operate in two modes: Prevent cut-off or AGC (automatic gain control). The modes are described below.

- Automatic input level control mode “Prevent cut-off”:
  - In “Prevent cut-off” mode, input signal is automatically attenuated if the signal level rises near to the cut-off level or if the cut-off occurs. Attenuation is automatically reduced and removed when input signal level returns under the cut-off level.
  - Cut-off warning is shown only if cut-off situation remains when all available internal attenuation is taken into use.
  - All inputs belonging to same MIMO group are measured and adjusted together to preserve MIMO balances.
  - As an example, in typical MS-BS test setup, MS can use high transmit power during link setup phase and reduce the power to nominal level after that. Using automatic input level control allows PROPSIM input level to be set to nominal MS transmit level and MS can still register to BS.
  - Example illustration of “Prevent cut-off” mode and the effect to path loss are shown in Figure 81 and Figure 82.

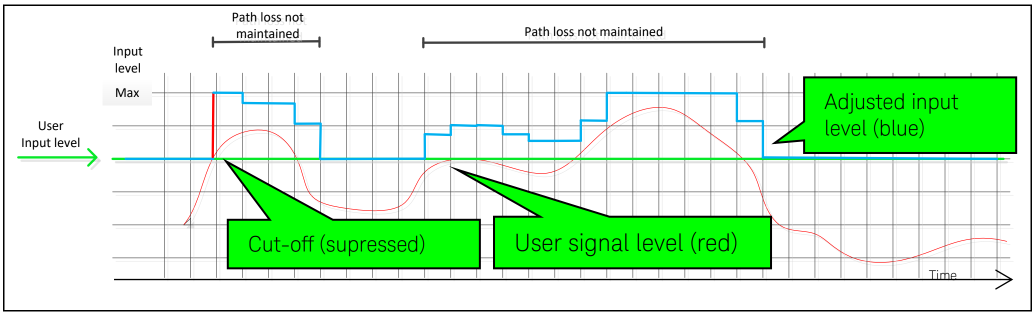


Figure 81 Illustration of automatic input level control operation in “Prevent cut-off” mode

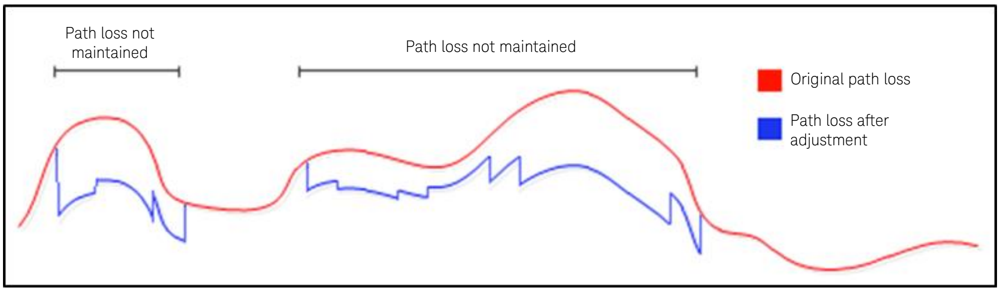


Figure 82 Path loss change due to signal cut-off prevention by automatic input level control in “Prevent cut-off” mode

- Automatic input level control mode “AGC”:
  - PROPSIM input gain changes from static mode to dynamic, so that the settings value follows the actual incoming signal power.
  - This means the input signal variation is compensated and the average output level remains constant (variating loss via PROPSIM).
  - All inputs belonging to same MIMO group are measured and adjusted together to preserve MIMO balances. The input with the highest measured power determines the AGC adjustment value for the MIMO group.
  - AGC operation is illustrated in Figure 83.
- Automatic input level control mode “AGC keep pathloss”:
  - Adjustment logic is similar to “AGC”, but input gain changes are compensated digitally to maintain pathloss via PROPSIM. Because digital amplification is not possible, user must set a suitable initial attenuation to link adjustment gain setting to allow maintaining the pathloss also when signal level at the PROPSIM input gets higher from its initial level.
  -

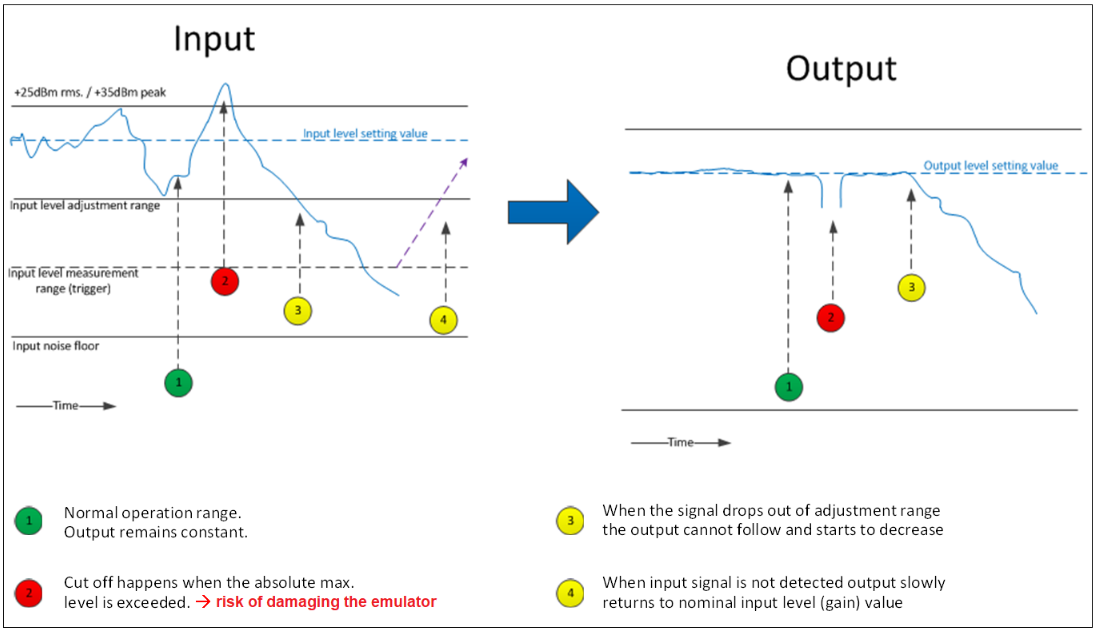


Figure 83 Illustration of automatic input level control operation in "AGC" mode

Auto adjust inputs

- PROPSIM measures and adjusts the input RF levels and crest factors

Measurement

- Device TX antenna power
  - Measured PROPSIM input power
- Device RX antenna power
  - Measured PROPSIM output power

RSRP measurement

- This settings pane area contains two LTE/5G specific operations: measuring of RSRP and tuning input phases according to incoming LTE/5G signals. Both operations are available when the device technology is either LTE or 5G. Required parameters are:
  - Frequency bandwidth:
    - LTE signal bandwidth (3, 5, 10, 15 or 20 MHz)
    - 5G signal bandwidth (20, 50 or 100 MHz)
  - Frequency offset: LTE/5G signal frequency offset compared to PROPSIM center frequency
  - Cell Id: Base station cell ID
  - Sub-carrier spacing (5G)
- Operation: Measure RSRP
  - If the signal with defined parameters is detected, measured RSRP appears to the RSRP measurement result field.
- Operation: Tune input phases
  - If the signal with defined parameters is detected, input phases are aligned with the primary input.
  - *Note: Phase tuning is supported with LTE base stations with 2 antennas and 5G base stations with 2, 4 or 8 antennas.*

RF Phase adjustment

- You can adjust the input/output phase with the slider or by entering a value to edit field. Adjustments are made immediately.
- Adjustments are hardware interface specific and changing the emulation will not affect them.

Connector info

- Shows the connector of each antenna on the PROPSIM front panel. Additionally, the tooltip shows the port number of the antenna.

Spectral inversion settings

- Spectral inversion setting for RX and TX.

Sub band phase and level adjustments

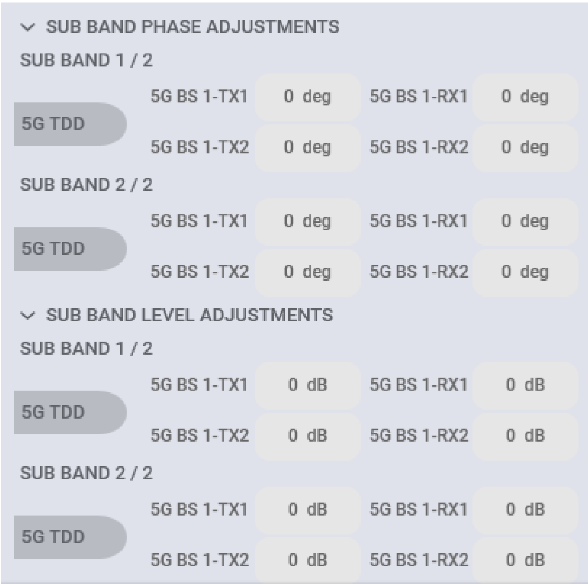


Figure 84 Sub band adjustment settings on BS7MS settings pane

- When the link bandwidth is greater than 160/200 MHz, these settings can be used to individually adjust each sub band's phase and gain.

Positioner

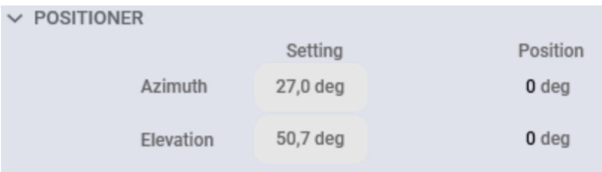


Figure 85 Positioner settings on BS/MS settings pane

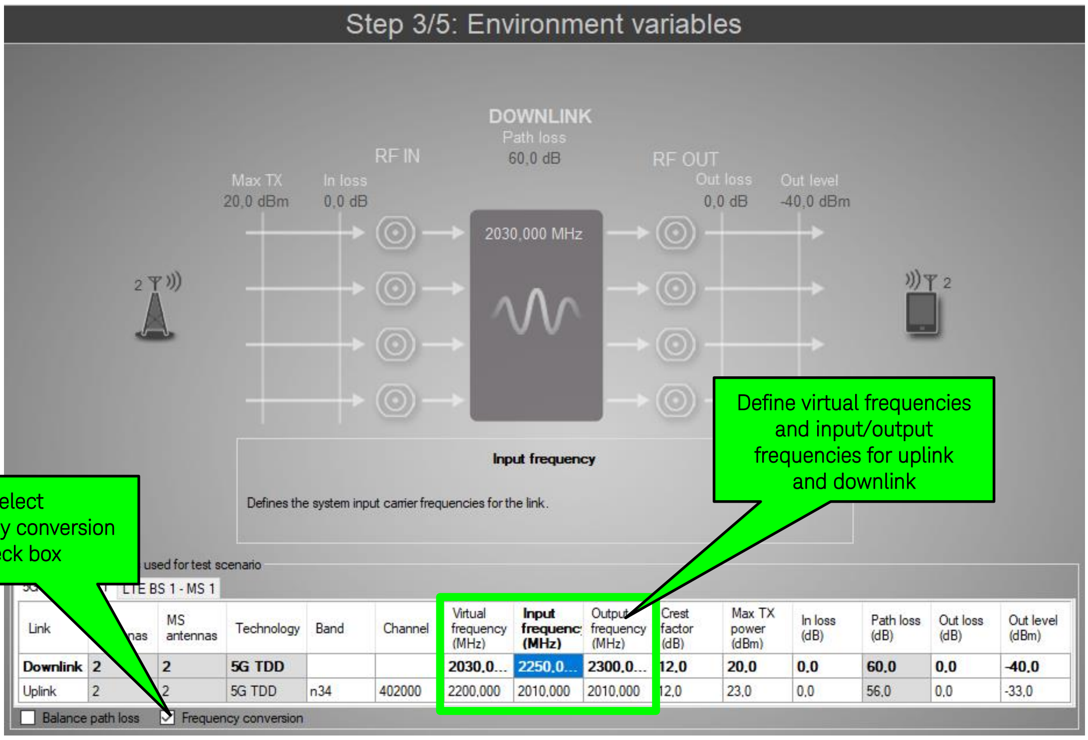
- Setting
  - You can move the positioner by adjusting the Azimuth and elevation angles in the **Setting** field.
  - PROPSIM can also read the positioner angles from an OTA orientation profile file created, for example, using GCM tools. In this case, the value in the **Setting** field is read-only.
- Position
  - Current Azimuth and elevation angle of the positioner (read-only).

4.3.4.1 Frequency conversion

The frequency conversion feature allows you to define the input and output frequencies separately for the uplink and the downlink.

By default, frequency conversion is disabled. To enable frequency conversion, select Configuration > Device Configuration in the Navigation bar, and select the Frequency conversion check box.

Frequency conversion in an emulation is defined in the Scenario Wizard, step 3/5 Environment variables:



Link	BS antennas	MS antennas	Technology	Band	Channel	Virtual frequency (MHz)	Input frequency (MHz)	Output frequency (MHz)	Crest factor (dB)	Max TX power (dBm)	In loss (dB)	Path loss (dB)	Out loss (dB)	Out level (dBm)
Downlink	2	2	5G TDD			2030,0...	2250,0...	2300,0...	12,0	20,0	0,0	60,0	0,0	-40,0
Uplink	2	2	5G TDD	n34	402000	2200,000	2010,000	2010,000	12,0	23,0	0,0	56,0	0,0	-33,0

Figure 86 Defining frequency conversion in Scenario Wizard

When frequency conversion is used, Virtual frequency of the link is used to calculate the relation between Doppler and mobile speed.

In the Emulation control view, MS and BS settings show that frequency conversion is in use. For example, with the input and output settings shown in Figure 86, the MS and BS settings appear as shown in Figure 87:

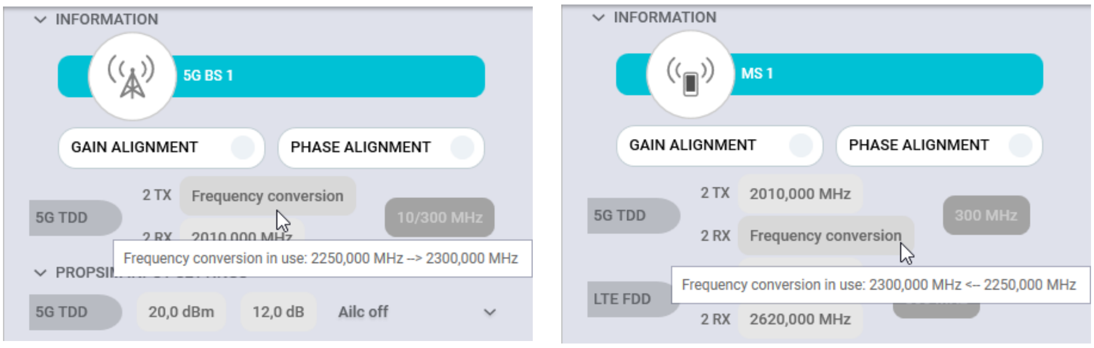


Figure 87 Frequency conversion shown in BS settings (left) and MS settings (right) in Emulation control view

If you change the frequency in BS/MS settings in the Emulation control view, the frequency changes only in the corresponding uplink or downlink.

4.3.5 Interference settings

Interference settings are shown when the interference element on the emulation diagram is selected.

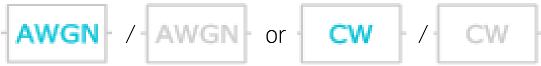


Figure 88 Interference element in emulation diagram. Blue icon = interference is enabled. Gray icon = interference is disabled.

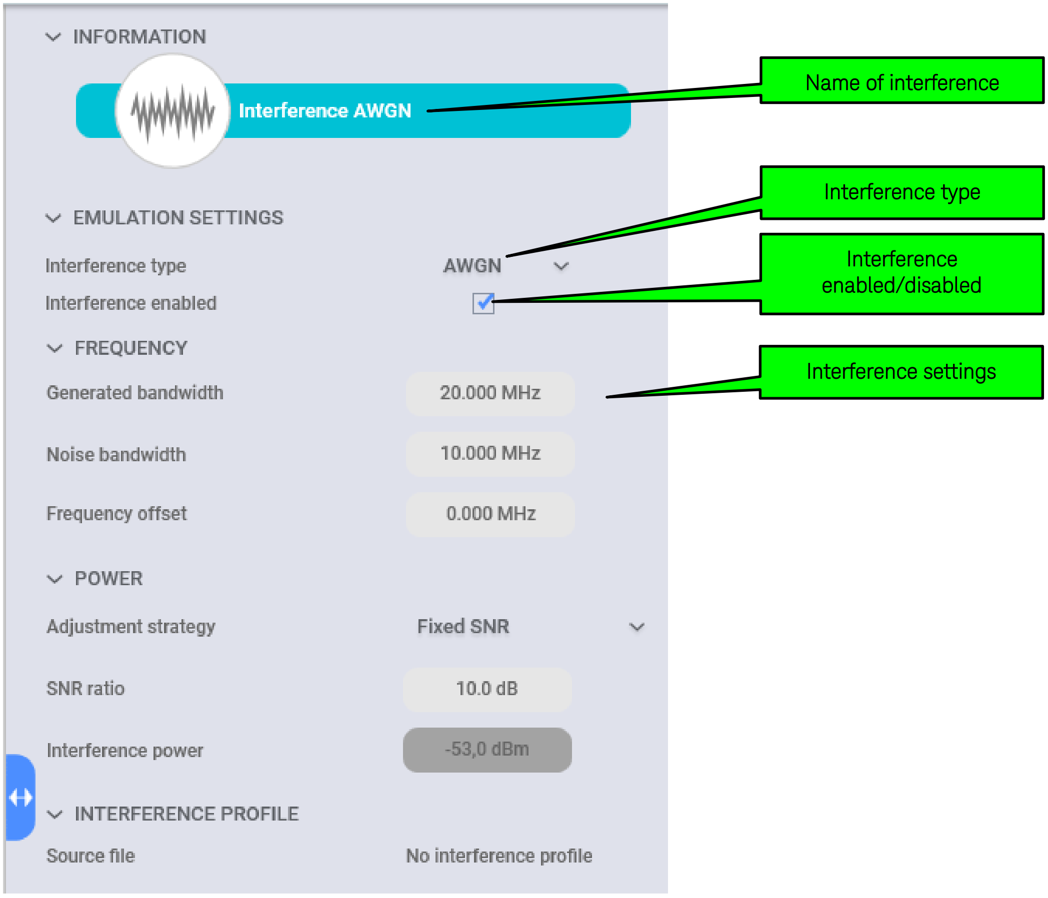


Figure 89 Interference settings pane

Name of interference

- Name of the interference (read-only)

Interference type

- Type of the interference: AWGN or CW
- Interference settings depend on the chosen interference type. See sections 4.3.5.1 AWGN interference and 4.3.5.2 CW interference.

Interference enabled/disabled

- Selection to enable or disable the selected interference. Interference status is indicated on the emulation diagram with color:
  - Blue interference icon: Interference is enabled.
  - Gray interference icon: Interference is disabled.

4.3.5.1 AWGN interference

AWGN interference is additive white gaussian noise, generated with user configurable bandwidth.

Settings

Frequency

- Generated bandwidth
  - User defines the generated noise bandwidth (15 kHz - [Emulation bandwidth]).
- Noise bandwidth
  - User defines the applied signal bandwidth (15 kHz - [Generated Noise Bandwidth]).
- Frequency offset
  - The available range depending on Emulation bandwidth and generated bandwidth ±([Emulation bandwidth] - [Generated Noise Bandwidth])/ 2.

Power

- Adjustment strategy
  - Fixed SNR
    - Ratio between signal and noise power is set by the user.
  - Fixed power and SNR
    - Noise is set by the user and kept at constant level, and SNR ratio is set by the user and the signal level adjusted accordingly.
  - Fixed power
    - Noise power level is set by the user.
- SNR ratio
  - Ratio of the strength of a signal to its background noise. Editable when the adjustment strategy is "Fixed SNR" or "Fixed power and SNR".
- Interference power
  - Output power level of the interference signal in dBm. Editable when the adjustment strategy is "Fixed power" or "Fixed power and SNR".

Interference profile

- Source file
  - User defined power profile file for interference. Uses the same file format as shadowing profiles (.shd). Only "Fixed power" mode is supported when using interference profile.

4.3.5.2 CW interference

CW interference is a carrier wave with user defined amplitude and frequency.

Settings

Frequency

- Frequency offset
  - The available range is ± Emulation Bandwidth / 2

Power

- Adjustment strategy
  - Fixed SNR
    - Ratio between signal and noise power is set by the user.
  - Fixed power
    - Noise power level is set by the user.
- SNR ratio

    - Ratio of the strength of a signal to its background noise. Editable when the adjustment strategy is "Fixed SNR".
- Interference power
    - Output power level of the interference signal in dBm. Editable when the adjustment strategy is "Fixed power".

Interference profile

- Source file
    - User defined power profile file for interference. Uses the same file format as shadowing profiles (.shd). Only "Fixed power" mode is supported when using interference profile.

4.3.6 Phase noise settings

Phase noise settings are shown when the phase noise element (PHN) on the emulation diagram is selected. Arrow icons under the phase noise element indicate the direction where the phase noise is applied.

Figure 90. Phase noise element in the emulation diagram.

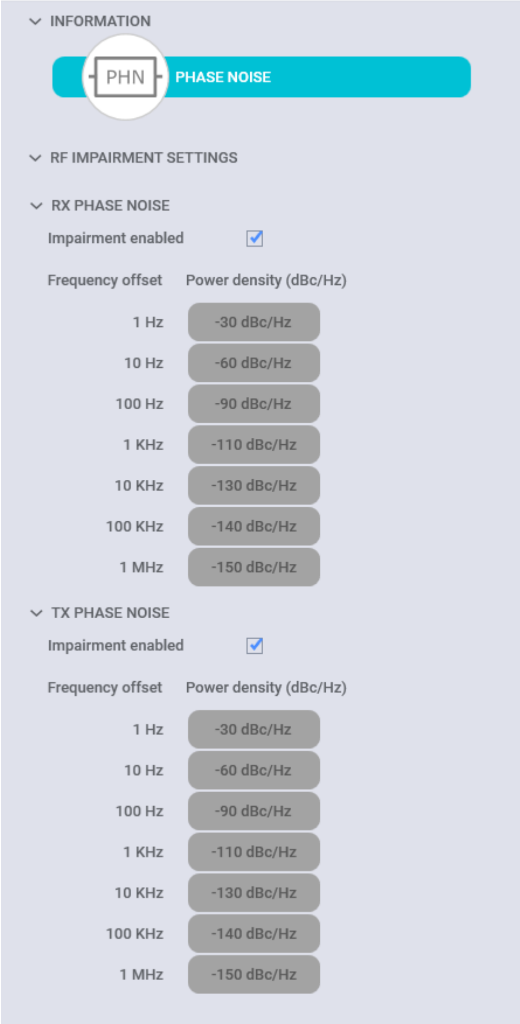


Figure 91 Phase noise settings pane

Settings

RX/TX Phase noise

- Impairment enabled
    - Phase noise enable / disable

- Frequency offset, Power density (dBc/Hz)
    - Phase noise profile (read only, can be edited in Scenario Wizard)

4.3.7 View and edit all settings

To open a list of all settings, click the **Edit all settings** button in the emulation settings pane.

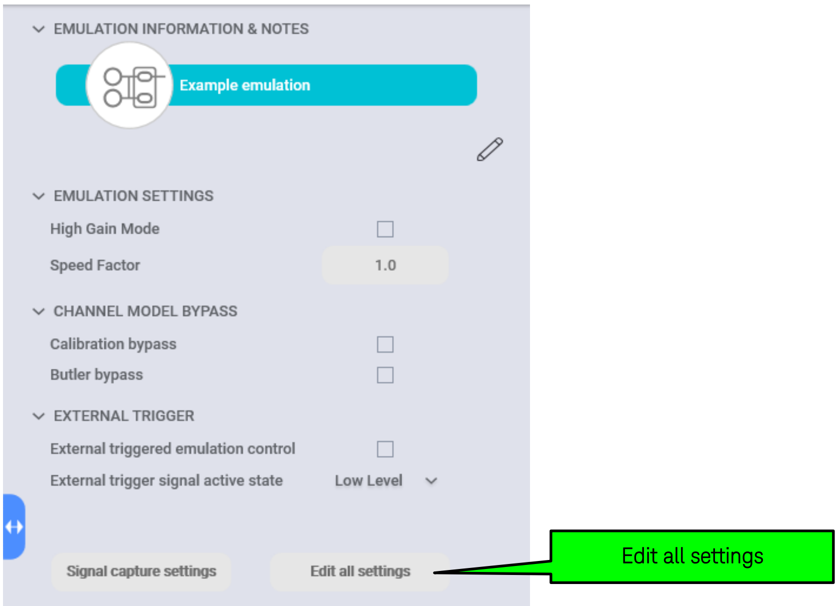
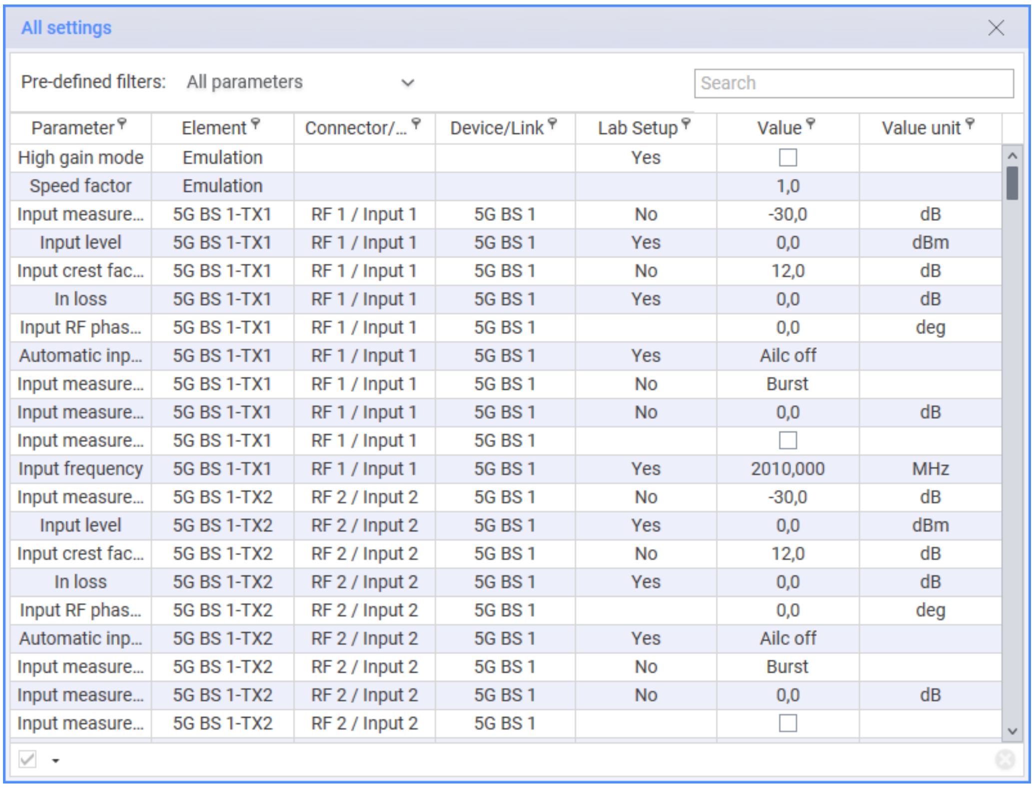


Figure 92 Opening list of all settings



Parameter	Element	Connector/...	Device/Link	Lab Setup	Value	Value unit
High gain mode	Emulation			Yes	☐	
Speed factor	Emulation				1,0	
Input measure...	5G BS 1-TX1	RF 1 / Input 1	5G BS 1	No	-30,0	dB
Input level	5G BS 1-TX1	RF 1 / Input 1	5G BS 1	Yes	0,0	dBm
Input crest fac...	5G BS 1-TX1	RF 1 / Input 1	5G BS 1	No	12,0	dB
In loss	5G BS 1-TX1	RF 1 / Input 1	5G BS 1	Yes	0,0	dB
Input RF phas...	5G BS 1-TX1	RF 1 / Input 1	5G BS 1		0,0	deg
Automatic inp...	5G BS 1-TX1	RF 1 / Input 1	5G BS 1	Yes	Ailc off	
Input measure...	5G BS 1-TX1	RF 1 / Input 1	5G BS 1	No	Burst	
Input measure...	5G BS 1-TX1	RF 1 / Input 1	5G BS 1	No	0,0	dB
Input measure...	5G BS 1-TX1	RF 1 / Input 1	5G BS 1		☐	
Input frequency	5G BS 1-TX1	RF 1 / Input 1	5G BS 1	Yes	2010,000	MHz
Input measure...	5G BS 1-TX2	RF 2 / Input 2	5G BS 1	No	-30,0	dB
Input level	5G BS 1-TX2	RF 2 / Input 2	5G BS 1	Yes	0,0	dBm
Input crest fac...	5G BS 1-TX2	RF 2 / Input 2	5G BS 1	No	12,0	dB
In loss	5G BS 1-TX2	RF 2 / Input 2	5G BS 1	Yes	0,0	dB
Input RF phas...	5G BS 1-TX2	RF 2 / Input 2	5G BS 1		0,0	deg
Automatic inp...	5G BS 1-TX2	RF 2 / Input 2	5G BS 1	Yes	Ailc off	
Input measure...	5G BS 1-TX2	RF 2 / Input 2	5G BS 1	No	Burst	
Input measure...	5G BS 1-TX2	RF 2 / Input 2	5G BS 1	No	0,0	dB
Input measure...	5G BS 1-TX2	RF 2 / Input 2	5G BS 1		☐	

Figure 93 All settings dialog

The parameter list contains the following columns:

- **Parameter:** All available parameters for all emulation elements
- **Element:** Base station transmitters and receivers, mobile station transmitters and receivers, individual channels, and interferences to transmitters and receivers
- **Connector/Port:** Number of the connector and port to which the parameter is associated.
- **Device/Link:** Base stations, mobile stations, links from base stations to mobile stations (downlinks), links from mobile stations to base stations (uplinks), interferences to base stations or mobile stations
- **Lab setup**: Shows whether the parameter is controlled by the current lab setup. This column is visible only when a lab setup is in use.
- **Value:** Editable or read-only values of the parameters. The read-only values have a grey background in the settings list. Possible limits to the values are shown as tooltips.
- **Value unit:** Possible unit of the parameter value

The settings list can be sorted and filtered:

- To sort the list, click on one of the column headings.
- To use one of the pre-defined filters, click the **Pre-defined filters** drop down list.
- To filter the list, click the filter icon and select the item you want to use to filter the list. The selected filter appears at the bottom of the dialog.
- To clear the filter, unselect the filter at the bottom of the dialog.
- Columns can be re-ordered by dragging from the column header.

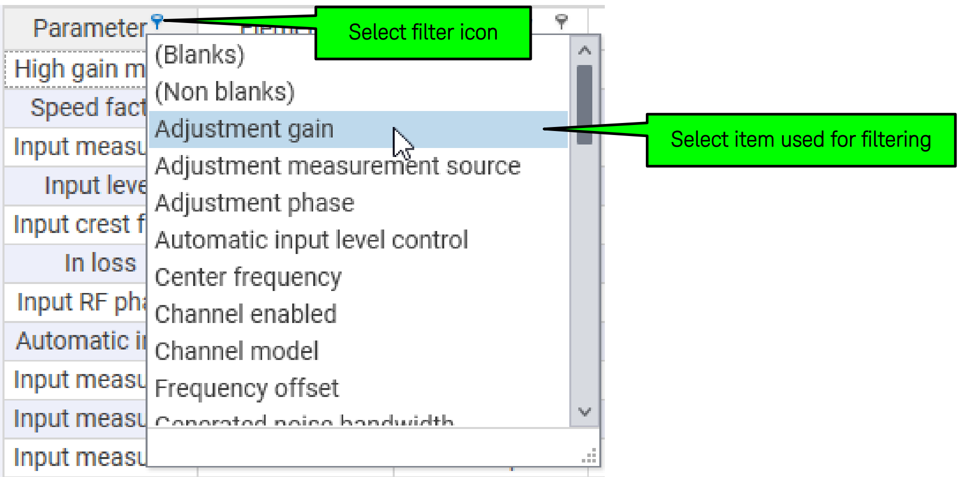


Figure 94 Filtering the settings list

4.3.7.1 Changing multiple values at once

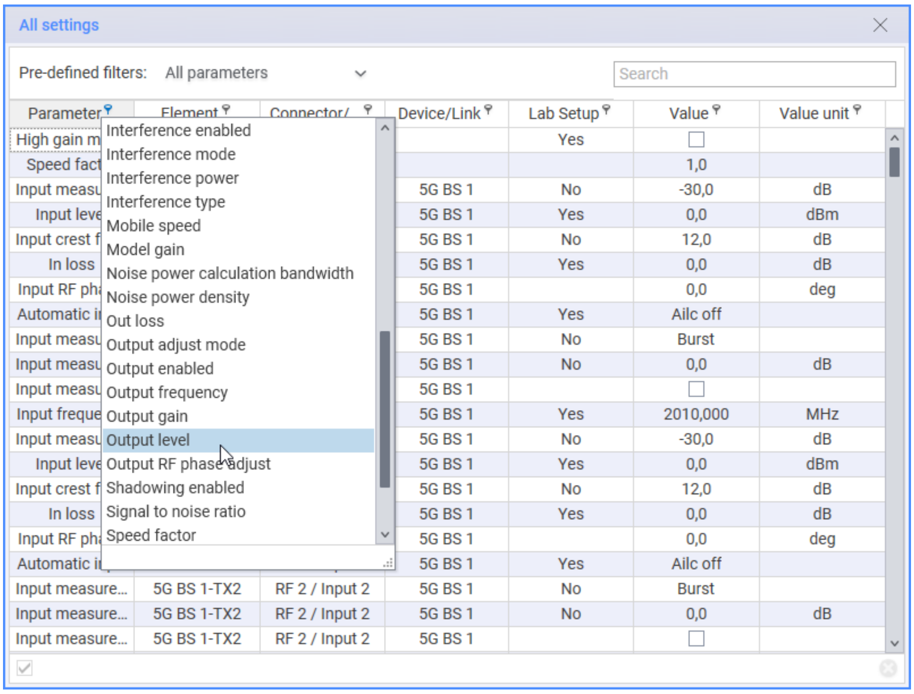
In the **All settings** dialog, it is also possible to change the values of multiple parameters at once.

Note: This is not possible with the values that are selected from a drop-down list.

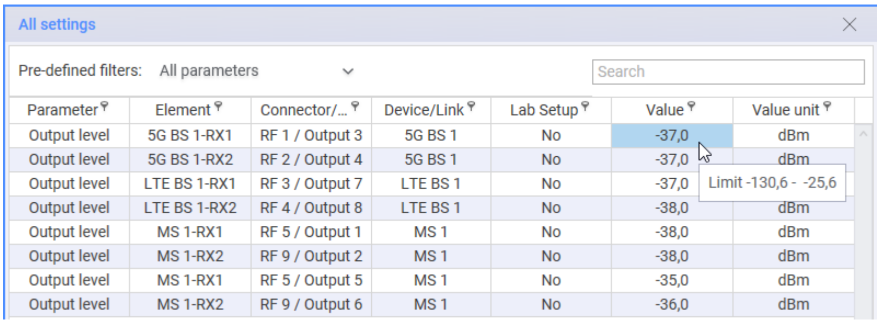
The following example shows how to change the output level of all the base stations and mobile stations in the emulation that is open:

4. Make sure none of the elements are selected in the emulation setup diagram.
5. In the emulation settings pane, click **Edit all settings**.

6. Filter the **Parameter** list by the parameter “Output level”.



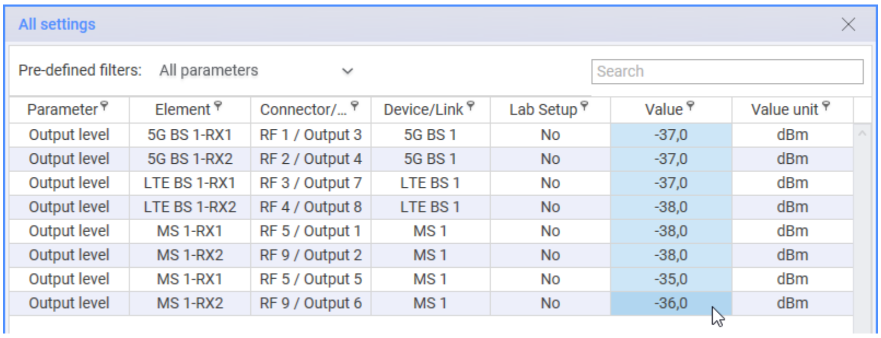
7. Select topmost value in the value range you want to change.



Parameter	Element	Connector/...	Device/Link	Lab Setup	Value	Value unit
Output level	5G BS 1-RX1	RF 1 / Output 3	5G BS 1	No	-37,0	dBm
Output level	5G BS 1-RX2	RF 2 / Output 4	5G BS 1	No	-37,0	dBm
Output level	LTE BS 1-RX1	RF 3 / Output 7	LTE BS 1	No	-37,0	
Output level	LTE BS 1-RX2	RF 4 / Output 8	LTE BS 1	No	-38,0	dBm
Output level	MS 1-RX1	RF 5 / Output 1	MS 1	No	-38,0	dBm
Output level	MS 1-RX2	RF 9 / Output 2	MS 1	No	-38,0	dBm
Output level	MS 1-RX1	RF 5 / Output 5	MS 1	No	-35,0	dBm
Output level	MS 1-RX2	RF 9 / Output 6	MS 1	No	-36,0	dBm

8. Press the Shift key continuously.

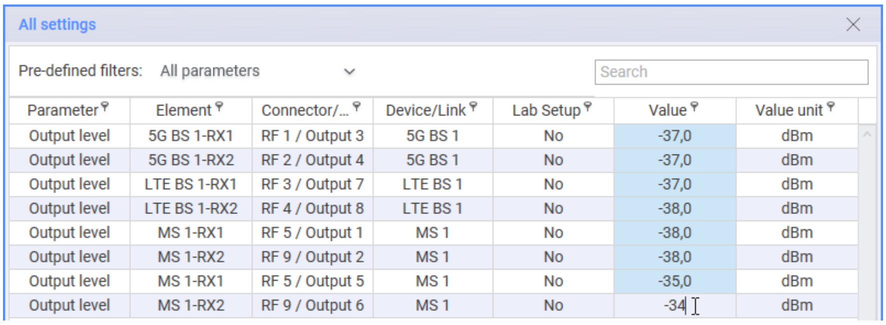
9. Select the bottom-most value in the value range you want to change. This will select all the values in the value range.



Parameter	Element	Connector/...	Device/Link	Lab Setup	Value	Value unit
Output level	5G BS 1-RX1	RF 1 / Output 3	5G BS 1	No	-37,0	dBm
Output level	5G BS 1-RX2	RF 2 / Output 4	5G BS 1	No	-37,0	dBm
Output level	LTE BS 1-RX1	RF 3 / Output 7	LTE BS 1	No	-37,0	dBm
Output level	LTE BS 1-RX2	RF 4 / Output 8	LTE BS 1	No	-38,0	dBm
Output level	MS 1-RX1	RF 5 / Output 1	MS 1	No	-38,0	dBm
Output level	MS 1-RX2	RF 9 / Output 2	MS 1	No	-38,0	dBm
Output level	MS 1-RX1	RF 5 / Output 5	MS 1	No	-35,0	dBm
Output level	MS 1-RX2	RF 9 / Output 6	MS 1	No	-36,0	dBm

Note: Alternatively, to steps 4–6, you can select the value range by clicking and dragging the mouse cursor over the range. You can also select/unselect multiple separate values by keeping the Ctrl key pressed and clicking the values.

10. Type in the new value.

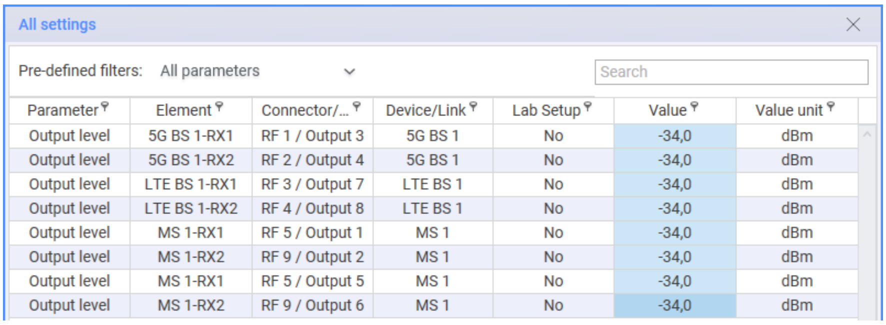


All settings

Pre-defined filters: All parameters | Search

Parameter	Element	Connector/...	Device/Link	Lab Setup	Value	Value unit
Output level	5G BS 1-RX1	RF 1 / Output 3	5G BS 1	No	-37,0	dBm
Output level	5G BS 1-RX2	RF 2 / Output 4	5G BS 1	No	-37,0	dBm
Output level	LTE BS 1-RX1	RF 3 / Output 7	LTE BS 1	No	-37,0	dBm
Output level	LTE BS 1-RX2	RF 4 / Output 8	LTE BS 1	No	-38,0	dBm
Output level	MS 1-RX1	RF 5 / Output 1	MS 1	No	-38,0	dBm
Output level	MS 1-RX2	RF 9 / Output 2	MS 1	No	-38,0	dBm
Output level	MS 1-RX1	RF 5 / Output 5	MS 1	No	-35,0	dBm
Output level	MS 1-RX2	RF 9 / Output 6	MS 1	No	-34	dBm

11. Press Enter. The new value is automatically copied to all the selected value fields.
Note: If the new value is lower than the minimum limit of a parameter, the value of that parameter is changed to the minimum limit value. Similarly, if the new value is higher than the maximum limit of a parameter, the value of that parameter is changed to the maximum limit value.



All settings

Pre-defined filters: All parameters | Search

Parameter	Element	Connector/...	Device/Link	Lab Setup	Value	Value unit
Output level	5G BS 1-RX1	RF 1 / Output 3	5G BS 1	No	-34,0	dBm
Output level	5G BS 1-RX2	RF 2 / Output 4	5G BS 1	No	-34,0	dBm
Output level	LTE BS 1-RX1	RF 3 / Output 7	LTE BS 1	No	-34,0	dBm
Output level	LTE BS 1-RX2	RF 4 / Output 8	LTE BS 1	No	-34,0	dBm
Output level	MS 1-RX1	RF 5 / Output 1	MS 1	No	-34,0	dBm
Output level	MS 1-RX2	RF 9 / Output 2	MS 1	No	-34,0	dBm
Output level	MS 1-RX1	RF 5 / Output 5	MS 1	No	-34,0	dBm
Output level	MS 1-RX2	RF 9 / Output 6	MS 1	No	-34,0	dBm

4.3.7.2 Copying values from external table

You can copy parameter values into the **All settings** list from an external table such as a spreadsheet or text editor table.

To copy values into the **All settings** list:

1. Make sure the values you want to copy are in the same order as the parameters in the **All settings list**.
Note: The values must be in a continuous list.
2. Select and copy the values from the external table.
3. Select the corresponding values in the **All settings** list.
4. Paste the values by pressing Ctrl+C or by right-clicking and selecting **Paste**.
Note: If the new value is lower than the minimum limit of a parameter, the value of that parameter is changed to the minimum limit value. Similarly, if the new value is higher than the maximum limit of a parameter, the value of that parameter is changed to the maximum limit value.

4.3.7.3 All settings list

Note: Some of the parameters listed in Table 10 are shown only if the relevant feature is used in the emulation. For example, the parameters "Interference enabled", "Interference mode", and "Interference type" are shown only if interference has been added to a BS or MS in the Scenario Wizard.

Table 10 List of all settings

Parameter	Description
Adjustment gain	Imbalance adjustment value, used to adjust the gain of separate channels.
Adjustment measurement source	Selects which inputs of the link are taken into account when adjusting the interference power in fixed SNR mode. Selection options: • All inputs (all inputs of the link's opposite end are applied) • Single input (only the first input of the link's opposite end is applied)
Adjustment phase	Imbalance adjustment value, used to adjust the phase of separate channels.

Parameter	Description
Automatic input level control (Ailc)	When enabled, automatic input level control can operate in two modes: Prevent cut-off or AGC (automatic gain control). See section 4.3.4 Base station/mobile station settings - Automatic input level control (Ailc).
Center frequency	Center frequency of the channel.
C/I ratio	Carrier to Interference ratio (CW interference).
Channel enabled	Selection whether the channel is enabled or not.
Channel model	File name of the channel model in use.
Frequency offset	In AWGN interference: The available range depending on Emulation bandwidth and generated bandwidth ±([Emulation bandwidth] - [Generated Noise Bandwidth])/ 2. See section 4.3.5.1 AWGN interference. In CW interference: The available range depending on bandwidth ±([Emulation bandwidth])/ 2. See section 4.3.5.2 CW interference.
Generated noise bandwidth	Noise bandwidth in AWGN interference. See section 4.3.5.1 AWGN interference.
High gain mode	Enabling this feature scales signal up +5 dB digitally. See section 4.3.2 Emulation settings – High Gain Mode.
In loss	Test setup loss between the transmitting device and PROPSIM input (external cables, attenuators and other equipment) in [dB]. See section 4.3.4 Base station/mobile station settings – In loss.
Input crest factor	Difference between the average power level and the peak power in the PROPSIM input signal. See section 4.3.4 Base station/mobile station settings – Crest factor.
Input frequency	PROPSIM input frequency for the device. If the up/down conversion feature is enabled, Input frequency and Output frequency settings can be used to adjust input and output frequencies separately to perform up/down conversion.
Input level	RF input level setting for the device (PROPSIM input).
Input measurement burst trigger level	PROPSIM input power measurement trigger level for burst measurement mode.
Input measurement freeze	Measurement taken from an input can be “frozen” as the last measured value or the measurements can be continued.
Input measurement mode	Selection of measurement mode: • Disabled: Measurement of selected input is disabled completely. • Continuous: Continuous signal measurement. Offset setting is applied to measurement, and result can be frozen. • Burst: Measurement is done during duty period of signal. Offset and freeze are available.
Input measurement offset	Offset value for measurement fine-tuning. The value set here is directly added to measurement result, and also affects C/I setting.
Input RF phase adjust	Adjustment of PROPSIM input RF phase. See section 4.3.4 Base station/mobile station settings – RP Phase adjustment.

Parameter	Description
Interference enabled	Selection whether the interference is enabled or not.
Interference mode	Selection of interference adjustment strategy: • Fixed power • Fixed SNR • Fixed power and SNR (AWGN interference type only) See sections 4.3.5.1 AWGN interference and 4.3.5.2 CW interference.
Interference power	Interference power settings. In case of AWGN, interference power is defined in the bandwidth defined by “Noise power calculation bandwidth”
Interference type	Selection of interference type: • AWGN • CW See sections 4.3.5.1 AWGN interference and 4.3.5.2 CW interference.
Link enabled	Enables / disables signal of the specified link.
Mobile speed	Mobile speed affects the CIR update rate. See section 4.3.3 Link settings – Mobile speed.
Model gain	Average gain of the channel model over the emulation bandwidth. ***Note:*** *There is always some variation due to the interaction of the input signal and the frequency selectivity of a channel model.*
Noise power calculation bandwidth	Bandwidth for AWGN power calculation.
Noise power density	AWGN power density.
Out loss	Test setup loss between the PROPSIM output and the receiving device (external cables, attenuators, and other equipment) in [dB]. See section 4.3.4 Base station/mobile station settings – Out loss.
Output adjust mode	Selection whether output gain or level is used for controlling PROPSIM output power.
Output enabled	Shows whether the output is enabled or not.
Output frequency	PROPSIM output frequency for the device. If the up/down conversion feature is enabled, Input frequency and Output frequency settings can be used to adjust input and output frequencies separately to perform up/down conversion.
Output gain	PROPSIM output gain for the device. ***Note:*** *If you change this value, the Output adjust mode is automatically changed to “Gain” for the corresponding element.*
Output level	Setting to define PROPSIM output gain for the device. ***Note:*** *If you change this value, the Output adjust mode is automatically changed to “Level” for the corresponding element.*
Output RF phase adjust	Adjustment of output RF phase. See section 4.3.4 Base station/mobile station settings – RP Phase adjustment.
Phase imbalance	Imbalance adjustment value, used to adjust the phase of separate channels.
Shadowing enabled	Selection whether shadowing is enabled or not. In case of output (analog) shadowing this parameter can be changed in runtime and emulation rebuild is not needed.
Shadowing model	File name of the shadowing model in use.

Parameter	Description
Signal to noise ratio	Signal to noise ratio of the AWGN interference.
Speed factor	Multiplication factor for increasing the emulation running speed from the original. See section 4.3.2 Emulation settings – Speed factor.
Total channel gain	Total channel gain from input to output.
Virtual center frequency	Used for calculating fading parameters (i.e. mobile speed, Doppler) in up/down conversion scenarios where RX and TX frequencies of the links are not equal or if application RF frequency is different from PROPSIM RF frequency (application IF frequency).
Virtual center frequency in use	Selection whether virtual center frequency is enabled or not.

5 DATA VIEWS

The PROPSIM data views offer different kinds of data about the emulation that is open or running:

- Graph with measured level, shadowing, and interference curves of the links (see 5.2)
- Live numeric data of antenna input/output powers and link Dopplers (see 5.3)
- Connectors on the emulator front panel (see 5.4)
- Channel impulse response (CIR, see 5.5)
- ATE LAN monitor (see 5.6)
- ASO Graph and ASO Line graph (see 5.7)
- Port RF level graph (see 5.8)
- GCM View (Map view for GCM emulations)

To open the Data views window, click the **Data view** button on the top right corner of the PROPSIM software.

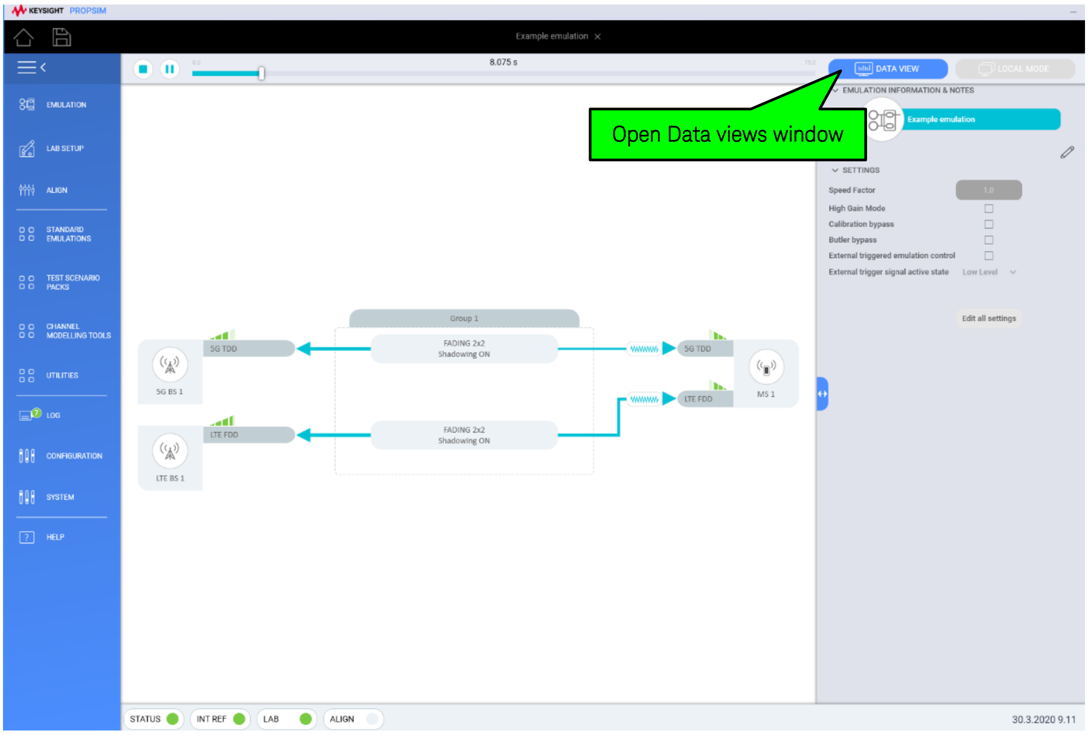


Figure 95 Opening Data view

If your PROPSIM software is installed on a computer with multiple displays, by default, the Data views window opens on the secondary display.

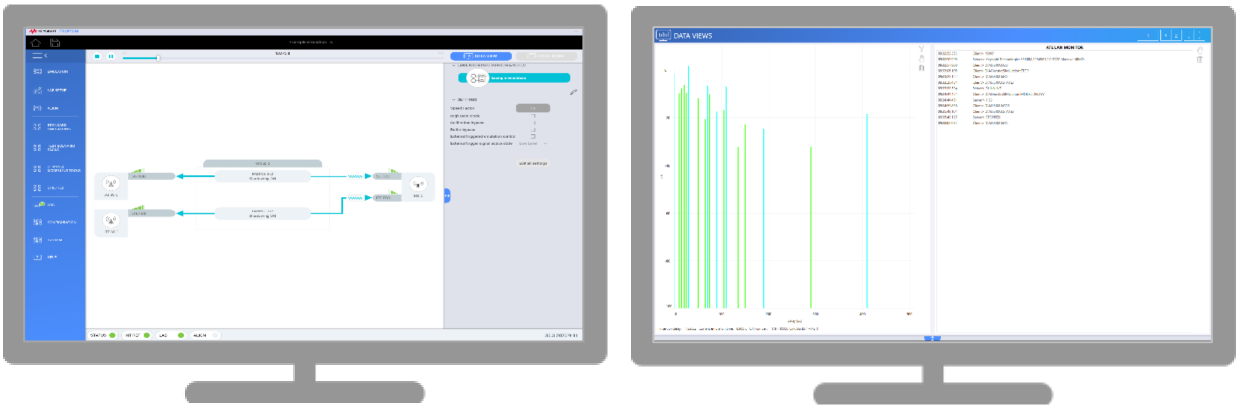


Figure 96 Emulation on main display and Data views window on secondary display

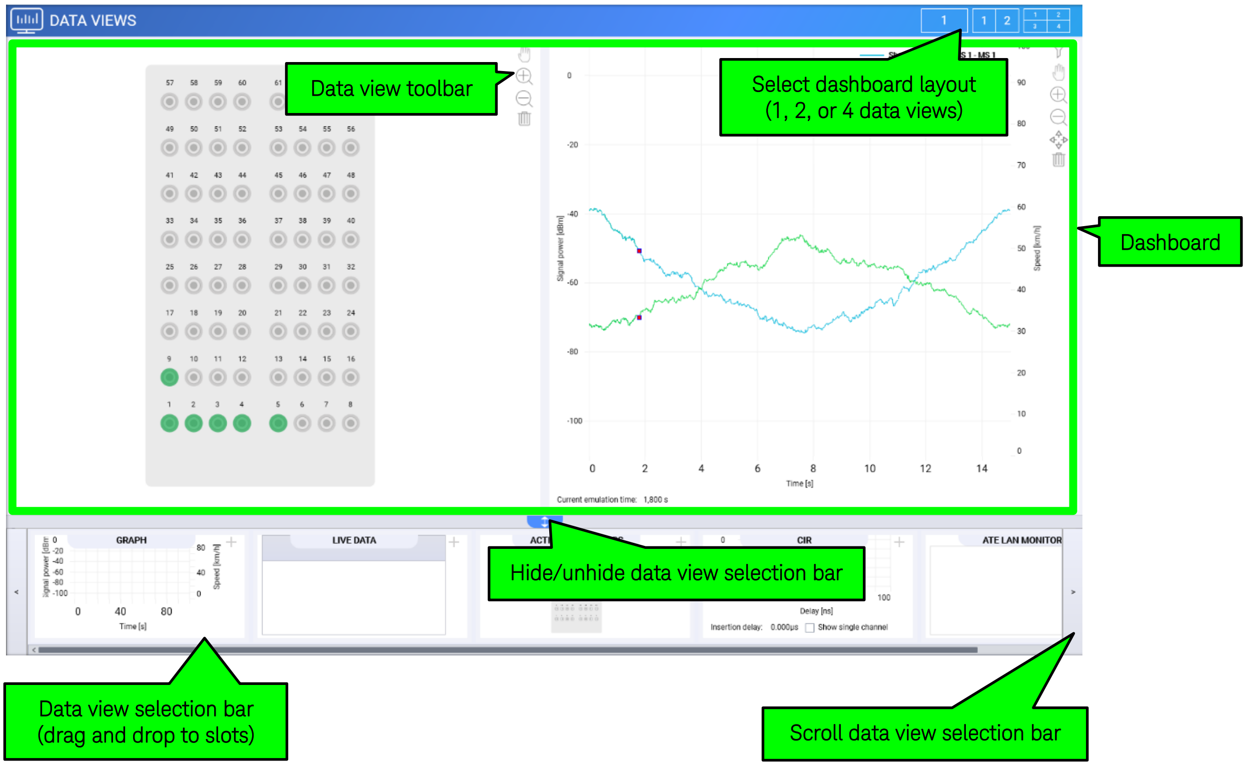


Figure 97 Data views window

5.1 Modifying data view dashboard

Note: When you close the emulation, the data view dashboard is stored for the next session, but the filters in each individual data view are cleared.

The dashboard layout options include dashboards with one, two, or four data views. Select the layout in the top right corner of the Data views window.

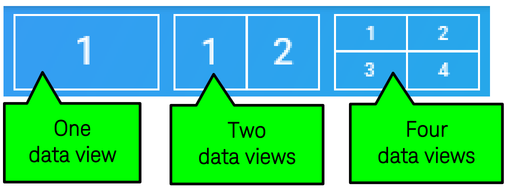


Figure 98 Selecting dashboard layout

To add data views on the dashboard, drag and drop the data views from the data view selection bar into the slot you want. You can add data views into empty slots or into slots that are already populated with a data view, replacing the existing data view with another one. You can also add the same data view to multiple slots in the dashboard and use the data view filters to show the same data about different elements in the emulation (for example, to show uplink curves in one graph view and downlink curves in another graph view).

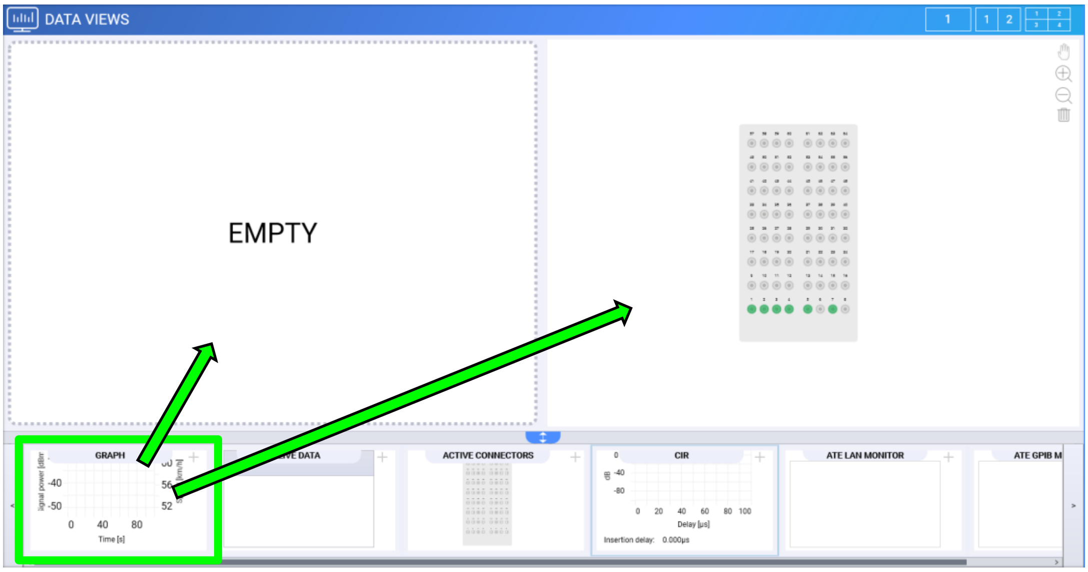


Figure 99 Adding data view on dashboard (drag-and-drop)

To move a data view from one slot to another slot, click and hold the (Drag) button and drag the data view to the slot you want.

To remove a data view from the dashboard, click the (Delete) button in the toolbar of that data view.

5.2 Graph view

The Graph view shows the curves for measured level, shadowing, and interference of the links in the emulation.

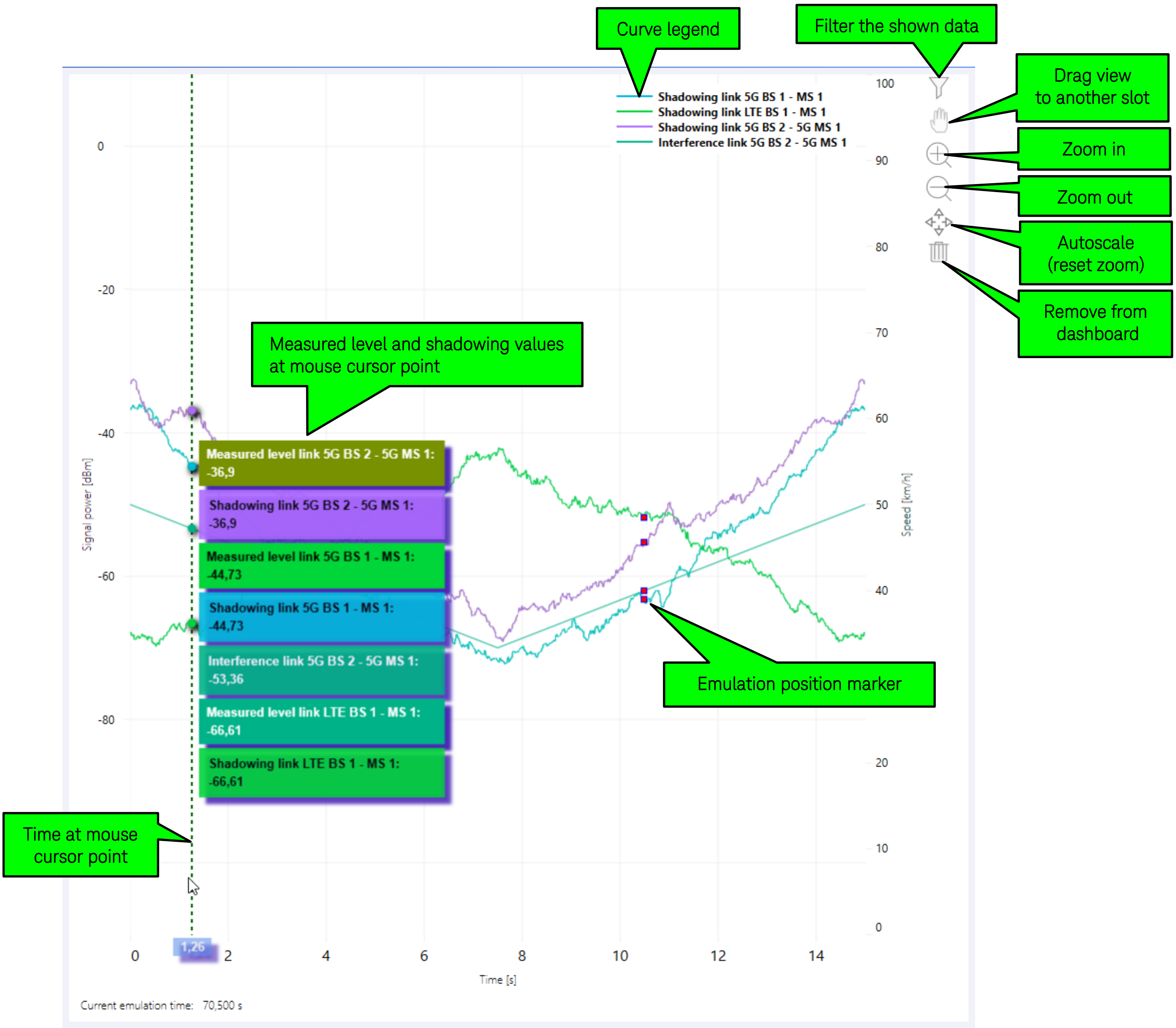


Figure 100 Graph data view with downlink curves

To select which links are shown on the graph, click the (Filter) button on the data view toolbar.

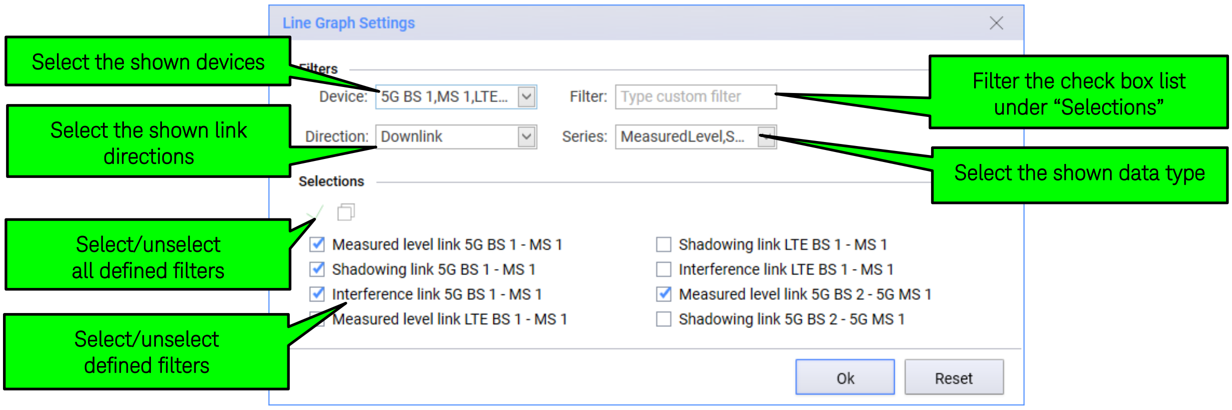


Figure 101 Line Graph Settings dialog

If your emulation contains several base stations and mobile stations, you can use two or more graph views to fit the curves more comfortably on the graph. For example, in Figure 102 below, one view shows the downlink curves, and the other view shows the uplink curves.

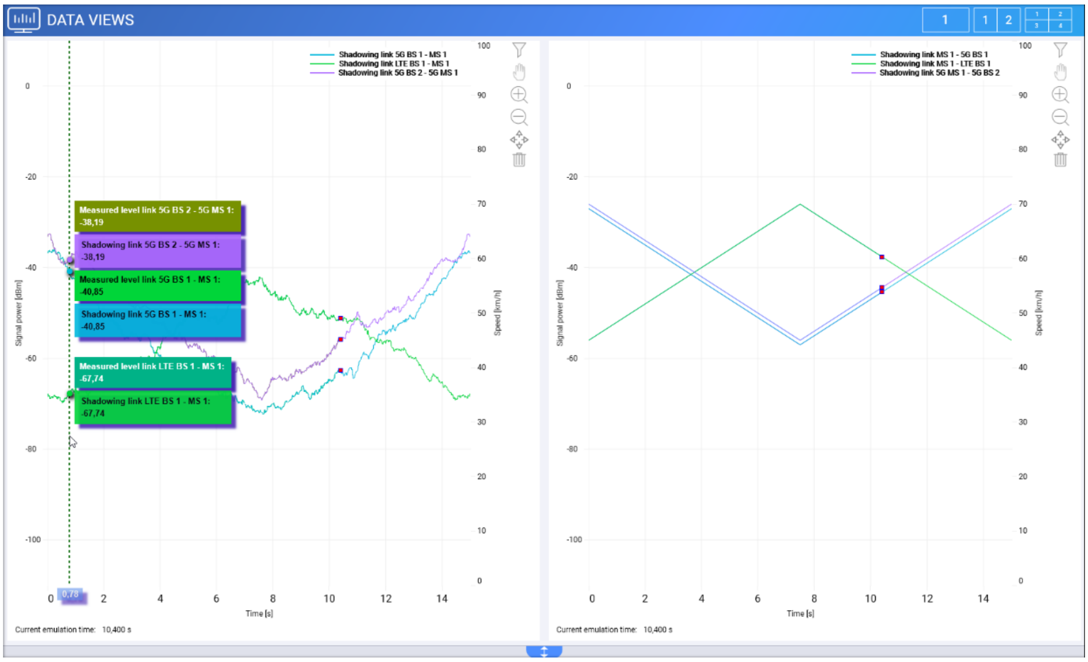


Figure 102 Downlink and uplink curves separated on two graph views using data view filters

5.3 Live data view

The live data view shows the input and output powers of antennas and the Dopplers and RSRP (LTE only) of links. You can also log the data to a .csv file.

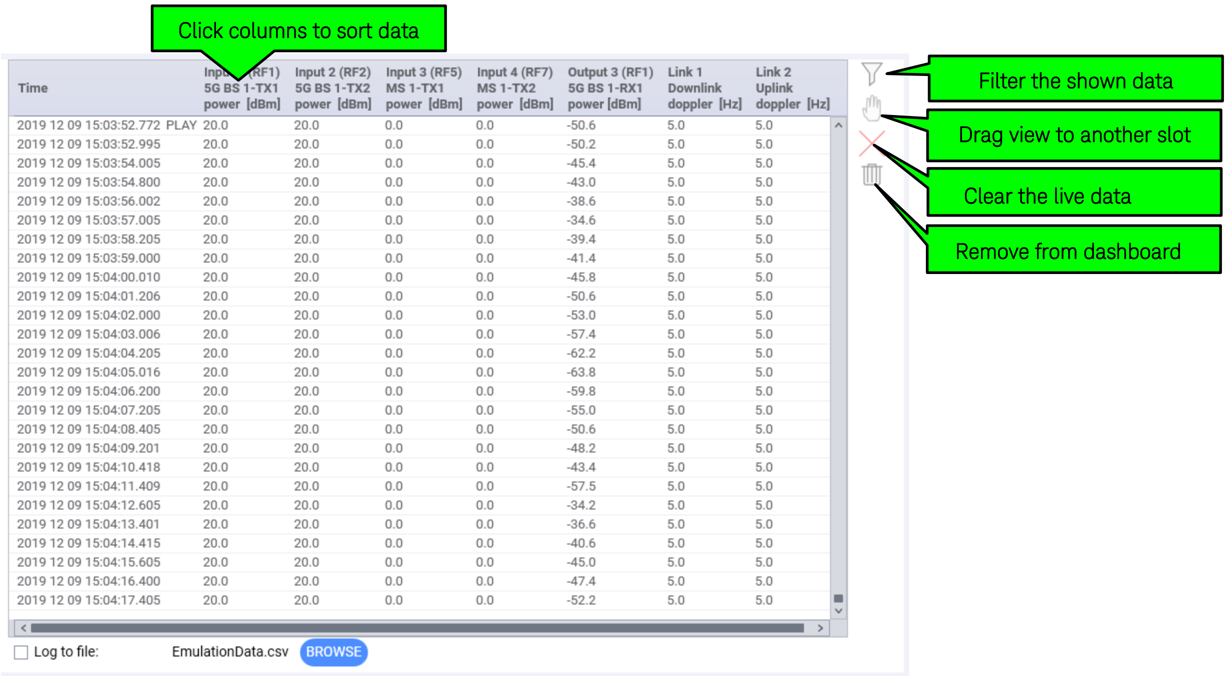


Figure 103 Live data view

To select which data is shown in the live data view and how often the view is updated, click the (Filter) button in the data view toolbar.

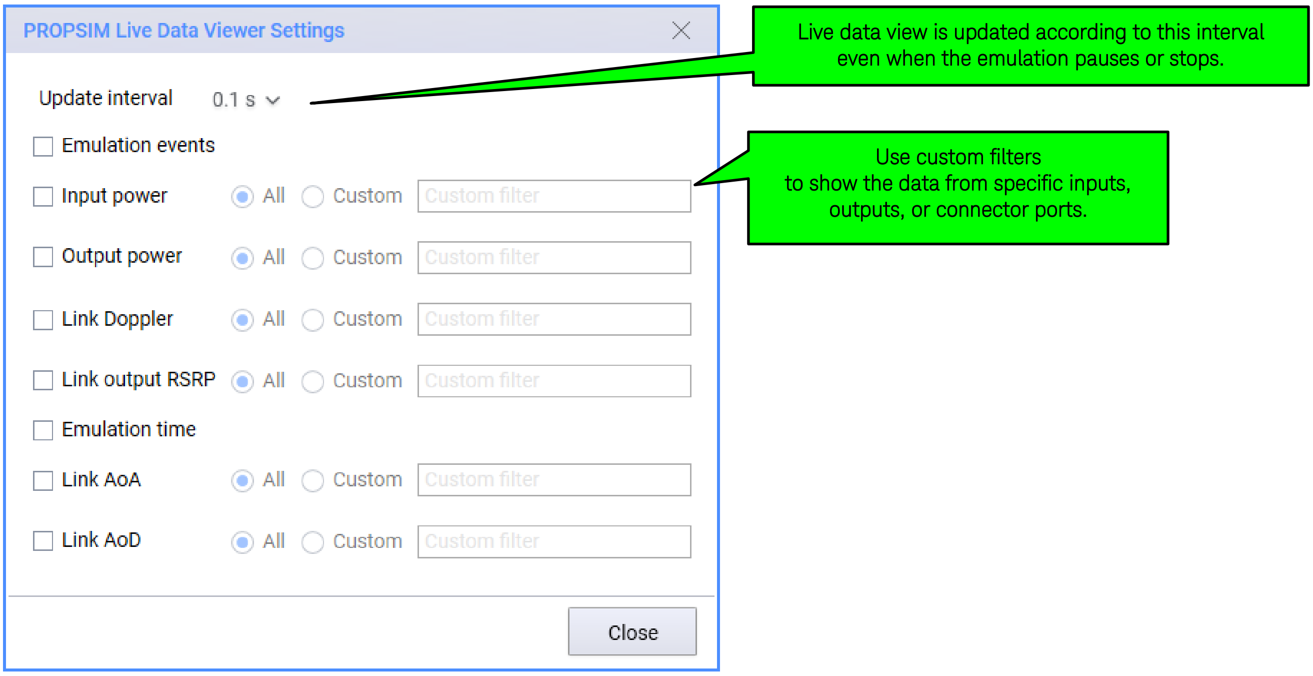


Figure 104 Live Data Viewer Settings

5.3.1 Defining custom filters for live data view

Using custom filters, you can show data from specific inputs, outputs, or PROPSIM front panel connector ports.

To show data from specific inputs/outputs, enter the input/output ID number in the Custom field next to the data type you want. You can also enter ranges and lists, for example, “1-4” or “1,3,6” (without spaces).

To show data from specific PROPSIM front panel connector ports, enter “RF[port number]” in the Custom field next to the data type you want. You can also enter ranges and lists, for example, “RF1-RF4” or “RF1,RF3,RF6” (without spaces).

5.3.2 Logging data to local file

To save the data that is shown in the live data view to a local file, select the **Log to file** checkbox.

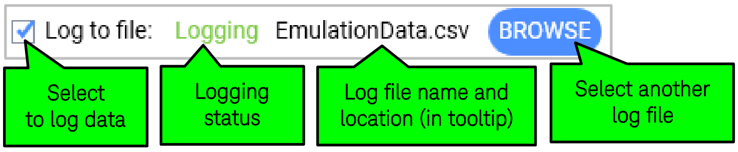


Figure 105 Logging data to file

Note: Only the data that is currently shown in the live data view is logged. Use the data view filters to select which data is shown in the view.

The log file format is a semicolon-separated file (.csv) with a dot (.) as the decimal separator in the values. The log file is always appended (not cleared). Logging stops automatically and a warning is given if the log file size exceeds 1 GB. Figure 106 shows an example log file. A header row containing element labels (starting with “time”) is always repeated when new data elements appear or disappear in the incoming data.

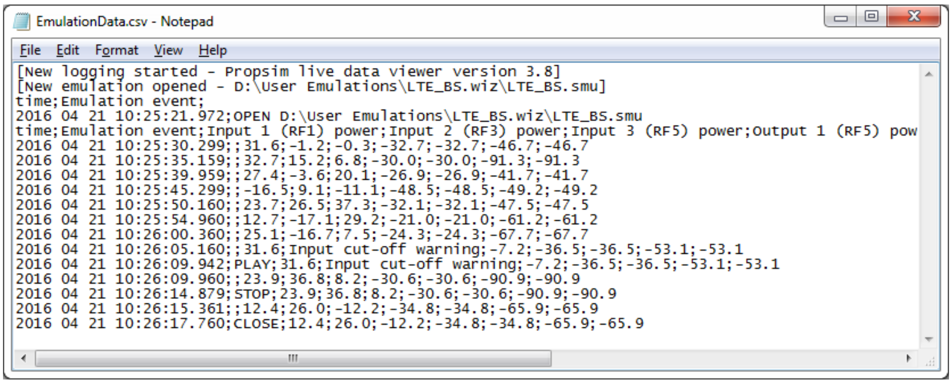


Figure 106 Example logfile of PROPSIM live data viewer

To change the log file, click the **Browse** button and select another file.

For instructions on sending measurement data to a network address, see chapter 8 Data Logging.

5.4 Active connectors view

The Active connectors view shows how the connectors are allocated in the emulator front panel. The tooltip on each connector shows the device and antenna information of the connector. For more information about the connectors and LEDs, see Table 2 and Table 3.

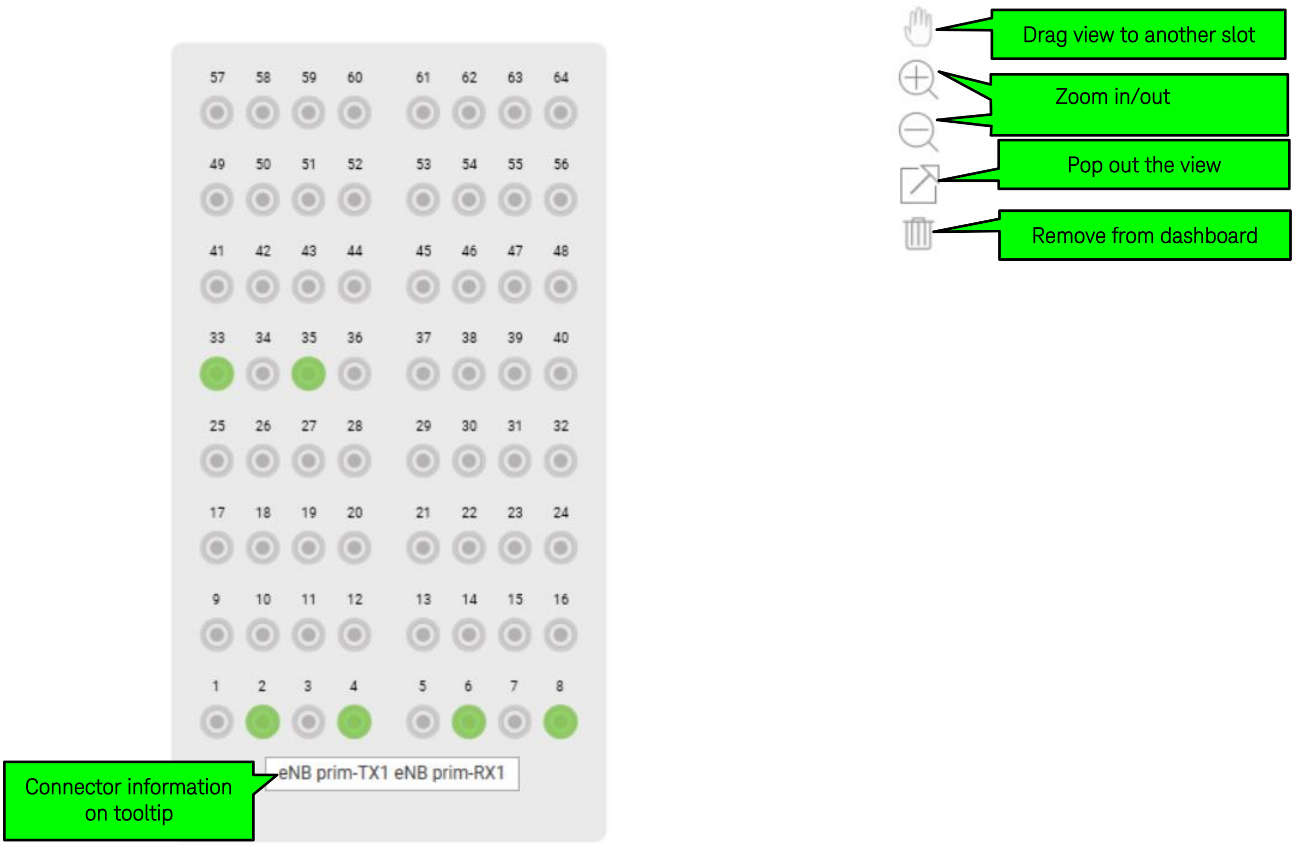


Figure 107 Active connectors view

5.5 CIR view

The CIR view shows the change of the channel input responses (CIRs) during the emulation. The signal paths of the channel model are visualized in the CIR graph as a power delay profile. Relative and absolute delay and gain values for each path can be seen in a tooltip.

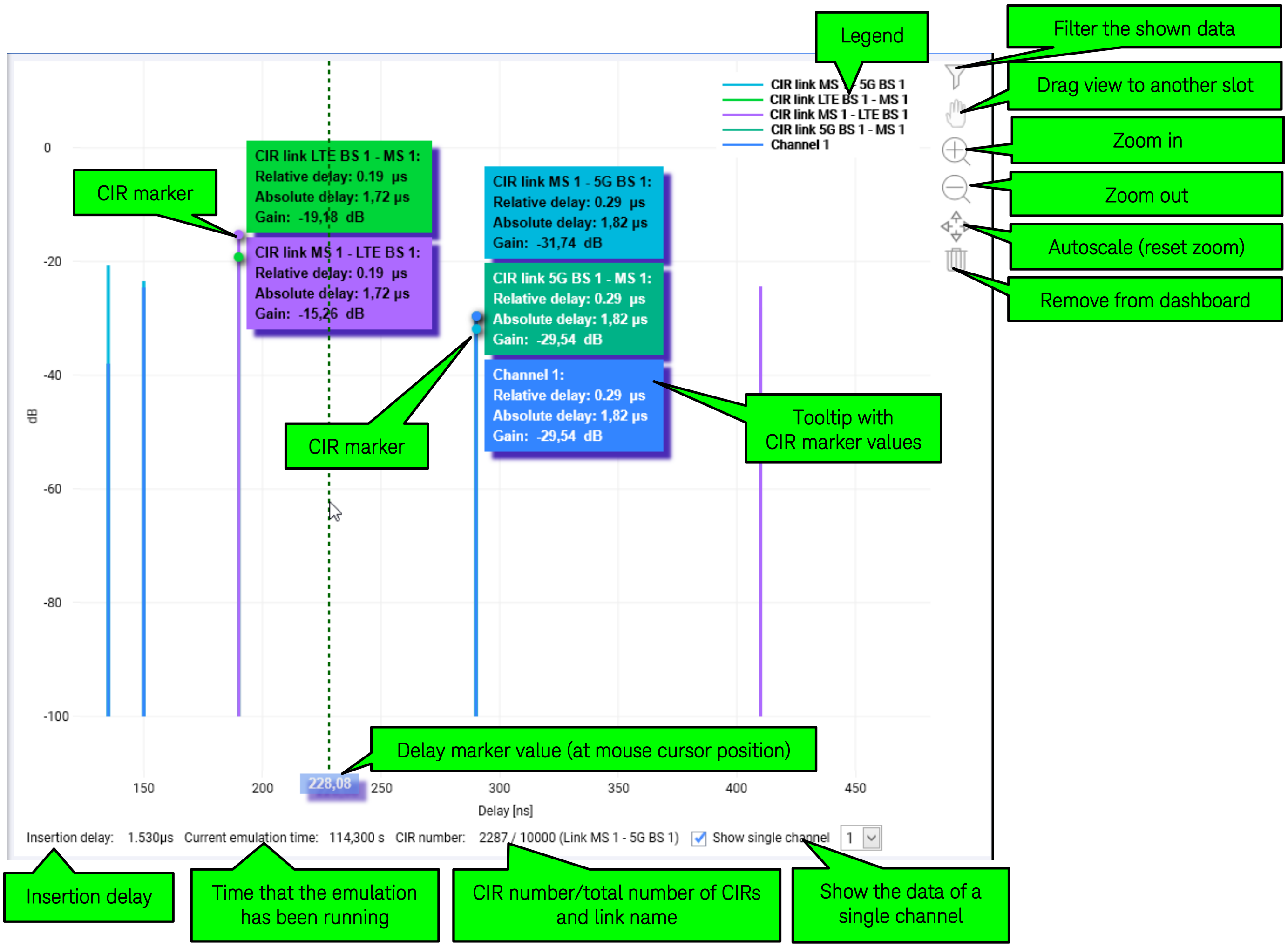


Figure 108 CIR view

To select which links are shown on the graph, click the (Filter) button in the data view toolbar.

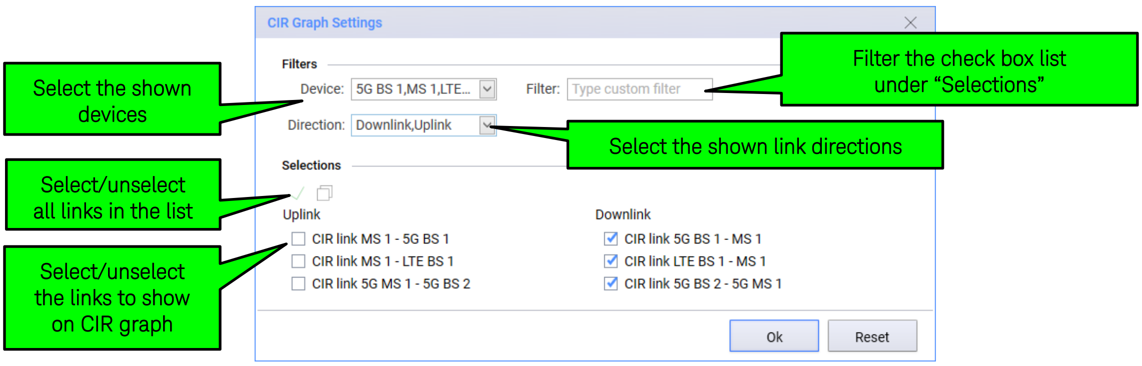


Figure 109 CIR Graph Settings dialog

In addition to links, you can also view the CIR data of a single channel by selecting the **Show single channel** option and the channel you want to see in the drop-down list.

5.5.1 Insertion delay

Insertion delay describes the internal hardware delay with the used channel model. The total real delay of a tap is the sum of the insertion delay and the delay value defined in the channel model. If the smallest delay in the channel model is greater than the insertion delay, the insertion delay is zero.

5.6 ATE LAN monitor view

The ATE LAN monitor view shows the commands and responses going via ATE connection.

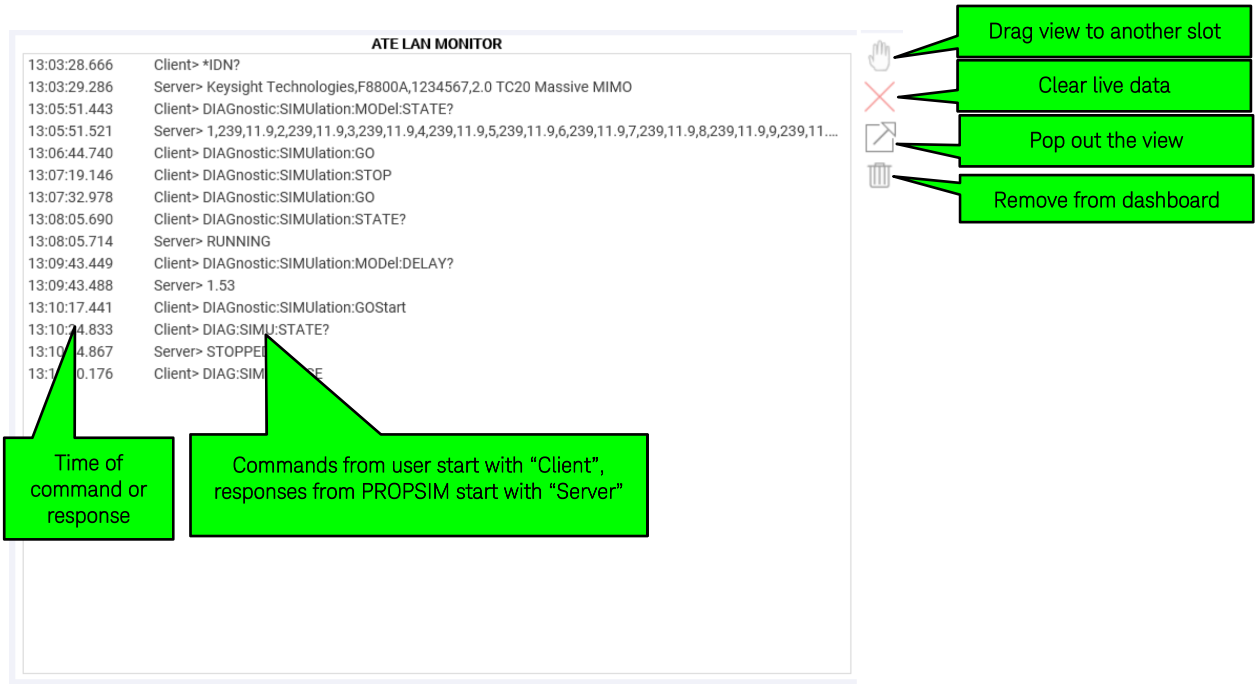


Figure 110 ATE LAN monitor view

For instructions on using the ATE connection and the available commands, see chapter 20.

5.7 ASO Graph and ASO Line Graph

ASO graph and ASO Line Graph are specific views created only for aerospace emulation. The views shows detailed information of the aerospace link paths.

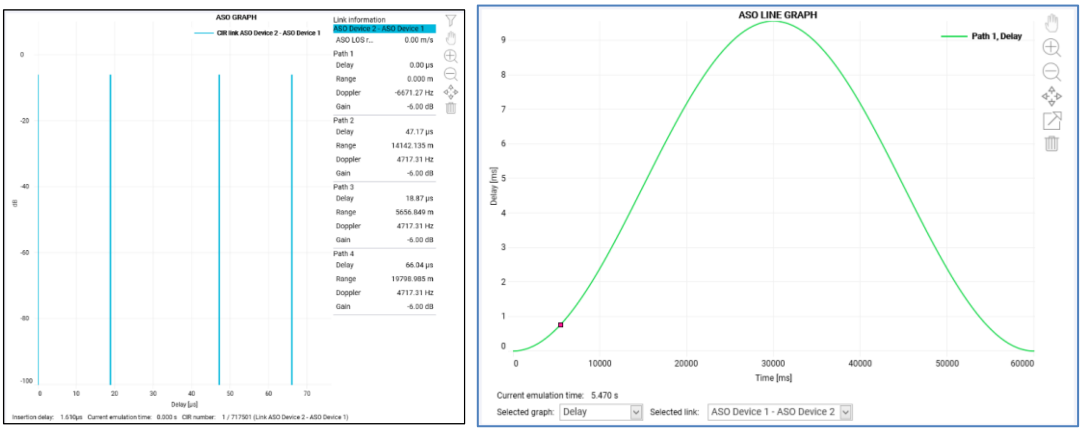


Figure 111 ASO graph and ASO Line Graph

5.8 Port RF level graph

Port RF level graph shows all the measured input power levels in a single graph. Port numbering can be selected to be shown as logical or physical. This selection can be changed under filter settings ( ).

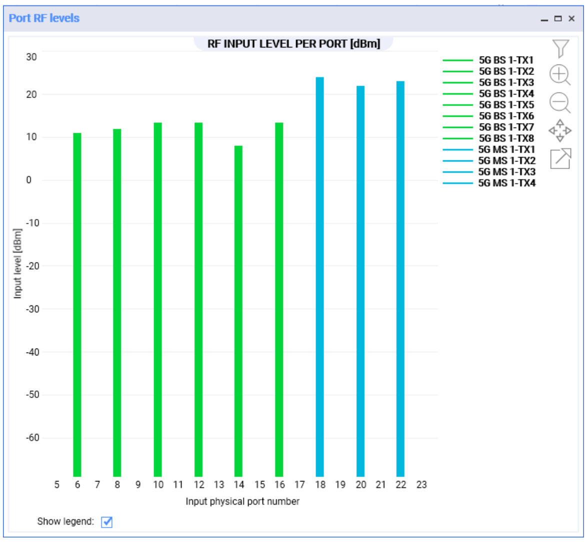


Figure 112 Port RF level graph

6 LAB SETUP FEATURE

When PROPSIM is connected as a part of a complete test setup, some of the operational parameters of PROPSIM become fixed. These parameters can be for example:

- PROPSIM connectors where devices under test (DUTs) are connected
- transmit and receive frequencies
- cable losses
- typical transmit and receive power levels
- DUT names
- etc.

Lab Setup feature is used to store these fixed parameters to ‘lab setups’. The stored lab setups can then be used together with any suitable emulation, eliminating the need for defining the common parameters all over again for all the used emulations.

Another benefit of using lab setups is that if any of the fixed parameters in the environment changes, it can be changed in a single place (lab setup) instead of all the used emulations. One example could be the situation, where some of the test setup RF cables need to be replaced. New cable loss values can be put into the used lab setup and everything continues working as before.

One lab setup can contain several different devices and technologies. It is not necessary to create a new lab setup for every emulation, instead, one lab setup can be used for as many emulations as practical.

Table 11 below shows an example of a lab setup with several devices. In the example, the lab setup can be used with emulations 1 and 2, but not with emulation 3 because the lab setup does not contain all the devices used in emulation 3.

Table 11 Example of lab setup suitability for different emulations

Lab setup devices	Emulation 1 Devices	Emulation 2 Devices	Emulation 3 Devices
BS 1 + Technology A	= BS 1 + Technology A	= BS 1 + Technology A	
BS 2 + Technology B		= BS 2 + Technology B	=BS 2 + Technology B
MS 1 + Technology A	= MS 1 + Technology A		
MS 2 + Technology B			= MS 2 + Technology B
MS 3 + Technology A and B		= MS 3 + Technology A and B	
			≠ BS 3 + Technology C
			≠ MS 4 + Technology C
LAB SETUP COMPATIBILITY WITH EMULATION	**OK**	**OK**	**NOT OK**

To use a lab setup with a certain emulation, the lab setup must contain the same devices and technologies that are used in that emulation:

- Base station:
  - Name
  - Radio technology
  - Equal or greater number of antennas compared to the emulation
- Mobile station:
  - Name
  - Radio technologies
  - Equal or greater number of antennas per technology compared to the emulation

6.1 Creating new lab setup

You can create multiple lab setups, and one lab setup can contain devices and technologies for many different emulations.

Selecting **Lab setup > New** in the navigation menu launches the lab setup editor.

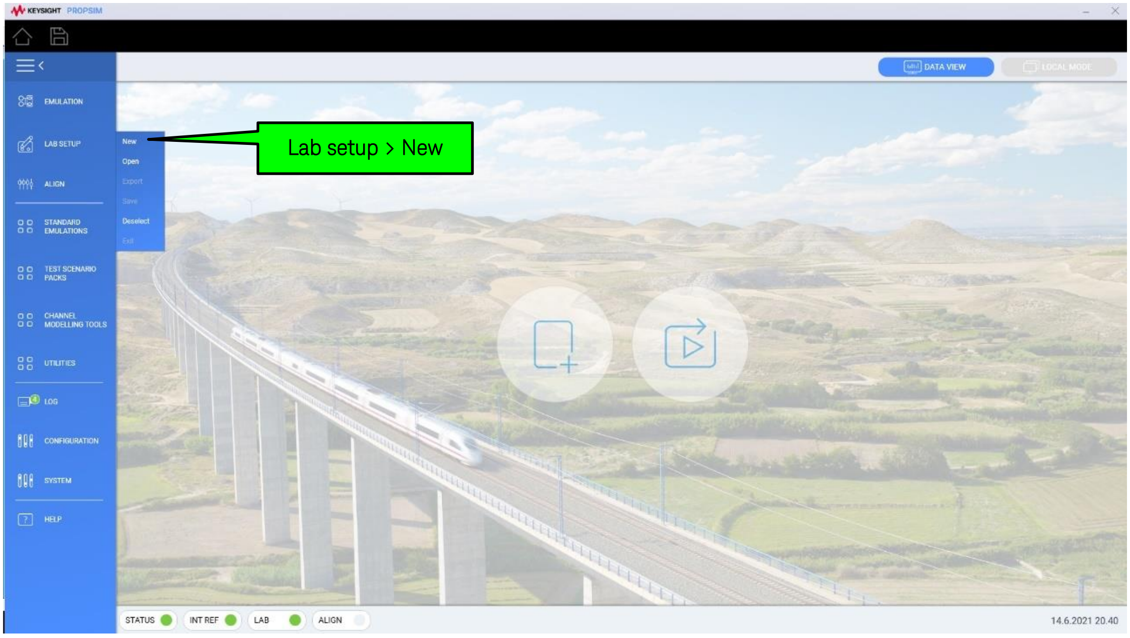


Figure 113 Create new lab setup

In the **Create new lab setup** dialog, give a name for the lab setup, and click **OK**.

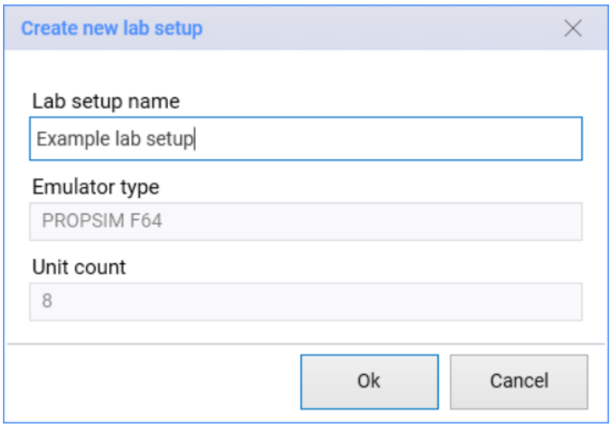


Figure 114 Create new lab setup dialog

The lab setup editor opens with an empty lab setup.

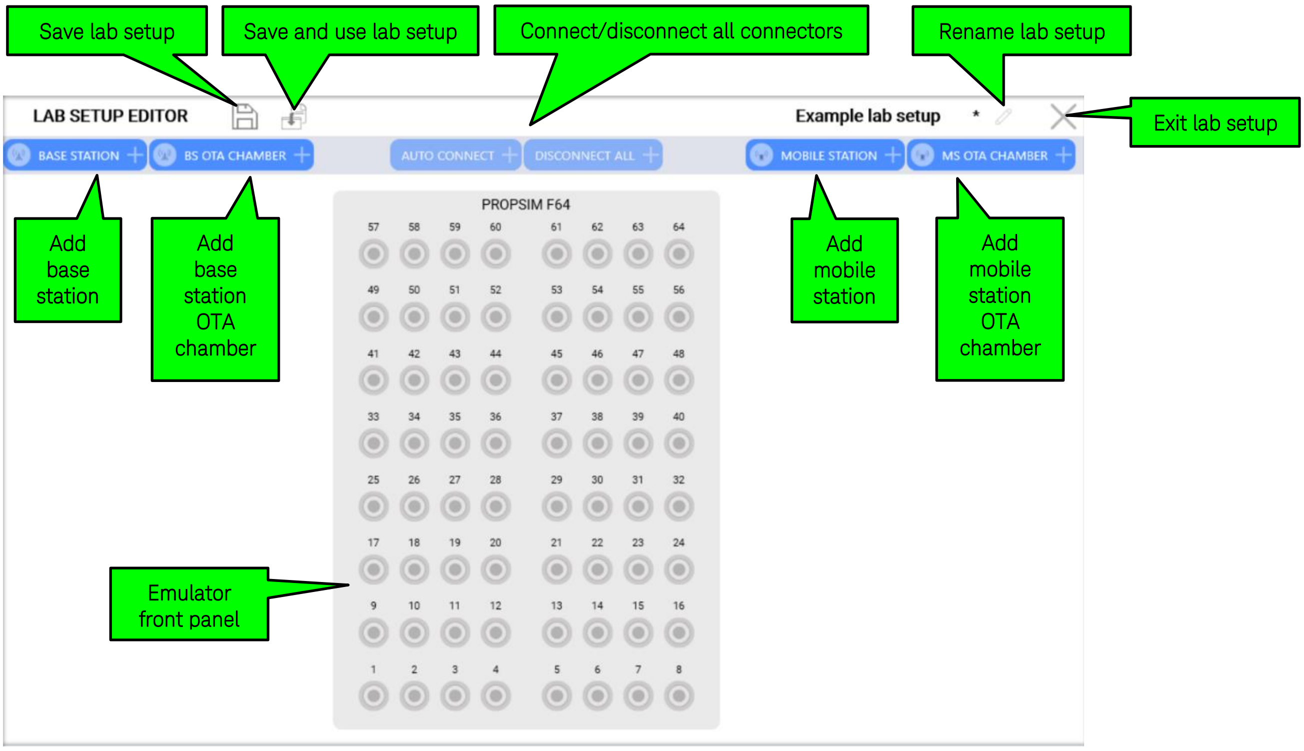


Figure 115 Empty lab setup

The lab setup button in the bottom right corner of the PROPSIM software indicates that a lab setup is being edited. You can switch to other views during editing and return to the lab setup editor by clicking the lab setup button. For example, you can open an emulation in the Emulation control view to check device information and return to editing the lab setup.

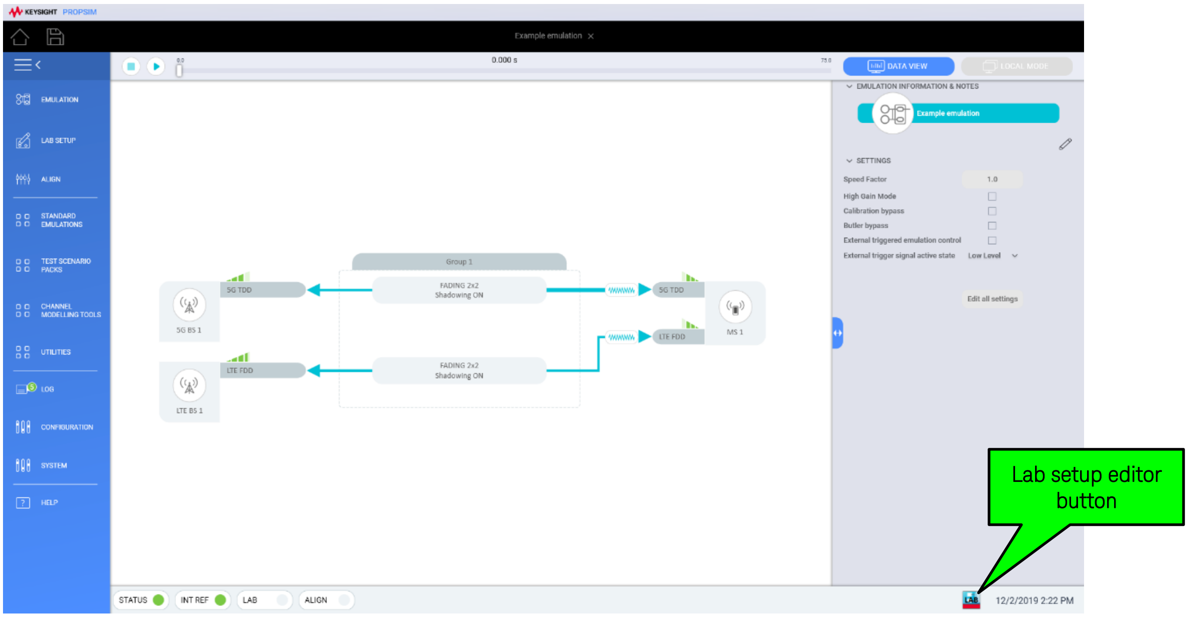


Figure 116 Emulation control view with lab setup open in background

6.1.1 Adding base station or base station OTA chamber

To add a base station in the lab setup, click the 

(Base station) button.

To add a base station OTA chamber in the lab setup, click the 

(BS OTA chamber) button.

Figure 117 below shows an example of 5G base station settings. The available settings for a base station OTA chamber are the same as for a base station.

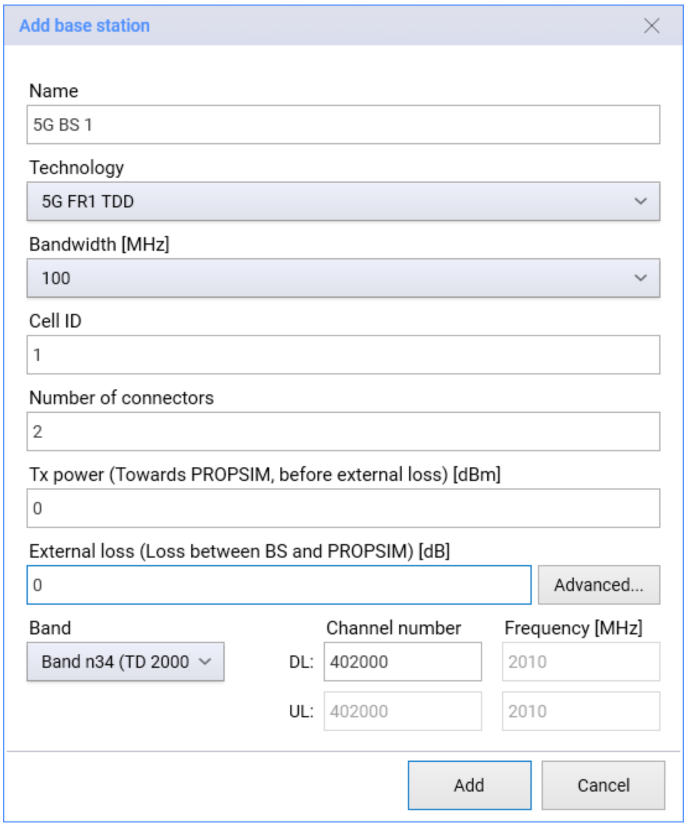
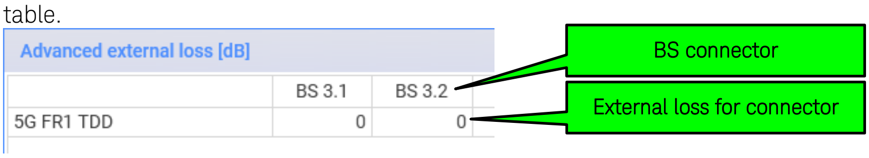


Figure 117 Add base station to lab setup

Settings

- Name
  - Base station or OTA chamber name.
- Technology
  - Radio technology used in the base station/OTA chamber
- Bandwidth [MHz] (only for 5G and LTE)
  - 5G/LTE modulation bandwidth
- Cell ID (only for 5G and LTE)
  - Base station cell ID
- Number of connectors
  - Number or physical connectors (antenna type can be changed in lab setup editor, options: TRX, TX, RX)
- TX power (PROPSIM input power) [RSRP]
  - Maximum transmit power of the DUT, i.e. maximum expected power level in the PROPSIM inputs
- External loss (Loss between BS and PROPSIM) [db]
  - Attenuation of cables and external components between BS and PROPSIM

- To define the external loss separately for each connector, click **Advanced** and enter the values in the table.



- Band (only for 5G, LTE, and WCDMA)
  - Technology specific band number
- Channel number (only for 5G, LTE, and WCDMA)
  - Technology specific band number
- Frequency
  - Actual uplink/downlink frequency

After adding a base station or BS OTA chamber, the device appears on the left side of the lab setup editor.

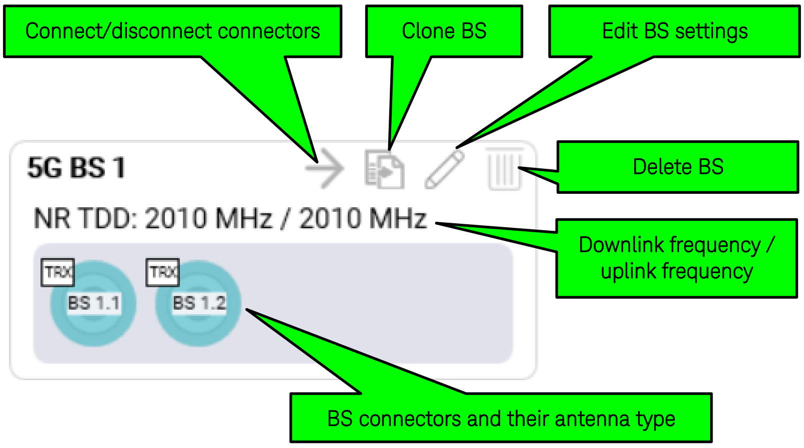


Figure 118 Base station added to lab setup

The connectors are named as follows: "BS [running number of BS].[running number of connectors]".

6.1.1.1 Changing antenna type

To change the antenna type of the connector, click the **TRX/TX/RX** icon on the connector:

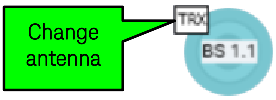


Figure 119 Changing antenna type of connector

6.1.2 Adding mobile station or mobile station OTA chamber

Note: Before mobile stations can be added in the lab setup, the lab setup must contain at least one base station (see section 6.1.1).

To add a mobile station in the lab setup, click the 

(Mobile station) button.

To add a mobile station OTA chamber in the lab setup, click the 

(MS OTA chamber) button.

Figure 120 below shows an example of mobile station settings. The available settings for a mobile station OTA chamber are the same as for a mobile station.

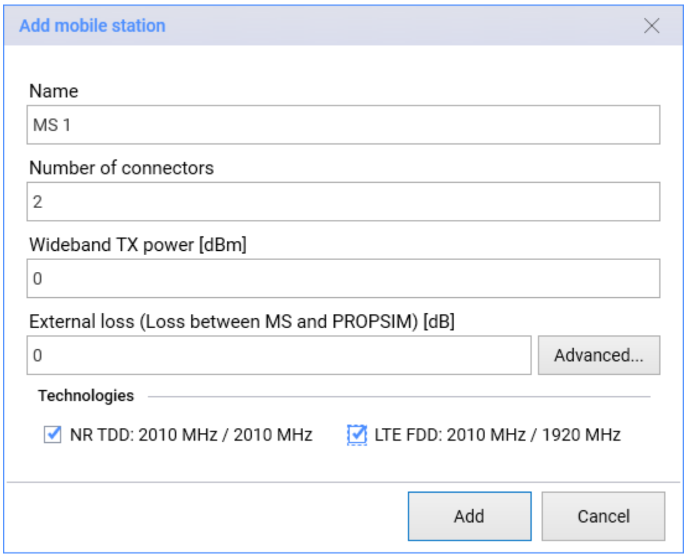
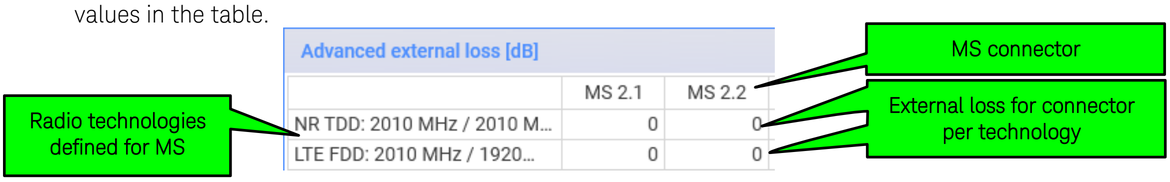


Figure 120 Add mobile station to lab setup

Settings

- Name
  - Mobile station or MS OTA chamber name
- Number of connectors
  - Number or physical connectors (antenna type can be changed in lab setup editor, options: TRX, TX, RX)
- Wideband TX power [dBm]
  - Maximum wideband transmit power of the mobile station
- External loss (Loss between MS and PROPSIM) [dB]
  - Attenuation of cables and external components between MS and PROPSIM
  - To define the external loss separately for each connector/technology, click **Advanced** and enter the values in the table.



- Technologies
  - Radio technologies used in the mobile station/MS OTA chamber. The options correspond to the radio technologies and frequencies/bands of the base stations/BS OTA chambers in the same lab setup. If two base stations use the same technology but with different frequencies, these are shown as separate technology options for the MS.

After adding a mobile station or MS OTA chamber, the device appears on the right side of the lab setup editor.

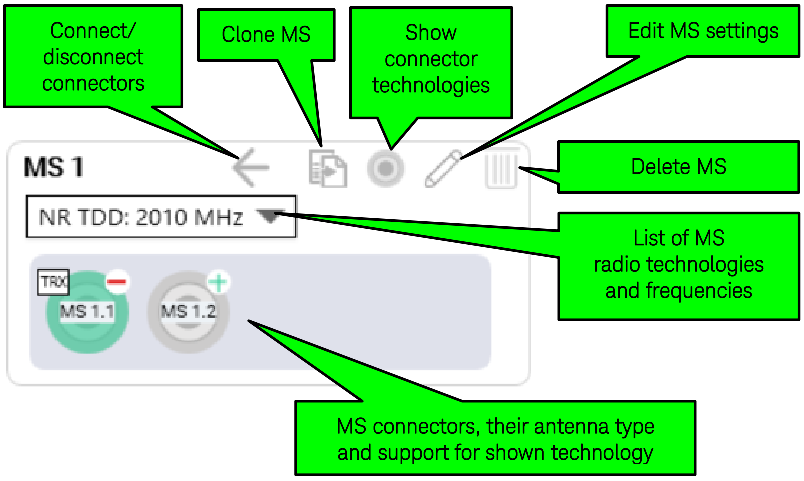


Figure 121 Mobile station added to lab setup

The connectors are named as follows: “MS [running number of MS].[running number of connectors]”.

6.1.2.1 Changing antenna type

To change the antenna type of the connector, click the TRX/TX/RX icon on the connector:

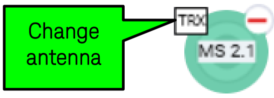
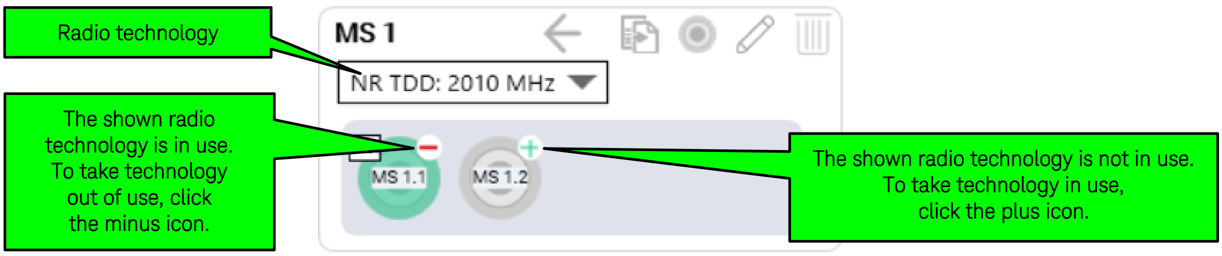


Figure 122 Changing antenna type of connector

6.1.2.2 Defining radio technologies for connectors

Radio technologies can be defined for each individual mobile station/MS OTA chamber connector.

The radio technologies list shows the technologies supported by the MS. When you select a technology, a plus or minus icon on the connector indicates whether that technology is used on that connector. To add or remove a technology from a connector, click the plus or minus icon.



You can also define the radio technologies for all connectors at once without having to change the shown radio technology. Clicking the  (Show connector technologies) button on the mobile station / MS OTA chamber box opens the Connectors and Technologies dialog. To define the radio technologies for each connector, select or unselecti the check boxes.

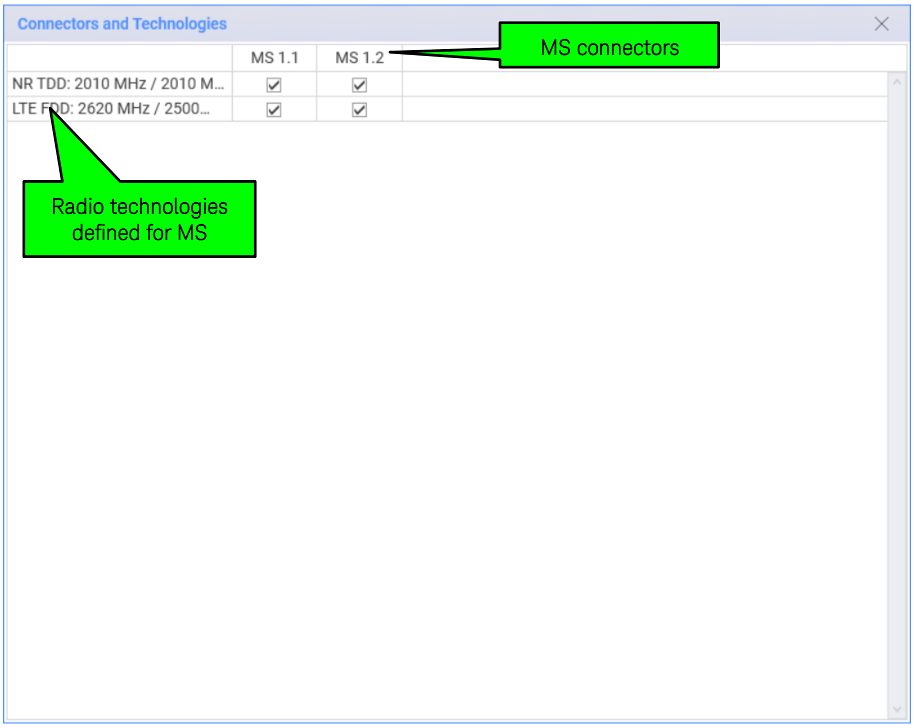
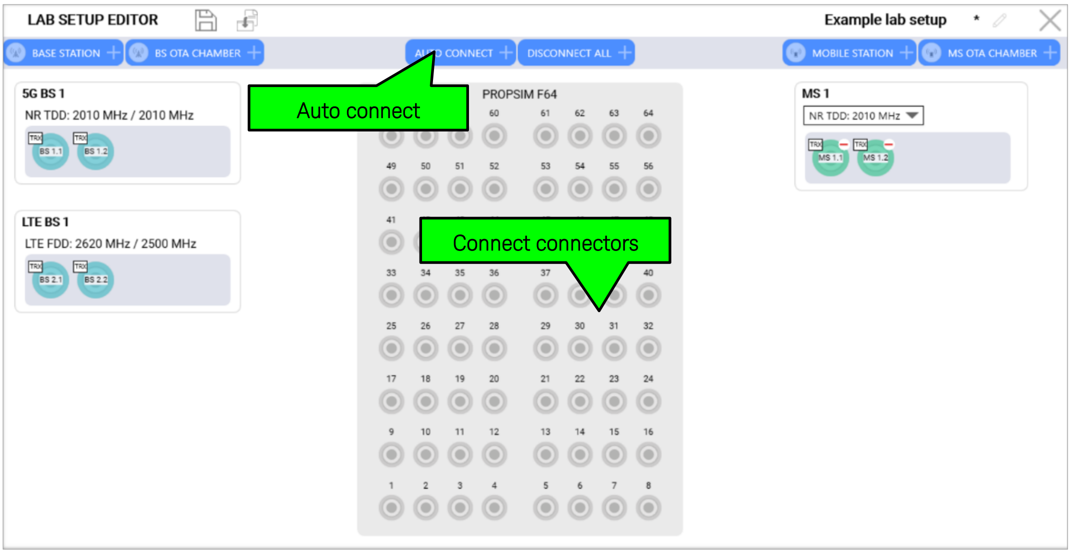


Figure 123 Connectors and Technologies dialog

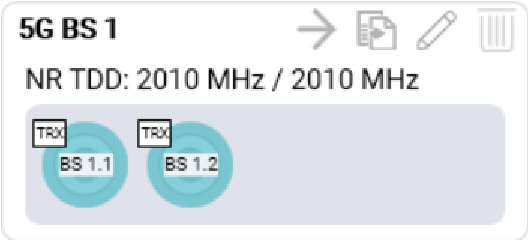
6.1.3 Connecting connectors

The connectors can be connected to the emulator front panel in three ways:

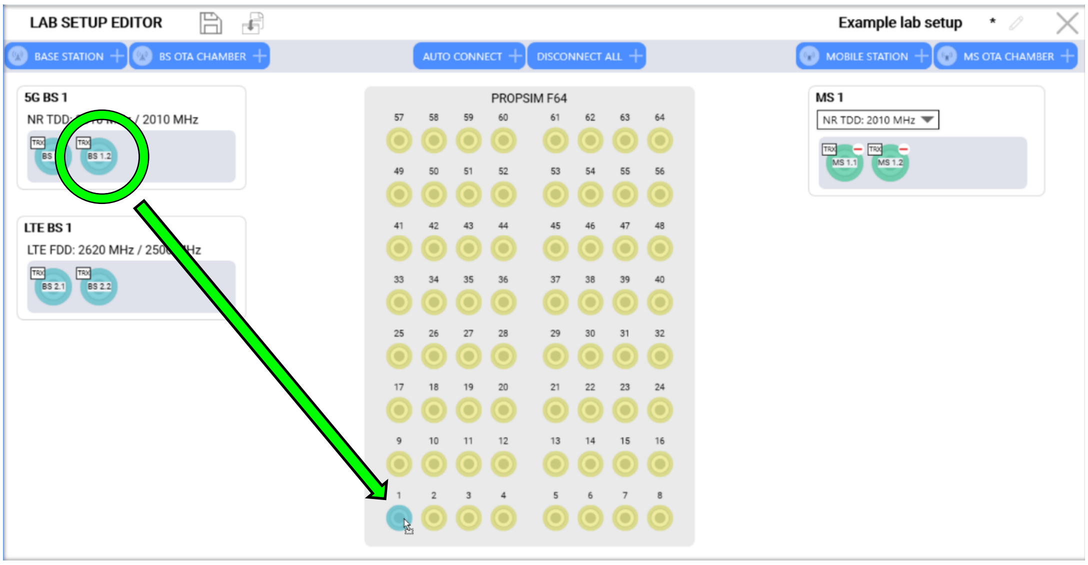
- Automatic allocation of all connectors
  - To automatically allocate all connectors of all devices in the lab setup, click the Auto connect button on the top of the lab setup editor. The connectors are allocated in their default positions on the emulator front panel.



- Automatic allocation of all connectors on an individual device
  - To automatically allocate all connectors of an individual device, click the arrow button on the device. The connectors are allocated in their default positions on the emulator front panel.



- Manual allocation of individual connectors
  - To manually allocate an individual connector, drag and drop the connector on the emulator front panel. The available positions on the panel are indicated with yellow color.



When the connector is allocated, the connector icon turns gray:

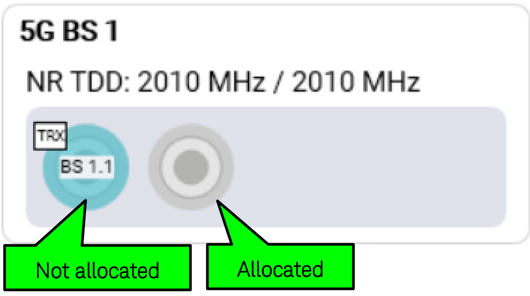
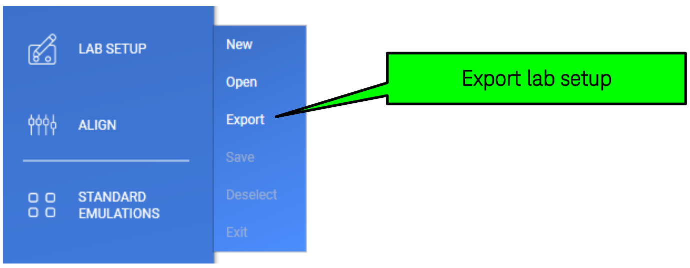


Figure 124 Connector appearance when allocated or not allocated

6.2 Exporting lab setup from an emulation

A new lab setup can be exported from the currently open emulation by selecting Lab Setup -> Export in the navigation menu. Exporting the lab setup is supported for the emulations created with Scenario Wizard, GCM Tool or any other tool that includes device and technology information in the emulation file. Lab setup export is available when there is no lab setup currently selected.



Exporting the lab setup allows defining the name, description and selecting individual parameters from the emulation that will be included in the exported lab setup. Lab setup export dialog is shown in Figure 125.

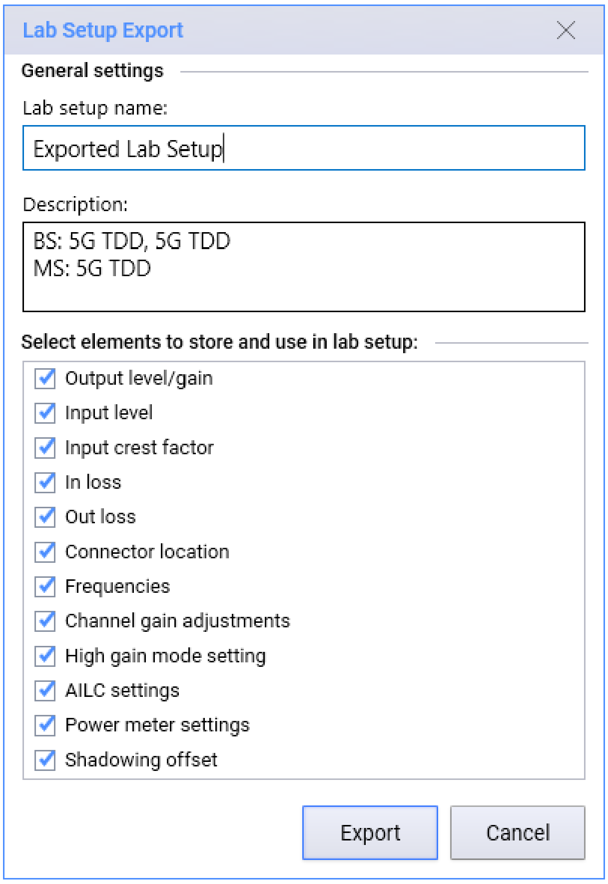


Figure 125 Selections for exporting the lab setup from the emulation

6.3 Taking lab setup in use

Note: Make sure all the emulations are closed before attempting to take the lab setup in use. The lab setup cannot be taken in use if an emulation is open.

The lab setup can be taken into use with an emulation when all the necessary devices are defined (names, technologies, number of antennas) and the necessary connectors are allocated on the emulator front panel.

When the lab setup editor is open with the lab setup you want to use, click the Save and use lab setup button to take the lab setup in use.

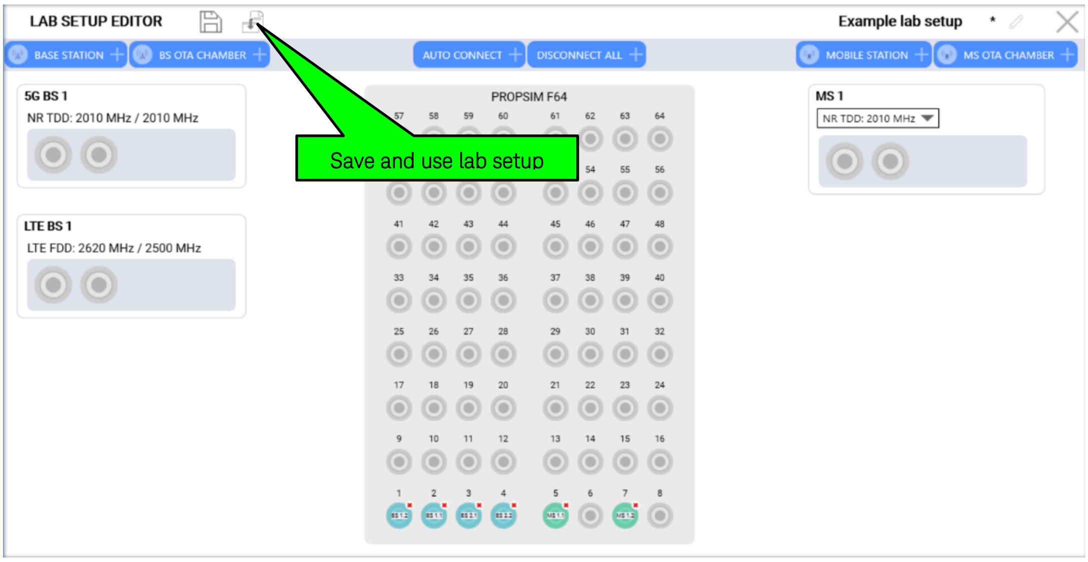


Figure 126 Save and use lab setup

The LAB button at the bottom of the PROPSIM window turns green to indicate that a lab setup is in use. The Lab button tooltip shows the name and BS/MS radio technologies of the lab setup in use.

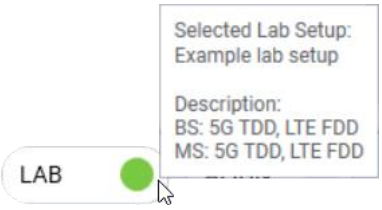


Figure 127 Lab setup button: lab setup in use

6.3.1.1 Opening emulation with lab setup

To use the lab setup with an emulation, first take the lab setup in use, then open the emulation.

The compatibility of the emulation with the lab setup is automatically verified when the emulation opens:

- If the emulation matches the lab setup in use, the lab setup settings are used in the emulation and the LAB button color stays green.
- If the emulation does not match the lab setup, the LAB button turns orange and the lab setup is not used with the current emulation. Check the LAB button tooltip and System log for more information.

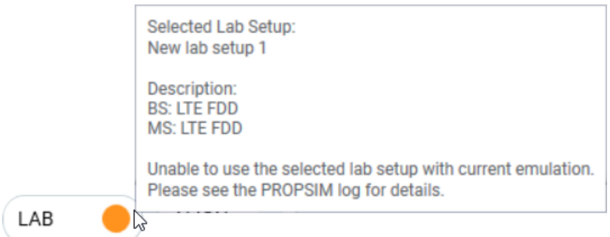


Figure 128 Lab setup cannot be used

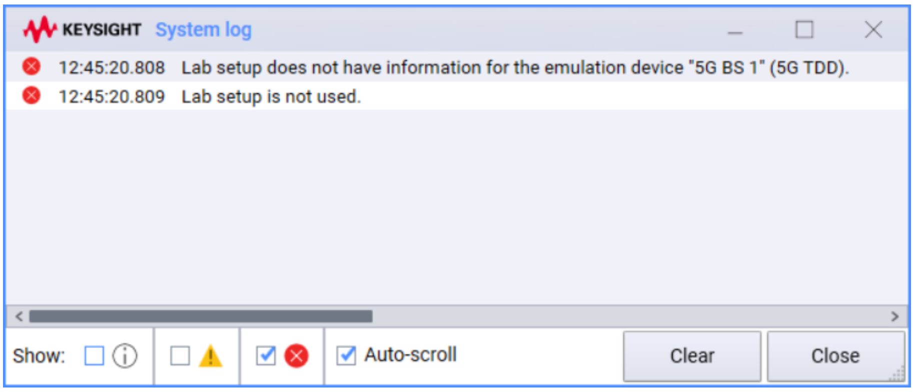


Figure 129 System log dialog for lab setup error

When the emulation is open in the Emulation control view and a lab setup is in use, the settings that are defined in the lab setup are indicated in the tooltip, in the editing dialog of that setting, and in the **All settings dialog**.

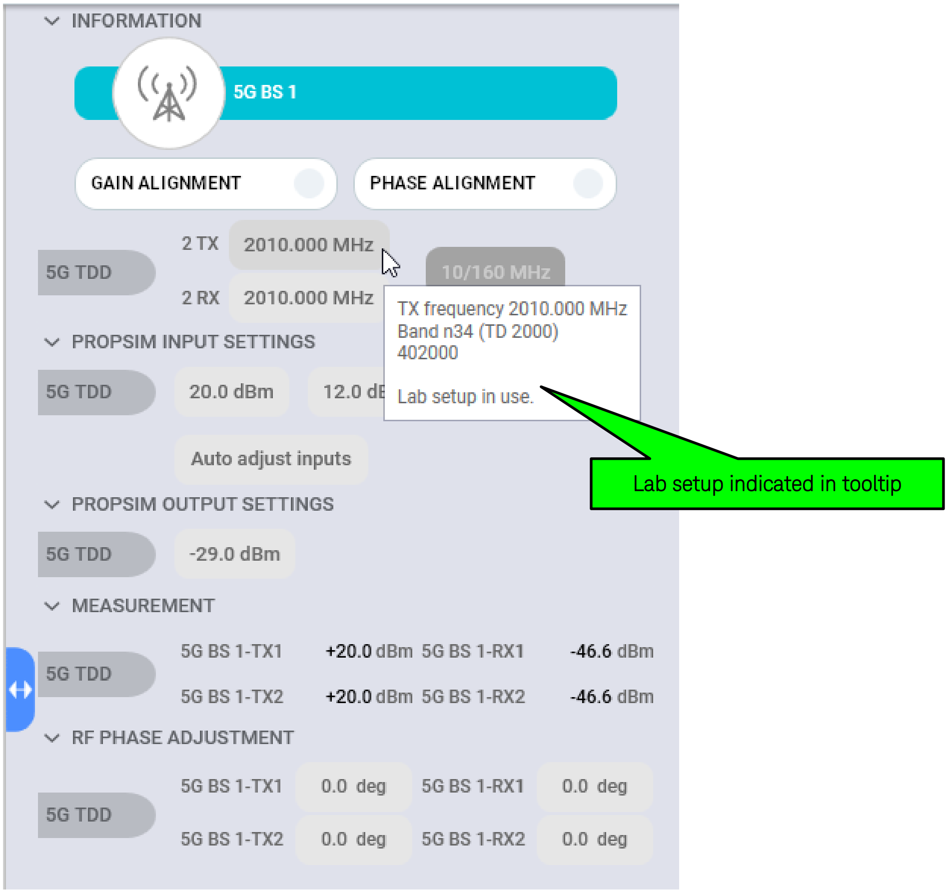


Figure 130 Lab setup usage indicated in BS settings tooltip

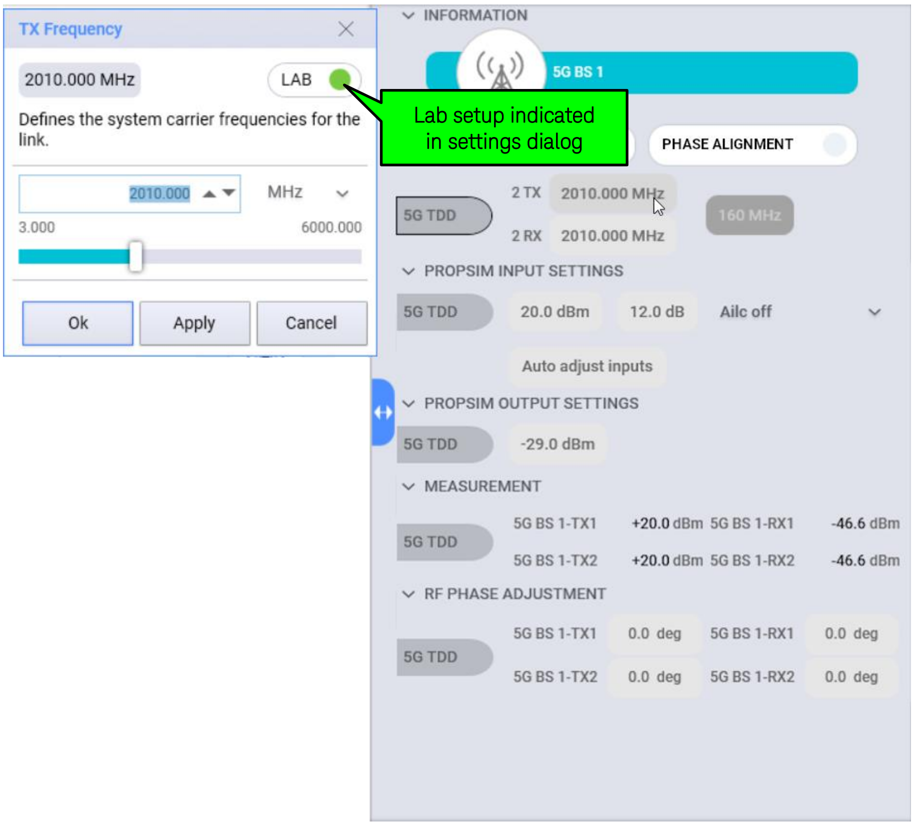
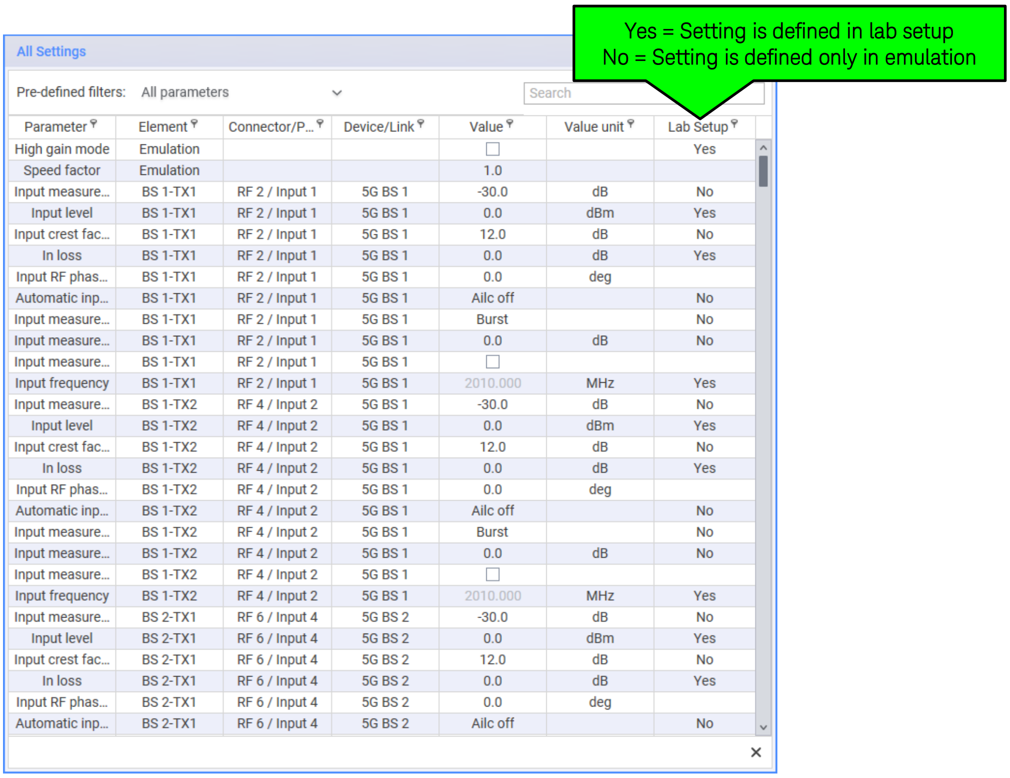


Figure 131 Lab setup usage indicated in BS settings dialog



Parameter	Element	Connector/P...	Device/Link	Value	Value unit	Lab Setup
High gain mode	Emulation			☐		Yes
Speed factor	Emulation			1.0		
Input measure...	BS 1-TX1	RF 2 / Input 1	5G BS 1	-30.0	dB	No
Input level	BS 1-TX1	RF 2 / Input 1	5G BS 1	0.0	dBm	Yes
Input crest fac...	BS 1-TX1	RF 2 / Input 1	5G BS 1	12.0	dB	No
In loss	BS 1-TX1	RF 2 / Input 1	5G BS 1	0.0	dB	Yes
Input RF phas...	BS 1-TX1	RF 2 / Input 1	5G BS 1	0.0	deg	
Automatic inp...	BS 1-TX1	RF 2 / Input 1	5G BS 1	Ailc off		No
Input measure...	BS 1-TX1	RF 2 / Input 1	5G BS 1	Burst		No
Input measure...	BS 1-TX1	RF 2 / Input 1	5G BS 1	0.0	dB	No
Input measure...	BS 1-TX1	RF 2 / Input 1	5G BS 1	☐		
Input frequency	BS 1-TX1	RF 2 / Input 1	5G BS 1	2010.000	MHz	Yes
Input measure...	BS 1-TX2	RF 4 / Input 2	5G BS 1	-30.0	dB	No
Input level	BS 1-TX2	RF 4 / Input 2	5G BS 1	0.0	dBm	Yes
Input crest fac...	BS 1-TX2	RF 4 / Input 2	5G BS 1	12.0	dB	No
In loss	BS 1-TX2	RF 4 / Input 2	5G BS 1	0.0	dB	Yes
Input RF phas...	BS 1-TX2	RF 4 / Input 2	5G BS 1	0.0	deg	
Automatic inp...	BS 1-TX2	RF 4 / Input 2	5G BS 1	Ailc off		No
Input measure...	BS 1-TX2	RF 4 / Input 2	5G BS 1	Burst		No
Input measure...	BS 1-TX2	RF 4 / Input 2	5G BS 1	0.0	dB	No
Input measure...	BS 1-TX2	RF 4 / Input 2	5G BS 1	☐		
Input frequency	BS 1-TX2	RF 4 / Input 2	5G BS 1	2010.000	MHz	Yes
Input measure...	BS 2-TX1	RF 6 / Input 4	5G BS 2	-30.0	dB	No
Input level	BS 2-TX1	RF 6 / Input 4	5G BS 2	0.0	dBm	Yes
Input crest fac...	BS 2-TX1	RF 6 / Input 4	5G BS 2	12.0	dB	No
In loss	BS 2-TX1	RF 6 / Input 4	5G BS 2	0.0	dB	Yes
Input RF phas...	BS 2-TX1	RF 6 / Input 4	5G BS 2	0.0	deg	
Automatic inp...	BS 2-TX1	RF 6 / Input 4	5G BS 2	Ailc off		No

Figure 132 All settings dialog with lab setup indication

The settings defined in the lab setup can be edited in the Emulation control view and in the lab setup editor. For more information, see section 6.4.

6.3.1.2 Changing to another lab setup

Note: Make sure all the emulations are closed before attempting to change to another lab setup. The lab setup cannot be changed if an emulation is open.

To change the used lab setup, select Lab setup > Open in the Navigation bar, or click the LAB button at the bottom of the PROPSIM window. In the Open Lab Setup dialog, select the lab setup you want to use, and click Select.

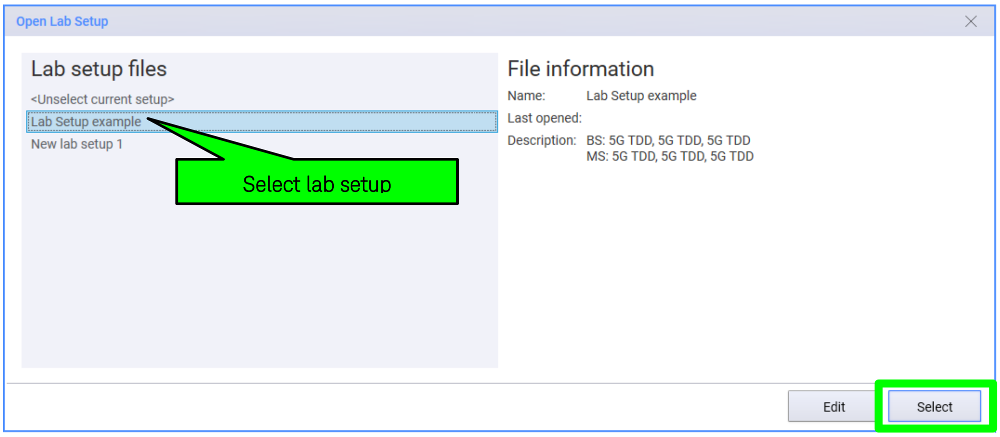


Figure 133 Open lab setup dialog

6.4 Editing lab setup

Lab setups can be edited in the lab setup editor and in the emulation control view.

6.4.1 Editing lab setup in lab setup editor

To edit a lab setup in the lab setup editor, select Lab setup > Open in the Navigation bar, or click the LAB button at the bottom of the PROPSIM window. In the Open Lab Setup dialog, select the lab setup you want to use, and click Edit.

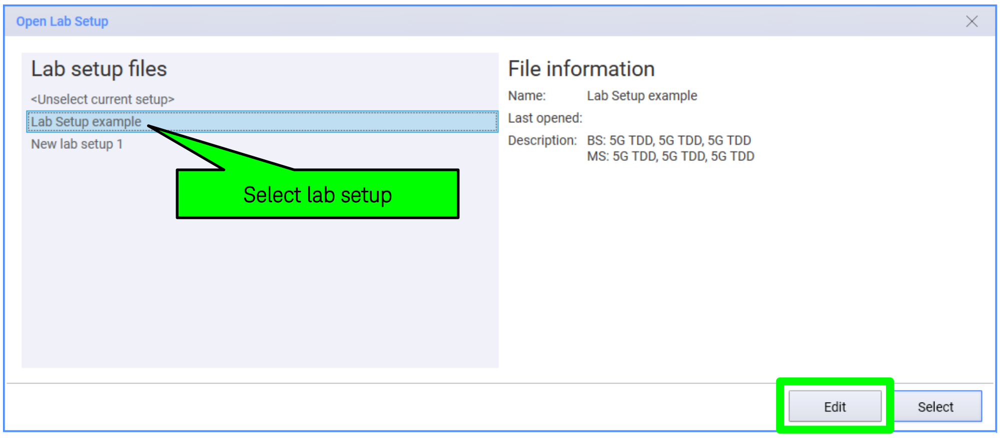


Figure 134 Opening lab setup for editing

For more information on the lab setup settings, see:

- 6.1.1 Adding base station or base station OTA chamber
- 6.1.2 Adding mobile station or mobile station OTA chamber
- 6.1.3 Connecting connectors

6.4.2 Editing lab setup in Emulation control view

When you open an emulation that uses a lab setup, the settings that are defined in the lab setup are indicated in the Emulation control view settings pane in the setting's tooltip and editing dialog.

You can edit the settings directly in the Emulation control view. If you edit settings that are defined in the lab setup, you can select whether you want to save the changes in the lab setup when you save the changed emulation:

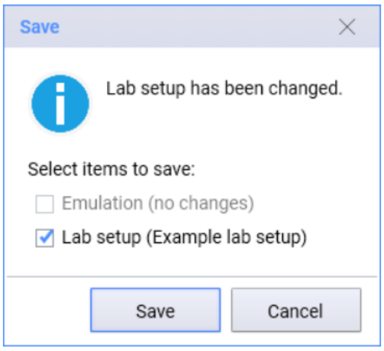
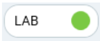


Figure 135 Saving changes to lab setup in Emulation control view

6.5 Applying only part of the parameters from the existing lab setup

When the lab setup is selected and emulation is not open, you can select which of the parameters from the selected lab setup are taken into use and applied for the emulations. This can be done by opening the lab setup selection window from the menu (Lab setup -> open) or pressing  button.

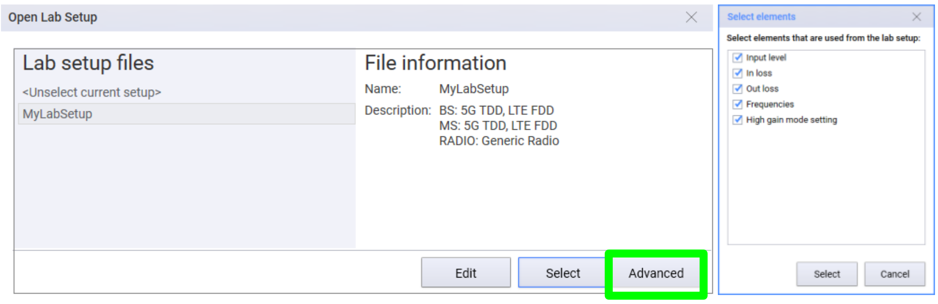


Figure 136. Selecting active parameters from the selected lab setup

6.6 Taking lab setup out of use

Note: Make sure all the emulations are closed before attempting to take a lab setup out of use. The lab setup cannot be taken out of use if an emulation is open.

To take a lab setup out of use, select Lab setup > Deselect in the Navigation bar, or click the LAB button at the bottom of the PROPSIM window, select the <Unselect current setup> option and click Select.

The LAB button at the bottom of the PROPSIM window turns grey to indicate that a lab setup is not in use.

Figure 137 Lab setup button: lab setup not in use

6.7 Deleting lab setup

To delete a lab setup, select **Lab setup > Open** in the Navigation bar, or click the **LAB** button at the bottom of the PROPSIM window. In the **Open Lab Setup** dialog, right-click the lab setup you want to delete and click **Delete lab setup**.

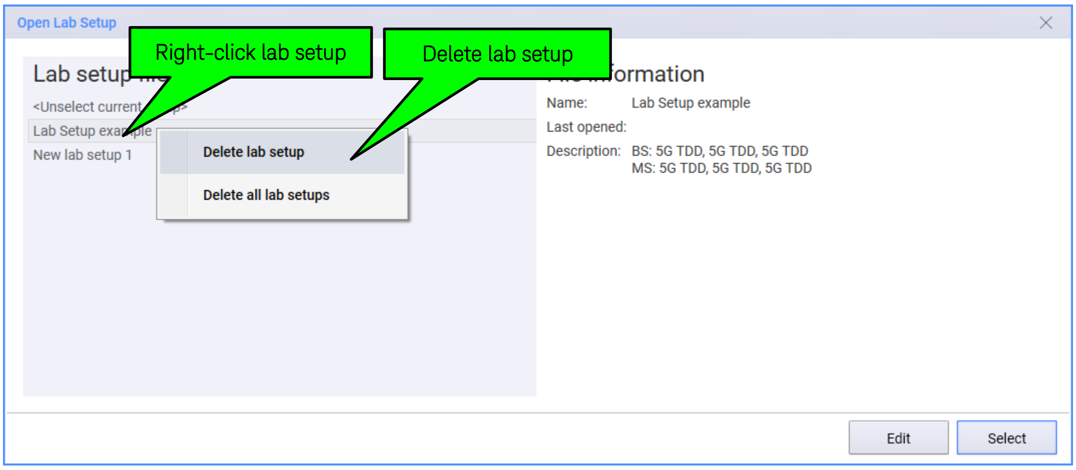


Figure 138 Deleting lab setup

7 UTILITIES

7.1 Channel model view

The Channel model view is targeted for creating a simplified statistical channel model for stationary scenarios and scenarios where the mobile station is moving away or towards the base station at a constant speed.

The main function of the Channel model view is to create and edit channel model files.

The view can be launched by selecting Utilities > Channel model view in the navigation bar.

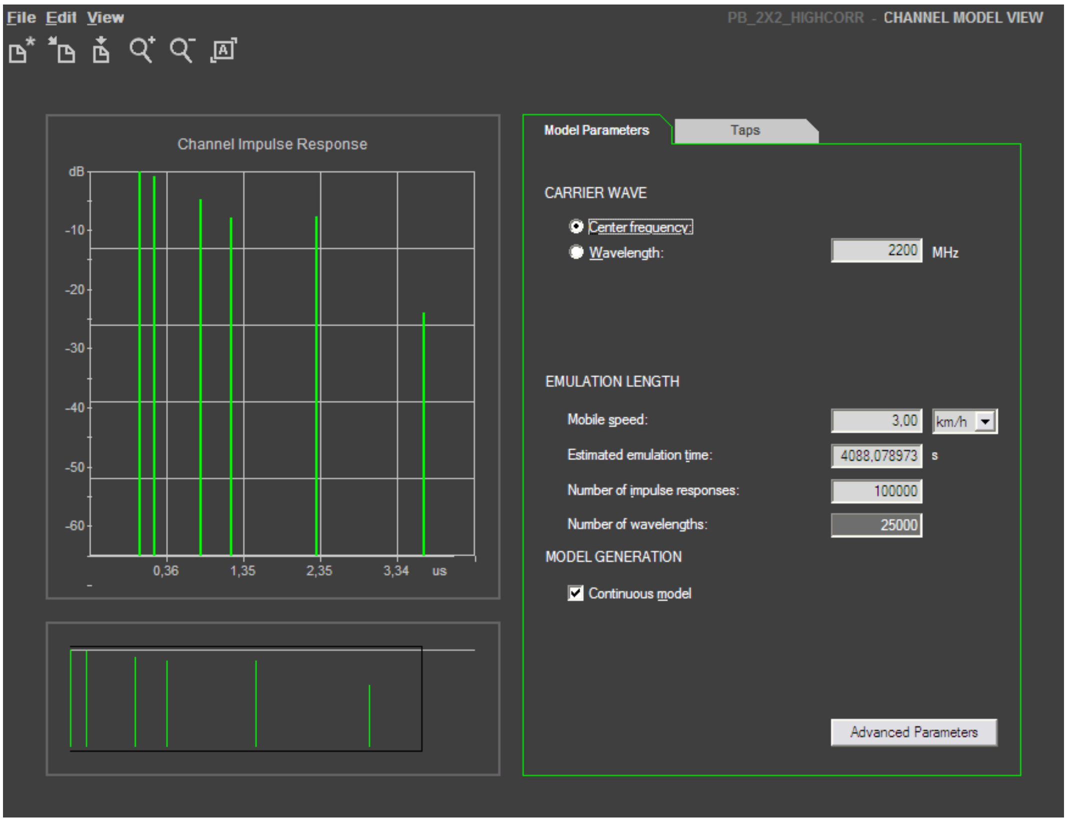


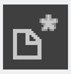
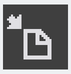
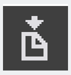
Figure 139 Channel model view

The Channel Model view contains a menu bar, a toolbar, a channel impulse response graph window and two tabs for channel model settings (Model parameters and Taps).

7.1.1 Channel model view toolbar

The tools available in Channel model view toolbar are described below.

Table 12 Channel model view toolbar

Symbol	Tool	Description
	Create new channel model	Creates new channel model file
	Open	Opens a dialog where you can browse and select an emulation file to be opened.
	Save	Saves edited channel model with the same name or prompts you to define a name, if the channel model has not been saved previously.
	Zoom in	Can be used to scale the CIR graphics in CIR Graphics View.

Symbol	Tool	Description
	Zoom out	Can be used to scale the CIR graphics in CIR Graphics View.
	Autoscale	Selects such a scaling in CIR Graphics View that all taps in all channels are visible.

7.1.2 Channel model view menus

The Menu bar contains the following menus:

Table 13 Channel model view menus

Menu	Submenu	Shortcut	Description
File	New	Ctrl-N	Creates new channel model, see chapter 7.1.2.1
	Open	Ctrl-O	Opens channel model
	Save	Ctrl-S	Saves channel model
	Save As...		Saves channel model with new name
	Change model type	Ctrl-G	Enables model type change
	Recent file list		Recently used channel model files
Edit	Cut	Ctrl-X	
	Copy	Ctrl-C	
	Paste	Ctrl-V	
	Delete	Del	
	Select All	Ctrl-A	
	Invert Selection		
	Add Tap		Adds a new tap
	Remove Tap		Removes selected tap
	Properties...		Opens Tap Properties dialog
View	Zoom In		
	Zoom Out		
	Autoscale		

7.1.2.1 New Model Generation

Selecting File > New opens New Model Generation Wizard dialog, see Figure 140. The wizard first asks for the type of the model. The model can be a single-channel model, MISO- or SIMO-type correlative model or a MIMO-type correlative model.

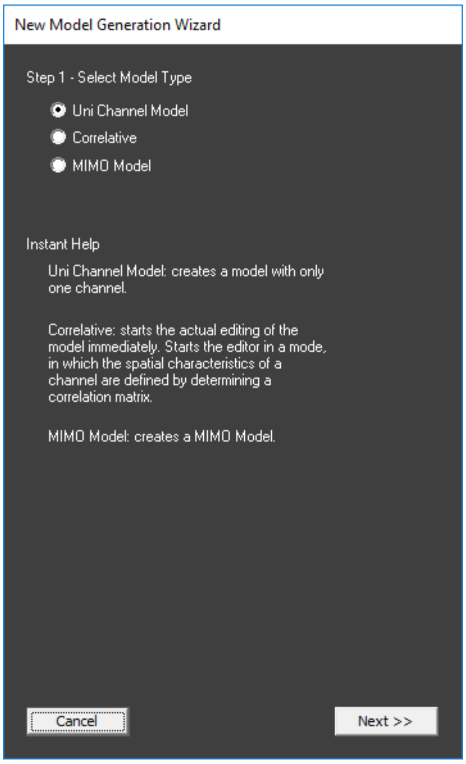


Figure 140 New Model Generation Wizard

Next step is to give the number of channels. If the model is correlative, single number is required. If a MIMO emulation is to be created, both number of transmitter and receiver antenna elements must be specified.

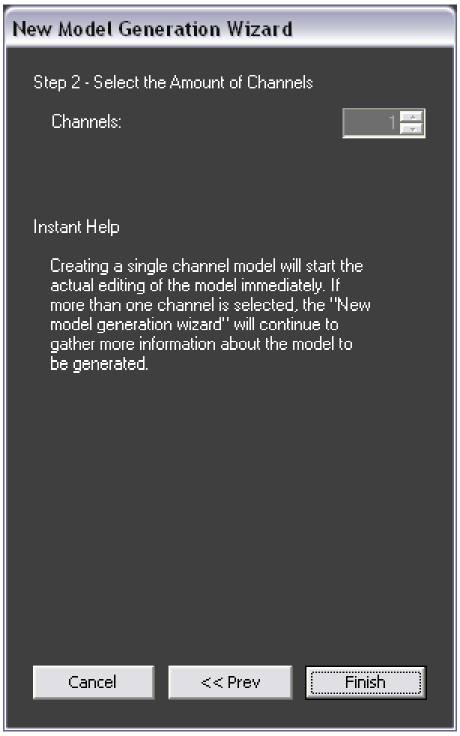


Figure 141 Number of channels

In case of a MIMO model, the wizard asks for the number of Transmitters and Receivers.

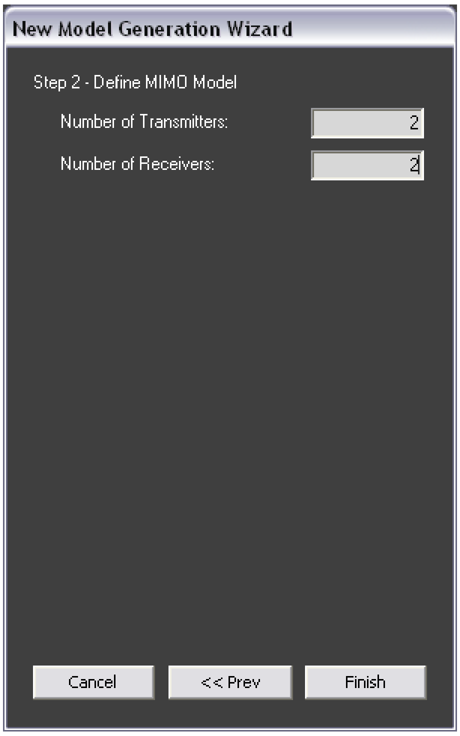


Figure 142 MIMO model definition

7.1.3 Channel Impulse Response Graph

The Channel Impulse Response graph of Channel model view consists of two windows. The lower window is fixed to show the whole impulse response with all existing taps and the upper window the zoomed section of the whole IR.

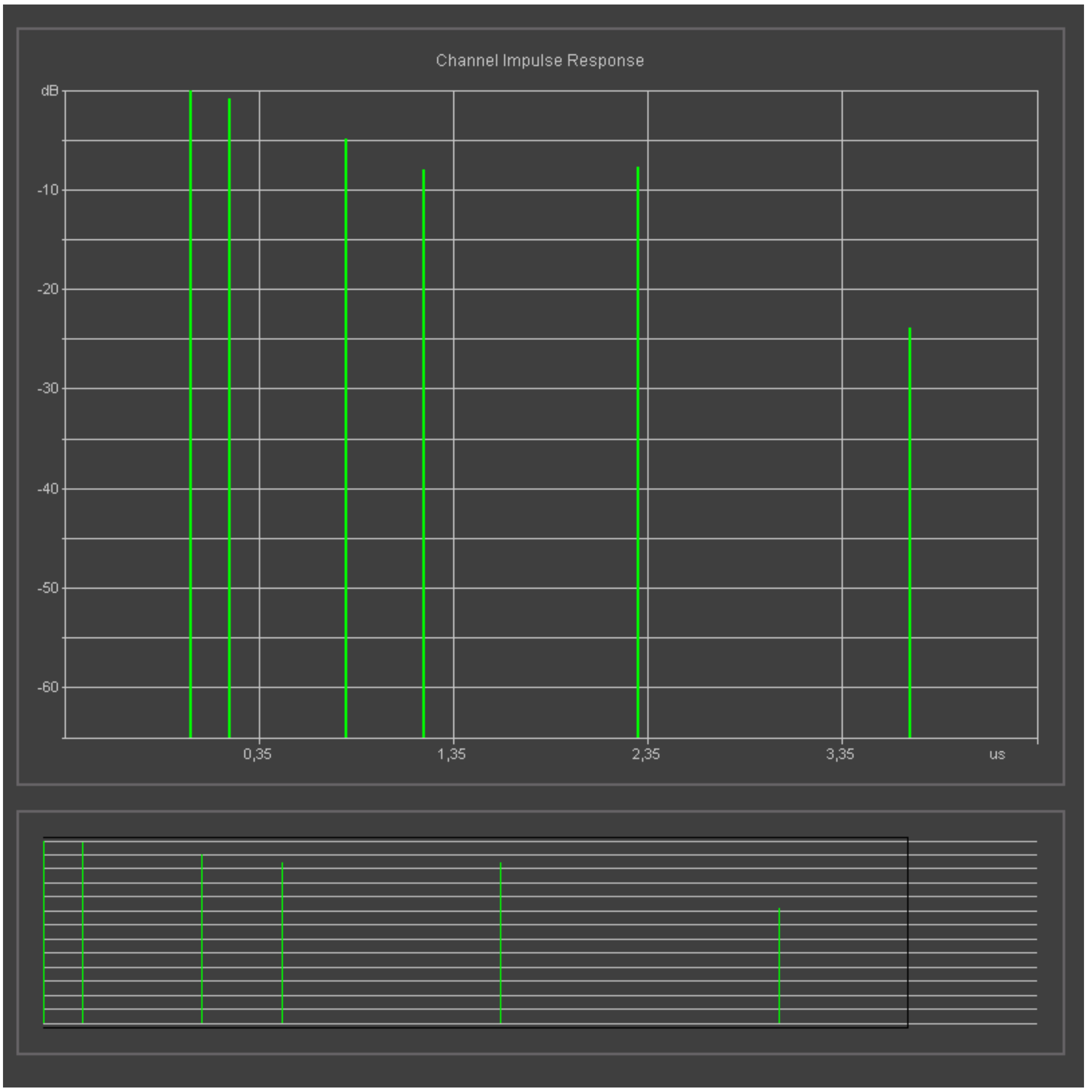


Figure 143 The channel impulse response graph

7.1.4 Channel model settings – Model parameters

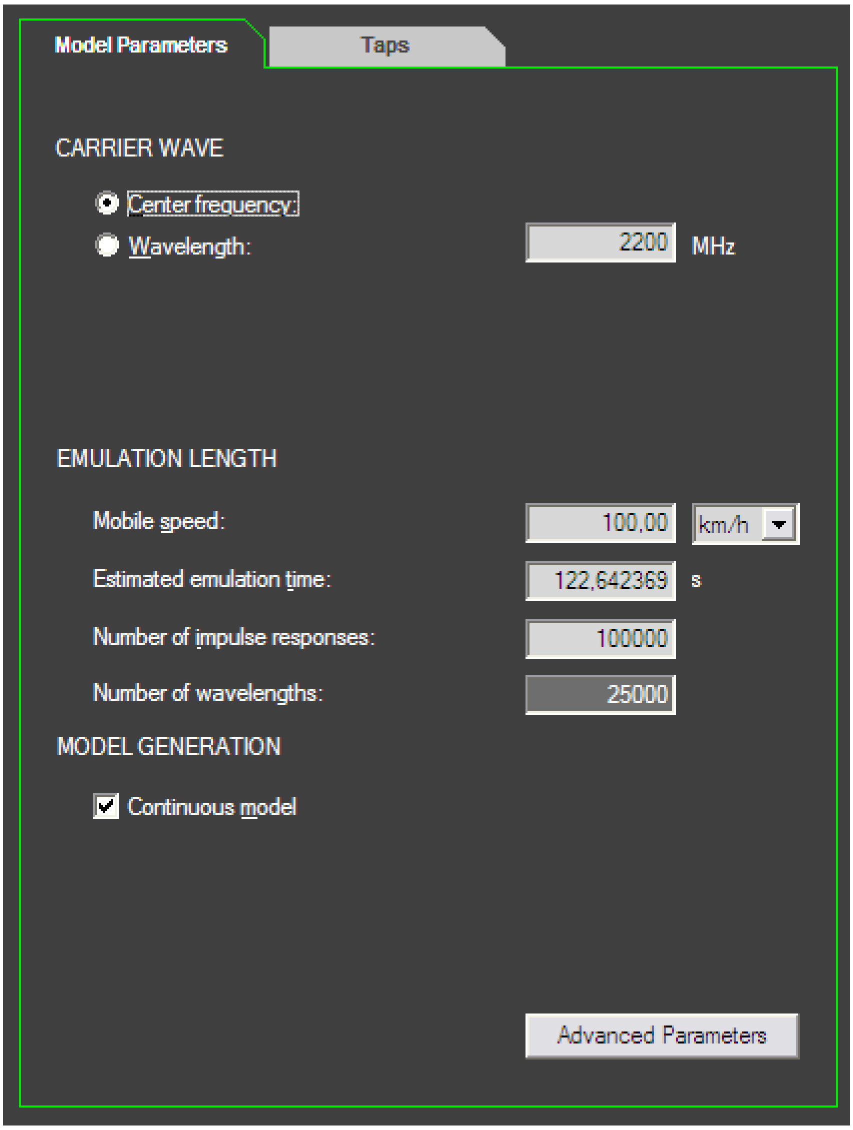


Figure 144 Model parameters

Carrier Wave

The carrier wave parameter can be given either as

- Center frequency [MHz] or
- Wavelength [cm]

(Sampling)

(Sample density / Sample distance)

- Fields are visible when opening models generated earlier with sample density other than 2. Default sample density is 2. Using greater sample density doesn't improve fading quality but increases the model size and building time (and decreases available model length). HW interpolator increases the sample density when emulation is running.

Emulation Length

Mobile speed

- Specifies the emulated mobile speed.

Estimated emulation time

- Estimated emulation time (length), based on number of impulse responses and mobile speed.

Number of impulse responses

- Number of impulse responses in emulation.

Number of wavelengths

- Number of wavelengths in emulation, based on the number of impulse responses and sample density.

Model Generation

Continuous model

- The amount of CIRs is adjusted so that the channel model is continuous from the last CIR to first.

Advanced Parameters

Channels

  - Shows the number of channels.

Distribution seed

- Set the channel model seed. When the same channel model is re-generated with the same seed, the result is the same as the original. If the seed is changed, new channel(s) are not correlating with the original ones.

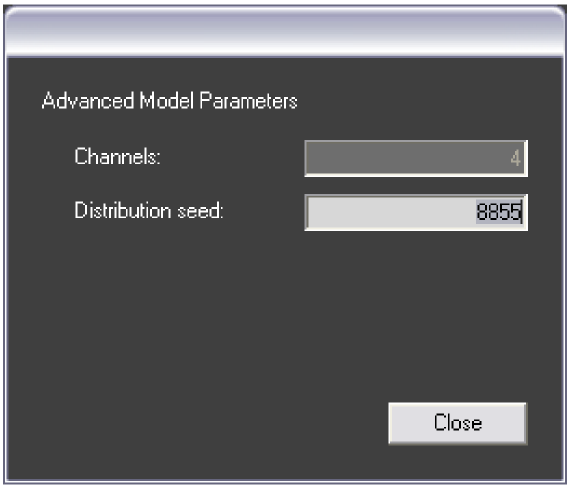


Figure 145 Advanced Model Parameters dialog with one single-channel channel model

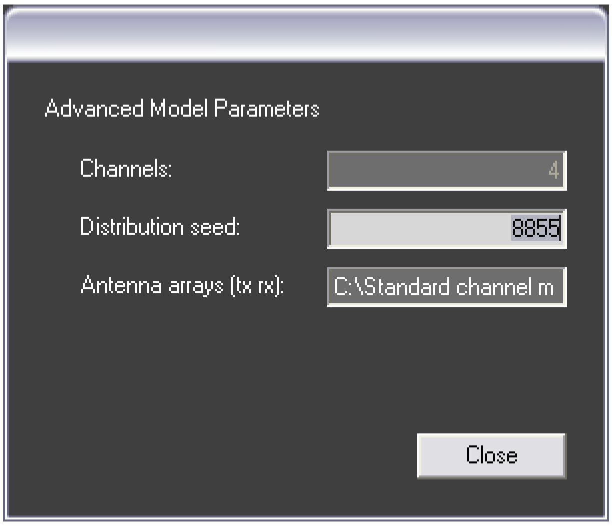


Figure 146 Advanced Model Parameters dialog in Geometric Model mode

7.1.5 Channel model settings – Taps

The Taps tab has a list of channel model taps with three columns:

- Delay shows the delay of the tap. Text "Rand" indicates hopping delay.
- Mean Relative Power shows the mean relative power of the tap.
- Fading drop-down list allows you to change distributions. If the distribution function requires parameters, defaults are used.

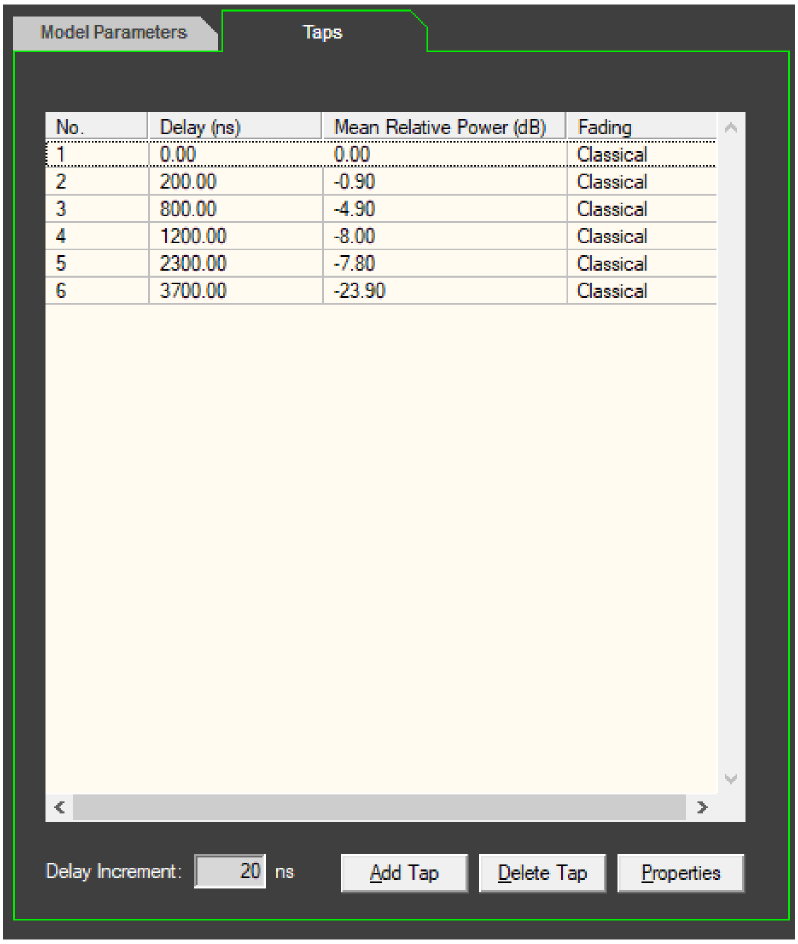


Figure 147 Channel model tap list

Delay increment

- Delay increment for next tap

Add Tap control adds one tap, i.e. one row, to the tap list and sets the default parameters for it:

- Delay: next multiple of Delay increment
- Mean relative power: 0 dB
- Distribution: Classical

Delete Tap removes currently selected tap from the list.

7.1.5.1 Tap Properties

To open Tap Properties dialog, click Properties in channel model tap list.

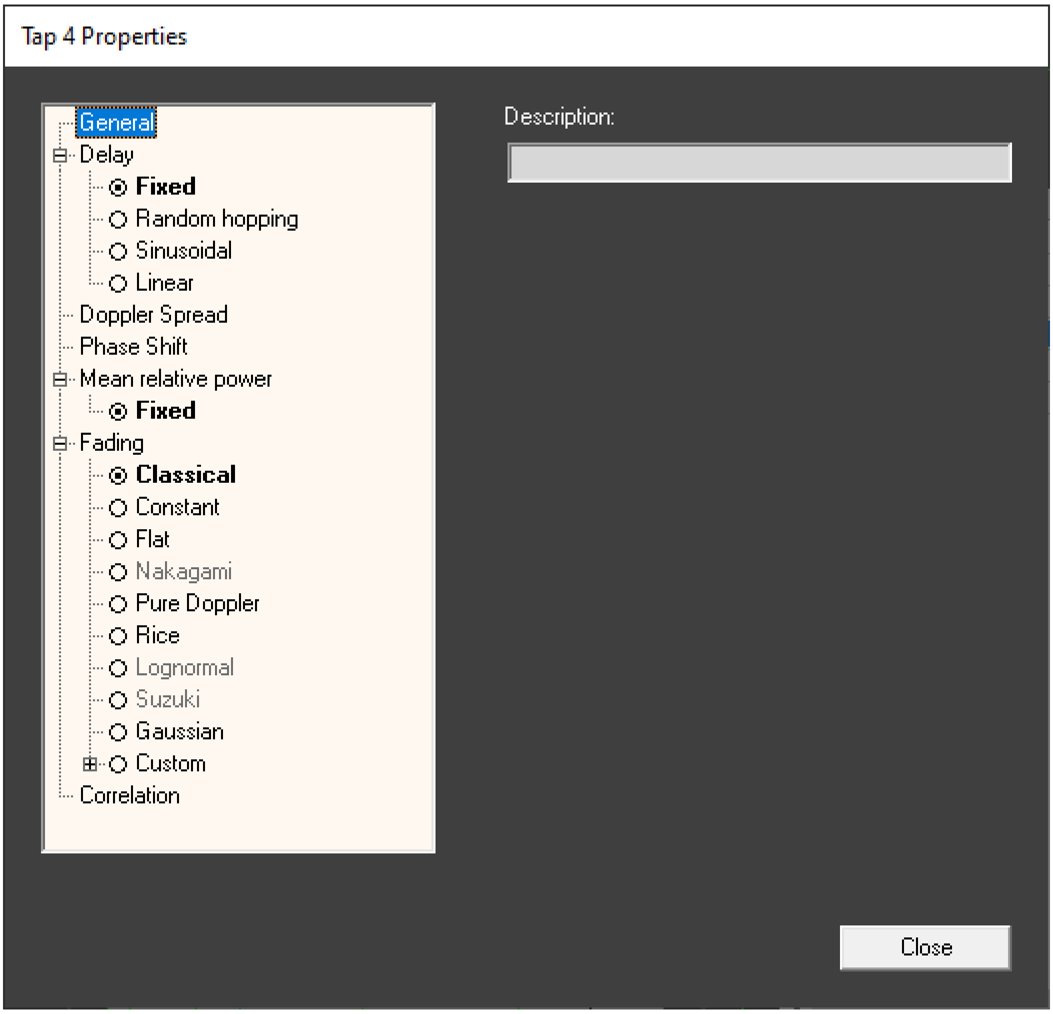


Figure 148 Detailed tap properties dialog

The Properties dialog title shows the number of the selected tap. When this dialog is open, you can select taps on the grid and edit properties.

You can set different properties in the list. The values are as follows:

- General
  - Description field (only informative)

- Delay
    - Delay function indicates the function according to how the delay behaves. Alternatives are: Fixed, Random Hopping, Linear and Sinusoidal.
- Doppler Spread
- Phase Shift
- Mean relative power
- Fading
- Correlation (appears in MIMO and correlating multi-channel cases)
- Geometric Model (appears in geometric multi-channel case only)

If a property is selected and Enter is pressed, the controls related to the active property appear on the right side of the dialog. Controls in the properties are described in the following subchapters.

7.1.5.1.1 Delay

Fixed

With fixed delay, the delay value of the current tap is fixed to the same value during the whole emulation period.

- Parameters:
    - Delay [ns]

Random Hopping

In random hopping, the user defines the delay positions, which are repeated throughout the emulation. Each hopping tap generates a pair of hopping taps. Only one tap is moved at a time and a new time position is selected so that it is neither of the previous ones.

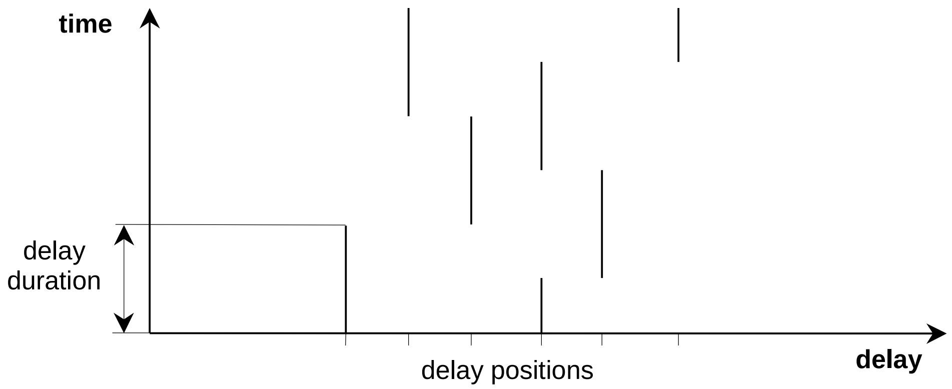


Figure 149 Random hopping delay parameters

- Parameters:
    - Delay duration as milliseconds. The period for keeping taps on one delay position. Note that when lifetime expires, only one tap is moved at a time. This way both taps have an "individual" period of 2x lifetime
    - Table of discrete delay values for tap positions

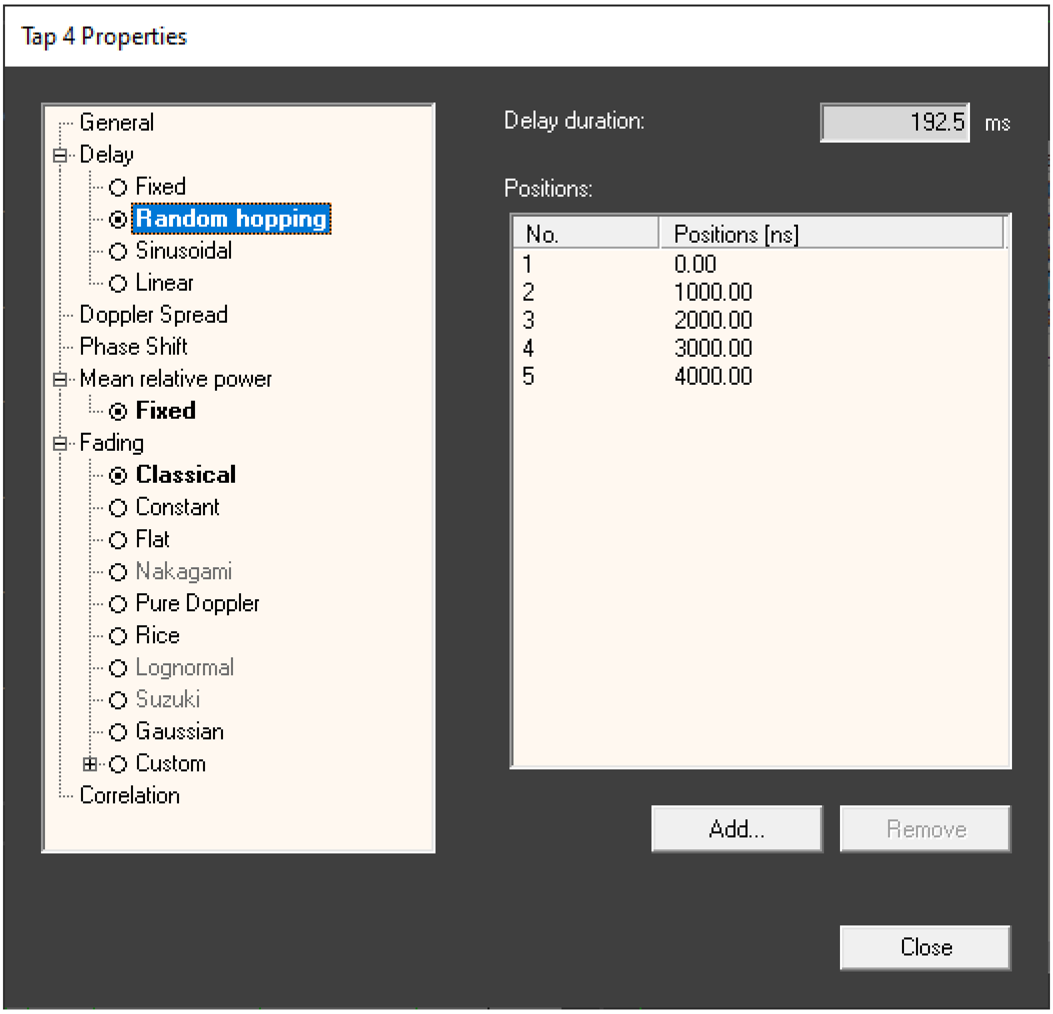


Figure 150 Random hopping properties

Sinusoidal

With sinusoidal delay, the delay of the current tap slides between the defined minimum and maximum delays sinusoidally. The start point of sinusoidal sliding tap is defined by "start phase" parameter. With a default start phase (0 degrees) sliding starts from the middle point of the sliding area. Speed of sliding can be adjusted with the period time, which is the time when the tap has been slid a full period from beginning to end.

Note: Changing mobile speed or system carrier frequency later in Emulation control view changes the sliding period time.

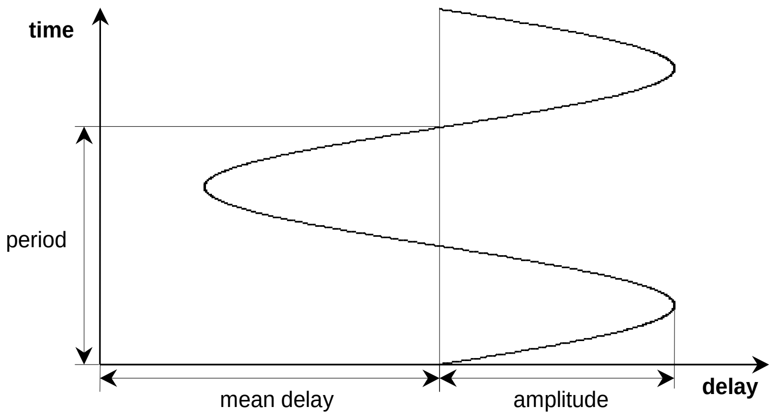


Figure 151. Sinusoidal delay parameters

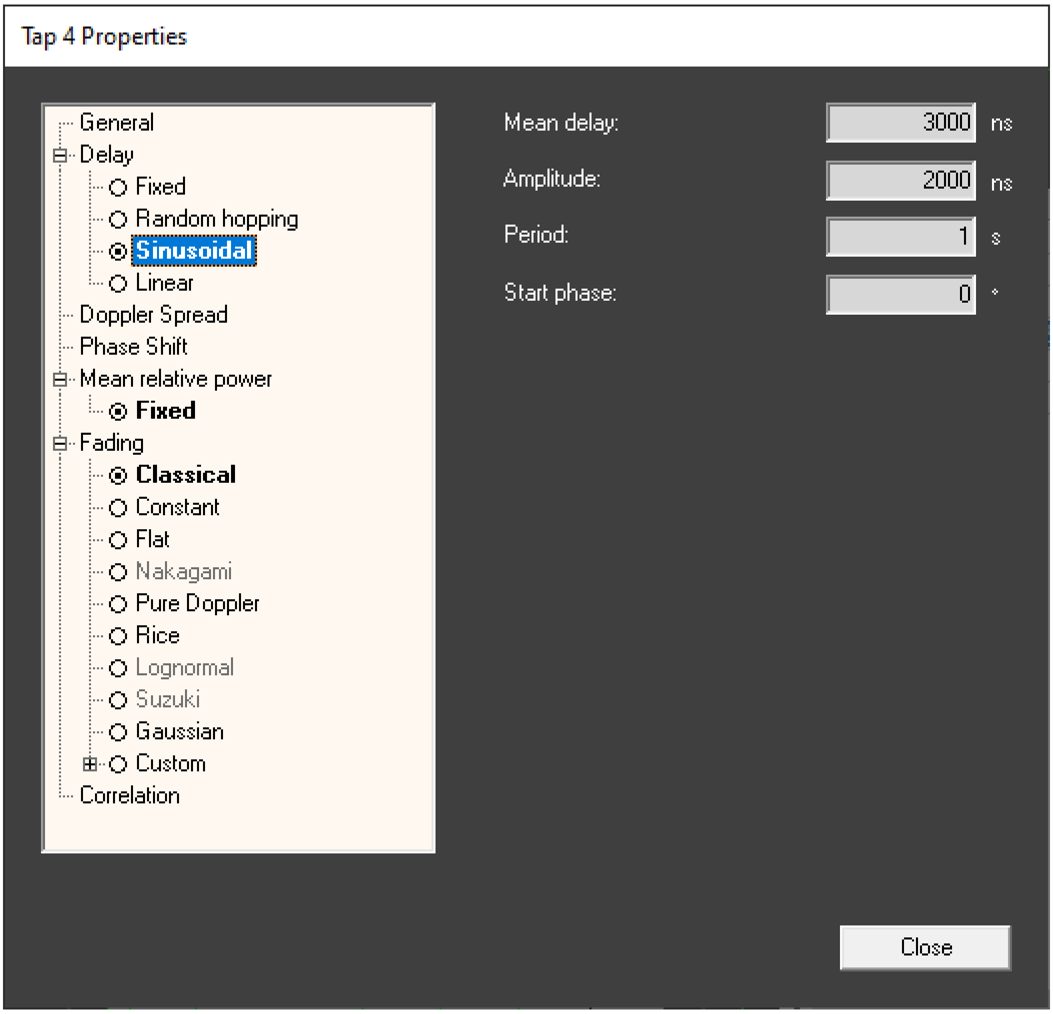


Figure 152 Sinusoidal delay form

Linear

With linear delay, the delay of the current tap slides between the defined minimum and maximum delays linearly. The start point of a linearly sliding tap is the defined minimum delay. Speed of sliding can be adjusted with the period time, which is the time when the tap has been slid a full period from beginning to end.

Note: Changing mobile speed or system carrier frequency later in Emulation Control view will change sliding period time.

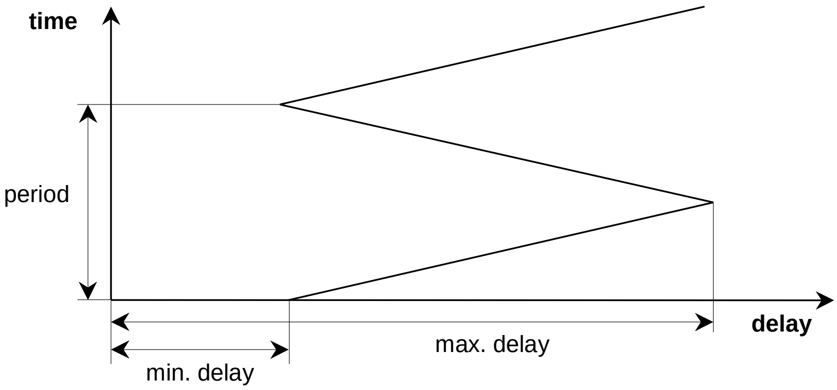


Figure 153 Linear delay parameters

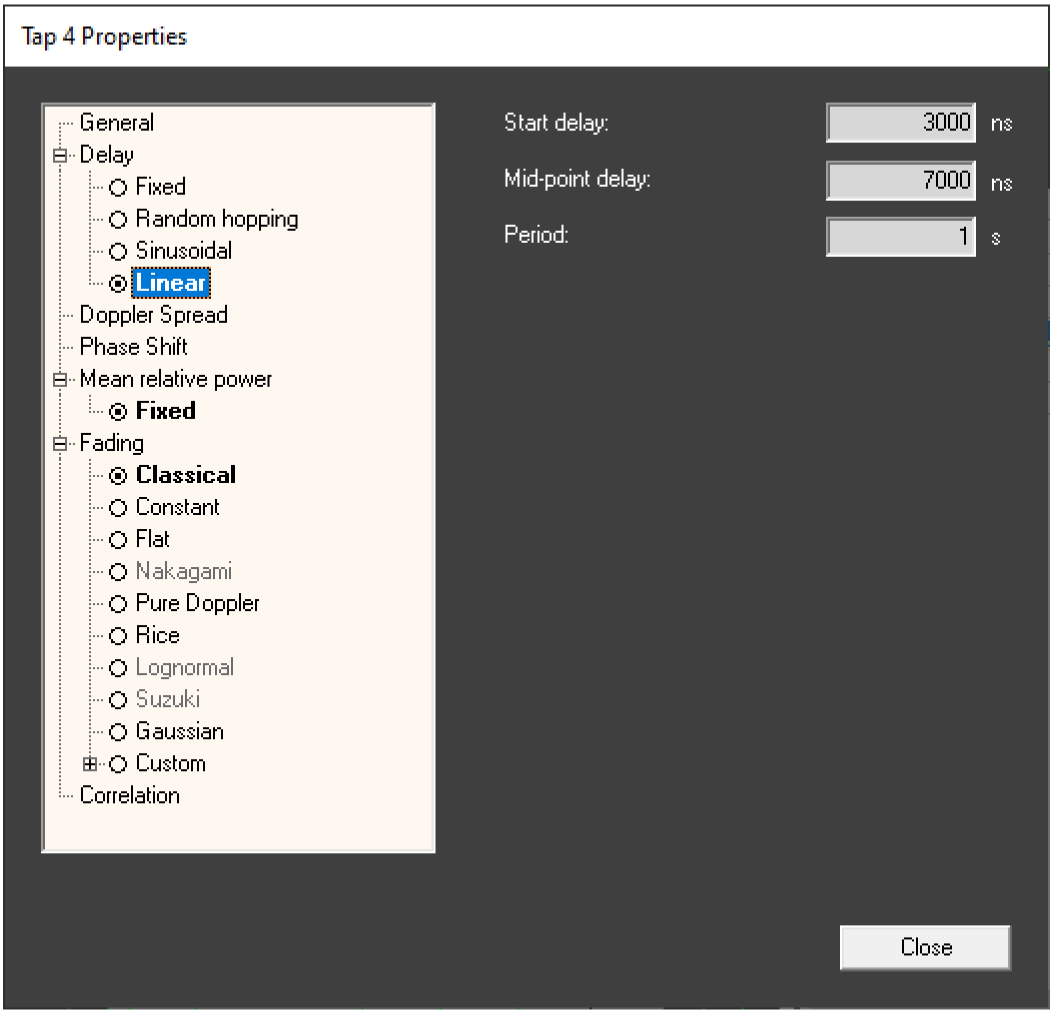


Figure 154 Linear delay form

7.1.5.1.2 Doppler Spread

Tap specific Doppler spread allows narrowing the Doppler of the individual taps compared to the Doppler spread of the model. If taps are desired to have different Doppler spreads, set the model Doppler spread (**Mobile speed** on the **Model Parameter** page, see chapter 7.1.4) to match the tap with the widest Doppler spread. **Reset to default** button sets the Doppler spread to match the model mobile speed.

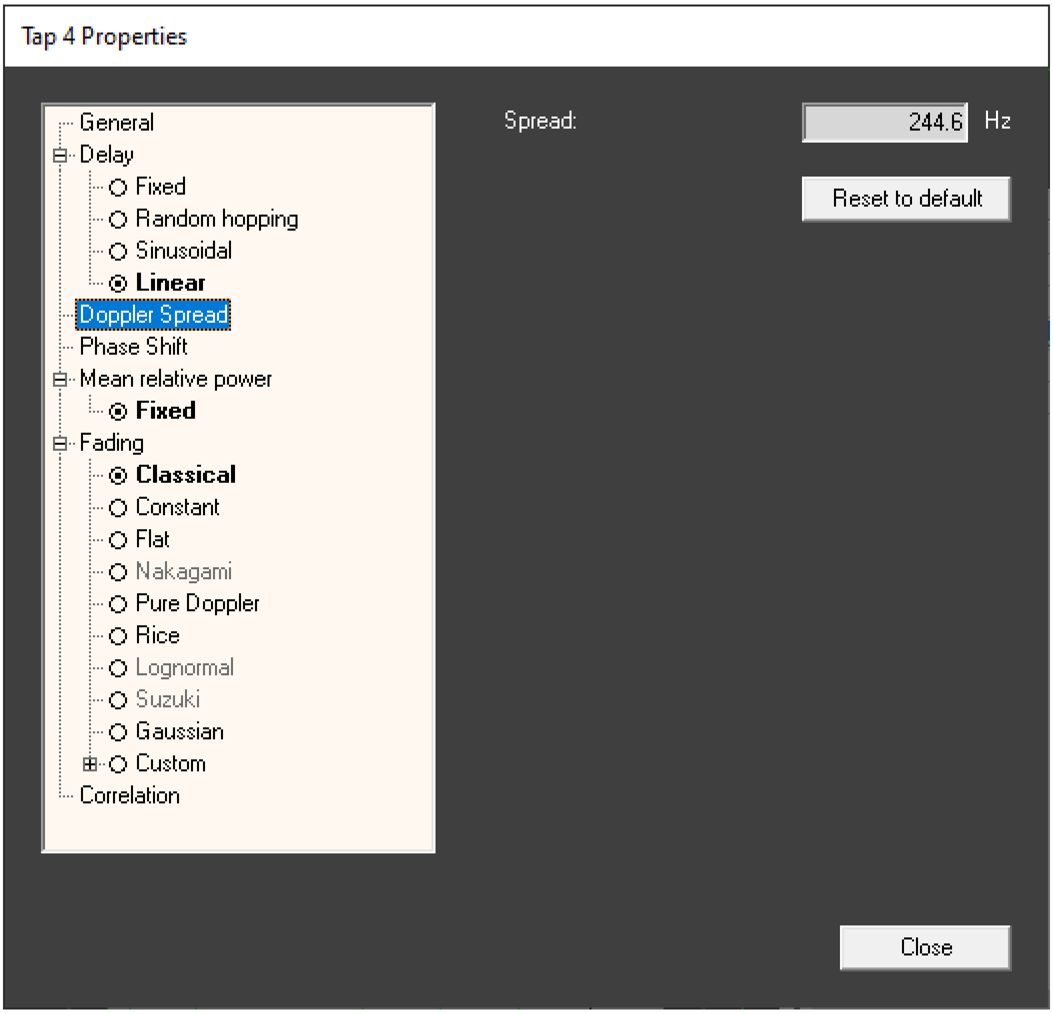


Figure 155 Tap specific Doppler spread

7.1.5.1.3 Phase Shift

Constant phase shift can be applied to any tap, in addition to other fading parameters. The phase shift is given in degrees.

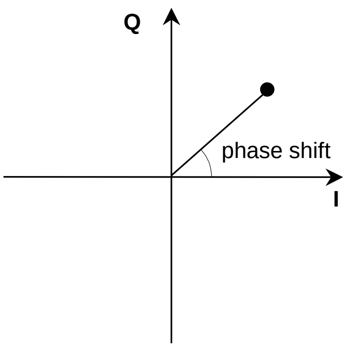


Figure 156 Phase shift

7.1.5.1.4 Mean relative power

The power of a tap in a specific channel impulse response is determined by two factors:

- Fast-fading characteristics determine the short-term variations in signal level.
- Overall mean power determines the level over several fast-fading cycles.

The long term behavior of individual taps can be specified with mean relative power. Currently only a fixed mean relative power can be specified. The specified power is relative to all other taps in all channels that belong to the emulation. For example, in a case of two channel emulation, the real tap power depends on tap powers of both channels: the relative differences between channels are preserved.

7.1.5.1.5 Fading

The user has to select one of the radio channel fading models shown under category. More information about fading types, see Wireless Propagation Environment application note document.

Note: Fading type cannot be selected in geometric model case.

Classical

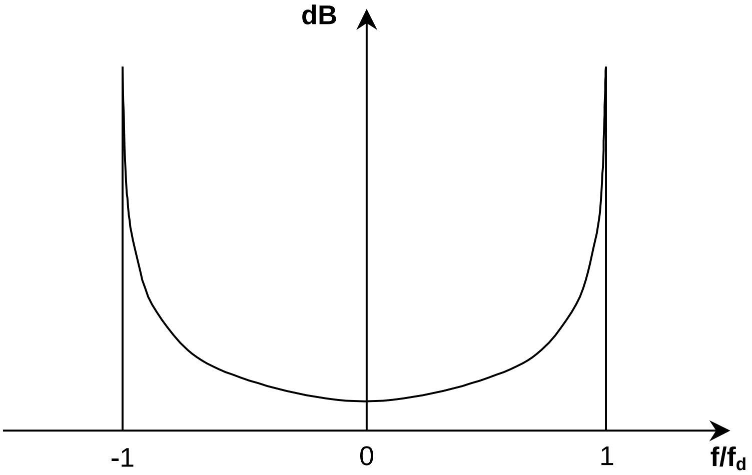


Figure 157 Doppler spectrum of classical fading model (Jakes)

No parameters.

Constant

No fading is applied: tap has constant power.

No parameters.

Flat

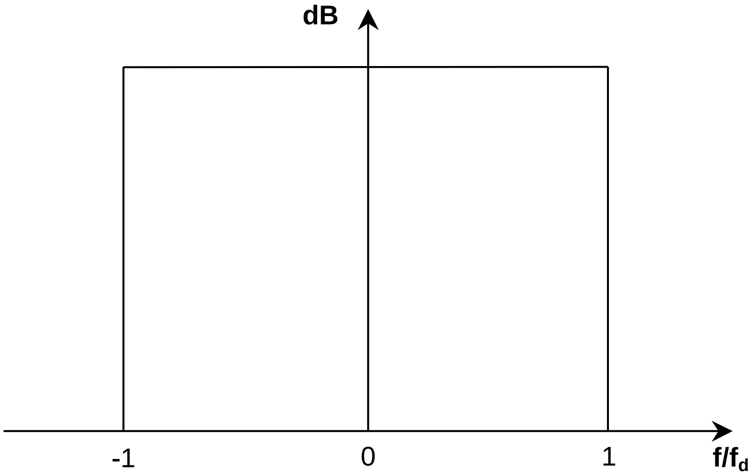


Figure 158 Doppler spectrum of flat fading model

No parameters.

Nakagami

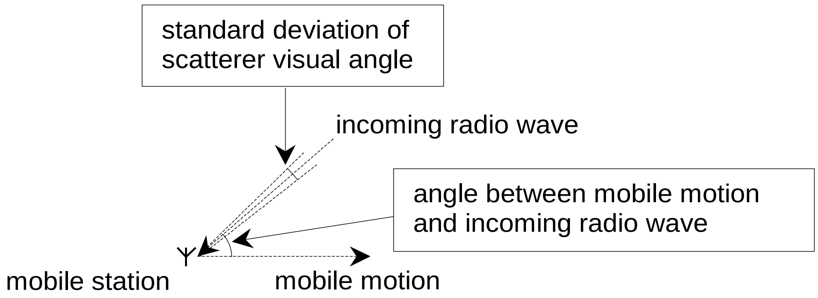


Figure 159 Nakagami parameters

Parameters:

- Angle between mobile motion and incoming radio wave [deg]
- M-parameter, describes the severity of fading, the bigger the M-parameter is the less the channel is fading. More information can be found in Wireless Propagation Environment application note document.
- Standard deviation of scatterer visual angle [deg]

Pure Doppler

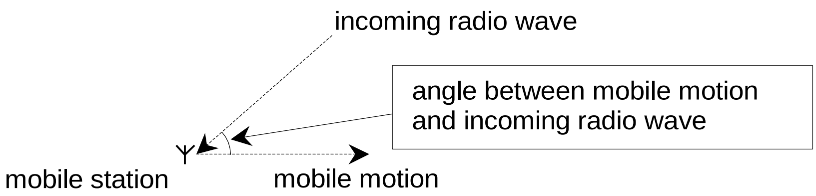


Figure 160 Pure Doppler parameters

Parameters:

- Angle between mobile motion and incoming radio wave [deg]

Rice

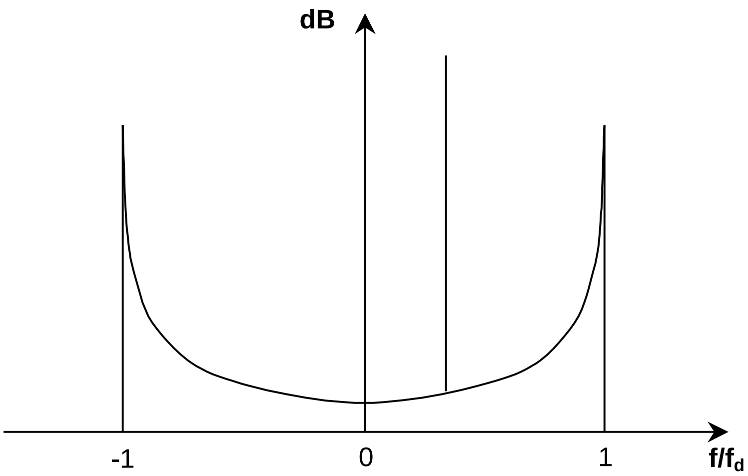


Figure 161 Doppler spectrum of rice fading model

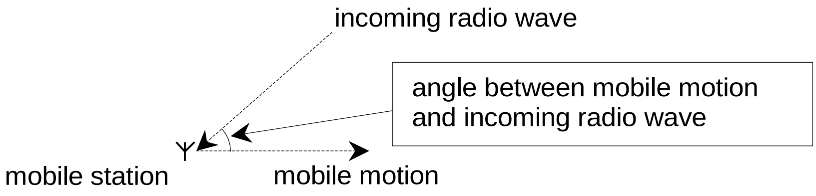


Figure 162 LOS component parameters with rice distribution

Parameters:

- Angle between mobile motion and direct ray [deg]
- Power ratio direct / scattered [dB]

Lognormal

Parameters:

- Standard deviation [dB], describes the amount of fading. Bigger values produce deeper fading.
- Correlation length [m], describes the time correlation of the channel model. After this length correlation is low, i.e. the channel has changed a lot.

Suzuki

Parameters:

- Standard deviation [dB]
- Correlation length [m]

Gaussian

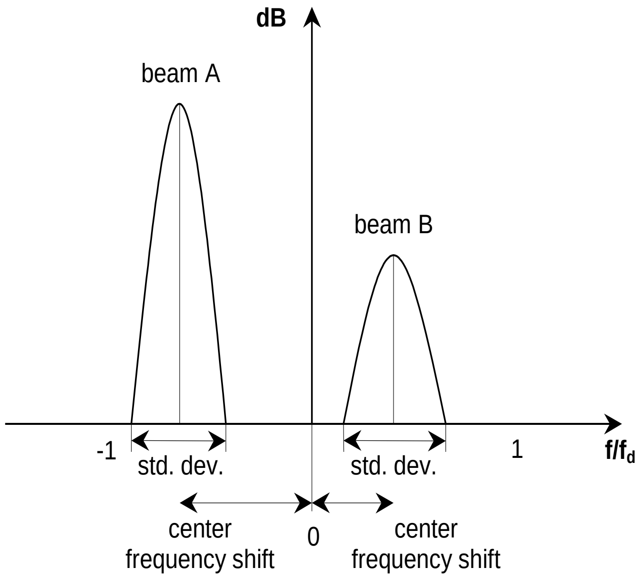


Figure 163 Doppler spectrum of gaussian fading model

Parameters:

- Beam A center frequency shift
- Beam A standard deviation
- Beam B center frequency shift
- Beam B standard deviation
- The power ratio of the beams A and B [dB]

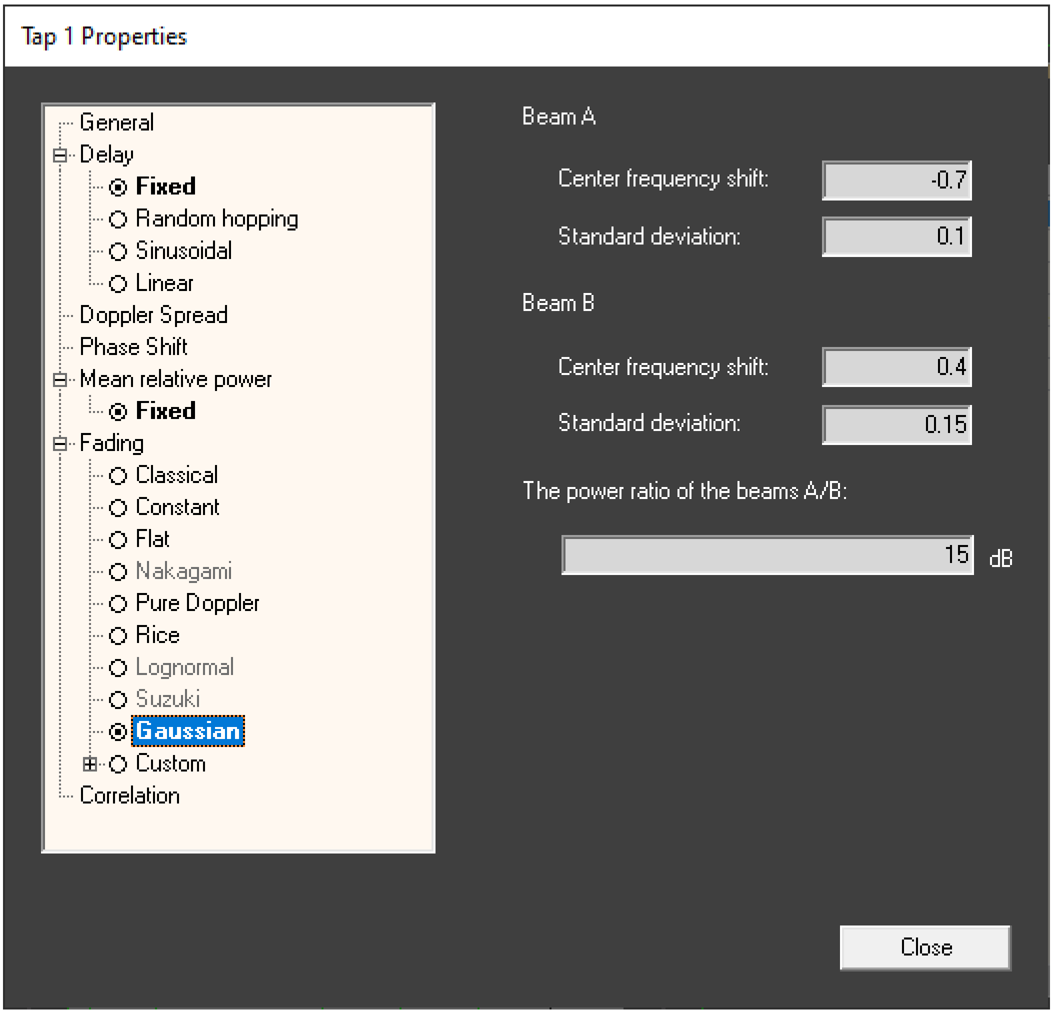


Figure 164 Gaussian fading form

Custom

Amplitude distributions:

- Constant
- Rayleigh
- Rice

Doppler spectrums:

- Pure Doppler
- Jakes
- Gaussian
- Flat
- Butterworth
- Rounded

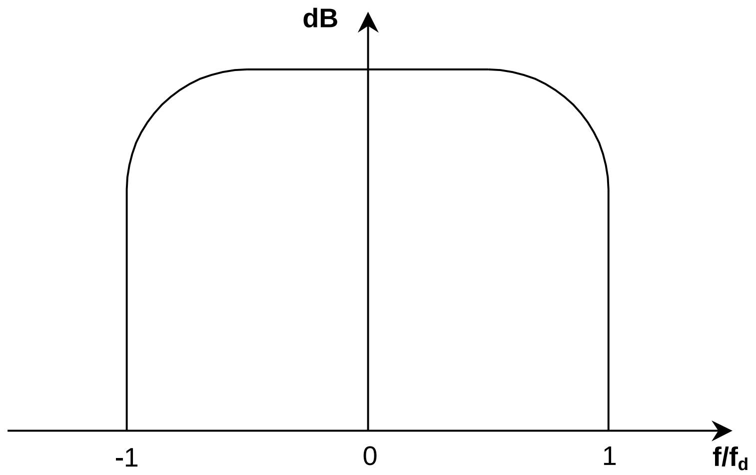


Figure 165 Rounded Doppler spectrum

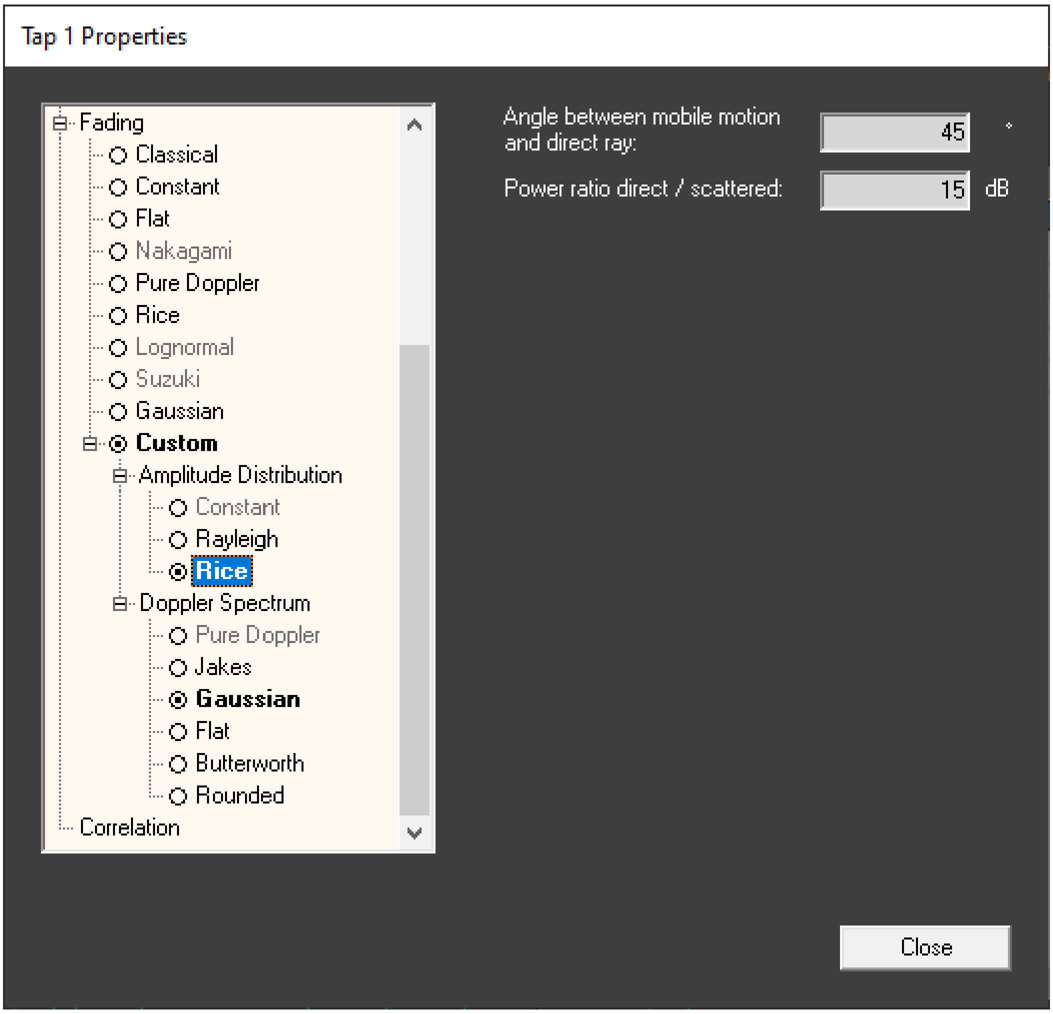


Figure 166 Custom fading (Rice with gaussian Doppler)

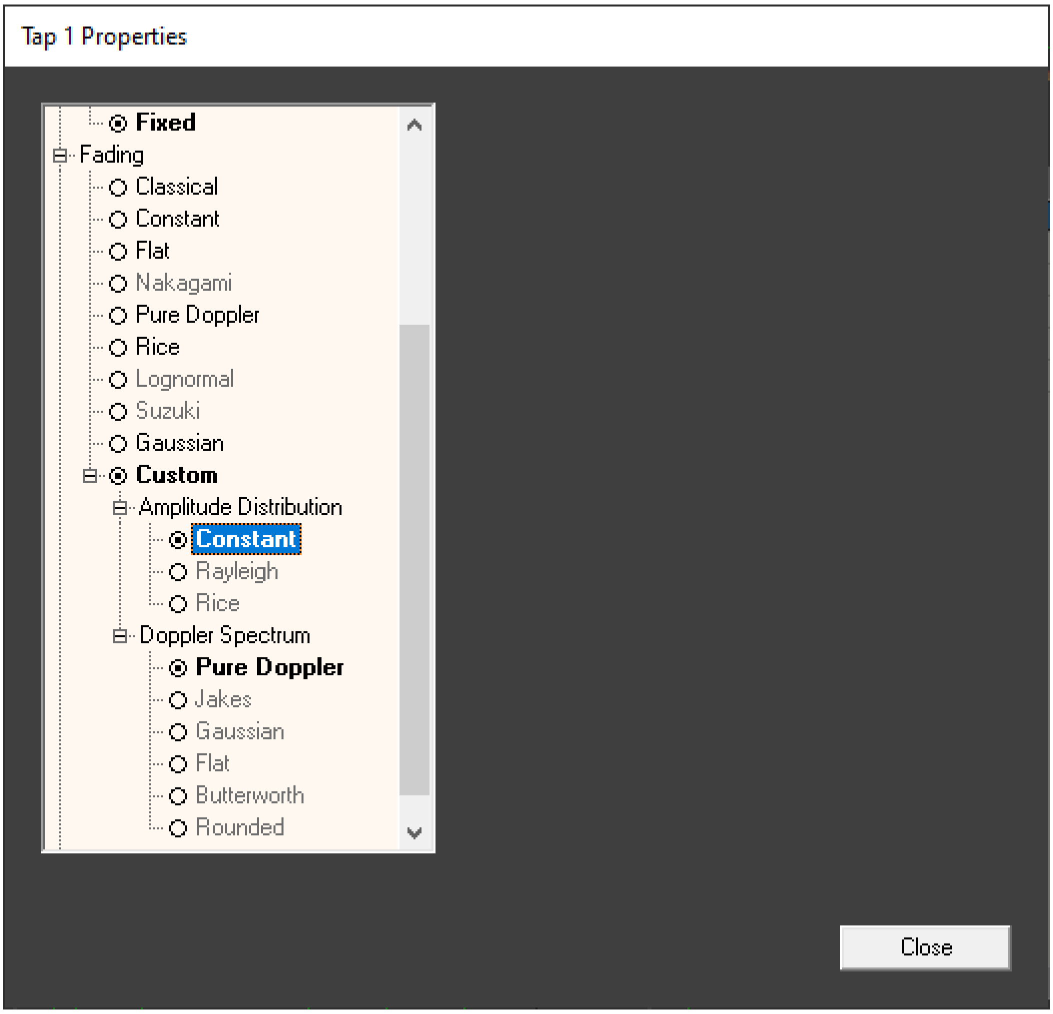


Figure 167 Custom fading (Constant)

7.1.5.1.6 Correlation

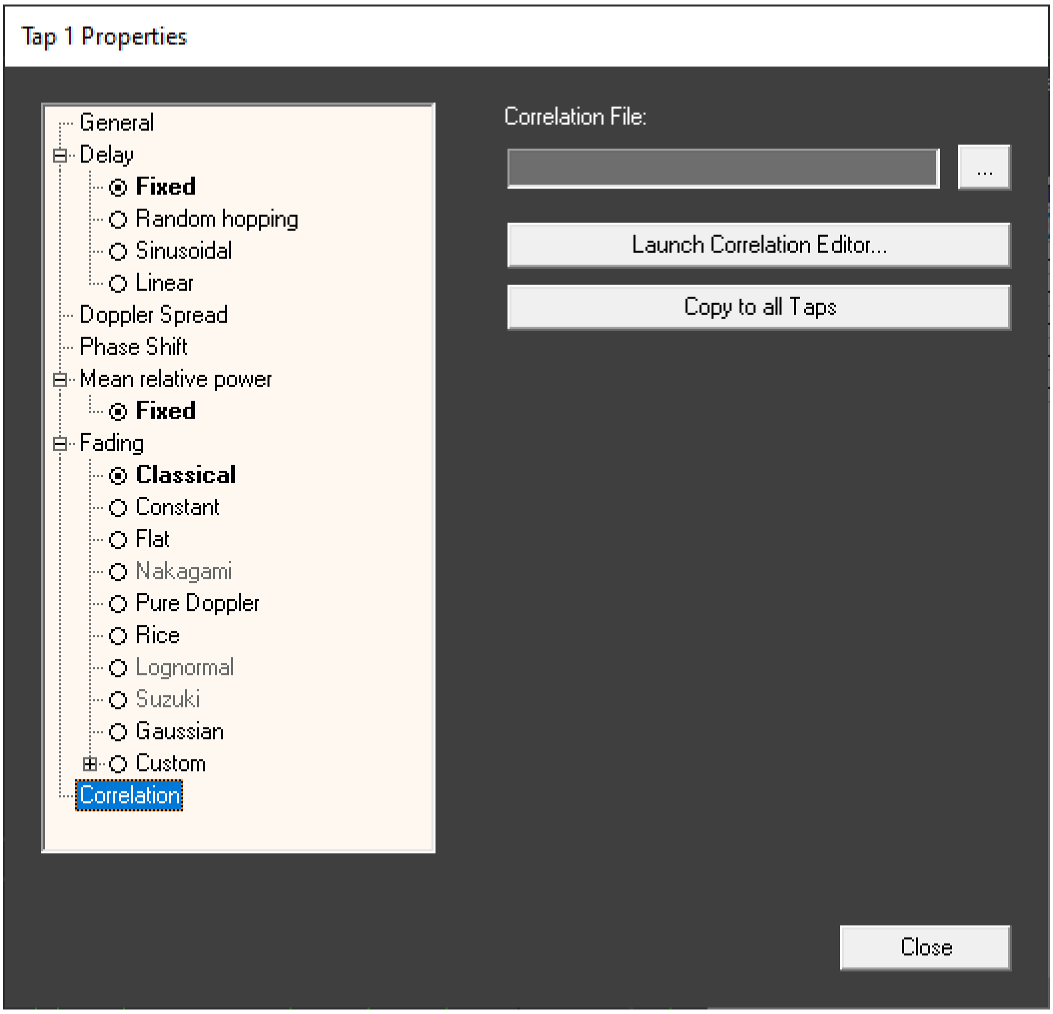


Figure 168 Correlation parameters

These parameters are available, if the current model is a correlation matrix multi-channel model (number of channels > 1) or MIMO model.

Correlation parameters:

- File name. Name of the file where the correlation matrix is stored
- The correlation matrix can either be selected from the existing correlation matrices or created from scratch. If the existing matrix is loaded, the type of the model is checked. If correlation file type does not match, editor refuses to load it.

To create a new correlation matrix, use **Launch Correlation Editor** button. To copy single tap's correlation matrix to all taps, use **Copy To All Taps** button, if the same matrix is used by all channels.

7.2 Correlation Editor

The Correlation Editor (Figure 169) is used to create and edit an amplitude or complex correlation matrices. The correlation matrix defines correlation coefficients between channels.

To start the Correlation Editor, select **Utilities > Correlation Editor** in the navigation bar or click **Launch Correlation Editor...** in the **Tap Properties** dialog of the **Channel model** view. See Figure 168.

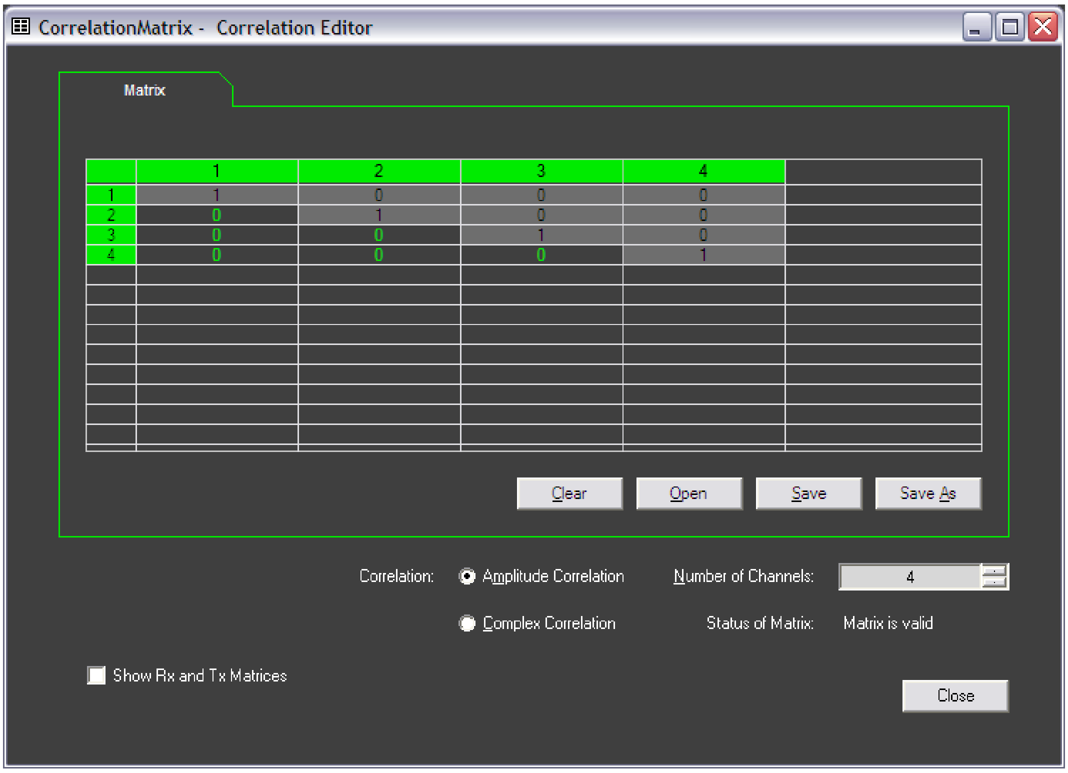


Figure 169 Correlation Editor

Matrix

- The correlation coefficients are viewed and edited in this diagonally symmetric matrix.
- Correlation matrix is Hermitian (NxN), where N is the number of channels. Values on the diagonal [c11 c22 c33 c44] are '1' and in general case other entries are complex. Therefore, it is enough to set the correlation values to the lower triangular of the matrix. The upper triangular values in Correlation Editor are read-only.

$$C = \begin{bmatrix} 1 & c_{21}^* & c_{31}^* & & c_{N1}^* \\ c_{21} & 1 & c_{32}^* & \cdots & c_{N2}^* \\ c_{31} & c_{32} & 1 & & c_{N3}^* \\ & \vdots & & \ddots & \\ c_{N1} & c_{N2} & c_{N3} & \cdots & 1 \end{bmatrix}$$

- Any coefficient values defined on the matrix will be retained even if the number of channels is increased afterwards.

Correlation

- Either Amplitude Correlation of Complex Correlation can be selected.
- Number format:
  - Amplitude Correlation 0.1234
  - Complex Correlation 0.1234+0.3220i
  - The decimal separator can be a '.' or ',' depending on the country settings.
  - Note that the editor does not accept complex numbers in Amplitude Correlation mode.

Number of Channels

- The size of a matrix can be set from 1 to 128 channels.

- Note that when Correlation Editor is launched from Channel model view, channel count is locked to number of channels in channel model and cannot be changed.

Status of Matrix

- Indicates whether the matrix entered is valid:

Not Valid	Matrix is not nonnegative definite, which means that correlation between channels is not realistic. Change some of the values, so that the matrix becomes valid.
Valid	Matrix is nonnegative definite, which means that correlation between channels is OK, and the matrix can be used by the TapToIrCompiler.

Show Rx and Tx matrices

- Show or hide Rx and Tx matrices.

Calculate Kronecker

- Kronecker calculation of the result matrix can be enabled or disabled.

7.2.1 Editing the correlation coefficients

To enter or replace a coefficient value, click on the desired item in the editable area (indicated with graphite grey background by default). An edit field and text cursor will appear.

An alternative method is to select the desired line with the up and down arrow keys and then press Enter. Any subsequent press of Enter will select the next editable location. The mouse and key controls are listed in Table 14.

Table 14 Correlation editor mouse and key controls

Mouse	Operation
Left click	Select a cell
Double click	Edit a cell

Key	Selection mode	Editing mode
Up	advance cell up	advance cell up
Down	advance cell down	advance cell down
Left	advance cell left	advance char/cell left
Right	advance cell right	advance char/cell right
Enter	edit next cell down	edit next cell down, wrap
Tab	---	edit next cell right, wrap
Shift+Tab	---	edit next cell left, wrap
Escape	no operation	cancel edit
F2	toggle edit mode	Toggle select mode
Ctrl+Tab	no operation	next dialog item

Notes:

- These commands only apply when the number of channels is 3 or more.
- While in Edit mode, the Left and Right Arrow keys advance the cursor one character at a time until the beginning or end of the text is reached. When there is no more text in the desired direction, the next cell will be edited.

7.2.2 Correlation Editor in MIMO mode

When a MIMO model has been defined in Channel model view, the Correlation Editor opens in MIMO mode, see Figure 168. This differs from the normal model so that matrices can be entered for both transmitter and receiver arrays and the editor calculates the correct result as Kronecker product from these matrices.

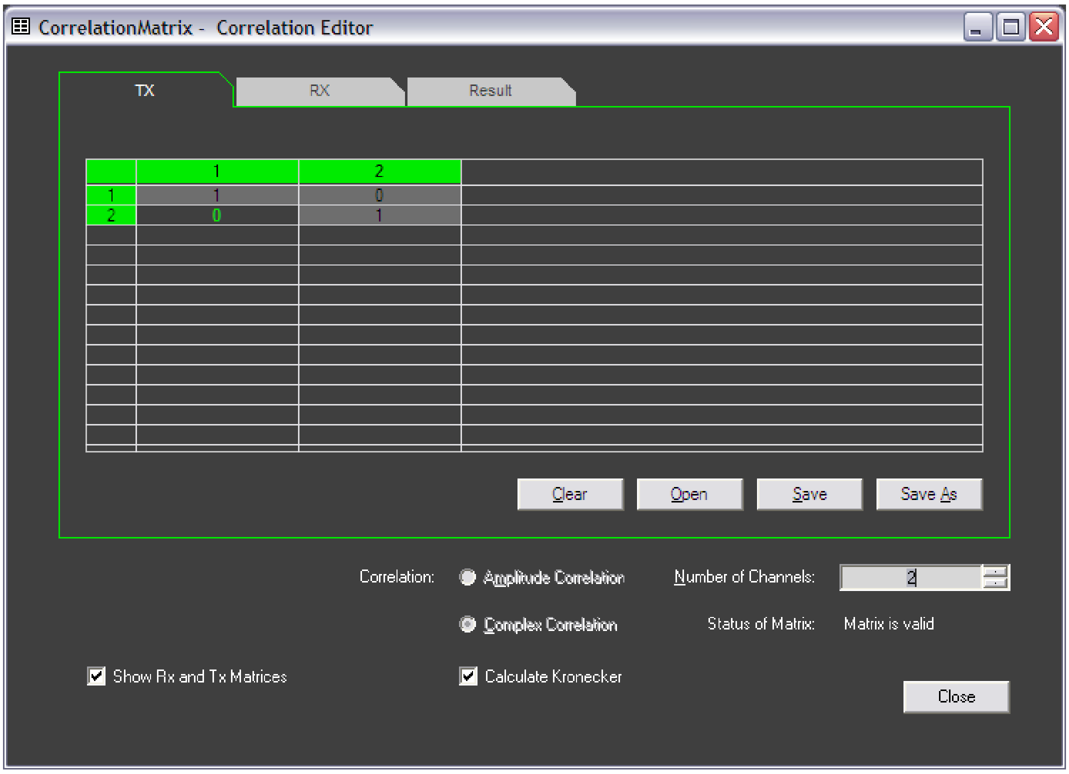


Figure 170 MIMO correlation editing

You can also enter the MIMO mode by opening the Correlation Editor in stand-alone mode and by loading a MIMO result matrix saved earlier with the editor. This loads all the three matrices simultaneously and editing can be continued.

The editor has three tabs: **Tx** Matrix, **Rx** Matrix and **Result** Matrix. The **Result** Matrix is a Kronecker Product of the Tx and Rx matrices and it is recalculated when you switch to the **Result** Matrix page. To disable the calculation of Kronecker product, uncheck the **Calculate Kronecker** check box. The calculation of the Kronecker can be disabled when a ready-calculated final result matrix is defined directly to **Result** Matrix, instead of using **Tx** and **Rx** source matrices.

All the matrices can be edited when page view is visible, but the **Result** Matrix will revert to calculated values when you switch back to the **Result** Matrix page, unless **Calculate Kronecker** has been disabled.

As a result of Kronecker product calculation, result matrix is a special MIMO matrix (when saved to file as .COR file) that contains all the **Tx**, **Rx** and **Result** matrices that can be loaded to the Channel model view. When MIMO matrix is saved, all matrices are saved to the result file. It is not possible to save only **Tx** or **Rx** matrices.

Because MIMO matrix file contains three matrices - **Rx** matrix, **Tx** matrix and the **Result** matrix - the editor asks you which of the matrices you want to load to current page view. See Figure 169. Note that it is not possible to load a normal matrix as MIMO Tx or Rx matrices; loading the matrix will lose MIMO matrix.

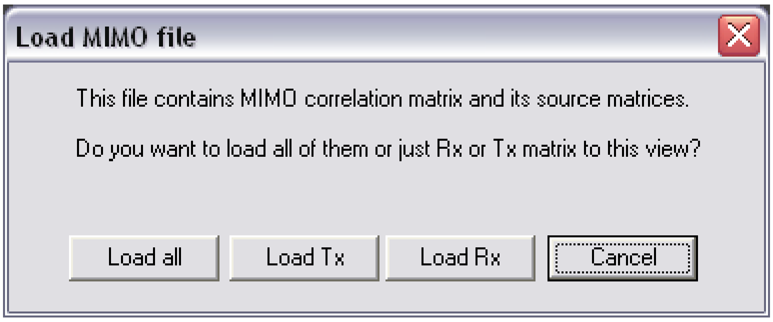


Figure 171 Loading MIMO file

To make the editor load all three matrices and transfer them to MIMO mode, select the **Load all option**.

If you change **TX** or **RX** matrix and click **Result matrix** tab, you must choose whether to update result matrix or not.

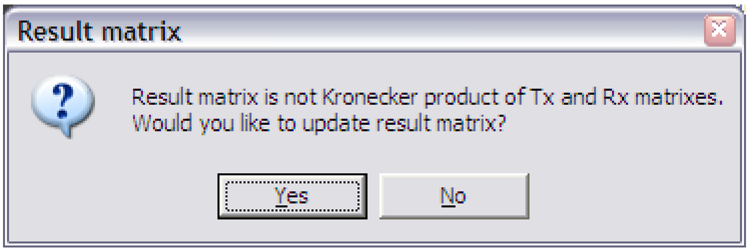


Figure 172 Result matrix update

7.3 Multi Emulator Scaler

PROPSIM scales the output gain of the emulator optimally for each emulation when the emulation is built. The scaling is dependent of the emulator hardware characteristics and the results can thus vary between emulators even for the same emulation. The Multi Emulator Scaler (Figure 173) can be used to scale the output gains of different emulators to match for multi-emulator use.

The emulation files (.SMU) of each emulator in the multi-emulator setup are provided to the Multi Emulator Scaler. The Scaler creates new .SMU files with modified output gain value for each emulation.

Optionally also scaler calibration file (.XML) can be provided for each emulator, see chapter 21.4. The calibration file includes calibration values (dB) for each input and output of the emulator. It can be used e.g. to compensate cable losses for different channels.

The generated .SMU files are stored in the same folder the original .SMU files were stored in. A backup folder is created for the original files “<path to .SMU files>\Backup”.

Multi Emulator Scaler can be started by selecting **Utilities > Multi Emulator Scaler** in the navigation bar.

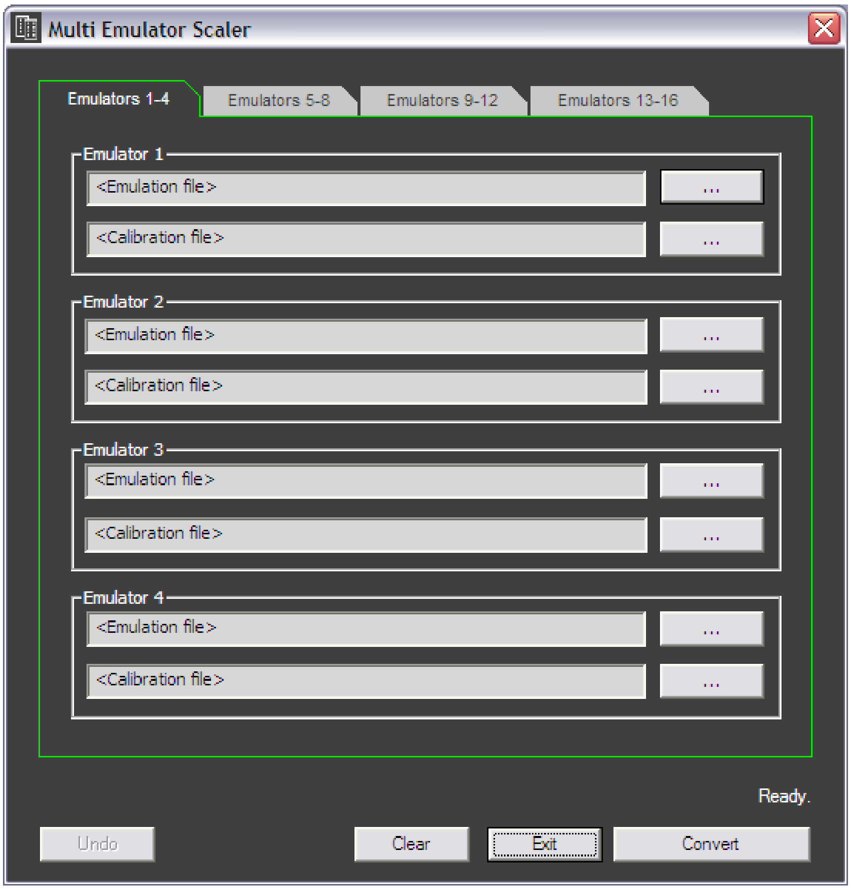


Figure 173 Multi Emulator Scaler

7.3.1 Using scaler tool

Scaler tool can be used in GUI or CLI mode. To start program in CLI mode, you must provide emulation files and optional calibration files as command line parameters. Without command line parameters, program starts in GUI mode.

In both operation modes, the scaler tool writes a new .SMU file for each emulation. Original .SMU files are stored inside the folder "<path to .SMU files>\Backup".

7.3.1.1 Command line usage

In Command Line Interface (CLI) mode, you must provide emulation files as command line parameters. Optionally, also the calibration files can be provided. If no parameters are provided, the program starts in GUI mode (see chapter 7.3.1.2).

Scaler tool usage in CLI mode:

```
C:\>Scaler.exe <SMU for emulator 1> <XML for emulator 1> … <SMU for
emulator n> <XML for emulator n>
```

Calibration files (.XML) are optional.

7.3.1.2 GUI usage

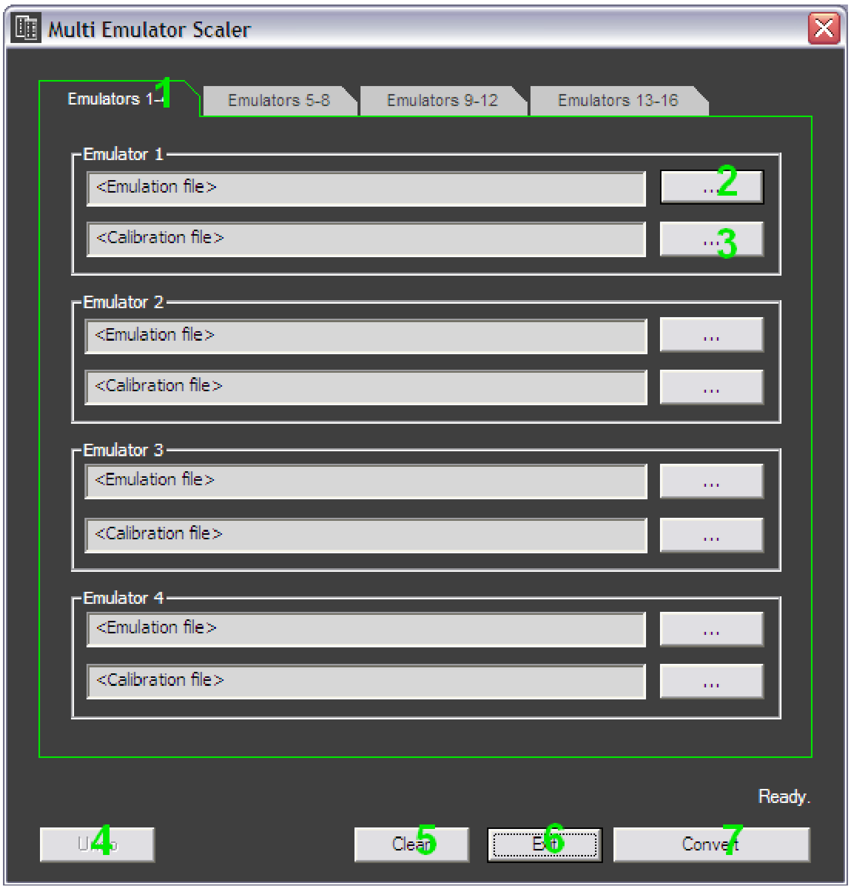


Figure 174 Multi Emulator Scaler GUI components

Scaler tool GUI components in Figure 174:

1. Tab pages for emulations. Each tab page contains input fields for 4 emulators. Currently maximum number of emulators is 16.
   Note: When using synchronous start of multiple PROPSIM F64 emulators, supported emulator count is 2.
2. Button for browsing .SMU file for selected emulator.
3. Button for browsing .XML calibration file for selected emulator. For more information on the file formats, see chapter 21.
4. Undo converting.
5. Button to clean text boxes.
6. Button to exit application.

7. Button to convert .SMU file.

7.4 Emulation Batch Builder

Emulation Batch Builder (Figure 175) is a tool for building multiple emulations at the same time. The Emulation Batch Builder can be started by selecting Utilities > Batch Builder in the navigation bar.

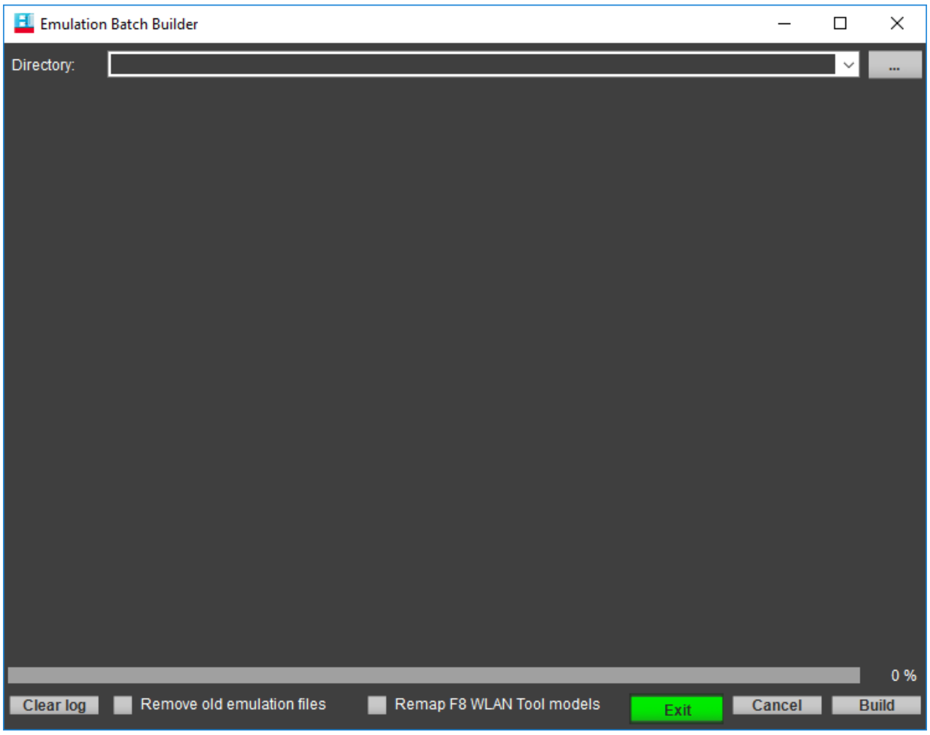


Figure 175 Emulation Batch Builder.

7.4.1 Usage

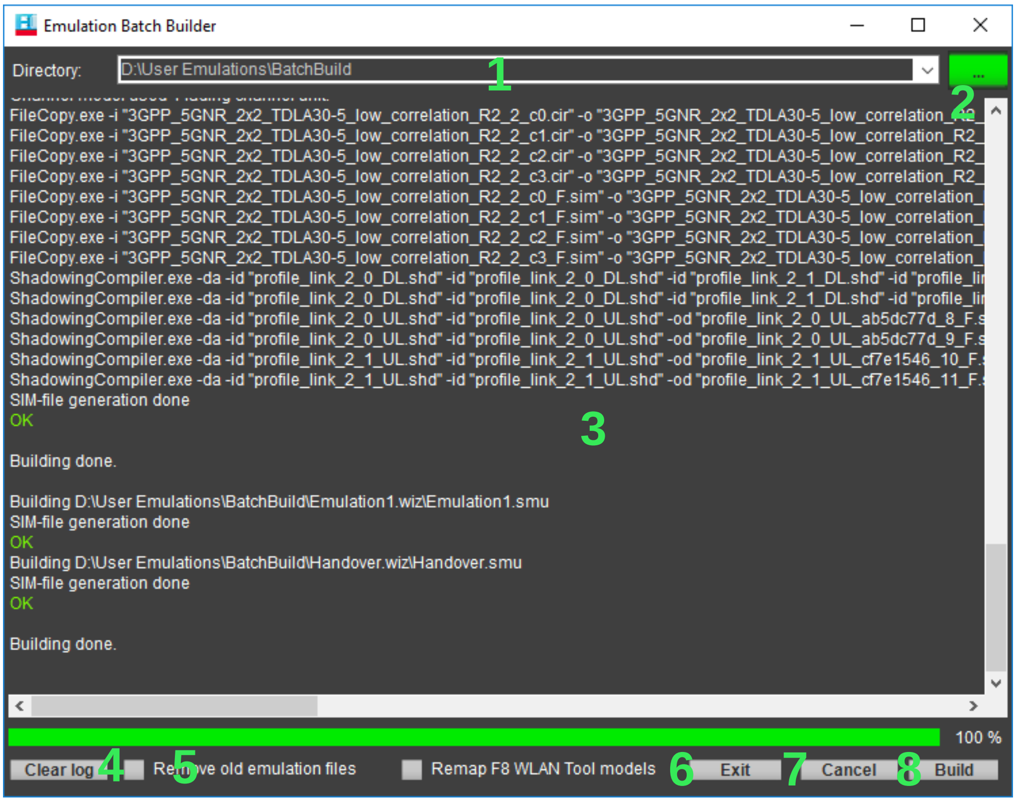


Figure 176 Emulation Batch Builder GUI components.

Emulation Batch Builder GUI components in Figure 176:

1. Currently selected directory. Use drop down menu for directory history.
2. Button for browsing emulation directory.
3. Build log.
4. Button for clearing the build log.
5. Check box for option to delete previous emulation files before building.
6. Button for exiting the application.
7. Button for canceling current build process.
8. Button for starting build process for selected directory.

7.5 IR and ASC converter

IR and ASC converter allows converting between .IR and .ASC files. Figure 177 shows the dialog for the conversion. Conversion can be done to both directions: IR to ASC and ASC to IR. After selecting files and pressing convert -button, conversion status, result and possible errors are shown in the Conversion log -field.

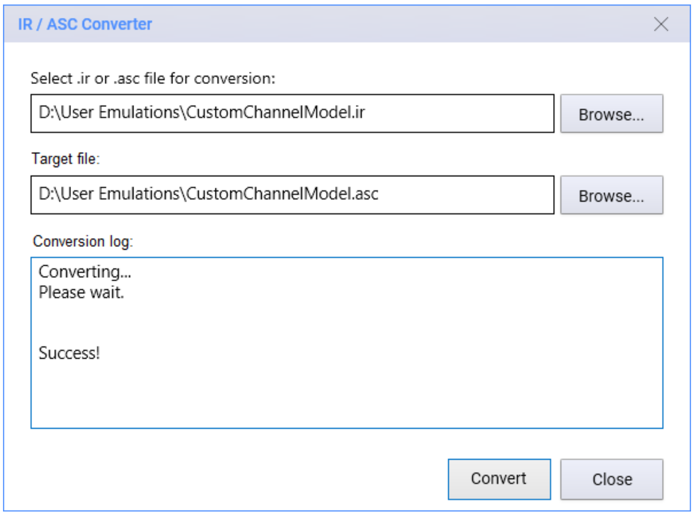


Figure 177 IR / ASC Converter settings

7.6 Creating and restoring a backup

PROPSIM Backup Tool can be used for backing up and restoring PROPSIM user data.

Following data is backed up:

- PROPSIM user data (“D:\” -drive contents)
- Lab setups (“LabSetups” -folder)
- Phase and gain auto alignments (“Autocalibration” -folder)
- Licenses
- Installed Scenario Packs
- Standard Emulations
- PROPSIM settings

To start the tool, select **Utilities > Backup/Restore** in the navigation menu. **PROPSIM Backup Tool** main menu is displayed when the tool is started (Figure 178).

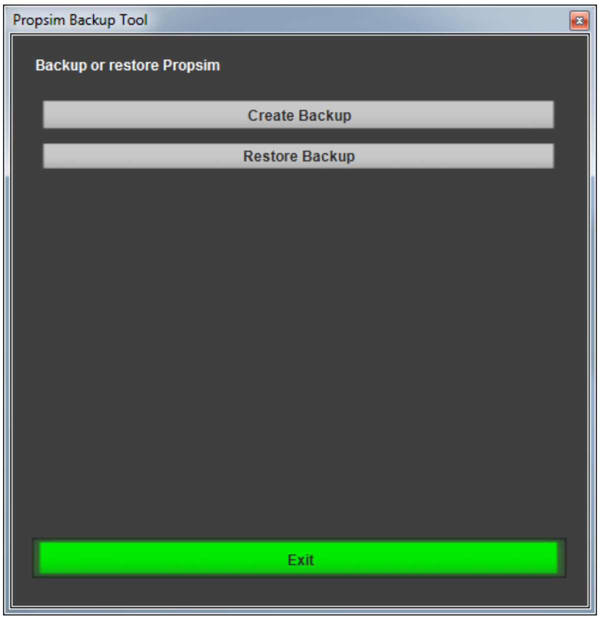


Figure 178 PROPSIM Backup Tool main menu.

7.6.1 Creating a backup

To create a backup, select **Create Backup** from **PROPSIM Backup Tool** main menu. The PROPSIM Backup Tool backup dialog (Figure 179) is shown. Set the location and name of the backup and click **Create Backup** to start the backup process.

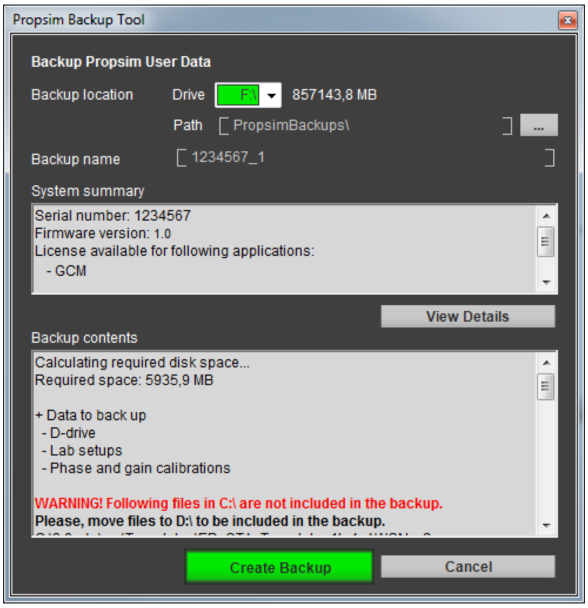


Figure 179 PROPSIM Backup Tool backup dialog.

PROPSIM Backup Tool **Create Backup** contains following settings and information:

- Backup location
  - Describes the drive and path where the backup is saved.
  - By default, the backup drive is set to the first found external drive, memory stick or network drive. To change the drive, select the drive in the combo box. Make sure you have enough disk space for the backup in the selected drive. The available space is shown beside the drive combo box.
  - By default, the backup path is set to “PROPSIMBackups” folder. To change the backup path, select the backup folder with the browse (“**...**”) button. “PROPSIMBackups” folder is always added at the end of the selected path if it isn’t already selected.
- Backup name
  - By default, the name of the backup is set as the serial number of the PROPSIM device added with a running number at the end e.g. “123456_1”

- You can change the name of the backup to anything you like. Note that the name should contain only characters that are accepted for a filename by the operating system.
- System summary
  - Contains information on the current PROPSIM system: serial number, firmware version, and available application licenses.
  - To view detailed system summary, click the **View Details** button. Detailed information of the system is opened in Notepad.
- Backup contents
  - Includes information on the backup; the required disk space, data to be backed up. If “C:\” -drive contains emulation files (".smu", ".tap", ".ir", ".ics"), a warning is shown that they are not included in the backup. If you want these files to be included in the backup, you must move them manually to the “D:\” -drive before starting the backup. “C:\” -drive is intended to be used only for the system, not user data.
- **Create Backup** button
  - Starts creating the backup and shows a dialog with progress bar and information on the creation of the backup. To cancel the backup, click the **Cancel** button.
- **Cancel** button
  - Closes the dialog without creating a backup.

7.6.2 Restoring a backup

To restore a backup, select **Restore Backup** from PROPSIM Backup Tool main menu. The PROPSIM Backup Tool restore dialog (Figure 180) is shown. Select the backup to be restored and click **Restore Backup** button to start restoring the backup.

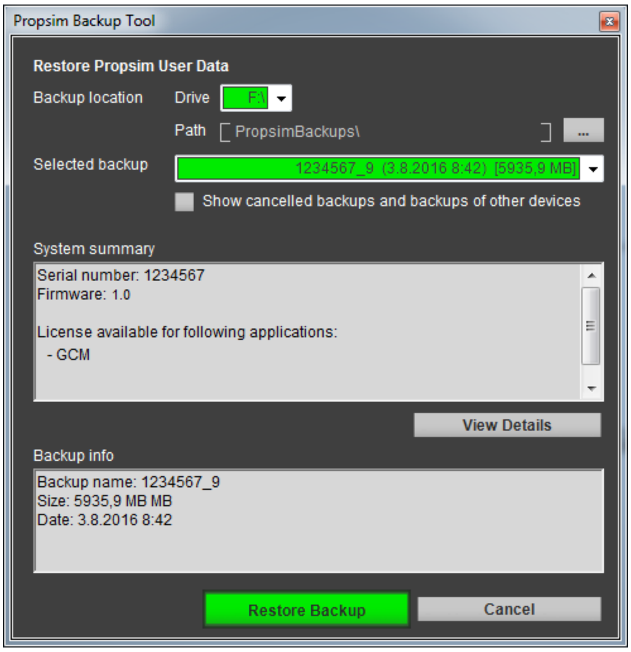


Figure 180 PROPSIM Backup Tool restore dialog.

PROPSIM Backup Tool “Restore Backup” contains following settings and information:

- Backup location
  - Describes the drive and path where the backup to be restored is located.
  - By default, the first found external drive, memory stick or network drive root is searched for the “PROPSIMBackups” folder. To change the drive, select the drive in the combo box.
  - By default, “PROPSIMBackups” folder is searched for the backups. To change the backup path, select the backup folder with the browse **(“...”)** button.
- Selected backup shows the backup to be restored. By default, the latest backup found from the backup location is selected. To change the backup, select the backup in the combo box list. Only completed backups created for current PROPSIM device are shown on the combo box list. To restore a backup created

for other PROPSIM device or a backup that was cancelled during creation, select the **Show cancelled backups and backups of other devices** check box.

- **Show all backups** checkbox shows all backups in the location including cancelled backups and backups created for other than the current PROPSIM devices.
- System summary shows information on the backed up PROPSIM device
  - To view detailed system summary, click the **View Details** button. Detailed information of the system is opened in Notepad.
- Backup info shows the backup name, serial number of the device backed up and the date when the backup was taken.
- To start restoring the backup to your PROPSIM device, click the **Restore Backup** -button. A dialog with progress bar and information on the restore is shown.
  - To cancel the restore, click the **Cancel** button on this form.
    Note: All the restoring that has been done before cancellation is final.
  - If your PROPSIM device has newer version of standard emulations than in the backup, a confirmation for overwriting with older version is asked.
- To close the dialog without restoring a backup, click the **Cancel** button.

7.7 Advanced emulation creation flow

7.7.1 Creating user defined emulations

Note: *PROPSIM Scenario Wizard is the recommended way of creating the PROPSIM emulations. Only some very specific cases require using of the advanced emulation creation flow described in this chapter.*

Creating user defined emulation with the PROPSIM is a three-step process. Figure 181 illustrates the general steps for running emulations with the PROPSIM. It also illustrates the applications that are used in the different steps.

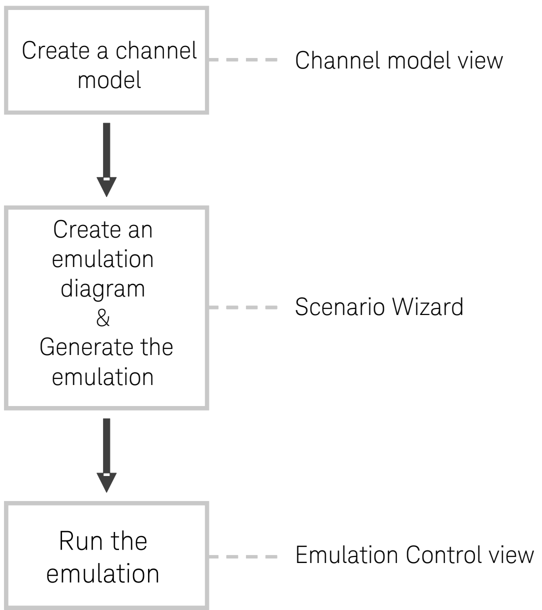


Figure 181 Advanced emulation creation flow

1. Create a channel model for the emulation. Those users of PROPSIM who want to use pre-defined standard channel models – as opposed to creating their own channel models – can skip the first step altogether and jump directly into creating the emulation diagram. Channel models are created in the **Channel model view** or by using the optional channel modelling tools like GCM, VDT or WLAN Tool. Most recent versions of these tools create also ready emulations which can be right away opened to Emulation Control view.
2. Create an emulation diagram for the channel model with the **Scenario Wizard**. Based on the created emulation diagram, the emulation itself can be generated.

3. Run the generated emulation in the **Emulation Control view**. The steps to run the emulation are the same regardless of the channel model and emulation diagram type. Cabling of the external equipment and the necessary interconnections with the PROPSIM emulator are emulation specific.

7.7.2 Creating channel models with the Channel model view

The Channel model view is used to define statistical and geometrical channel models. These models can be defined precisely using a few, easily understandable parameters. Channel model creation in the Channel model view is a three-step process, as illustrated in Figure 182.

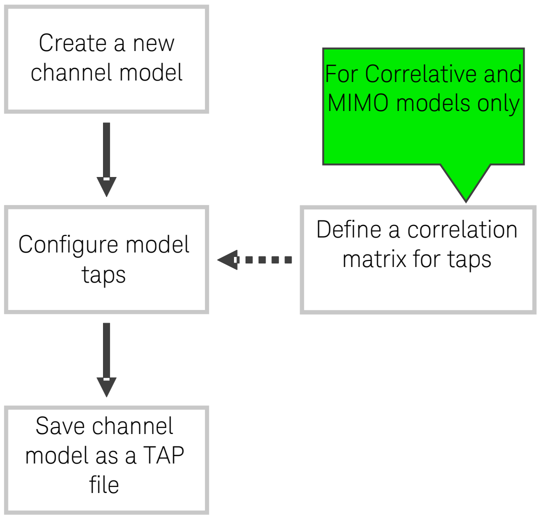


Figure 182 Channel model creation process

- First you must define what type of a model you wish to create. You can create four different types of channel models with the channel model view. These types are:
  - Uni-channel model
  - Geometry based model
  - Correlative model
  - MIMO model
- After selecting the model type the next step is to configure the model taps, or multipath components (MPC) for the model. Together all the taps define the channel impulse response (CIR). Additionally, if you are creating a Correlative or a MIMO multi-channel model, you need to define the statistical correlation between the channels of the model. For this purpose, the PROPSIM includes the Correlation Editor application.
- As a third step, you need to save the created channel model as a channel model file.

7.7.2.1 Launching the channel model view

To start the channel model view, select **Utilities > Channel Model View** in the navigation bar. This opens the Channel model view in the view area (see Figure 183). The view is the same for all channel model types.

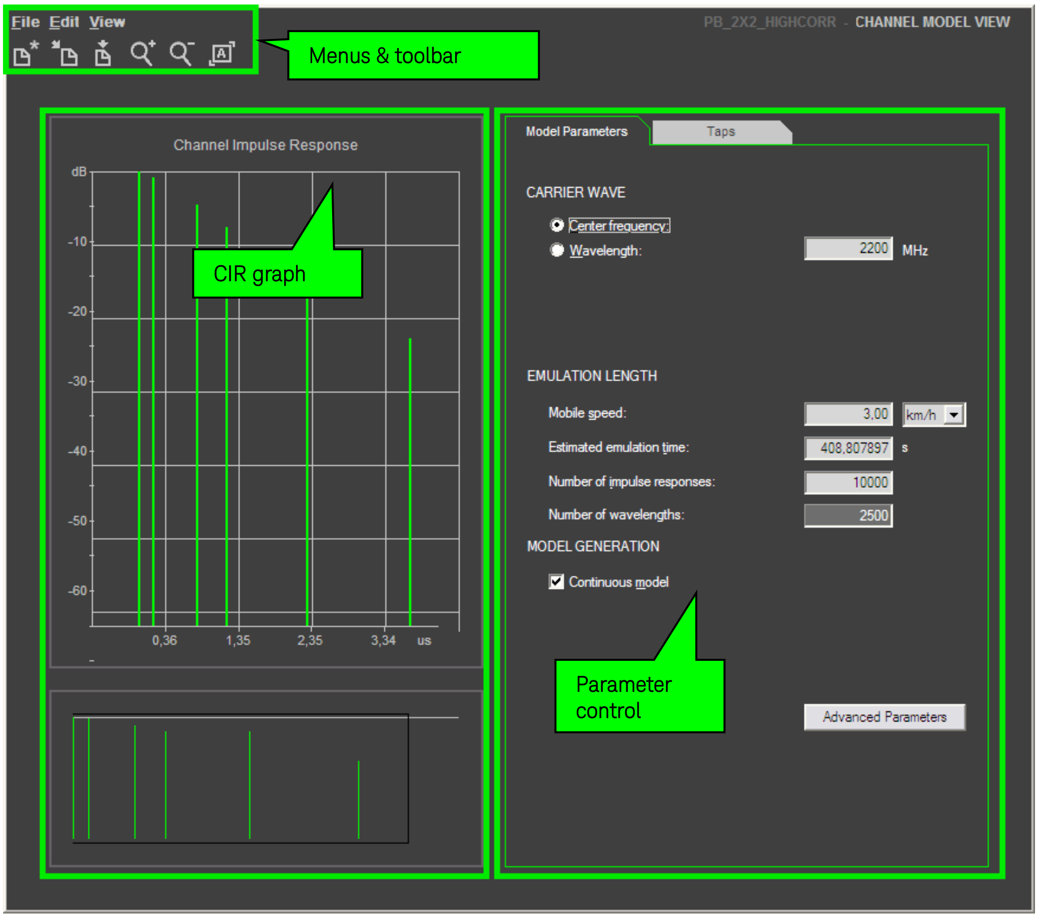


Figure 183 Channel model view

The channel model view is divided to the Toolbar, the Channel Impulse Response (CIR) graph and the Parameter control segments. The parameter control segment consists of Model Parameters and Taps tab.

The CIR graph area presents the taps that are configured for a specific model in a graphical format. The area consists of two windows. The lower window displays the entire impulse response (IR) of the model. This window is scaled to display all the taps. The upper window displays a chosen, zoomed section of the IR. To display certain taps of the IR, select the corresponding segment in the lower window. An example CIR graph area with six configured taps is shown in Figure 184.

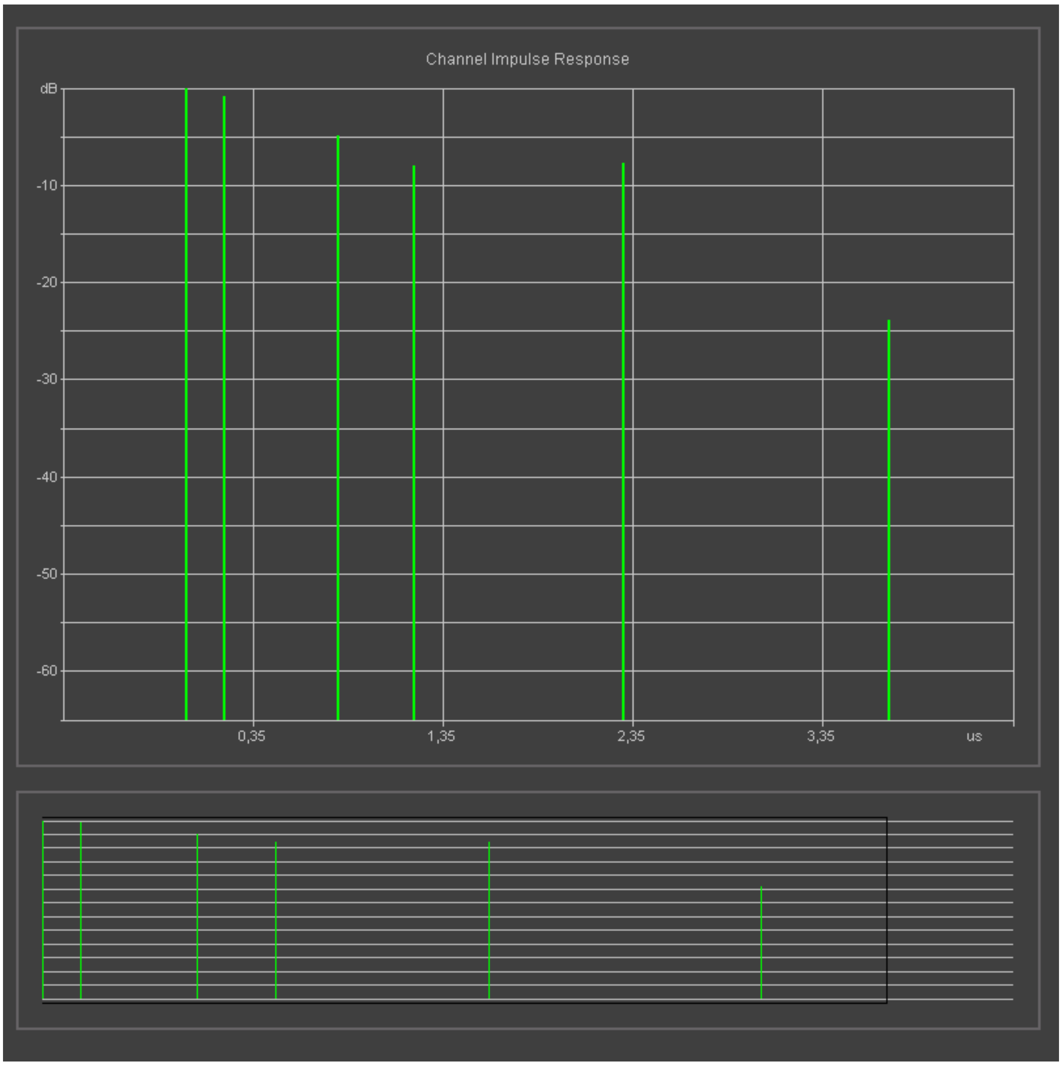


Figure 184 CIR graph

To change the zoomed section, slide the viewfinder in the lower window. The graph displays the delay and the mean relative powers of the defined taps. If the channel model includes any taps that have a sliding delay, the CIR graph shows these taps in their starting position.

7.7.2.2 Creating a uni-channel model

Use this model type to create simple channel models that only use a single channel. In uni-channel models, the fading and Doppler shift are based on statistical properties.

Figure 185 SISO environment

1. To start the creation of a new model, click **New** toolbar button or select **New** in the **File** menu. The **New Model Generation Wizard** is shown in Figure 186.

2. In Step 1, select Uni-Channel Model and click Next.
3. In Step 2, the number of channels is defined. Note that the Channels field is permanently set to a value of one for a uni-channel model.
4. Click Finish to close the wizard.

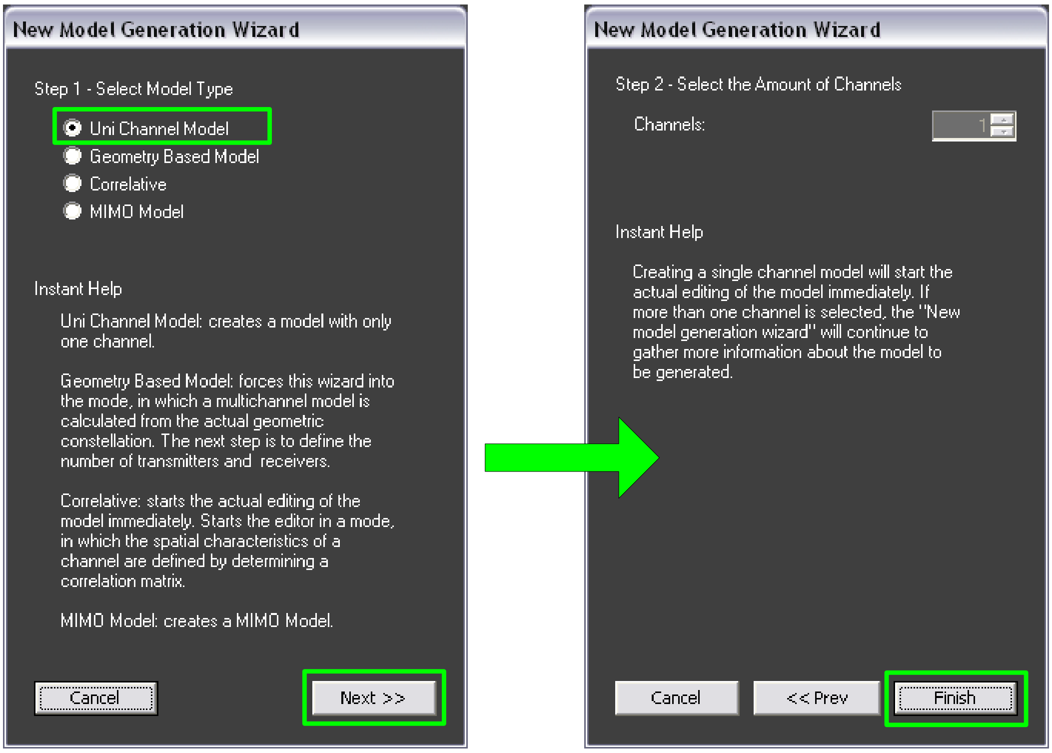


Figure 186 Uni-channel model creation

7.7.2.2.1 Configuring model parameters for a uni-channel model

The model parameters tab is used to configure properties that affect the entire channel model, see Figure 187.

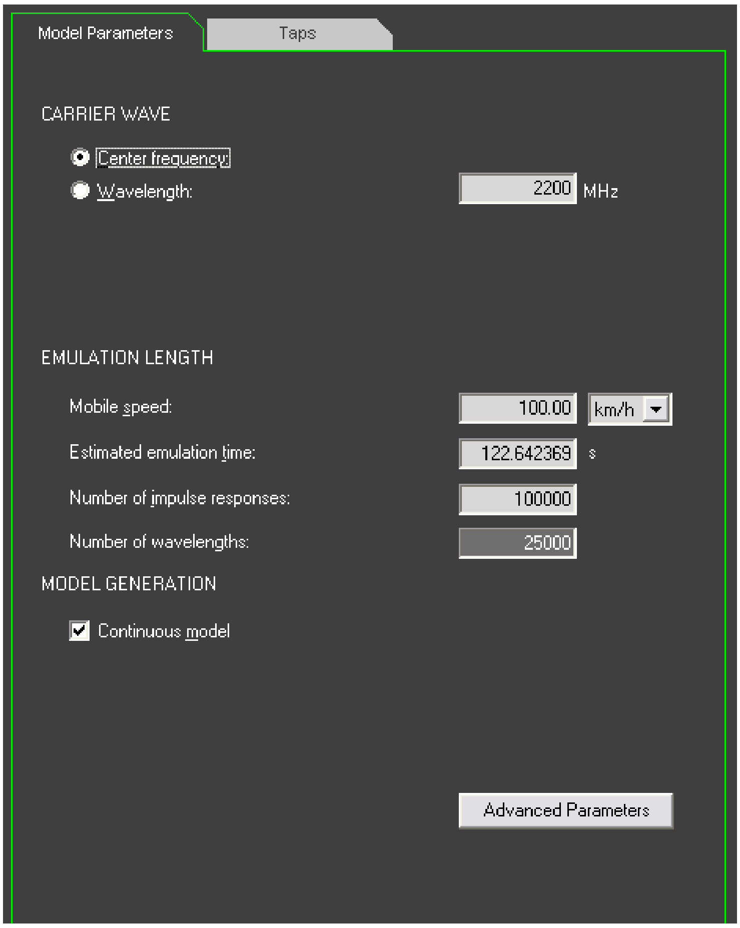


Figure 187 Model Parameters tab

- Carrier Wave:. This parameter defines the frequency of the input signal and is used by the emulator to calculate a frequency for the local oscillator. The carrier wave can be defined using the center frequency or wavelength of the input signal.
- Emulation Length: This includes defining the mobile speed, which affects the rate at which the fading environment changes. This can be defined as meters per second, kilometers per hour, miles per hour or as hertz (Doppler). Together the carrier wave and mobile speed effect on the number of impulse responses and estimated emulation time. Changing the value of the estimated emulation time affects the number of impulse responses, and the other way around. Changing these two parameters does not affect the other values.
- Model Generation:
  - The PROPSIM runs emulations continuously, so that the channel models are looped back to the first CIR when they end. If the first and last CIR of the channel model has a big difference in their respective values, the looping causes a discontinuity (noise peak) in the output signal. Check the Continuous model option to adjust the amount of CIRs so that the model is continuous. The emulator performs the adjustment automatically by interpolating new CIR values.
  - The PROPSIM enables full repeatability of all emulations. A channel model can be regenerated with the exact same fading channel and Doppler shifts. This is accomplished by using the same distribution seed for random number generation during channel model regeneration. To regenerate a channel model that is uncorrelating with the original one the distribution seed value has to be changed, as shown in Figure 188. If you wish to create multiple, uncorrelating channel model files e.g. for RX/TX diversity testing, based on the exact same parameters, click Advanced Parameters in the model parameter tab to open the advanced model parameters dialog. Change the Distribution seed field and click Close.

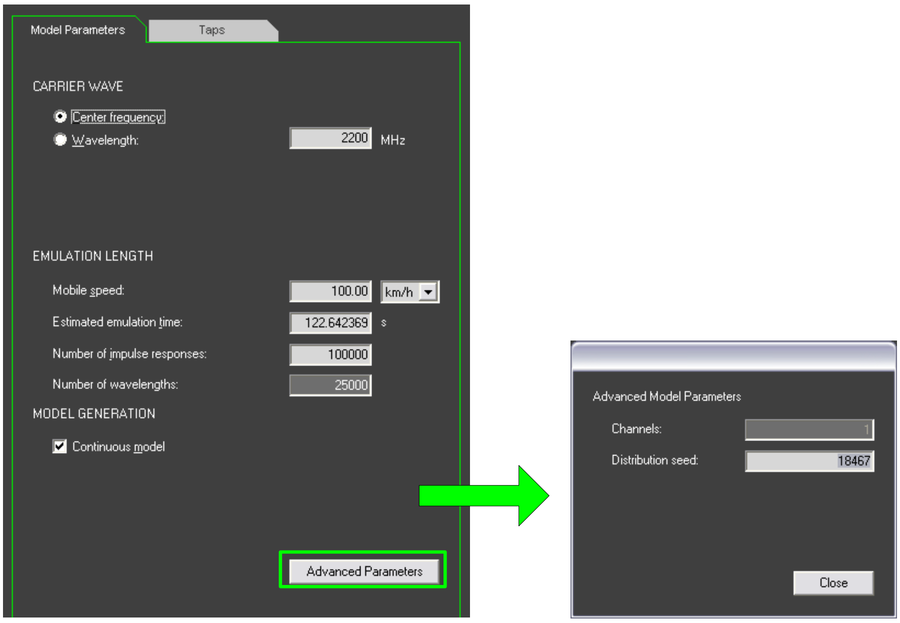


Figure 188 Advanced model parameters

7.7.2.2.2 Configuring taps for a uni-channel model

The **Taps** tab is used to define the taps, or multipath components (MPC) for the channel model. For each tap the page displays information about the tap number, tap delay in nanoseconds, tap mean relative power and tap fading type. The **Taps** tab is shown in Figure 189.

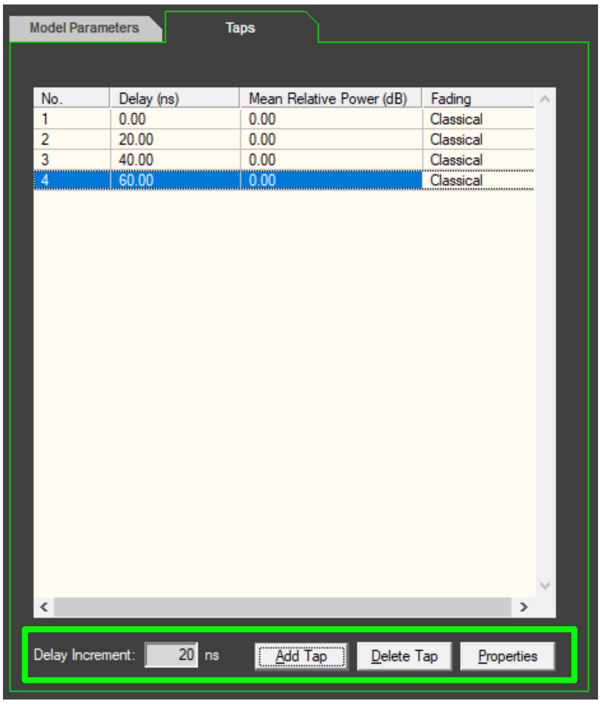


Figure 189 Taps tab

To add a new tap, click **Add Tap**. This creates a new tap in the list copying the parameters from the last tap in the list. Tap delay is automatically incremented by 20 ns, this increment can be changed with **Delay Increment** field. The tap can now also be seen in the CIR graph window.

To delete a tap, first select the tap to be deleted from the tap list and then click **Delete Tap**.

To define the tap properties in detail, select the tap to modify from the tap list and click **Properties**. The **Tap Properties** dialog opens as shown in Figure 190. Note that you can change the tap to be modified while the **Tap Properties** dialog is open. The properties of the selected tap are automatically presented.

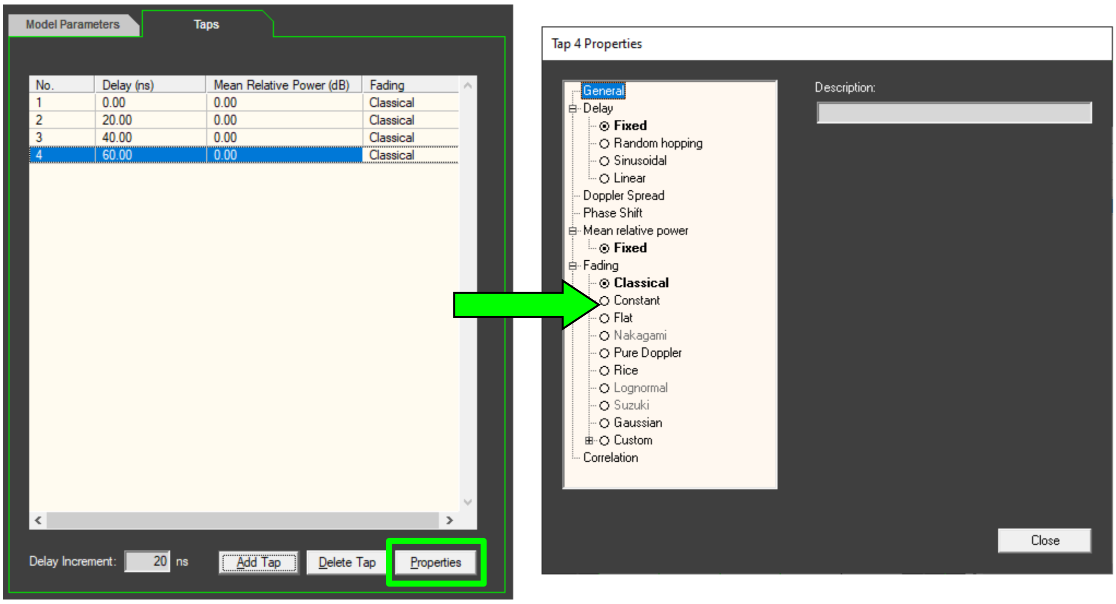


Figure 190 Modifying uni-channel model tap properties

The left side of the Tap Properties dialog has a property tree. When you select a specific property from the tree, the parameter controls of the property are opened to the right side of the Tap Properties dialog. You can configure the following properties for each tap:

- The General property allows you to give an informative description for the tap. This description has only local significance in the sense that it is displayed only on the properties page.
- The Delay property allows you to define how the delay for the tap behaves. Four different delay types are available: fixed, random hopping, sinusoidal and linear delay. Each type has its own parameters.
- The Doppler Spread property allows you to define the exact Doppler spread value in Hertz. The initial value for the Doppler spread is calculated based on the mobile speed and center frequency, defined in Model Parameters tab.
- A constant phase shift can be defined for the tap by modifying the Phase Shift property. This value is given in degrees.
- The Mean relative power property allows defining a mean attenuation factor for the tap. This value represents the average power of the tap compared to the other taps. Value is defined in decibels. Note that only negative values are accepted.
- The Fading property defines the statistical fading model for the tap. This parameter affects both the statistical amplitude distribution as well as the Doppler distribution. The Custom fading type allows you to set the fading model and Doppler distribution independent of each other.

Define similarly properties for each tap in the channel model.

7.7.2.2.3 Saving a uni-channel model

After you have defined the taps for the uni-channel model you need to store it in the PROPSIM.

Click Save toolbar button  or select the Save as option form the File menu to store the channel model as shown in Figure 191.

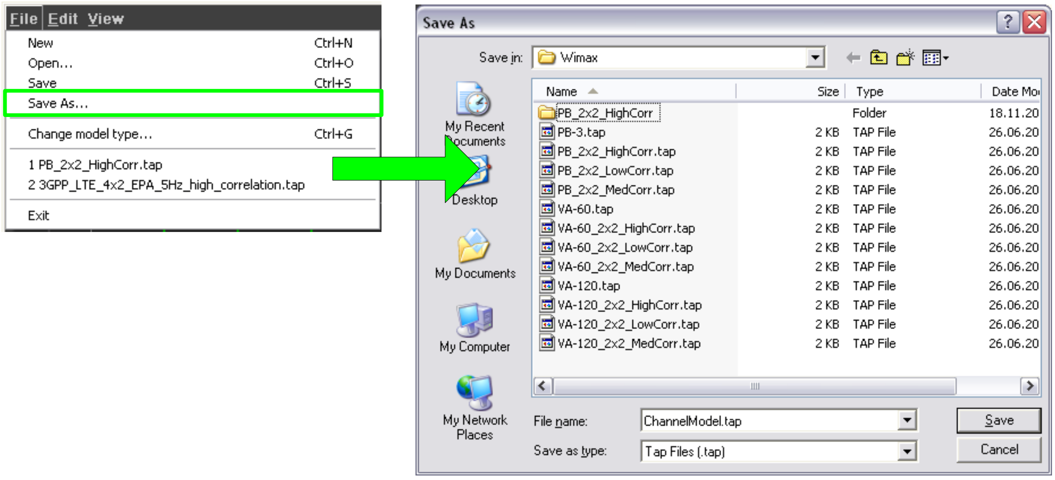


Figure 191 Saving a uni-channel model

7.7.2.3 Creating a MIMO channel model

Use this model type to create multichannel MIMO models. With MIMO models, fading and Doppler shift are based on statistical properties. You can also define the statistical correlation between different channels separately for the transmitters and the receivers. Up to 8x8 MIMO setups can be created with a single PROPSIM emulator.

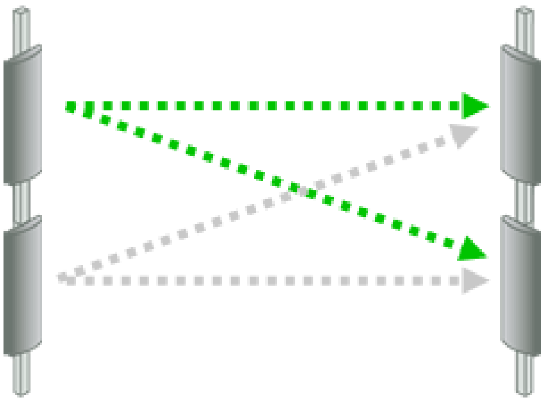


Figure 192 MIMO environment

MIMO channel model creation is a two-step process:

1. Select the **MIMO Model** radio button in the channel model type dialog and click **Next**. This opens the MIMO model dialog, as shown in Figure 193.

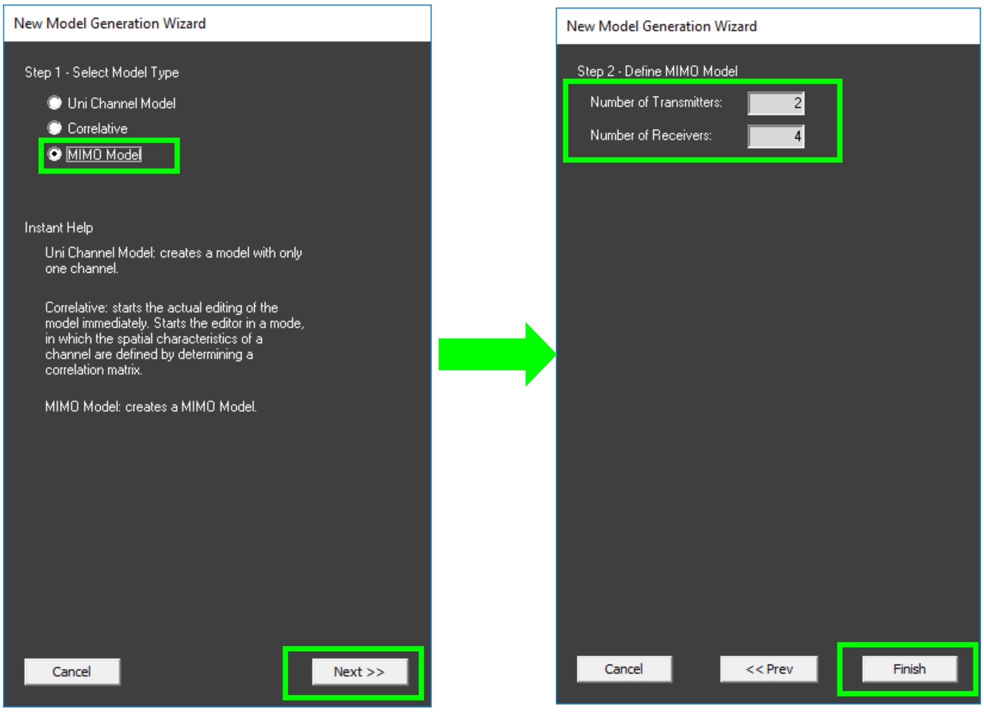


Figure 193 Creating a MIMO model

2. Select the number of transmitters and receivers for the model. Click **Finish** to finalize the MIMO channel model creation.

Note that a single PROPSIM can support up to 1024 logical channels according to installed MIMO extension option. The logical channels can be used for MIMO channels or just independent fading channels e.g. for diversity channels. With 128 logical channels you can create a MIMO setup, such as a bi-directional 8x8 MIMO.

7.7.2.3.1 Configuring model parameters for a MIMO channel model

Model parameter configuration is identical for all channel model types. Follow the same configuration steps as for a uni-channel model, explained in section 7.7.2.2.1, page 148.

7.7.2.3.2 Configuring tap properties for a MIMO channel model

The **Taps** tab is almost identical for all channel model types; the MIMO channel model tap properties include all the same properties as a uni-channel model does. Configure these as explained in section 7.7.2.2.2, page 150. Additionally, **Correlation** properties can be defined for the MIMO channel models, see Figure 194.

In the MIMO channel models the correlation between different channels is based on statistical properties. The correlation is defined with a MIMO correlation matrix. Each tap in the emulation must be mapped with a MIMO correlation matrix. The matrix can be unique for each tap.

Correlation is also defined for correlative channel models. Difference with the correlation matrices of a correlative channel model and a MIMO channel model is that for a MIMO model you must define a matrix for the transmitter and receiver channels separately.

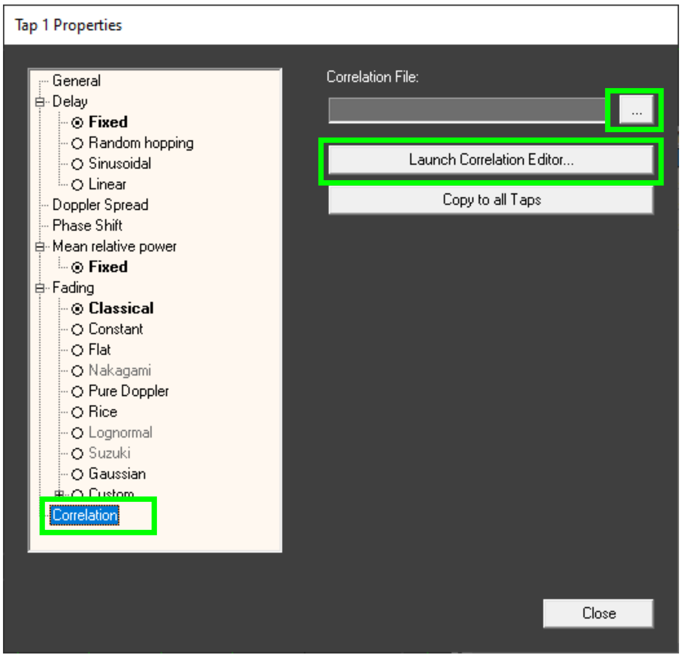


Figure 194 MIMO model tap properties dialog

The correlation matrix for the MIMO channel model can be created with the Correlation Editor application that is included in the PROPSIM GUI. Also predefined correlation matrices stored in the PROPSIM can be used.

7.7.2.3.3 Defining a correlation matrix for the MIMO channel model

1. To create a new correlation matrix, select the Correlation branch from the Tap properties dialog.
2. Click Launch Correlation Editor, see Figure 194. This opens the Correlation Editor application in the MIMO mode, as shown in Figure 195.

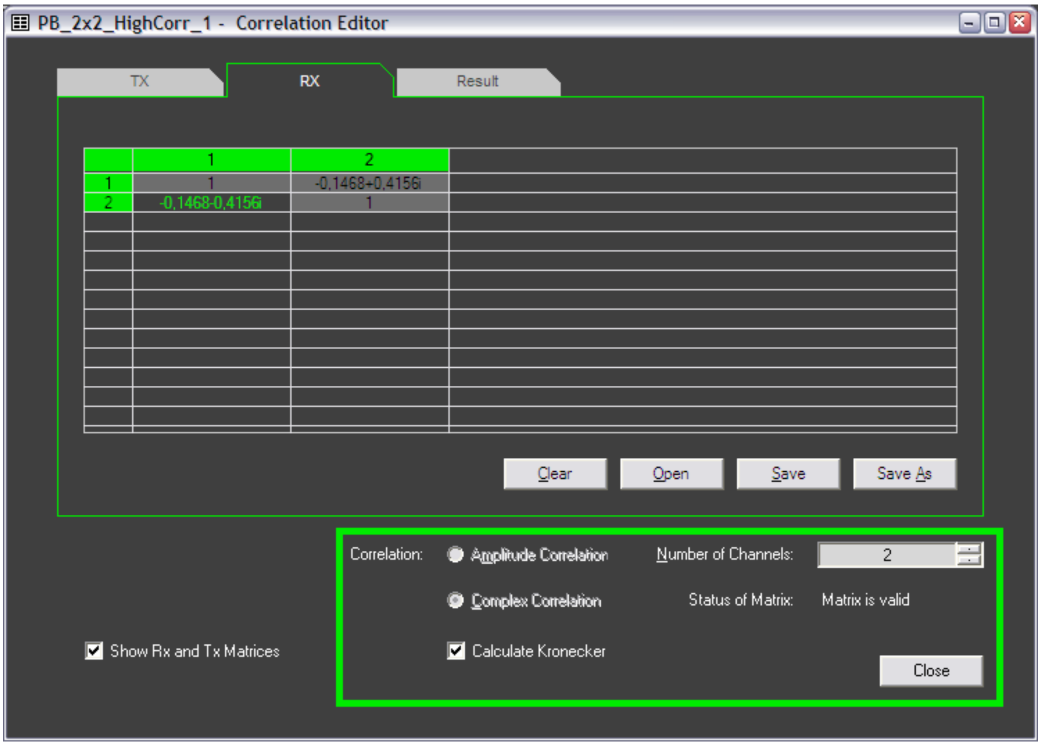


Figure 195 Creating RX correlation matrix for MIMO model

3. The correlation editor automatically sets the matrix size based on the number of channels defined for the MIMO channel model. In the example shown above, the number of channels for the MIMO model was set to 2. In MIMO model the receiver (RX), transmitter (TX), and result all have their own matrices, located on separate tabs. Select the tab you want to work with from the upper corner of the correlation editor.
4. The correlation matrix takes correlation coefficients as input. The correlation can be based on signal amplitude or amplitude and phase. In the latter case the coefficients in the matrix are defined as complex

numbers. First select either the Amplitude Correlation or Complex Correlation radio button. MIMO channel model uses always complex correlation. In Amplitude Correlation mode, the matrix will not accept complex values. You can add correlation coefficients to TX and RX matrix separately. The Result page contains the Kronecker product of the RX and TX matrices you have created. Correlation coefficients can also be added directly to Result page.

a. First define the values for the TX correlation matrix. Give values for the cells below the main diagonal. Double-click on a matrix cell to edit the cell value. As you set a value for a cell, the corresponding cells above the main diagonal are automatically given a value that is the complex conjugate of your input. Each cell on the matrix main diagonal – from top left to bottom right – always have a value '1'.

b. After you have given values for all cells, check the Status of Matrix field. If the status is "Not valid" it means that the correlation between the channels is not realistic with the current values. This indicates that you need to change some of the values, in order to make the matrix valid.

c. Next select the RX tab and repeat the above-mentioned steps to input values for the RX correlation matrix. The user interfaces for the TX and RX tabs are identical, as shown in Figure 196.

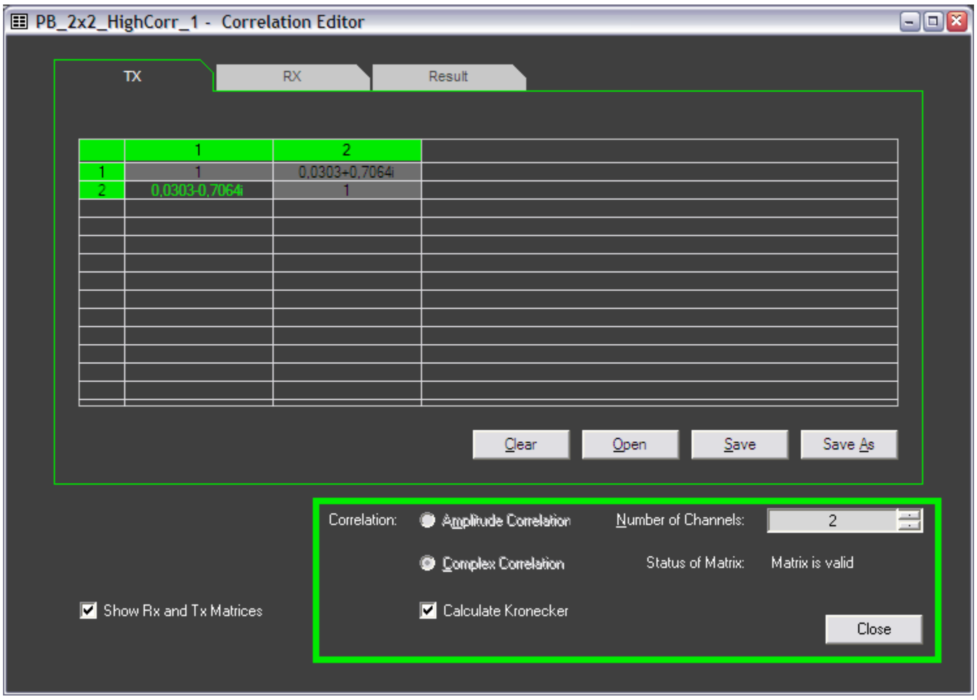


Figure 196 Creating TX correlation matrix for MIMO model

d. After you have given values for both the TX and RX matrices, select the Result tab. It contains the Kronecker product of the TX and RX matrices you have created. The resulting matrix is automatically calculated based on the input in the other two matrices. The resulting matrix is shown in Figure 197.

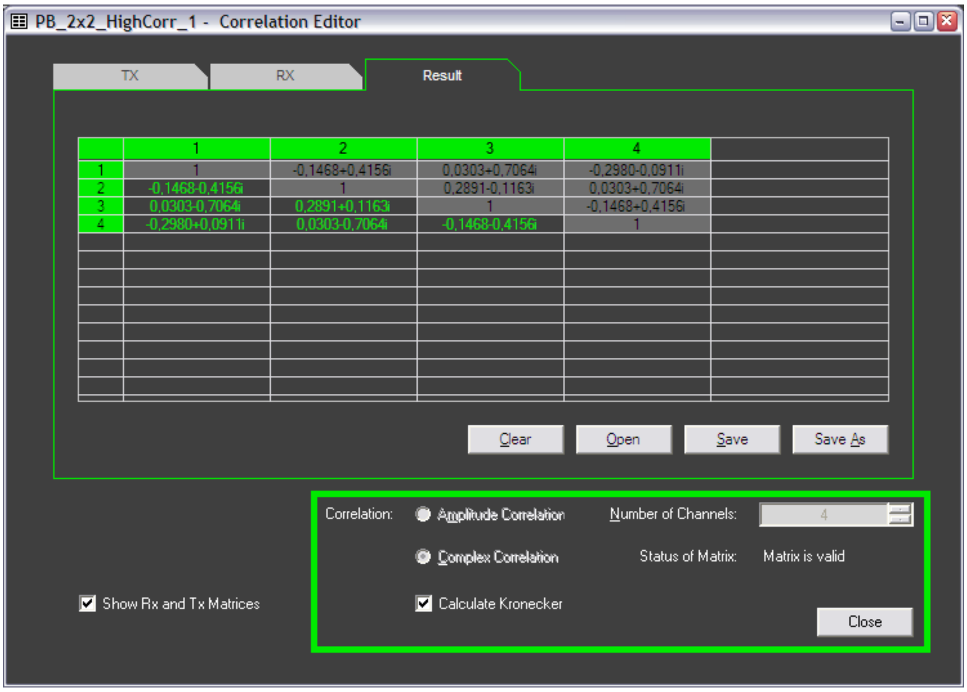


Figure 197 Result matrix

e. Check the **Status of Matrix** field in the **Result** tab. If the status is Valid, you can save the matrices. All three matrices are saved to a single file. It is not possible to save an individual matrix.

f. Click **Save As** to save all the matrices, as shown in Figure 198. The correlation matrices are stored as COR files. This file format is identified by the .cor suffix in the file name.

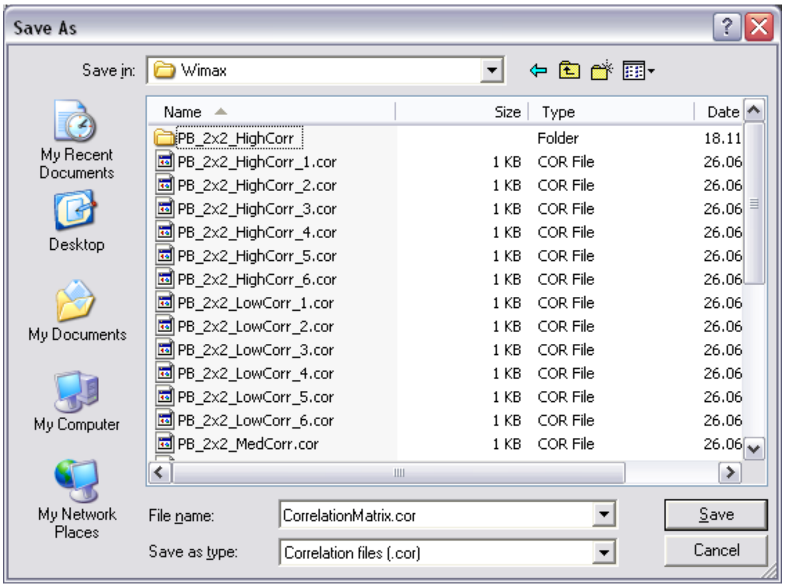


Figure 198 Saving the MIMO matrices

g. After you have saved the correlation matrix, click **Close** in the correlation editor to close the application.

5. Next, map the correlation matrix with the tap by clicking the "..." button in the tap properties dialog, see Figure 194. This opens the load dialog.

6. Select the COR file you have just created and click **Open**. The path of the file becomes visible in the **Correlation File** field.

7. If you want to use predefined correlation files, follow the same steps for mapping the COR file with the tap. Note that the predefined COR file must have a compatible channel configuration with your channel model.

8. Perform the same process for all the other taps in the channel model. If you plan to use the same correlation matrix for all taps, click **Copy to all Taps**. This automatically maps the same correlation file for all the other taps in the channel model.

7.7.2.3.4 Saving a MIMO channel model

After you have defined the taps and correlation matrices for the MIMO channel model you need to store it to the PROPSIM.

Click **Save** toolbar button  or select the **Save** or **Save As** option from the **File** menu to store the channel model, as shown in Figure 199. The saved file includes information about the mapping of MIMO correlation matrices.

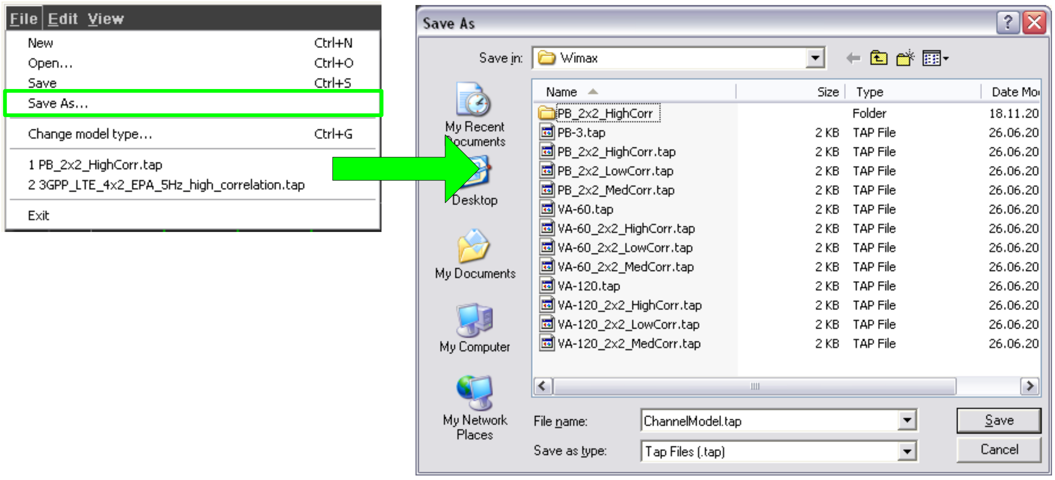


Figure 199 Saving a MIMO channel model

More information on using a predefined MIMO model in creating a multi-channel emulation can be found in chapter 3.1.3.1.

7.7.3 Creating emulation diagrams in Scenario Wizard

Use the Scenario Wizard to create emulations that use the channel model you created. For instructions on creating emulations, see section 3.1.

Note: In the Scenario Wizard, you select the channel model in the Link properties dialog, as instructed in section 3.1.3.1.2.

8 DATA LOGGING

PROPSIM supports logging of emulation data, either to a local file (.csv) or by sending it to a specified network address (UDP protocol).

8.1 Configure data sending

You configure the settings for sending measurement data in the Device configuration dialog as follows:

1. To open the dialog, select Configuration > Device Configuration in the navigation bar.
2. To enable measurement data sending and to show the relevant settings, select the Enabled checkbox under Measurement data sending.

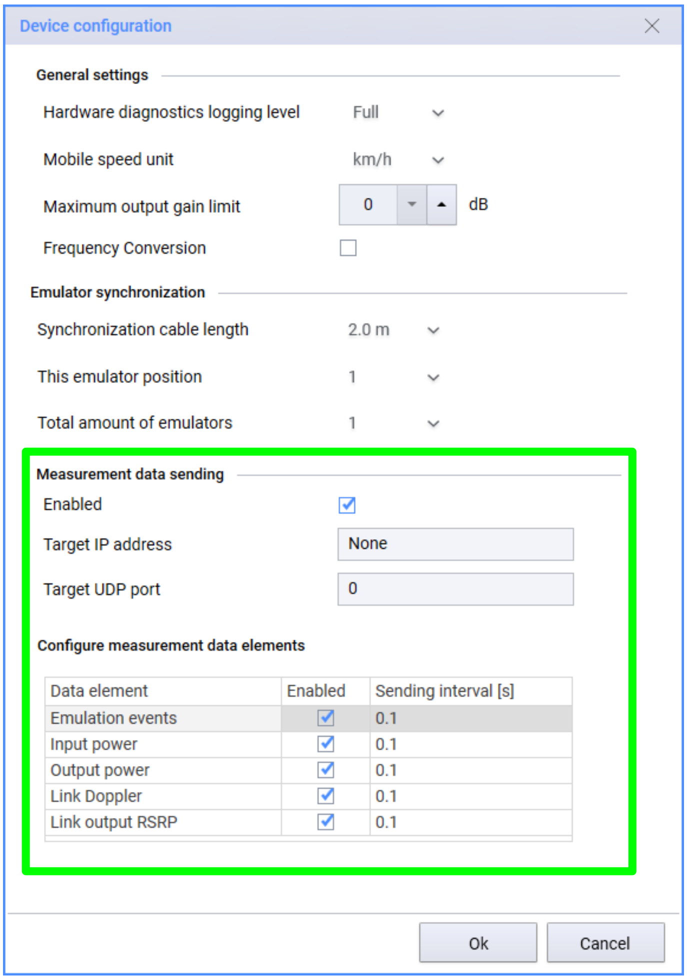


Figure 200 Measurement data sending configuration

3. Select remote IP address and port where the emulation data is sent.
4. Enable the sending and define the sending interval for each element type separately.
5. Click OK.

Data sending format is specified in chapter 8.3. Logging the data to a local file can be done in the Live data view, see section 5.3. Corresponding ATE commands for configuring the data sending are specified in chapters 20.4.2.26 - 20.4.2.27.

Note: Getting Link Output RSRP data requires performing RSRP measurement at PROPSIM input(s) after opening each emulation. For more information, see section 4.3.4 (Base station/mobile station settings - LTE RSRP measurement).

8.2 Saving measurement data to local file

You can configure measurement data logging to a local file in the Live data view. For more information about the live data view, see section 5.3.

8.3 Data format (UDP)

Following format is used for sending emulation data via UDP. Data is ASCII only.

T=[timestamp]#D=[datatype]#C=[connector]#E=[element id]#N=[element name] #V=[value][CR+LF]

Timestamp

Local time timestamp in ISO 8601 format: YYYY-MM-DDThh:mm:ss.ttt

Datatype

1=Emulation event

3=Emulation time

101=Input power measurement (dB)

201=Output power measurement (dB)

401=Link Doppler (Hz), from the first channel of the link

402=Link output RSRP (dBm)

403=Link AoA angle (deg)

404=Link AoD angle (deg)

801=Link speed profile data (see chapter 8.4)

802=Link shadowing profile data (see chapter 8.4)

803=Link interference profile data (see chapter 8.4)

804=Link AoA angle profile data (see chapter 8.4)

805=Link AoD angle profile data (see chapter 8.4)

811=Channel Speed profile data (see chapter 8.4)

812=Channel Shadowing profile data (see chapter 8.4)

813=Interference profile data (see chapter 8.4)

814=Channel AoA angle profile data (see chapter 8.4)

815=Channel AoD angle profile data (see chapter 8.4)

Note: AoA and AoD angle data requires the modeling tool (GCM) support for writing the angle data when creating the emulation.

Connector (optional)

RF1 ... RF64

Element id (optional)

1 ... N (input, output or link number)

Element name (optional)

Text, with # characters escaped with ##. Includes a link name or user defined name for input/output.

Value

Value in textual format, without unit.

When Datatype=1 (Emulation event), value is one of the following:

- OPEN <file.smu> *(emulation is opened)*
- PLAY *(emulation is played)*
- STOP *(emulation is stopped and rewound to start)*
- PAUSE *(emulation is paused)*
- CLOSE *(emulation is closed)*
- GOTO *(emulation goto operation is performed)*
- STEP *(emulation step operation is performed)*

If the value (i.e. emulation name and path) contains # characters, they are escaped with ##.

Example data:

```
T=2016-04-12T08:41:04.102#D=1#V=OPEN d:\User Emulations\LTE 2x4.smu\r\n
T=2016-04-12T08:41:05.506#D=101#C=RF1#E=1#N=BS1-TX1#V=-23.4\r\n
T=2016-04-12T08:41:05.519#D=101#C=RF3#E=2#N=BS1-TX2#V=-24.4\r\n
```

8.4 Emulation profile data sending

Data logging feature supports logging/sending of shadowing-, interference- or speed profile data points with a user defined time step. Operation is requested from the ATE interface. See chapter 20.4.2.29 for the command details.

9 SYSTEM CONFIGURATION AND SHUTDOWN

9.1 Device information

To open the Device Information dialog of the PROPSIM, select Configuration > Device Information in the Navigation bar. Device Information dialog contains e.g. device name and serial number, IP address, and general channel information. External units (Auto Calibration Unit, Auto Switching Unit) are also shown, if they are powered on and connected to PROPSIM during PROPSIM startup.

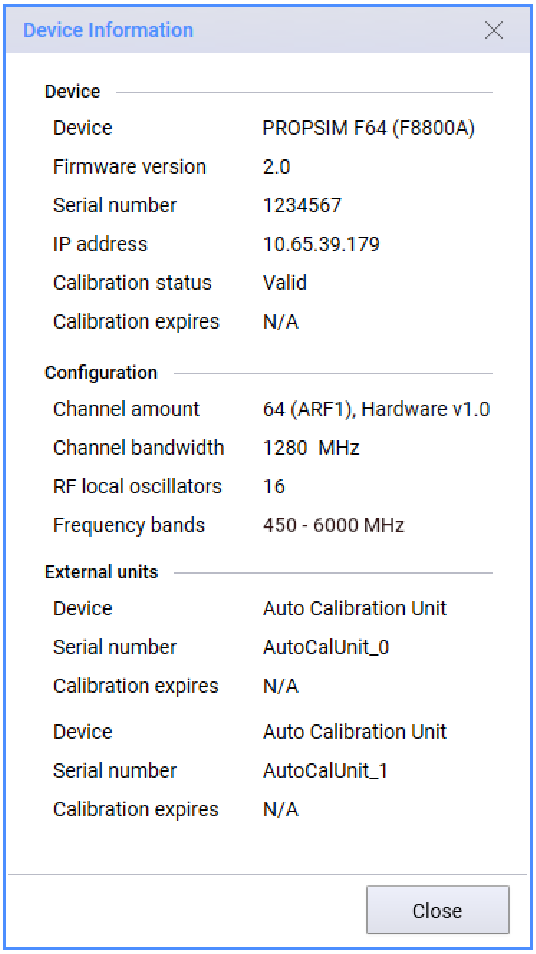


Figure 201 Device Information dialog

9.2 Device configuration

To open the configuration dialog of the PROPSIM, select **Configuration** > **Device Configuration** on the Navigation bar. The Device Configuration dialogue contains e.g. HW diagnostic logging level, frequency conversion activation, emulator synchronization, and measurement data sending settings.

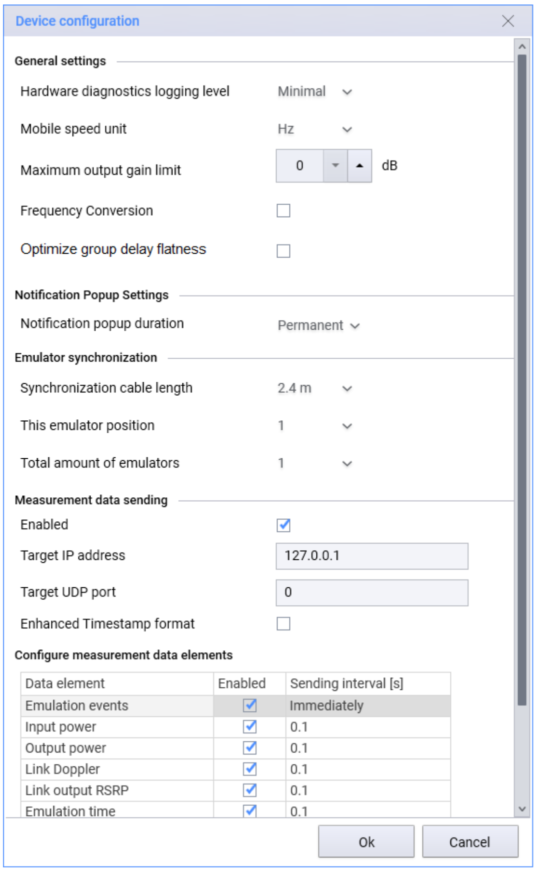


Figure 202 Device configuration dialog

General settings

Hardware diagnostics logging level

- The logging level setting is for troubleshooting purposes only and does not directly affect the emulator functionality. The default setting (minimal) should not be changed unless explicitly instructed by Keysight.

Mobile speed unit

- Defines the default speed unit used in user interface. Supported units are km/h, ms/s and Hz (Doppler)

Maximum output gain limit

- Defines the maximum output gain (amplification) device is allowed to use

Frequency Conversion

- Enables/disables use of frequency conversion (input and output and output of the same link is allowed to operate with different frequency)

Optimize group delay flatness

- This setting optimizes group delay flatness with a cost of other parameters and specifications. Performance of this feature might change between firmware versions. Emulation must be reloaded when changing this setting.

Emulation synchronization

Synchronization cable length

- Synchronization cable length specifies the length of cables used for synchronization when multiple emulators are used. Available options depend on device hardware model. The cables need to be of the equal length, emulation must be closed and re-opened for applying the changed value.

This emulator position

- Defines the position of the emulator in multi-emulator configuration. Emulation must be closed and re-opened for applying the changed value.

Total Amount of Emulators

- Defines the total number of emulators in multi-emulator configuration. Emulation must be closed and re-opened for applying the changed value.

Measurement data sending

Settings for measurement data sending. See chapter 8.1 for more information.

9.3 License importing

To add or change licenses, open the **Import License** dialog by selecting **Configuration** > **Import license**.

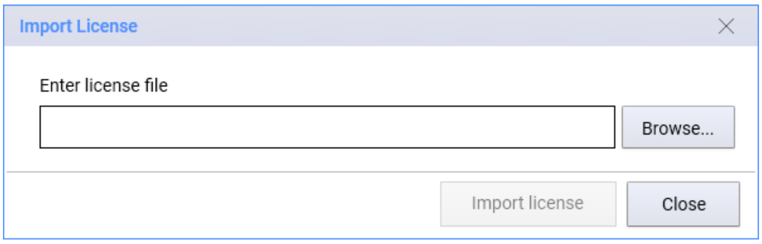


Figure 203 Import License dialog

To import licenses, you need to have an “lservrc” file containing license changes and/or additions. Browse and select the license file and click the **Import license** button. The file containing license changes is compared to the current license files and a list of additions or modifications shown (Figure 204).

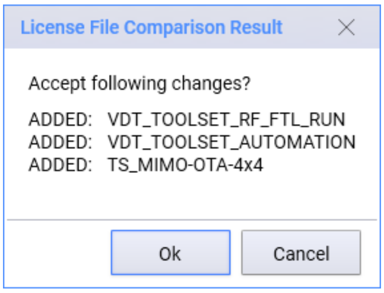


Figure 204 List of license changes and additions to be done

You can accept changes by clicking **OK** in the **License Files Comparison Result** dialog. If you accept the changes, the original license files are first backed up in their current folder and the changes are written to the original license files. The backup files are named as “lservrc.backup_YYMMDD_HHMMSS”, where YYMMDD describes the date of the backup and HHMMSS the time of the backup. Restart is required after license import.

Loan licenses might require an activation key from customersupport.di@keysight.com

9.4 Restart (power cycle) / Shutdown

To restart (power cycle) the PROPSIM, select **System** > **Restart / Shutdown** > **Restart (power cycle)** in the navigation bar and click **OK** in the confirmation dialog.

To shut down the PROPSIM, either press the Standby button in the emulator front panel or select **System** > **Restart / Shutdown** > **Shutdown** in the navigation bar and click **OK** in the confirmation dialog.

10 SNMP INTERFACE

SNMP protocol is used for exchanging information between clients and SNMP enabled entities. PROPSIM has an SNMP enabled application, which responds to the queries made by SNMP clients. Through this interface, PROPSIM provides basic device information such as name, model, and version.

10.1 SNMP information

Following Object Identifiers (OID) are available on the device:

1.3.6.1.4.1.43286.1.1 – vendor name - "Keysight Technologies"

1.3.6.1.4.1.43286.1.2 – model name - e.g. "PROPSIM F64"

1.3.6.1.4.1.43286.1.3 – serial number - e.g. "123456"

1.3.6.1.4.1.43286.1.4 – firmware version - e.g. "1.0"

1.3.6.1.4.1.43286.2.1 – current timestamp - "yyyy/mm/dd hh:mm:ss"

1.3.6.1.4.1.43286.2.2 – utilization information – one of the following values:

0 = Information not available

1 = Device has not been used since the last query

2 = Device has been used since the last query

10.2 Configuring SNMP

Community string of the SNMP protocol can be set by modifying the *snmpcommunitystring* file located at "C:\PROPSIM\". File is a simple text file, containing only the community string in use. As a default PROPSIM uses 'public'. Content of the file can be replaced with a user defined community string.

When the file is changed, the new community string is taken into use when the device is next time restarted.

11 USER ALIGNMENT

11.1 User alignment feature

User alignment feature allows creating phase/gain/delay -alignments for setups. Each alignment stores desired settings keyed by frequency, connector and bandwidth and thus can be re-used later with appropriate emulations.

User alignment stores and applies the following emulator-wide settings:

- In Loss
- Out Loss
- RF phase
- RF delay
- Sub-band adjustment gain (Emulation BW > 200 MHz)
- Sub-band adjustment phase (Emulation BW > 200 MHz)
- Sub-band adjustment delay (Emulation BW > 200 MHz)

User alignment feature is internally used by PROPSIM External Response Calibration Tool, which writes the response corrections as user alignment files. In addition to the settings above, these alignment files also contain frequency response correction data. User cannot modify the alignment files produced by PROPSIM External Response Calibration Tool.

ATE commands related to user alignment feature are listed in ATE command interface, starting from chapter 20.4.2.18.

11.2 Creating or selecting the user alignment

User alignment is created or selected from the Align -menu, as shown in Figure 205.

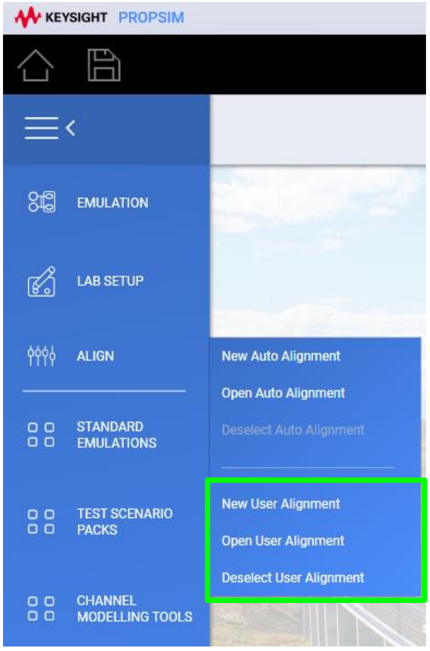


Figure 205. Creating or selecting user alignment

Creating a new user alignment allows user to define a name, description and optionally copy the existing values for each port and frequency from the currently open emulation to the new user calibration. New alignment creation is shown in Figure 206.

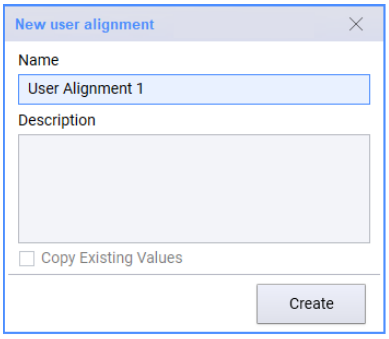


Figure 206. Creating a new user alignment

When user alignment is in use, it is shown in the status bar as shown in Figure 207.

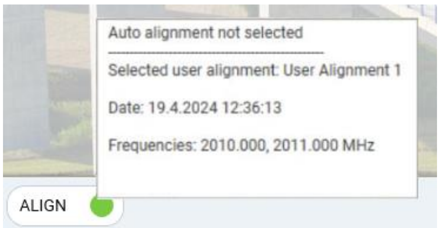


Figure 207. User alignment selected

11.3 Operation with user alignment in use

11.3.1 Emulation loading

When an emulation is loaded for running and the user alignment is in use, each input and output of the emulation will be automatically applied with the user alignment settings (listed in Chapter 11.1), according to the input and output physical port, bandwidth, and frequency.

11.3.2 Changing frequency in the emulation

When emulation center frequency (or individual port's frequency in case of frequency conversion) is changed and user alignment is in use, current settings (listed in Chapter 11.1) of each port are stored to user calibration and values corresponding to the new frequency are automatically taken into use.

11.3.3 User alignment together with auto alignment

It is possible to use user alignment together with the auto alignment. If the auto alignment measurement is done with user alignment enabled, both must also be enabled and used together for the correct operation. In this case their parameter values for each port are summed together.

12 SHADOWING (OPTIONAL FEATURE)

Shadowing is a phenomenon in which a large obstruction such as a hill or a large building obscures the main signal path between a transmitter and a receiver, causing attenuation in the received signal power. The Shadowing option of the PROPSIM enables modelling of the shadowing phenomenon with a variety of functions. Shadowing feature in PROPSIM has two operating modes: RF output shadowing and channel specific shadowing. The modes and their differences are discussed in the following chapters 12.1 and 12.2. The modes are exclusive – one emulation can contain only RF output shadowing or channel specific shadowing, not both.

12.1 RF output shadowing

RF output shadowing can be defined for each output and it uses analog RF output attenuator to create the shadowing curve. Using analog attenuator is always optimal for signal dynamics and performance, thus it has a limitation that at the same time it attenuates all the channels and interferers connected to the output. In certain duplex- or AWGN fixed noise power scenarios, this is an issue, and channel specific shadowing can be used instead, see chapter 12.2.

Shadowing profiles can be defined in the Scenario Wizard. In Scenario Wizard, "Single link" and "Multi-RAT" cases use RF output shadowing by default. Shadowing profile can be defined for each link as shown in Figure 208 and described more detailed in chapter 3.1.3. Selected shadowing profile is applied for all Rx antennas in the link.

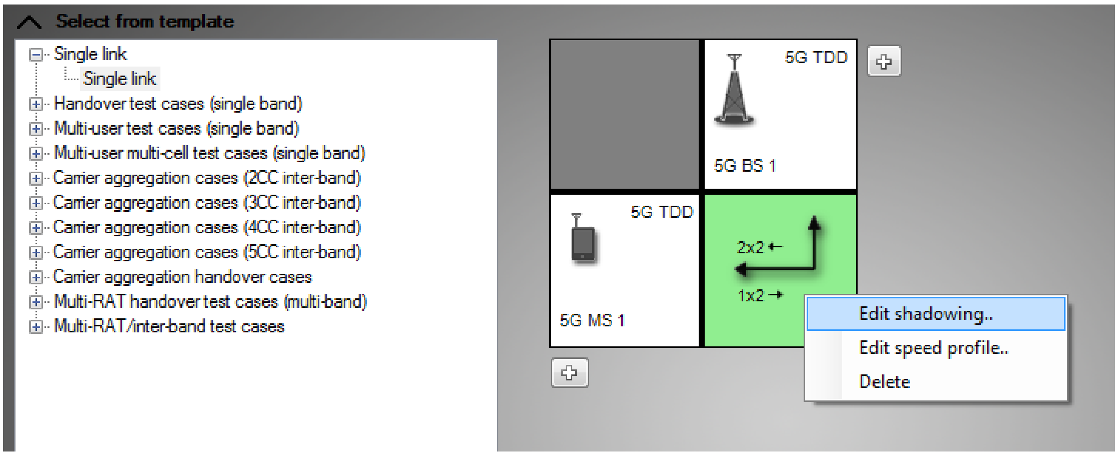


Figure 208 Adding shadowing profile field in the Scenario Wizard

12.2 Channel specific shadowing

Channel specific shadowing is otherwise like RF output shadowing, but the shadowing profile can be different for each channel. This is useful especially in following test cases:

- Handover or multi-user scenarios using PROPSIM internal summing
- Shadowing emulation with fixed AWGN power at output (only digital shadowing)

Scenario Wizard automatically utilizes channel specific shadowing in all the internally summed configurations. Multi-RAT scenarios using external summing are the only ones created with output RF shadowing. In MANET emulations, you can also manually define channel specific shadowing. For more information on defining shadowing in Scenario Wizard, see section 3.1.3.1.

By default, channel specific shadowing uses a combination of RF analog output attenuator and digital scaling to optimize the signal dynamics and RF performance. In practice, this is done during the model compilation: all the shadowing channels sharing the same output RF attenuator are handled together. Common part of the attenuation is done with analog RF attenuator and the rest with digital scaling.

In Scenario Wizard, you can configure channel specific shadowing to be "only digital" by disallowing the use of RF analog attenuator. This can decrease signal dynamics, but at the same time allows the usage of fixed noise power AWGN at output.

12.3 Running the shadowing emulation

After building the emulation, it can be opened for playback in the Emulation Control View or ATE. Shadowing playback starts when emulation is played. In the Data views window, the Graph view also shows the shadowing curves and the current position marker on the curve, see Figure 100 in section 5.2 Graph view.

12.3.1 Shadowing enable/disable

Shadowing can be disabled/enabled for each link individually at run time using the Emulation Control View or ATE. For more information, see section 4.3.3.

12.3.2 Shadowing offset

Shadowing offset can be used to achieve the expected signal level for the DUT at specific shadowing curve point. Typically, this is done in test setup calibration or validation phase. When RF output shadowing is used, i.e. multi-RAT setup, value can be changed using Emulation Control View or ATE at run-time. If the shadowing offset value is positive, shadowing curve is saturated (flattened) from high peaks, as much as needed. Saturation level varies due to hardware configuration and other emulation settings as bi/uni-directional emulation, TDD, FDD, and max output gain. The Graph view in the Data views window visualizes the saturation point, as shown in Figure 209.

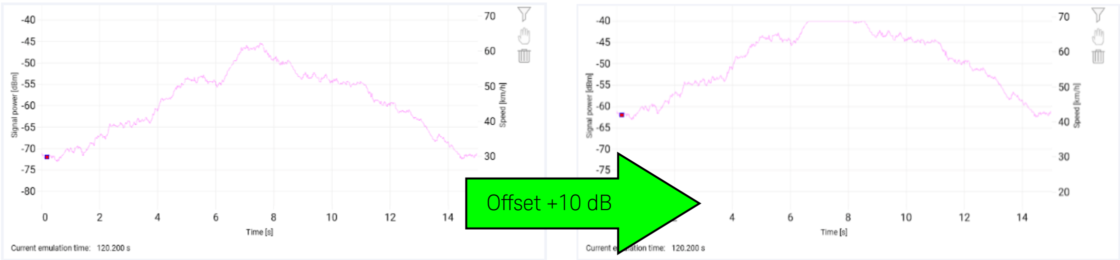


Figure 209 Shadowing offset and saturation, Graph view in Data views window

Shadowing offset setting can be adjusted for each link (uplink and downlink), see section 4.3.3.

In case of digital (channel specific) shadowing (i.e. internally summed inter-band handover), changing the offset requires model to be re-build and the change must be done in Scenario Wizard. In that case the saturation point is always 0 dB in the shadowing profile.

12.4 Shadowing profiles

When defining a MANET emulation, you can also create a new shadowing profile. Clicking **Create...** under the **Shadowing profile** field opens the Shadowing Editor, see Figure 210. The editor opens with the default Sinusoidal profile type when a new profile is created.

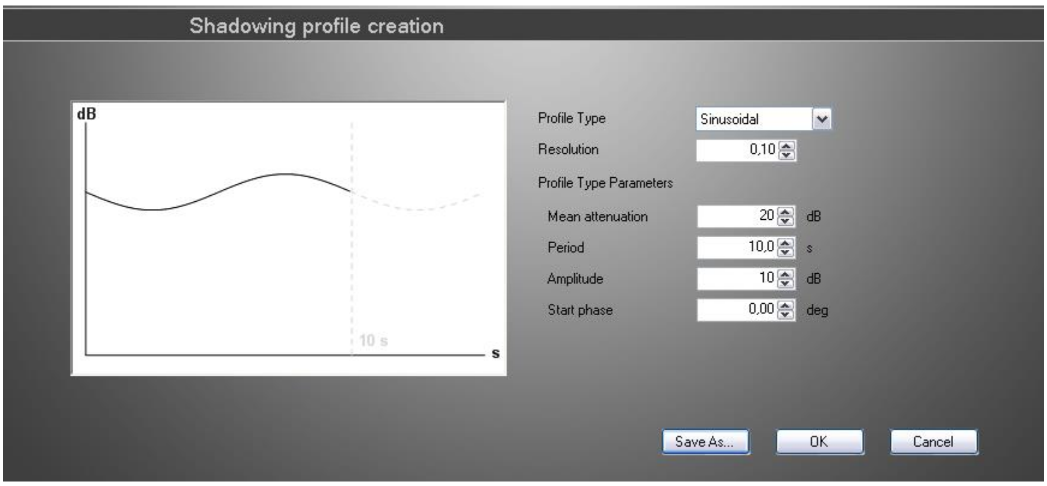


Figure 210 Shadowing profile creation in Scenario Wizard (MANET/Mesh)

Note of attenuation

- Under no circumstances can attenuation values be smaller than 0 dB or larger than 100 dB; if values that would cause such situation are entered, profile is changed automatically to match these limitations.

Profile type

- The following profiles types are available Lognormal, Sawtooth, Sinusoidal, Triangle and User defined. See the following subchapters for more information about each profile type.

Resolution (s)

- Specifies how often shadowing is updated.

12.4.1 Lognormal shadowing profile

Lognormal shadowing profile is a distribution of a random variable whose decibel value follows the normal distribution.

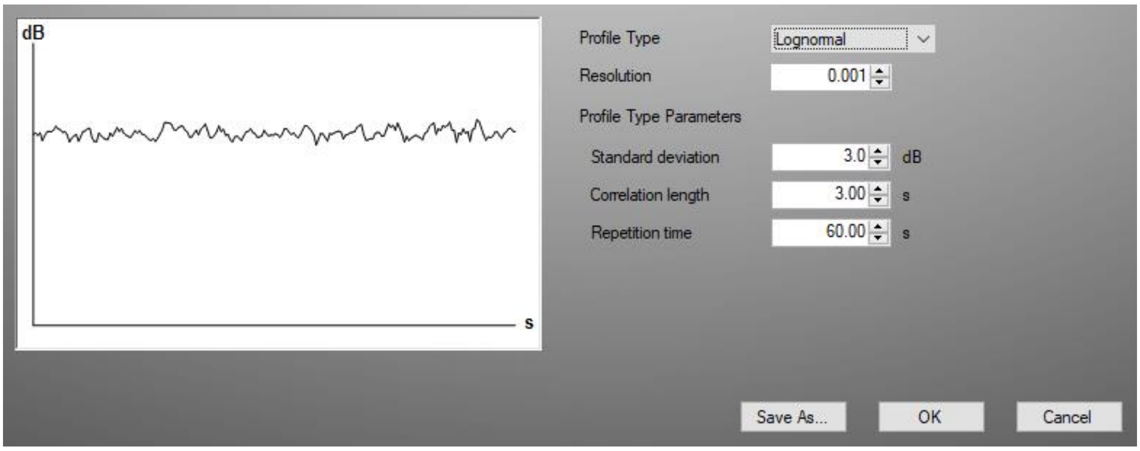


Figure 211 Lognormal shadowing profile

Profile Type Parameters

Standard deviation (dB)

- Standard deviation describes the dispersion of attenuation values. With larger values more deviation from mean value will occur. Default value is 5.0 dB.

Correlation length (s)

- Correlation length describes the time correlation of the Lognormal profile. Slow fading between points separated by correlation length is uncorrelated. Default value is 3.00 s.
- Mean attenuation of Lognormal profile depends on the standard deviation. The mean attenuation is automatically adjusted so that the profile is not cut.

12.4.2 Sawtooth shadowing profile

With a sawtooth profile, the attenuation increases or decreases linearly between defined start and end attenuations during a defined period time.

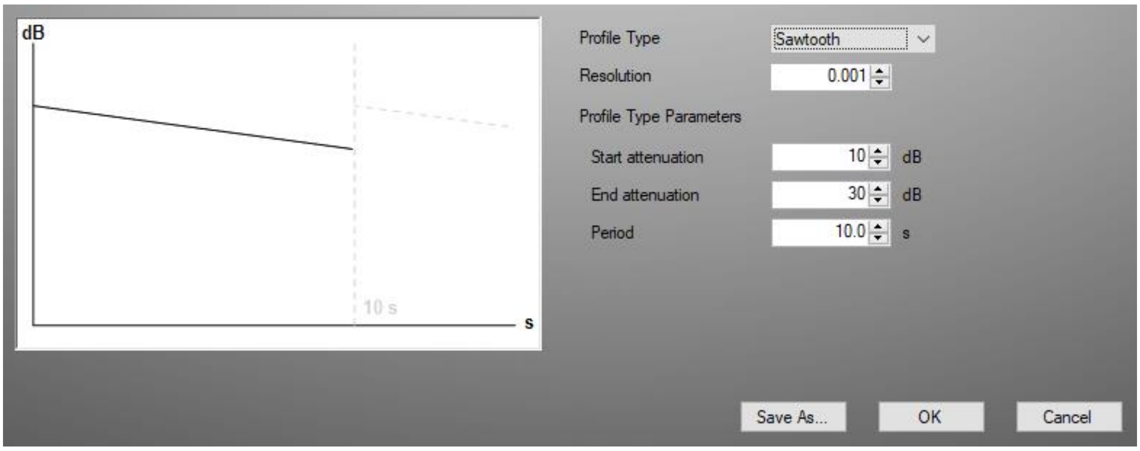


Figure 212 Sawtooth shadowing profile

Profile type parameters

Start attenuation (dB)

- The start attenuation of sawtooth profile. Default value is 10 dB.

End attenuation (dB)

- The end attenuation of sawtooth profile. Default value is 30 dB.

Period (s)

- The period time of sawtooth profile. After the period time the profile is started again from the start attenuation. Default value is 10.0 s.

12.4.3 Sinusoidal shadowing profile

With a sinusoidal profile, the attenuation varies according to the defined sinusoidal curve.

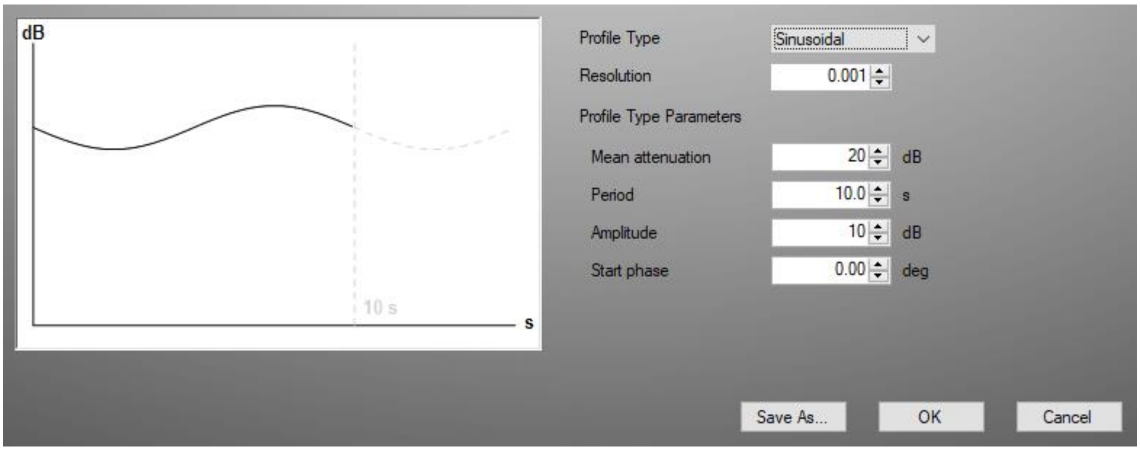


Figure 213 Sinusoidal shadowing profile

Profile Type Parameters

Mean attenuation (dB)

- Average attenuation of sinusoidal profile. Default value is 20 dB.

Period (s)

- The period time of sinusoidal profile. After the period time the profile is started again from the start point. Default value is 10.0 s.

Amplitude (dB)

- The amplitude of sinusoidal profile. Default value is 10 dB.

Start phase (deg)

- The phase angle in which the sinusoidal profile is started to run. Default value is 0.00 deg.

12.4.4 Triangle shadowing profile

With a triangle shadowing profile, the attenuation varies linearly from start attenuation to mid-point attenuation and back to start attenuation during the period time.

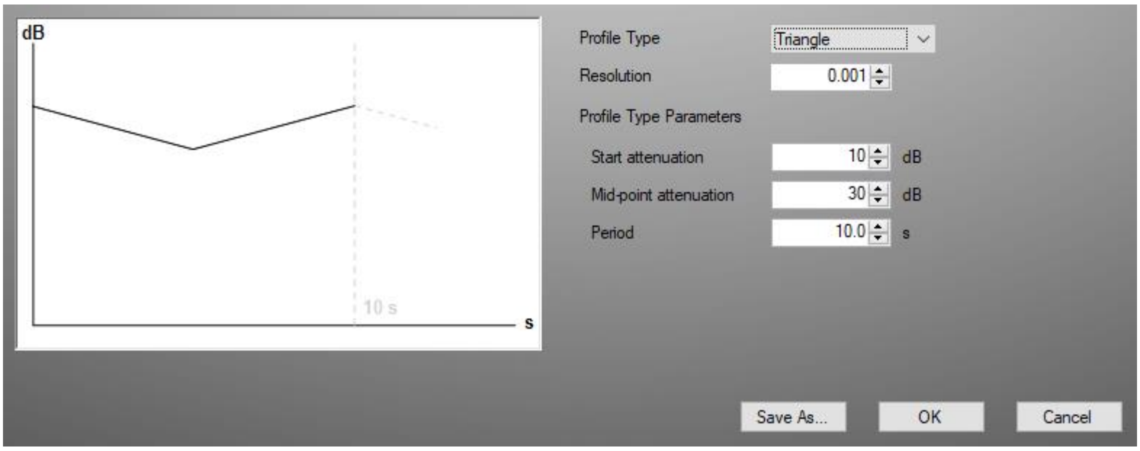


Figure 214 Triangle shadowing profile

Profile Type Parameters

Start attenuation (dB)

- The start attenuation of triangle profile. Default value is 10 dB.

Mid-point attenuation (dB)

- The attenuation where attenuation starts to vary back towards the start attenuation. Default value is 30 dB.

Period (s)

- The period time of triangle profile. After the period time the profile is started again from the start point. Default value is 10.0 s.

12.4.5 User defined shadowing profile

In a user defined profile, you can define the attenuation values and the time in seconds between value changes. Adding time positions and attenuation values in the normal fashion produces ramps from point to point. To add a fast step, add two points having the same time value, the latter having the attenuation after the step.

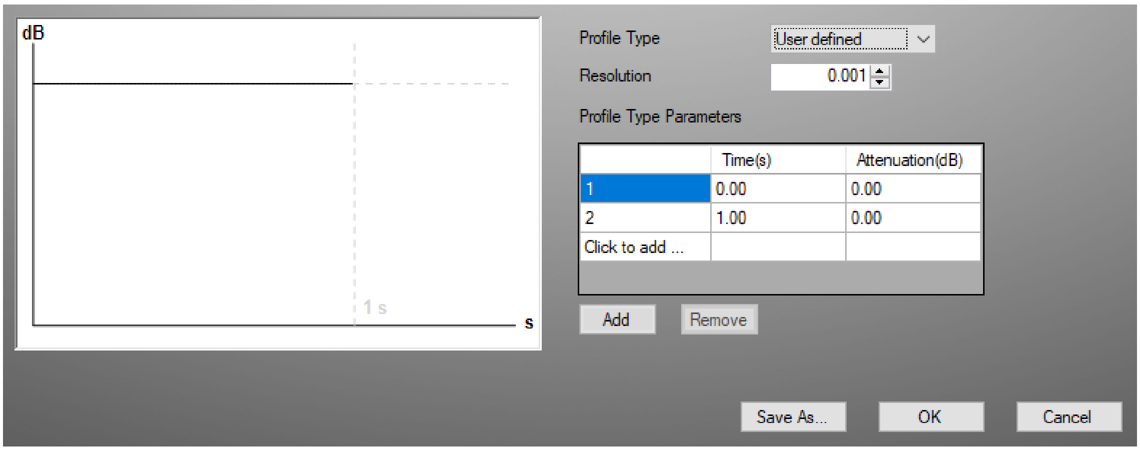


Figure 215 User defined shadowing profile

Profile Type Parameters

Time (s)

- The time from the start point of shadowing profile

Attenuation (dB)

- The attenuation value in the defined time

13 INTERNAL INTERFERENCE GENERATOR (OPTIONAL FEATURE)

13.1 Interference types

Supported interference types are AWGN (Additive white Gaussian noise) and CW (Carrier Wave).

Features of these interferences can be summarized as follows:

- Located at PROPSIM output
- Combined internally
- Can be added/removed in edit phase (before building) and at run time (ATE/Emulation Control View)
- Dynamic power profile is defined to interferer itself (before building)

Interference sources can be added to the emulation in the Scenario Wizard, see section 3.1.3.1.4. For instructions on editing an existing emulation in the Scenario Wizard, see section 3.2.

At runtime, interference generator can be added or controlled through the PROPSIM GUI or with ATE commands, see section 4.3.5.

13.2 Interference settings

Interference settings can be accessed by selecting the interference element in the emulation diagram in Emulation Control View. For more information, see section 4.3.5.

Interference settings can also be accessed in Scenario Wizard as described in 3.1.3.1.4.

The interferences support following power adjustment modes: constant power and constant C/I ratio.

14 PHASE NOISE GENERATOR (OPTIONAL FEATURE)

14.1 Definition

Definition for phase noise is *time-varying random phase error in a signal*. Phase noise can be thought as a modulation error, modulating the signal's phase (for example a carrier wave tone). In frequency domain, phase noise manifests itself as frequency spectrum spreading of the fundamental single tone.

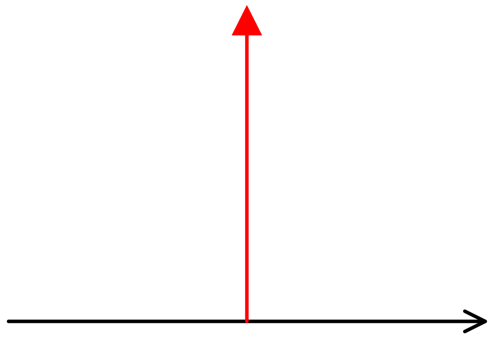


Figure 216 Theoretical oscillator output frequency spectrum

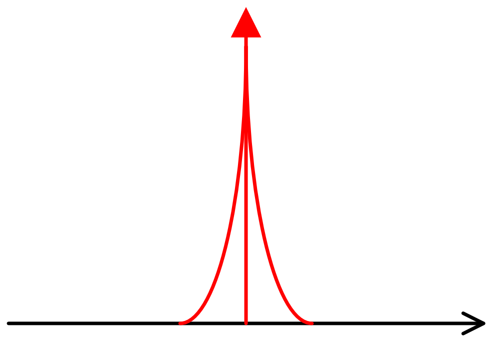


Figure 217 Real-life oscillator output frequency spectrum with phase noise

14.2 Phase noise generation in PROPSIM

Phase noise generator in PROPSIM is implemented as shown in the figure below. In any input's/output's signal chain there is a phase modulator to which the generated and configured phase noise is fed. This performs the phase noise phase modulation to the user signal.

Figure 218 Phase noise generation implementation in PROPSIM

The phase noise generator can be added to any input/output to emulate transmitter or receiver phase noise. The phase noise generator can be configured to an arbitrary phase noise profile within seven fixed frequency offset points with corresponding phase noise power density values, considering the limitation that each frequency offset point must have a power density value in a range of -20 ... + 40 dBc compared to the previous point. Absolute power density value limits can be seen from the tooltip. Emulations/scenarios containing phase noise generators can be created and edited with Scenario Wizard available within the PROPSIM UI.

14.3 Creating a phase noise emulation in PROPSIM

1. To create an emulation, select "new emulation" button at the center of the home screen. It creates a new emulation to Scenario Wizard.

2. In the Scenario Wizard page 2 (Device and link information), at the bottom left corner, select **Profiles and SNR**. **Edit link profiles and SNR** view opens.

3. Go to **RF Impairments** page.

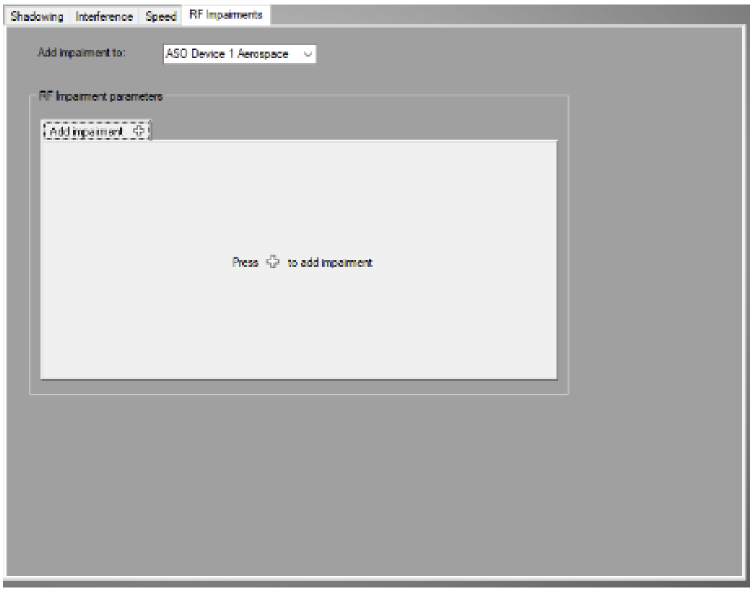


Figure 219 RF Impairments page

4. Select the device's technology in **Add impairment to** dropdown list.
5. Select **+** to add an impairment for TX and RX. TX phase noise is applied to PROPSIM input and RX phase noise to PROPSIM outputs. Power density values can be edited to achieve the desired phase noise profile.

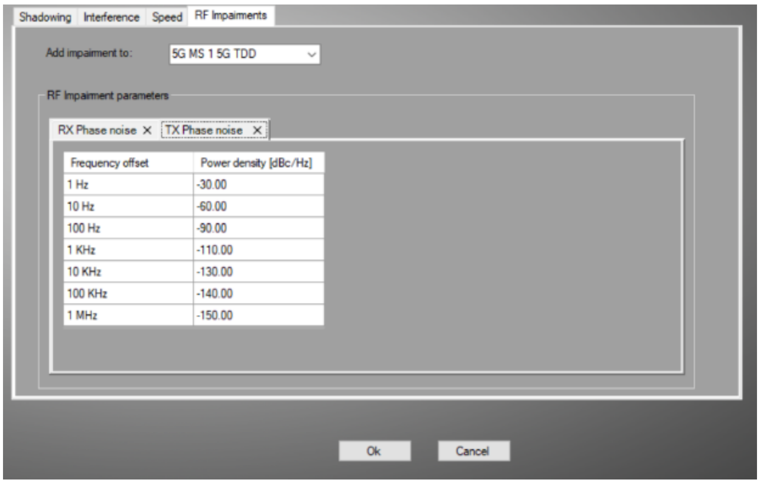


Figure 220 TX and RX impairments

6. To delete an impairment, close the page by clicking X. A confirmation dialogue opens.

Applied phase noise is shown as a “PHN”-symbol in the topology, according to the applied device’s technology.

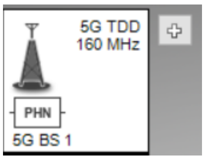


Figure 221: PHN symbol in chosen technology box

To edit the RF impairments (phase noise), right-click the associated device’s technology.

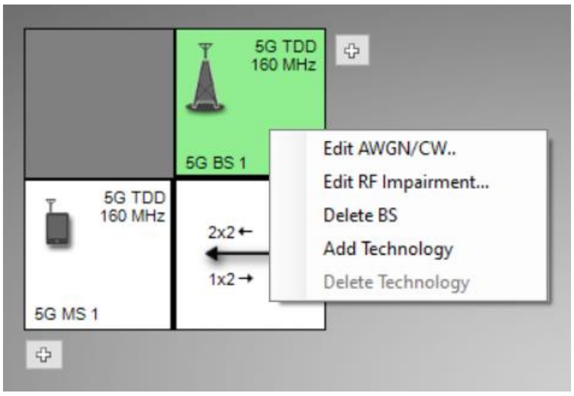


Figure 222 Editing RF impairments

Even if no RF impairments are defined, the RF impairments tab opens, where the user can add/remove phase noise to each device’s technology.

14.4 Running a phase noise emulation

1. Build and open the phase noise emulation for running (see chapter 4)
2. In the emulation control view, to view detailed information about the emulation, click the elements in the emulation graph.

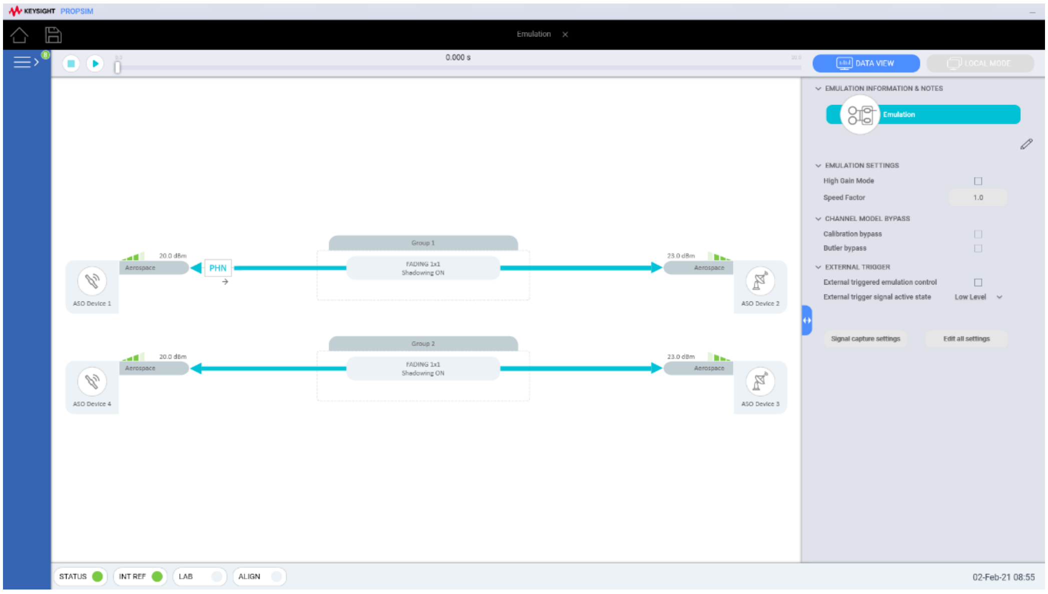


Figure 223 Emulation control view with a phase noise in the emulation

Each direction can be disabled or enabled separately from the UI (see chapter 4.3.6)

Arrows below the PHN symbol indicate the direction of the phase noise effect

Figure 224 Direction of the phase noise effect

Color of the symbol and/or arrows indicate the status of the phase noise.

- Light grey arrow means direction disabled

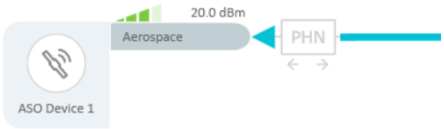


Figure 225 Both directions disabled

- Grey symbol means both/all directions disabled

15 SIGNAL CAPTURE (OPTIONAL FEATURE)

15.1 Definition

With signal capture feature you can capture raw IQ data from PROPSIM inputs and save it into desired location in text or binary format. Each channel unit can capture up to 8 inputs simultaneously, depending on device type and license configuration.

F8800B and F8820B support capturing 8 inputs of each channel unit simultaneously

F8800A and F8820A devices require option F88x0A-ME1 to support 8 simultaneous captures in each channel unit. Without it, 4 simultaneous captures are allowed where two of them must be located on the left side of the channel unit and two on the right, as illustrated in Figure 226.

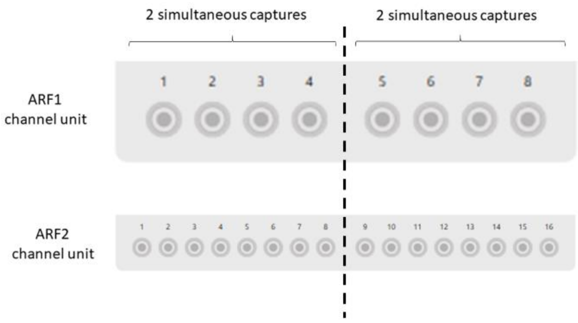


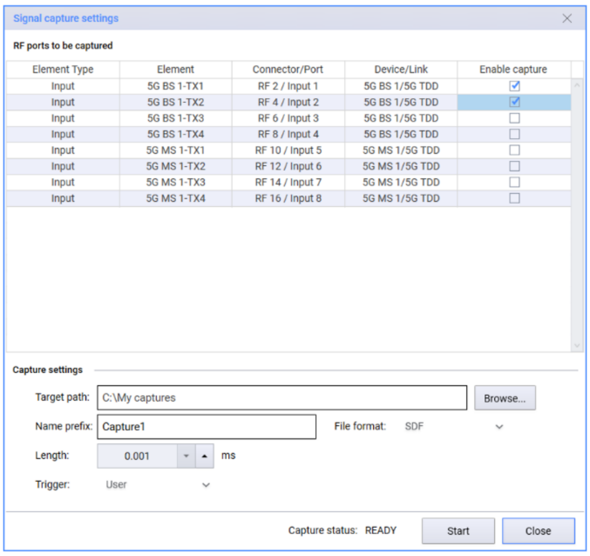
Figure 226 Capturing four inputs in a channel unit without memory extension option F8800A-ME1

Signal capture is supported for the inputs where the emulation bandwidth is 160 MHz or less (F8800A and F8820A) or 200 MHz or less (F8800B and F8820B).

15.2 Signal capture settings

To open Signal Capture settings,

1. In Emulation Control View, click on empty area on emulation graph to see “Emulation Information & notes”
2. In the information area on the right, click Signal Capture settings button. Signal Capture Settings dialog opens.


Signal capture settings

RF ports to be captured

Element Type	Element	Connector/Port	Device/Link	Enable capture
Input	5G BS 1-TX1	RF 2 / Input 1	5G BS 1/5G TDD	☑
Input	5G BS 1-TX2	RF 4 / Input 2	5G BS 1/5G TDD	☑
Input	5G BS 1-TX3	RF 6 / Input 3	5G BS 1/5G TDD	☐
Input	5G BS 1-TX4	RF 8 / Input 4	5G BS 1/5G TDD	☐
Input	5G MS 1-TX1	RF 10 / Input 5	5G MS 1/5G TDD	☐
Input	5G MS 1-TX2	RF 12 / Input 6	5G MS 1/5G TDD	☐
Input	5G MS 1-TX3	RF 14 / Input 7	5G MS 1/5G TDD	☐
Input	5G MS 1-TX4	RF 16 / Input 8	5G MS 1/5G TDD	☐

Capture settings

Target path: C:\My captures Browse...

Name prefix: Capture1 File format: SDF

Length: 0.001 ms

Trigger: User

Capture status: READY Start Close

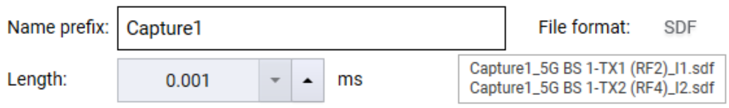
Figure 227 Signal capture settings

At the top of the dialog there are the RF ports that are used for capturing data.

To enable signal capture, select the check box of the desired port.

To choose a location for the captured data, click **Browse** at the bottom of the dialog and navigate to the desired destination.

To define prefix for the capture file(s), type text in **Name prefix** field. Hoovering mouse over the control will show the complete name of file(s) to be created (with port and technology information).



To define the file format to be used for the capture, choose the format in the **File format** dropdown list. The default file format is SDF (Keysight binary) that is a native format of Keysight PathWave Vector Signal Analysis (89600 VSA) software.

To define the length of the capture in µs, choose the length in the **Length** dropdown list. All the settings in this dialog are saved in the emulation file. The default capture length is 10 ms.

To define the trigger of the capture, choose trigger type in the **Trigger** dropdown list, see Table 15.

To start capturing, click **Start**. When Capture status becomes READY an individual IQ data files have been created for each capture sources and copied into **Target path** according to **Name prefix**.

Table 15. Signal capture trigger modes

Capture trigger mode	Description
User	Capture is started manually by pressing “Start”.
Rising edge BNC C3/C4	Rising edge in the front panel BNC connector C3/C4 triggers the capture. Additional trigger delay can be defined.
Falling edge BNC C3/C4	Falling edge in the front panel BNC connector C3/C4 triggers the capture. Additional trigger delay can be defined.
Time	Emulation time triggers the capture.

15.3 Capturing multiple sources into single file

Multiple sources can be captured into single file using SDF (Multi-signal) file format:

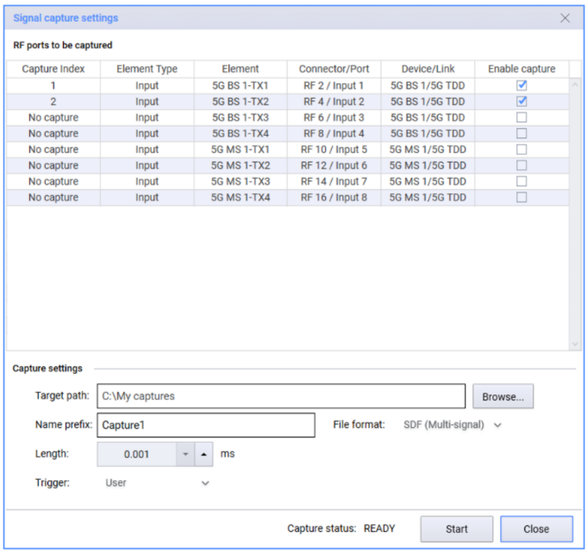


Figure 228 Signal capture settings (multi-signal capture)

The Signal capture settings dialog adds a new column with name **Capture Index** to indicate the order of how selected capture sources are stored into combined IQ data file. This provides a way to map the capture sources according to the cabling of setup in logical order.

It is also important to note that if **Enabled captures** do not share the same parameters, they will be automatically grouped and stored in separate files based on the used band, center frequency and technology.

Note: You can also configure and start capturing with an ATE commands. For details, see chapter 20.4.16

16 SIGNAL WAVEFORM PLAYBACK (OPTIONAL FEATURE)

16.1 Definition

Signal waveform playback feature can be used to stream out IQ waveform files (.wfm) created with Keysight Signal Studio. Data streams from the files are routed out via PROPSIM fading. This allows applying MIMO topology and MIMO fading channel models for the data streams before they come out from PROPSIM.

16.2 Licenses

Signal playback feature requires two separate option licenses:

1) PROPSIM Signal Waveform Playback Tool license, shows in Help->Technical Support -> My Software Support, as shown in Figure 229.

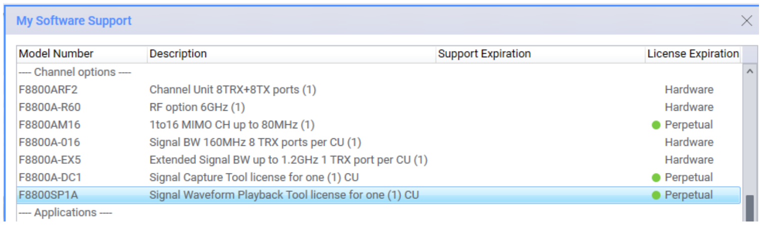


Figure 229 PROPSIM Signal Waveform Playback Tool license

2) Keysight Signal Studio hardware playback license. This license depends on the signal technology (LTE, 5G, WiFi, etc.) used when generating the waveform file in Keysight Signal Studio (N76xxEMBC). This license is installed into Keysight License Manager application, which can be opened from operating system Start -menu, as shown in Figure 230.

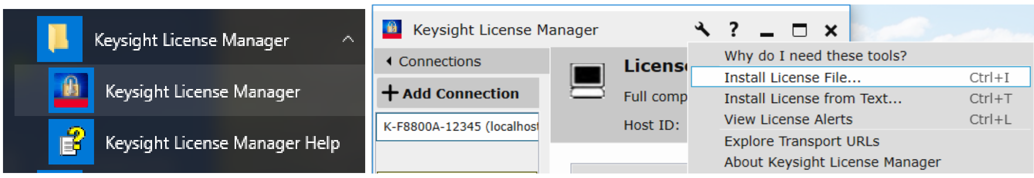


Figure 230 Importing Keysight License Manager license

16.3 Creating waveform playback emulation in PROPSIM

In Scenario Wizard page 2, playback source can be added with -button as shown in Figure 231. Depending on the use case, it can be added either in BS or MS side.

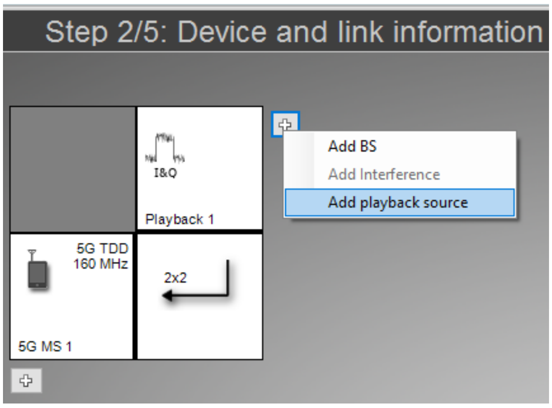


Figure 231 Adding a new playback source in Scenario Wizard

Selecting the playback source opens the playback source properties, as shown in Figure 232. Properties are described below.

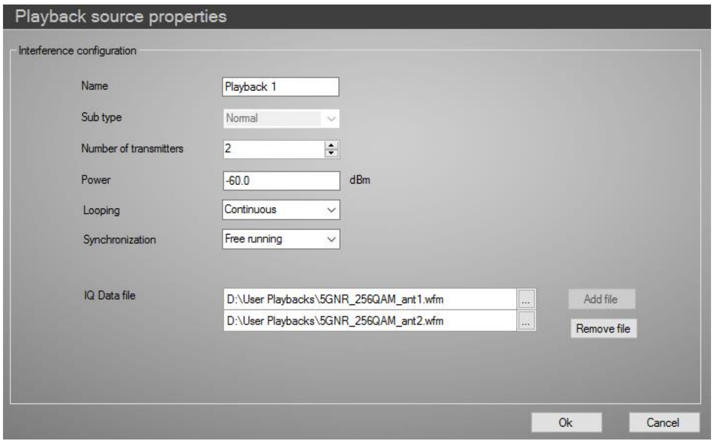


Figure 232 Playback source properties

Name

- User defined name for playback source.

Subtype

- Normal (default)

Number of transmitters

- Number of transmitters (streams). Parameter defines how many IQ Data files (streams) can be selected for the playback source and what will be the MIMO topology (IQ streams as MIMO inputs) of the fading via the playback source is routed to.

Power

- Power of waveform playback at PROPSIM outputs, including the power from MIMO combining of different streams.

Looping

- Continuous (default)

Synchronization

- Free running, playback starts immediately after emulation has been loaded
- Ext. trig rising edge, rising edge in the front panel BNC connector C3/C4 triggers the playback, additional trigger delay can be defined
- Ext. trig falling edge, falling edge in the front panel BNC connector C3/C4 triggers the playback, additional trigger delay can be defined
- Emulation time, defined emulation time trigger

IQ Data file

- .wfm files to be used in playback. Press ... to browse a waveform file. ADD and REMOVE -buttons add or remove additional source files. Maximum number of source files is limited by the parameter “Number of transmitters”. If there are less source files than playback streams (number of transmitters), remaining streams will be assigned the same files, repeating in the same order.

  Waveform file sampling rate must match the PROPSIM type and emulation bandwidth as listed in Table 16 and Table 17.

Table 16. F8800A and F8820A Waveform sampling rates for emulation bandwidths

Emulation/link bandwidth (MHz)	Waveform file sampling rate (MHz)
40	50
80	100
100/160	200
> 160	Waveform playback not supported

Table 17. F8800B and F8820B Waveform sampling rates for emulation bandwidths

Emulation/link bandwidth (MHz)	Waveform file sampling rate (MHz)
50	62.5
100	125
200	250
> 200	Waveform playback not supported

Each playback waveform stream will internally reserve one PROPSIM input connector. F8800A and F8820A devices require option F88x0A-ME1 to support 8 signal playback sources in single channel unit. Without it, 4 signal playbacks in single channel unit is supported. F8800B and F8820B support 8 signal playback sources in single channel unit.

Remaining steps of emulation creation and building follow the same steps as described in detail in the chapter 3.1 (Create emulation with Scenario Wizard).

16.4 Running waveform playback emulation

1. Create waveform playback emulation as described in chapter 16.3.
2. Build and run emulation, emulation opens to Running Graph as shown in Figure 233

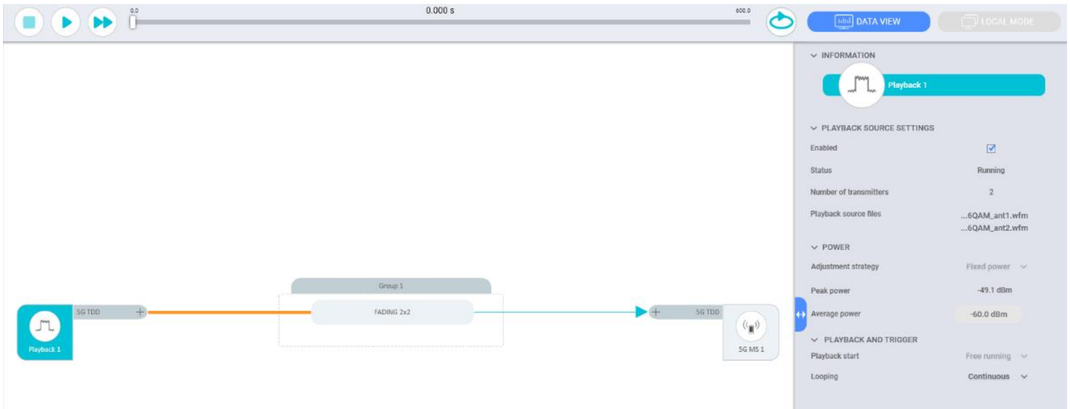


Figure 233. Waveform playback emulation

When playback source is selected, information and settings are shown in the settings view. If the playback is triggered to external signal or emulation time, “Status” -field indicates if the playback is “Waiting for trigger” or “Running”.

Parameters that can be adjusted in running mode:

Enabled

- Enable/disable playback signal

Average power

- Average power of waveform playback at PROPSIM outputs, including the power from MIMO combining of different streams.

Playback looping mode is always continuous. This means that if the waveform playback uses triggered start and it had been triggered (external trigger or emulation time) it continues looping until the emulation is closed and reloaded.

Remote commands for using waveform playback emulations are listed in chapter 20.4.18.

17 INTEGRATED SETUP ALIGNMENT FEATURE (OPTIONAL FEATURE)

- Auto Alignment (Integrated Setup Calibration) is an optional feature, which enables PROPSIM to align phases and levels between multiple channels in test setup. Auto alignment feature includes a wizard that is used to create alignment configurations and perform alignments. Alignment data is automatically saved, and the data can be restored afterwards during the PROPSIM power-on period. ATE remote interface support exists for restoring or status checks of the alignment data.

Note: The terms "calibration" and "auto calibration" are used in the wizard to mean the same as the term "auto alignment": the alignment of phases and levels between multiple channels in test setup. The terms "auto calibration" and "auto alignment" do not mean factory calibration.

Information and external equipment needed for alignment

Before performing auto alignment, the following information should be gathered from test setup to achieve the best possible adjustment result.

- Combined number of base station antennas and active PROPSIM connectors
- Combined number of mobile antennas and PROPSIM connectors, which antennas are cabled
- Used frequencies
- Needed adjustments
    - Phase
    - Gain
    - Group calibration (aligns BS and MS groups)

Alignment is performed by using a thru adapter for making a loop between the antenna cables.

Note: If the setup to be aligned uses FR1 and FR3 frequencies simultaneously (F8800B/F88020B BRF1 + BRF2 hardware), please contact support for additional instructions.

17.1 Starting new alignment

To start a new alignment, select **Align > New Auto Align** in the navigation bar, or click the **ALIGN** button in the status bar, click **AUTO ALIGNMENT** and then **New** in the **Open Auto alignment** dialog.

Figure 234 Start new alignment from

Selecting New Auto Alignment starts the alignment wizard, which is divided into two parts: defining alignment configuration (see chapter 17.2) and alignment procedure (see chapter 17.3).

If you are using Auto Calibration Unit (ACU) in the alignment, check the following items before starting a new alignment:

- ACU is powered on.
- Control cable is connected between PROPSIM connector C7 and ACU connector P1. Impedance of the BNC control cable must be 50 ohms.

17.2 Defining auto alignment configuration

17.2.1 Basic information

After auto alignment wizard is launched, a sheet of basic information for alignment configuration is shown (see Figure 235). If one or more ACU (Auto Calibration Unit) is connected to PROPSIM, Automatic alignment mode is available and the detected Auto Calibration Units are shown for base station and mobile group, as shown in Figure 236.

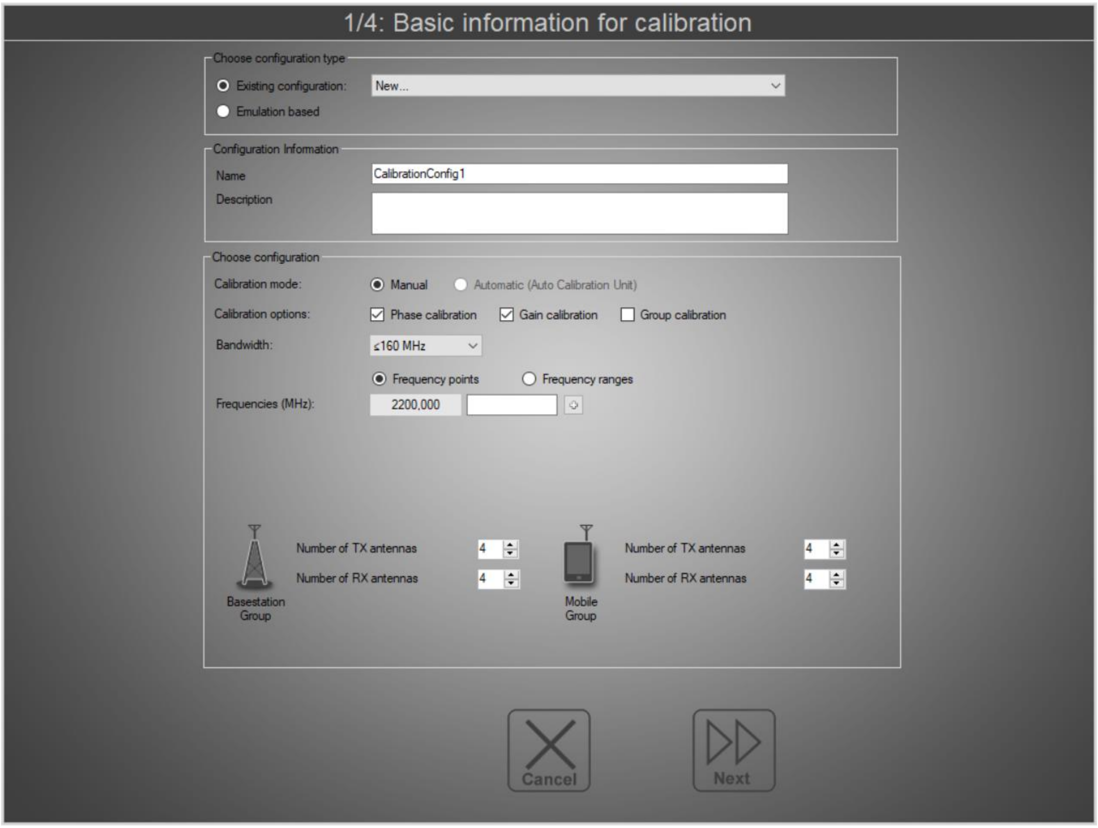


Figure 235 Basic information sheet of auto alignment wizard for manual alignment (without Auto Calibration Units)

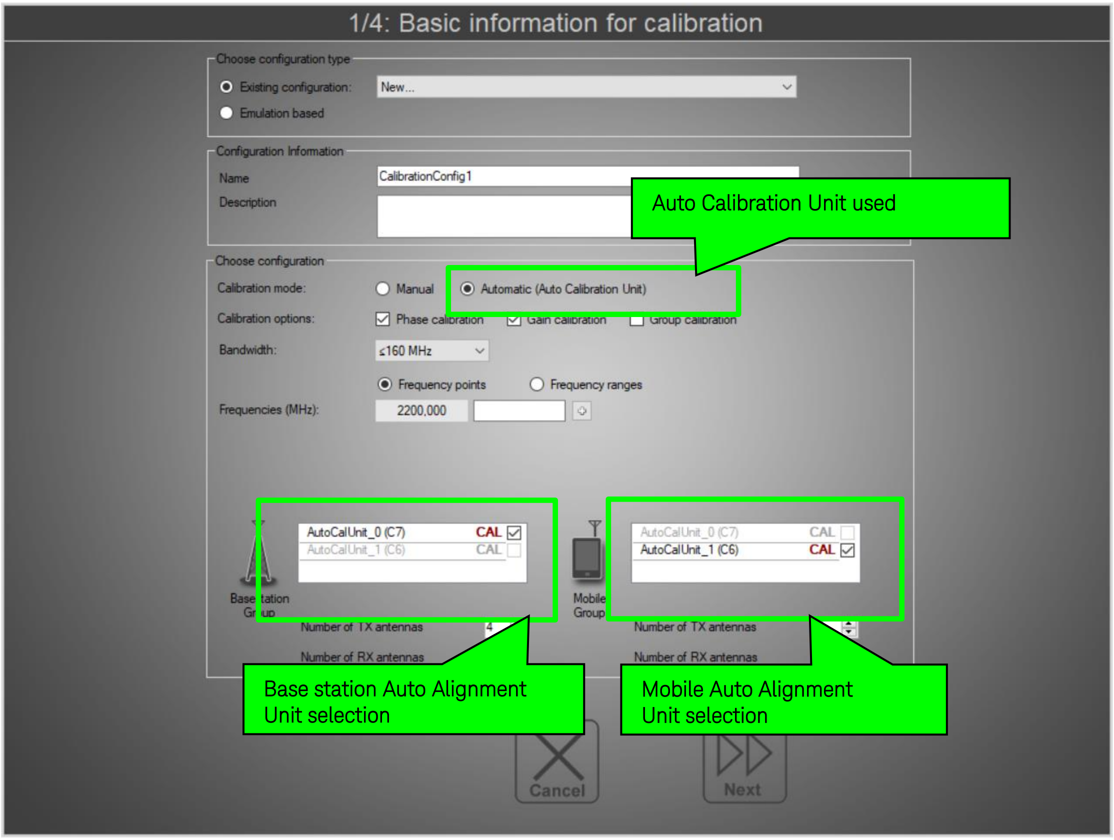


Figure 236 Basic information sheet of auto alignment wizard with use of Auto Calibration Unit

Choose configuration type

Existing configuration

- New...
    - Configuration is set to default values based on emulator hardware.
- Predefined configuration
    - Previously defined configuration could be selected from dropdown menu (see Figure 237).

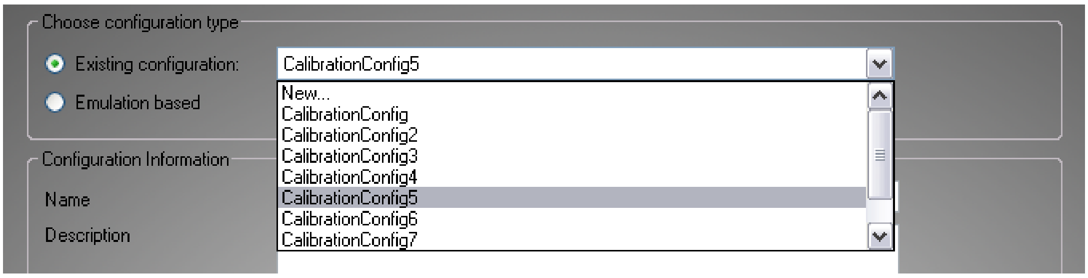


Figure 237 Selecting existing alignment configuration.

Emulation based

- Configuration is gathered from emulation files, see Figure 241.

Configuration information

Name

- Name of the configuration. Named configuration is saved and can be later reused by selecting it from existing configuration dropdown menu.
- Name of configuration is also default name for alignment (see chapter 17.4).

Description

- Description of configuration. Description is also the default description for alignment (see chapter 17.4).

Choose configuration

Calibration mode

- Manual
    - User handles the cable connections between alignment steps (see chapter 17.3.1).
- Automatic
    - PROPSIM automatic alignment device (Auto Calibration Unit) handles the cable connection between alignment steps.

Calibration options

- Phase calibration
    - Phases inside groups are adjusted.
- Gain calibration
    - Levels inside groups are adjusted.
- Group calibration

- Levels and phases are aligned between aligned groups.

By selecting the group calibration, additional alignment steps are added to align phase responses also between uplink and downlink connections. This guarantees fully reciprocal phase balance between uplink and downlink. Group calibration is usually not required but depending on the channel sounding and feedback methods used in DUT's, it may be necessary.

Frequencies

- Alignment frequencies are defined in MHz.
- Frequencies can be given as single frequency points or frequency ranges depending on selected mode (see Figure 238 and Figure 239).
- Additional duration estimate is displayed when frequency range mode is selected as seen in Figure 239.
- If frequency range mode is used, any frequency on the selected range is usable with models.

Figure 238 Frequency point mode

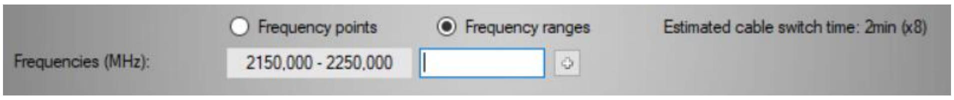


Figure 239 Frequency range mode

- Adding frequencies is done by giving a new frequency or frequency range to an empty field and clicking .
- Frequencies can be modified by double clicking the existing frequency field.
- Frequencies can be removed by clicking in the frequency field.
- Limits are shown in tooltip (see Figure 240).

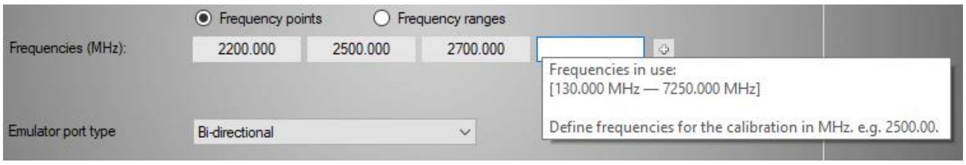


Figure 240 Frequency tooltip.

Alignment groups

- Base station group
  - Combined number of TX and RX antennas in base station group.
- Mobile group
  - Combined number of TX and RX antennas in mobile group.

Navigation pane

Cancel button

- Closes auto alignment wizard. Configuration is not saved.

Next button

- Proceeds to connector configuration (see chapter 17.2.2).

17.2.1.1 Emulation based configuration

In **Emulation based** configuration, alignment configuration data is gathered from emulations. Gathered data includes frequencies, port type, group information, and used emulator ports.

Notes:

- All emulations to be added must have the same connector type (unidirectional/bidirectional).
- Adding an emulation can fail if there is a conflict in the port group information with the previously added emulations.
- Added emulations are not stored in the configuration, only gathered information.

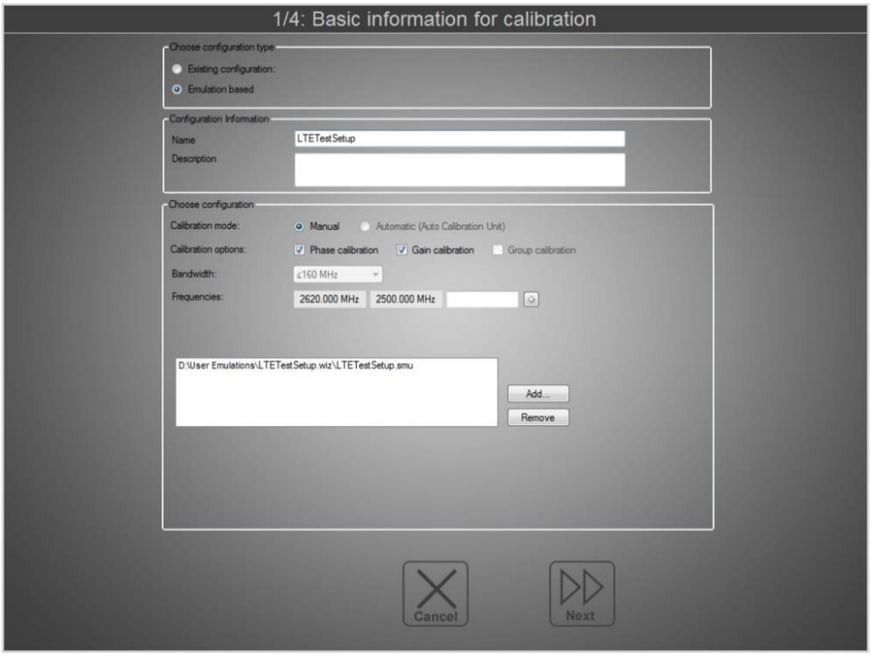


Figure 241 Emulation based configuration

Emulation list

Add... button

- Opens the file browse dialog for selecting emulation.

Remove button

- Removes selected emulation from configuration.

Configuration can be modified after emulations have been added by selecting **Existing configuration**. Existing configuration is now in “<Emulation Based>” mode (see Figure 241).

Figure 242 Selected configuration “<Emulation based>”

Notes:

- *If the emulation list is modified after configuration modification, modifications are lost.*
- *If external duplex components are used together with Emulation based alignment, select Existing configuration after adding all emulations (Figure 242) and then change Emulator port type to “Unidirectional with external duplexers”.*

17.2.2 Connector configuration

Places for base station and mobile antenna connectors are defined in Connector configuration page. When using manual alignment, configuration page appears as in Figure 243. When Auto Calibration Unit is used, setup page appears as in Figure 244. Figure 245 shows the Auto Calibration Unit connectors and their usages.

In User defined configuration case, connectors are allocated to default places, and in Emulation based case, connector places are imported from emulation files. Settings can be verified, or connector locations can be modified to match cable connections of the test setup.

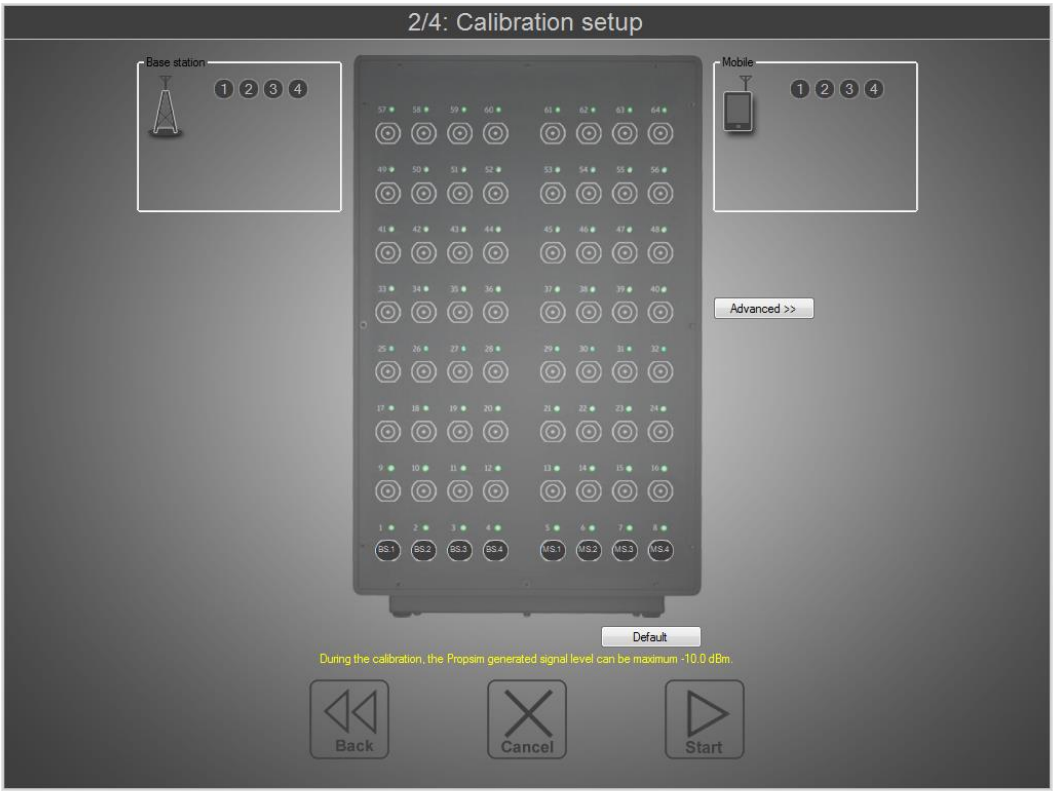


Figure 243 Connector configuration page, manual calibration.

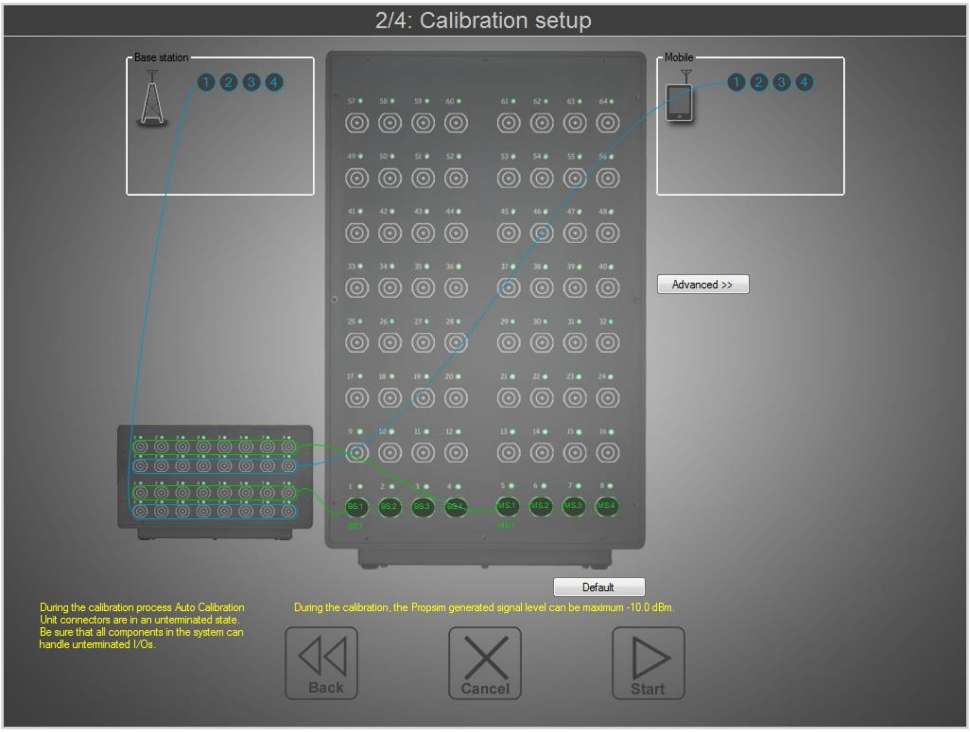


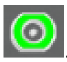
Figure 244 Connector configuration page, Auto Calibration Unit in use

Figure 245 Auto Calibration unit connector order.

Connector Information

Allocated connector information is based on device identification (BS or MS) and connected device antenna number 
.

Relocating connectors

Connector setup can be modified by dragging the wanted antenna from one RF connector to other. When dragging the antenna to a new connector, the suitable connector is shown as green in the front panel: .

Default button

Restores the default connector setup.

Advanced button

Shows the external phase offset editing window as shown in in Figure 246. Phase values for each connector and each frequency are embedded into the alignment data. They are used as additional phase adjustment values in emulation and updated automatically when the emulation center frequency is changed. External phase offsets can be used to compensate fixed phase offsets outside the alignment loop. If the external phase offsets are not defined, the additional phase adjustment values in the emulation are set to zero.

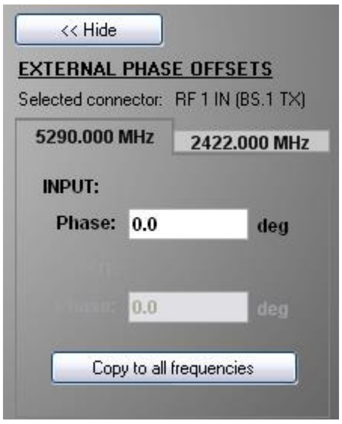


Figure 246 External phase offset edit

Navigation pane

Back button

- Returns to basic information sheet.

Cancel button

- Closes auto alignment wizard. The configuration is not saved.

Start button

- The configuration is saved.
- Begins the auto alignment procedure.
- The Auto alignment procedure initialization may take a few moments, especially the first time when the auto alignment wizard is used.
- The Auto alignment procedure initialization fails, if the emulation is open in the Emulation Control view or ATE.

17.3 Alignment procedure (manual calibration)

The alignment procedure is divided into two states: making the connection and the actual alignment measurement. These two states alternate until all the requisite information is gathered for calculating the phase and level alignments.

17.3.1 Making alignment connection

Auto alignment wizard shows the connector setup for the next alignment step by using blue leds and green cable connection (see Figure 247). The connector setup is also shown in PROPSIM by using blue leds and the connectors to be aligned are shown by green leds.

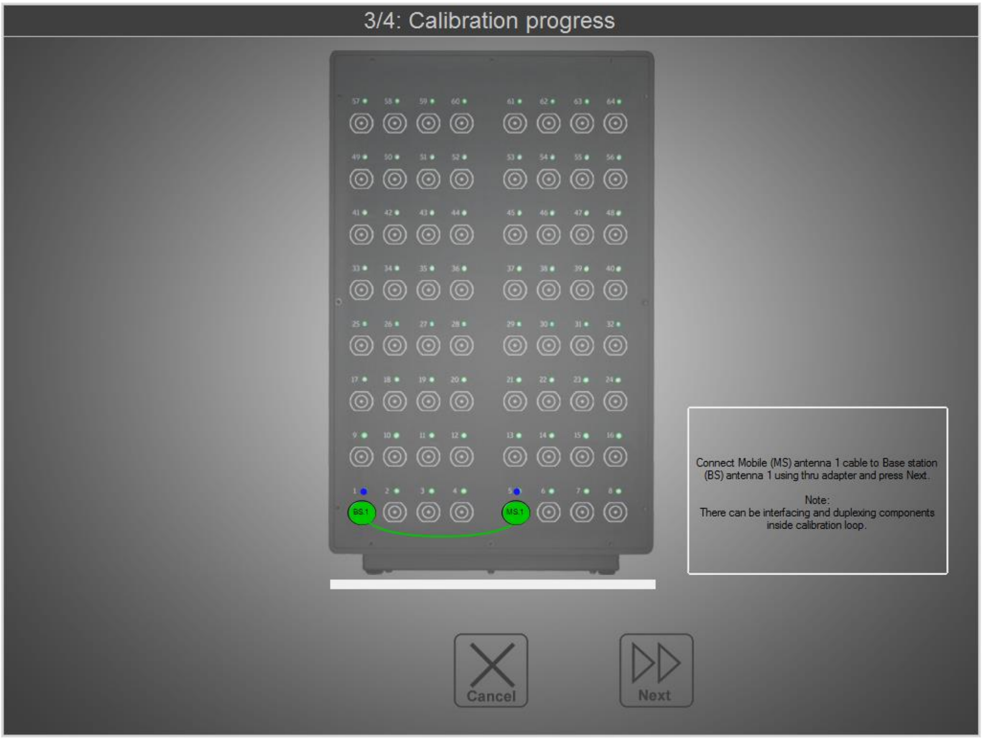


Figure 247 Connection change state of Auto alignment wizard

The connector setup is done by adding a thru adapter between the cables connected to mobile and base station antennas.

Note

- The best alignment result is achieved, if the connection is done as close as possible to DUTs.

Navigation pane

Cancel button

- Closes auto alignment wizard. The alignment is not saved.

Next button

- Proceeds to measurement state (see chapter 17.3.2).

17.3.2 Performing alignment measurement

After the proper cable connection is done and the Next button is clicked, the Auto alignment wizard performs the measurement of alignment values for connection. During measurement, the remaining time estimate and progress is given in a dialog (see Figure 248).

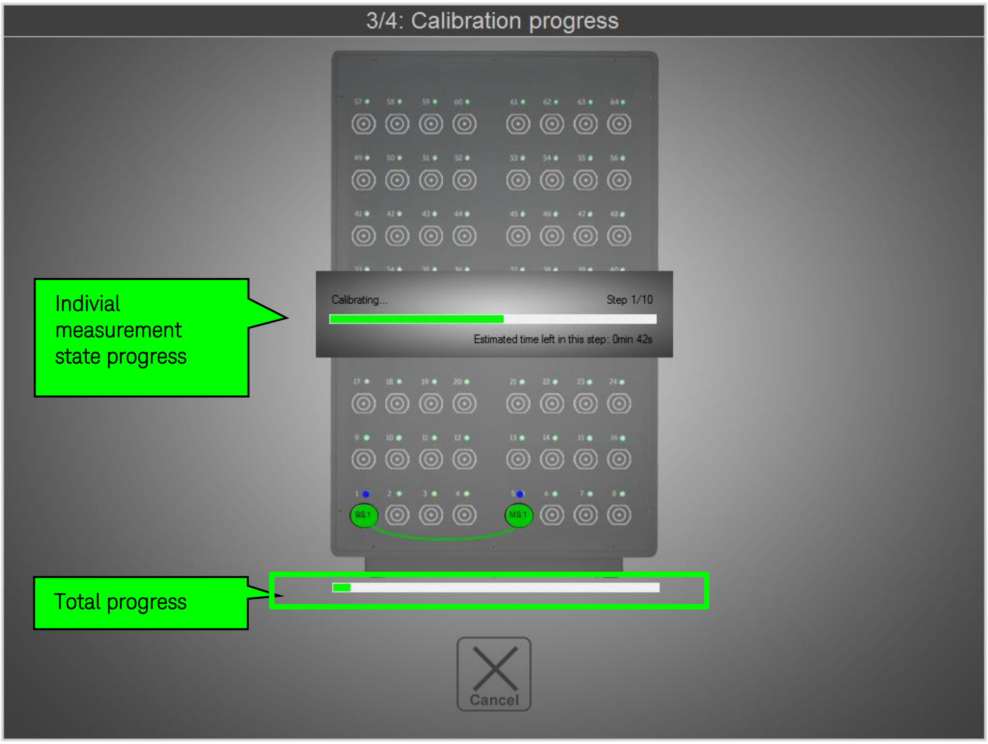


Figure 248 Auto alignment wizard is performing a measurement

Navigation pane

Cancel button

- Closes auto alignment wizard. Alignment is not saved.
- Cancel operation may take few moments during measurement.

17.3.2.1 Error during alignment

If an error occurs during the measurement, the Next button is activated and a red-colored error message is shown like in Figure 249. After the possible reason for error is solved, the Next button proceeds measurement from the last proper state.

Figure 249 Error occurred during the measurement

17.3.3 Performing alignment measurement

When Start is clicked, initializing actions take place and then the wizard starts performing the alignment measurement. If the cables have been properly connected, alignment does not require any user actions (e.g. changes in cable connections) during the alignment. During the measurement, the remaining time estimate with progress information is shown along with the currently measured connector (Figure 250).

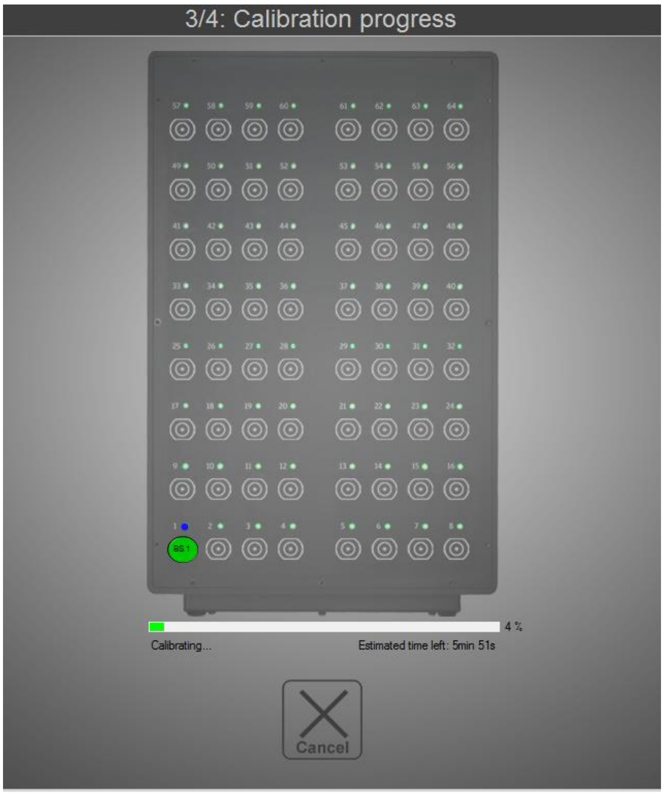


Figure 250 Calibration progress page

Navigation pane

Cancel button

- Closes the auto alignment wizard. The alignment is not saved.
- Cancel operation may take a few moments during the measurement.

17.3.3.1 Error during alignment

If an error occurs during the measurement, the Next button is highlighted in red color and a red colored error message is shown like in Figure 251. After the possible reason for the error has been solved, click the Next button to continue the alignment. The blue leds in both PROPSIM and Auto Calibration Unit mark the currently measured connectors for the easier verification of cable connections.

Figure 251 Error in auto alignment

17.4 Auto alignment finished

After the auto alignment procedure is finished, the last page of wizard is activated automatically showing a brief summary of the alignment. To see also the alignment results, click Show details to view the results in table format like in Figure 252.

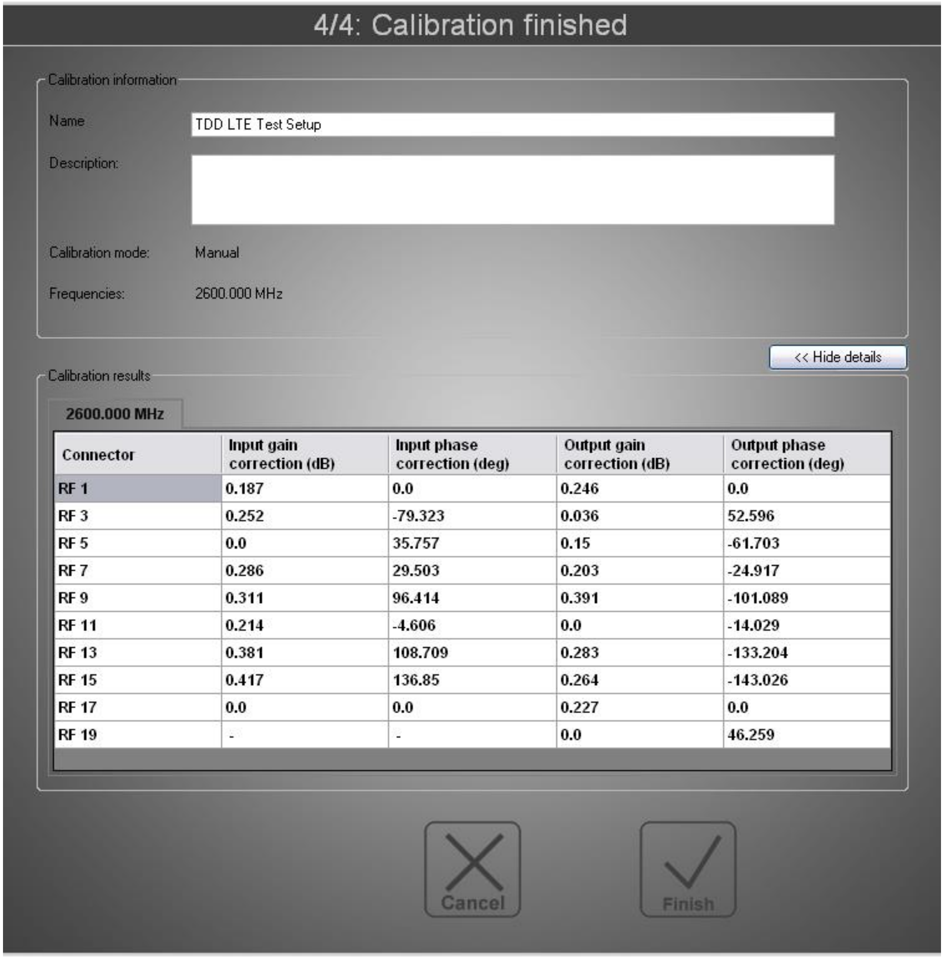


Figure 252 Finish page of auto alignment wizard.

Calibration information

Name

- The name of the alignment.
- The default name is the name of the used auto alignment configuration.

Description

- The description of the alignment.
- The default description is the description of the used auto alignment configuration.
- The description can contain information about the quality of the alignment.

Calibration mode

- The used alignment mode (Manual or Automatic).

Frequencies

- The aligned frequencies.

Calibration results (Show details)

- The table of amplitude and phase corrections

Navigation pane

Cancel button

- Closes the auto alignment wizard. The alignment results are not saved.

Finish button

- The alignment is saved and you can take the alignment into use. (see Figure 253).

Figure 253 Finishing auto alignment

Alignment finish dialog

OK button

- The alignment is saved and taken into use (see Figure 255).

Cancel button

- The alignment is saved. It can be taken into use later (see chapter 17.5).

17.5 Taking auto alignment in use

To take an auto alignment in use, select **Align > Open** in the navigation bar or click the **ALIGN** button. In the **Open Auto alignment** dialog, select the auto alignment file you want to use and click **Select**.

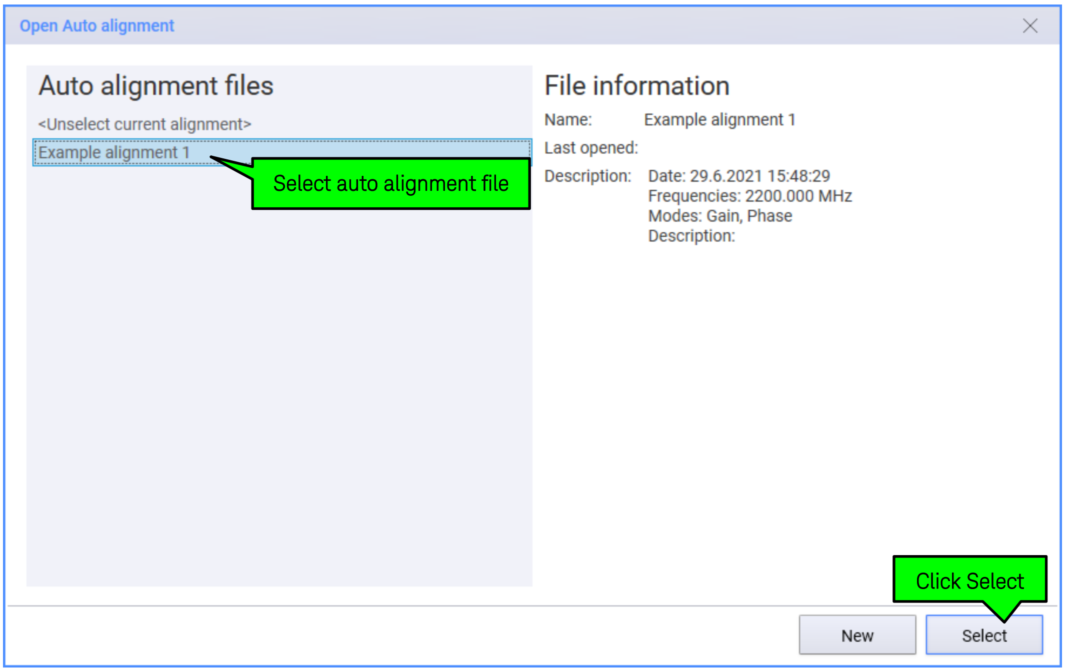


Figure 254 Taking auto alignment in use in Open Auto alignment dialog

Note: Auto alignment results become obsolete when the emulator shuts down. Obsolete alignment files are not shown in the list.

You can also use ATE commands to take the auto alignment in use (see section 20.4.2.10).

When the auto alignment is in use, the **ALIGN** button in the status bar turns green. The tooltip shows the name and the date of the alignment, as well as the frequencies and the auto alignment modes.

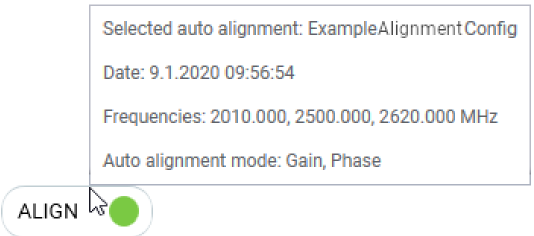


Figure 255 The ALIGN button and the tooltip when the auto alignment is in use

The selected auto alignment values are used in all emulations if the used frequencies and ports are found in the alignment data (see section 17.5.1).

17.5.1 Opening emulation when auto alignment is in use

The compatibility of an emulation with the auto alignment file is automatically verified when you open the emulation. In the BS and MS settings panes, the **Gain alignment** and **Phase alignment** buttons indicate the compatibility.

- If the emulation matches the auto alignment in use, the **Gain alignment** and/or **Phase alignment** buttons turn green.
- If the emulation does not match the auto alignment, the **Gain alignment** and/or **Phase alignment** buttons turn orange and auto alignment is not used with the current emulation. Check the tooltip for more information.

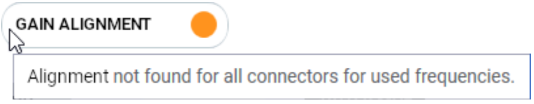


Figure 256 Auto alignment compatibility error and tooltip

Note: If the gain or phase alignment mode are not included in the auto alignment, the corresponding button in the BS and MS settings panes stays grey.

If the auto alignment affects the BS and MS settings, this is indicated in the tooltip of each setting.

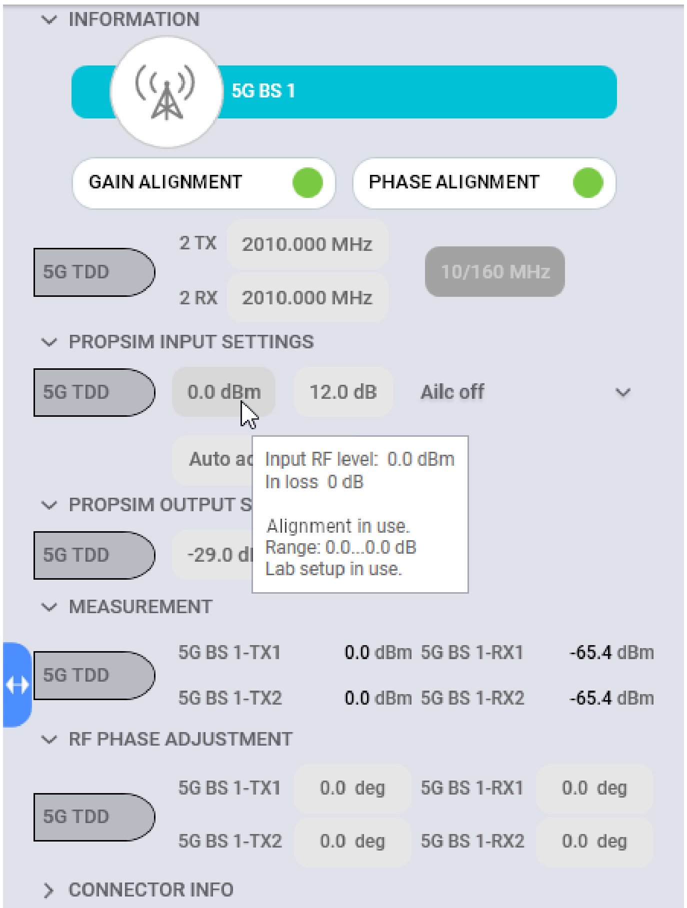


Figure 257 Base station settings, auto alignment status indicates in parameter tooltip

17.6 Taking auto alignment out of use

To take the auto alignment out of use, select Align > Deselect in the navigation bar or select "<Unselect current alignment>" from the Auto Alignment files list in the Open Auto alignment dialog.

18 EXTENDED FREQUENCY RANGE (OPTIONAL FEATURE)

The available PROPSIM base unit frequency range can be extended with external devices:

- E7770A Common Interface Unit (CIU) and M1740A remote radio heads (RRH).
- S9165A Unit and M1742A remote radio heads (RRH)

The CIU supports frequency ranges from 6 up to 15 GHz depending on hardware options. This frequency range covers the highest portion of the 3GPP FR1 frequency range and C/X-band applications. The CIU also supports testing through intermediate frequency (IF) interface (where DUT IF is supported) as this range is used as IF in many mmW radio systems.

The M1740A and M1742A are versatile remote radio heads (RRHs) covering mmWave frequency bands up to 43.5 GHz. The RRH upconverts the RF signal to the mmWave frequency range and downconverts the mmWave signal to PROPSIM frequency range. The CIU provides controls and power for the M1740A RRHs in mmW setups and the S9165A provides controls for the M1742A RRHs. One CIU can support up to eight M1740A RRHs and one S9165A can support up to eight M1742A RRHs.

The additional frequency ranges are:

- 6 ... 12 GHz or 7 .. 15 GHz with E7770A CIU. See section 18.2.
- 28 and 39 GHz mmWave with E7770A CIU and M1740A RRH. See section 18.3.
- 10 ... 32 GHz mmWave with S9165A and M1742A RRH. See section 18.4.

Application notes for extended frequency range:

[1] Dual OTA Test System Setup Guide, Application Note (F8800-93120)

[2] S9165A Test System Setup Guide, Application Note (F8800-93136)

For any assistance regarding the E7770A CIU, S9165A and/or RRH units contact Keysight technical support.

18.1 Taking External Units in Use

When adding the E7770A / S9165A and RRH units to an existing PROPSIM setup, the devices must be taken in use in the following order:

1. Import the appropriate licenses to activate the extended frequencies, see section 9.3.
2. Install the driver for the E7770A / S9165A and RRH units as well as the configuration file, see section 18.1.1 and 18.1.2.
3. Start up the devices, see section 18.1.3.

18.1.1 Installing Driver and Configuration File

You can download the external unit driver software from the Keysight Software Manager (http://www.keysight.com/my). This requires a myKeysight account. The installer also contains a configuration file with a default OTA testing setup with eight RRH units (M1740A) and E7770A CIU.

To install the external unit driver software, run “Propsim_External_Units_Driver_Installer.exe” as administrator. The External Unit Driver is installed into the folder *C:\PROPSIM\ExternalUnitManager*. The driver also installs the configuration file “externalUnit.config” into the folder *C:\PROPSIM\ExternalUnit*.

The configuration file must match the used setup (number of RRHs, RRH ports in CIU / S9165A /PROPSIM and possible chamber rotators). See section 18.1.2. For any assistance or information, please contact Keysight technical support.

18.1.2 Creating Configuration File

Use External Unit Manager to create configuration to match the used external unit setup.

Selecting **Configuration > External Unit Manager** in the navigation menu launches the External Unit Manager UI.

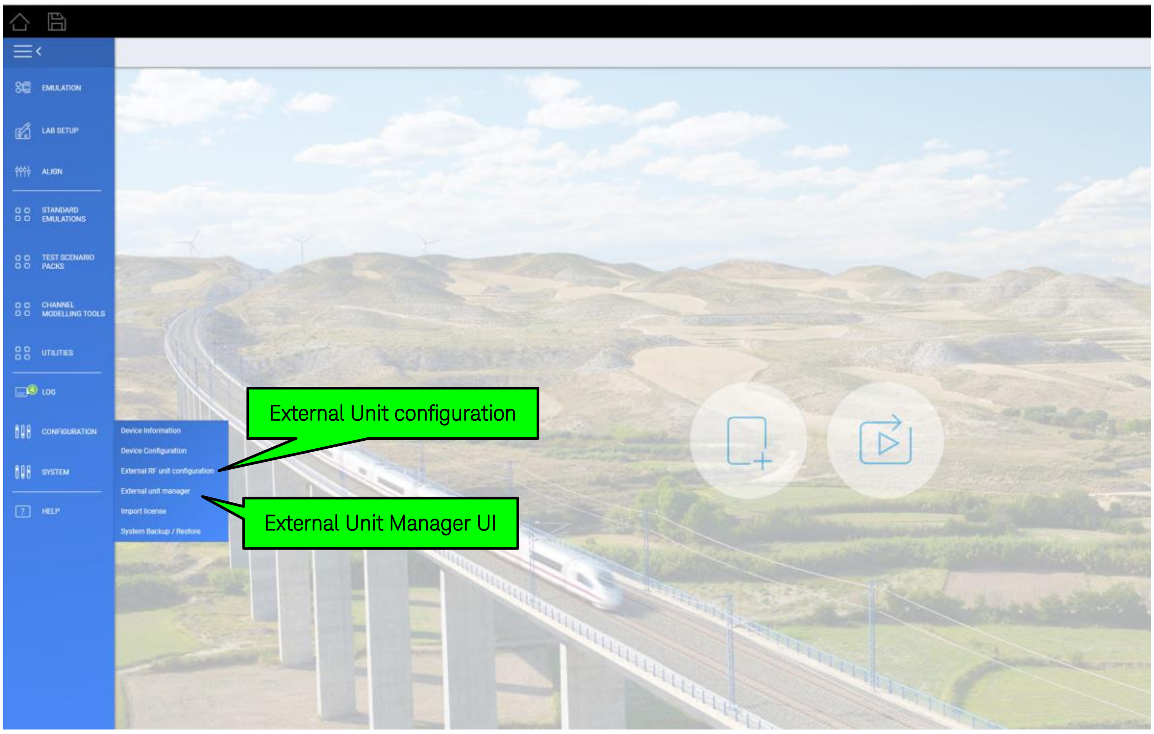


Figure 258 External Unit Manager UI

Select New button and give name for the configuration.

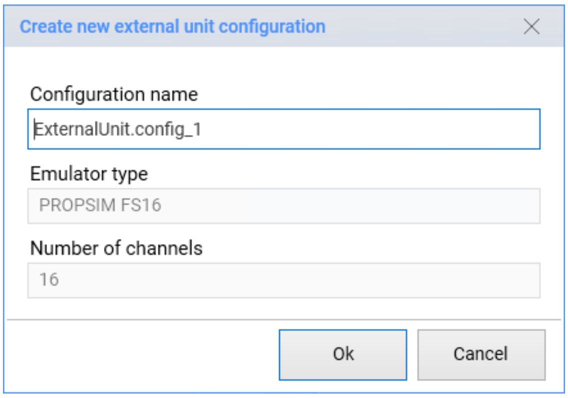


Figure 259 Create new configuration

External Unit Manager opens with an empty configuration as shown in **Figure 260**.

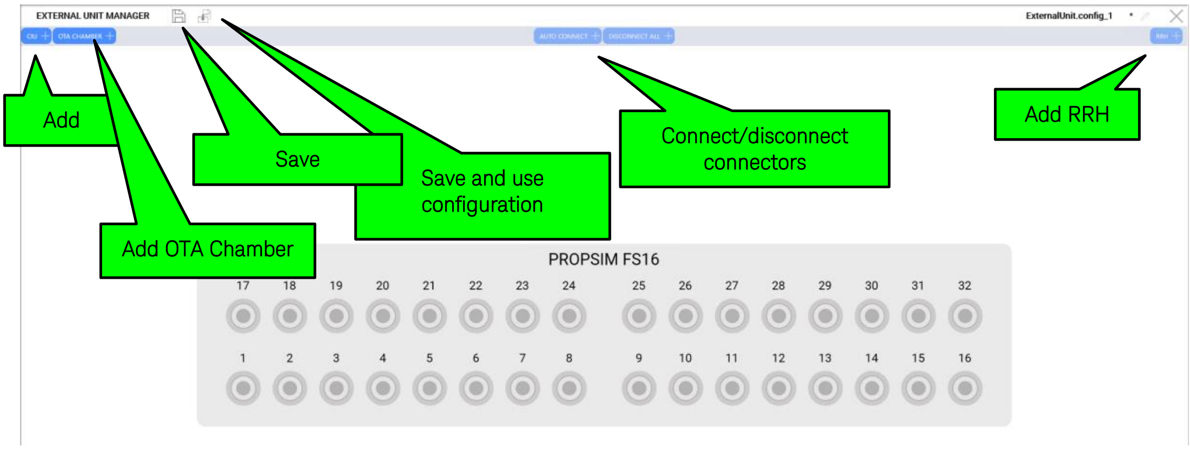


Figure 260 External Unit Manager user interface

18.1.2.1 Add CIU (E7770A)

Click Add CIU E7770 button to create CIU to the configuration. In the Add CIU dialog, choose the Frequency range configuration and Number of CIU channel cards depending on the setup in use. System supports up to two CIU devices in single setup.

For more information on CIU IF Setup see section 18.2.

For more information on CIU + RRH Setup see section 18.3.

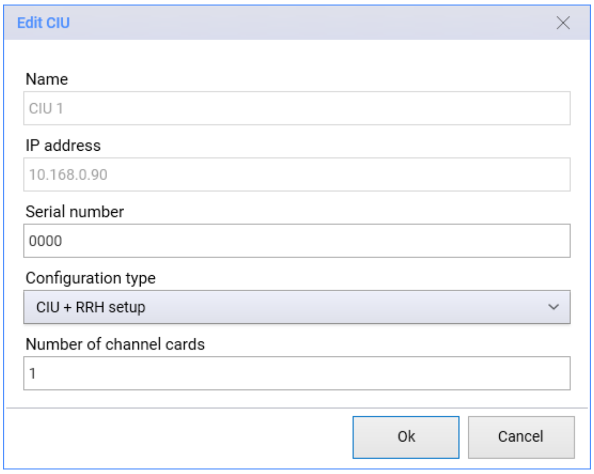


Figure 261 New CIU (E7770A) properties

18.1.2.2 Add S9165A

Click Add S9165A button to create S9165A to the configuration. In the Add S9165A dialog, choose the number of channel cards depending on the setup in use.

For more information on S9165A Setup see section 18.4.

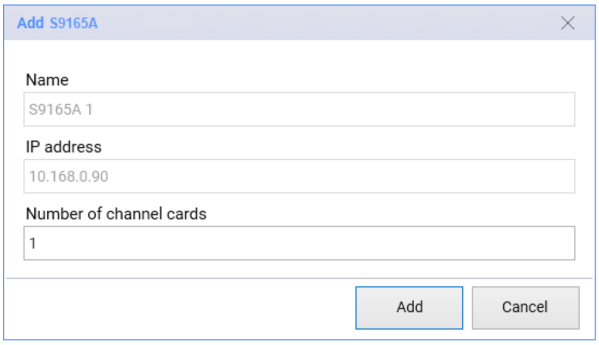


Figure 262 New S9165A properties

18.1.2.3 Add RRH (M1740A / M1742A)

To add RRH to the configuration, either E7770A CIU (28 ... 39 GHz) or S9165A (10 ... 32 GHz) must exist in the configuration.

Select Add RRH button, and in the dialog, set In / Out loss values. When adding RRHs, they are automatically numbered in ascending order.

RRH M1740A specific settings:

Trigger mode: None / CIU Trigger 1...4

When using RRH M1740A in transmit/receive switch mode with external trigger, select the used CIU Trigger input connector for Trigger source. In this mode, RRH RF Tx/Rx 1 port is used to transmit and receive signal.

When transmit/receive switch is not used, select None as Trigger source. In this mode RRH RF Tx/Rx 1 port is used to transmit signal and RF Tx/Rx 2 port is used to receive signal.

RRH LO/Pwr/Ctrl/IF In port is connected to the CIU channel shown in CIU LO/Ctrl/Pwr.

For more information on CIU + RRH Setup (28 ... 39 GHz) see section 18.3.

For more information on M1742A + RRH Setup (10 ... 32 GHz) see section 18.4.

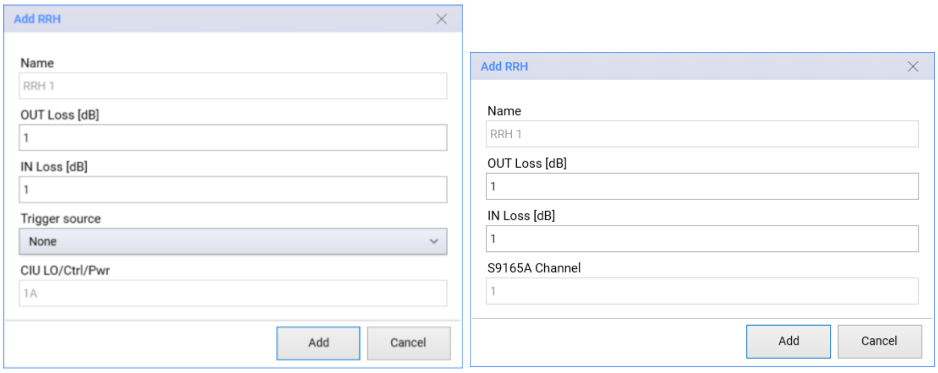


Figure 263 New RRH properties for M1740A and M1742A

18.1.2.4 Add OTA Chambers

To add OTA chamber to the configuration, click Add OTA chamber button.

In the configuration dialog, select chamber Role, Type and connection parameters. Connection parameters are filled with default values when adding a chamber.

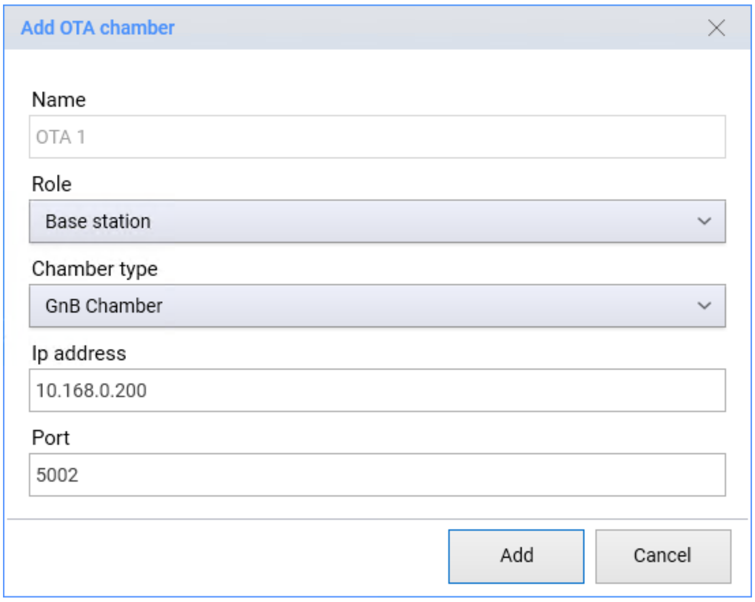


Figure 264 New OTA chamber properties

18.1.2.5 Connector selection

Use Auto connect button to make default RF connections between Propsim and external devices. Connections can also be made manually by using drag and drop with a selected connector.

Disconnect all button clears connections.

18.1.2.6 Managing configurations

Use the **Save configuration** button, to save configuration with a given name. You can have multiple configurations saved in the Propsim. Configurations can be opened for editing and taken in to use as needed.

Use the **Save and use configuration** button to take the currently opened configuration into use during the next boot.

Use **Configuration > External RF Unit Configuration** to enable/disable external units and to view currently used configuration.

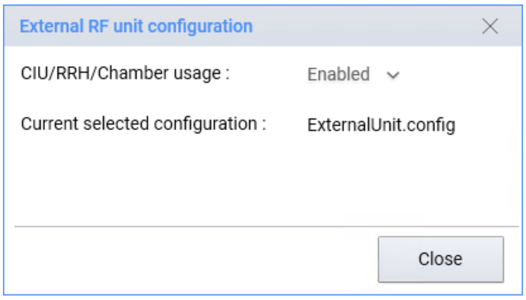


Figure 265 External RF unit configuration enabled

18.1.3 Starting Up Devices

After the licenses, external unit driver, and configuration file are installed, start up the devices exactly as instructed in the following steps:

1. Shut down the PROPSIM by selecting **System > Restart / Shutdown > Shutdown** in the navigation bar. Click **OK** in the confirmation dialog.
2. Check the cable connections to make sure the connections are OK.
3. Turn on the CIU/S9165A unit.
4. Turn on the chambers.
5. Turn the PROPSIM back on. In the first start-up, the PROPSIM will download calibrations from the RRHs. This takes some time, so the first start-up will take 10 ... 15 minutes longer than normal.

Note: The CIU and chambers must be powered on before PROPSIM in every start-up, not only in the first start-up.

When everything is working as expected, the PROPSIM GUI opens without errors. The **Device Information** dialog shows the extended frequencies (**Configuration > Device Information**). If the extended frequencies are used in an emulation, PROPSIM will automatically use the CIU/S9165A and RRH units.

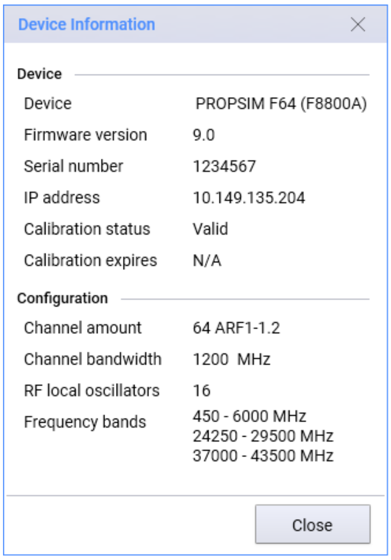


Figure 266 Device information dialog showing extended frequency ranges

At start-up, some ports in the emulator front panel are reserved for the extended frequencies (CIU and RRH units). When you create a new emulation in the Scenario Wizard, only the suitable ports are available for each connector, depending on the frequency (see section 3.1.5 Step 4: Active connectors selection). The Scenario Wizard automatically checks the external devices from the configuration file and the licenses.

18.2 6 ... 12 GHz and 7 ... 15 GHz Frequency Range (CIU)

This frequency range requires a Common Interface Unit (CIU). The PROPSIM operates at sub-6GHz frequency range and the CIU converts the signal to target frequency in range. The Common Interface Unit supports intermediate frequencies in the range 6 GHz to 15 GHz and works with TDD and FDD signals.

Hardware requirements:

- CIU E7770A with channel option E7770A-HB1 - converts signals (6 ... 12 GHz) from Keysight/customer test equipment (where DUT IF is supported).
- CIU E7770A with channel option E7770A-HB2 – converts signals (7 ... 15GHz) from Keysight/customer test equipment (where DUT IF is supported).
- OTA chamber (optional) – provides the environment for the over-the-air testing.

  Note: With CIU E7770A-HB1 option, there is a small gap (8.6 GHz ± 200 MHz) in the intermediate frequency spectrum supported by the Common Interface Unit. You must not assign a center frequency in this gap. For more information, contact Customer Support.

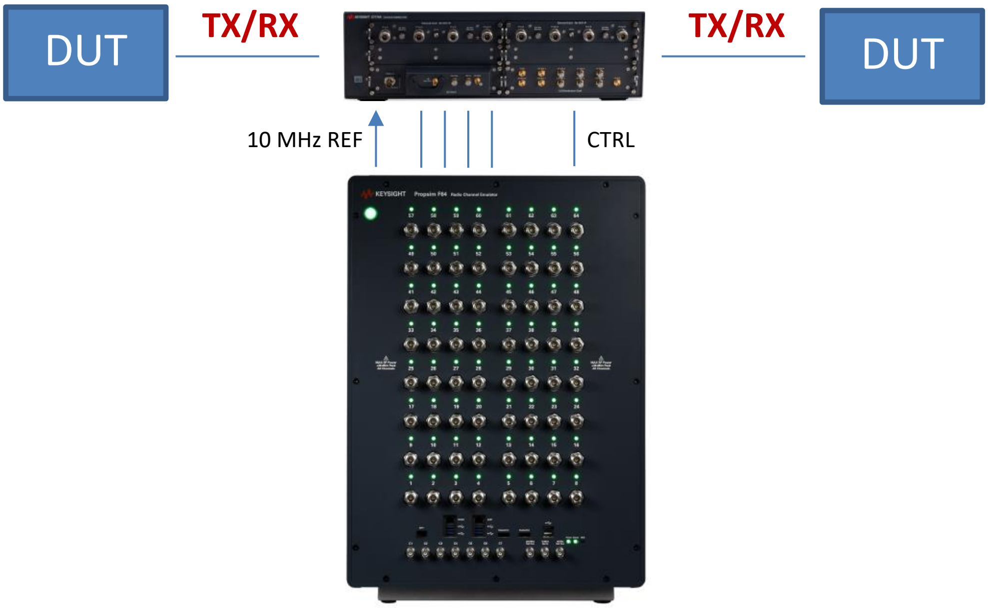


Figure 267 PROPSIM setup with Common Interface Unit (CIU)

18.3 28 GHz and 39 GHz mm Frequency Range (CIU and M1740A RRHs)

Testing in frequencies 28 GHz and 39 GHz requires devices to have antennas that support the 5G NR bands in the mmWave frequency spectrum. Testing at these frequencies uses RF antenna horns inside RF chambers to transmit the signals over-the-air (radiated testing).

Hardware requirements:

- M1740A RRH - up-converts output from CIU to mmWave frequencies, and down-converts mmWave inputs to the CIU.
- CIU - provides control, power, and local oscillator frequency input for mmWave remote radio frequency heads (RRHs). One CIU supports up to eight RRHs.
- OTA chamber (optional) – provides the environment for the over-the-air testing.

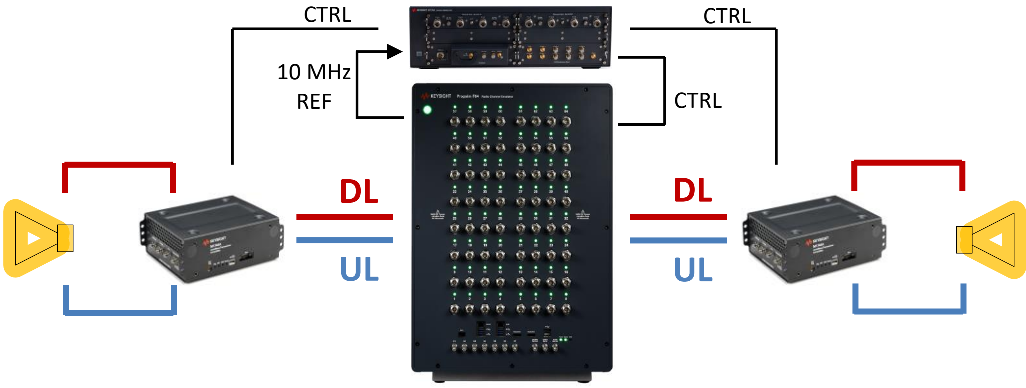


Figure 268 PROPSIM setup with Common Interface Unit (CIU), two Remote Radio Heads (RRH) and antennas

18.4 10 ... 32 GHz mm Frequency Range (S9165A Unit and M1742A RRHs)

Performing mmWave range testing in the frequency range 10 ... 32 GHz can be done with PROPSIM combined with S9165A Unit and M1742A remote radio heads.

Hardware requirements:

- M1742A RRH - up-converts output from S9165A to mmWave frequencies, and down-converts mmWave inputs to the S9165A.
- S9165A - provides control, power, and local oscillator frequency input for mmWave remote radio frequency heads (RRHs). One S9165A supports up to eight RRHs.
- OTA chamber (optional) – provides the environment for the over-the-air testing.

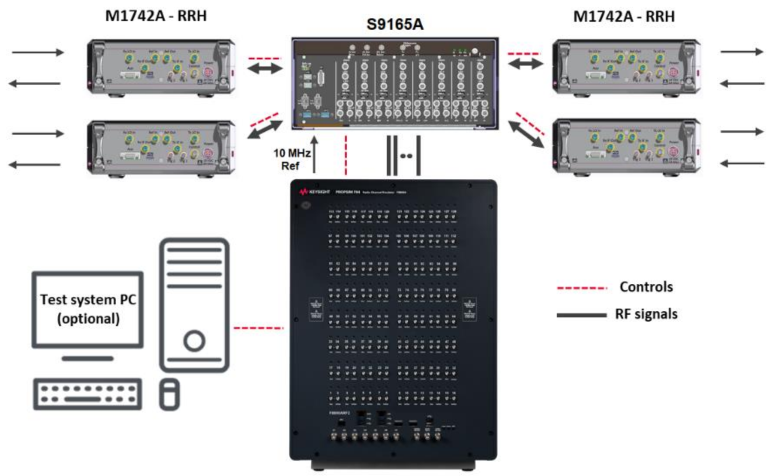


Figure 269 PROPSIM setup with S9165A with four M1742A Remote Radio Heads

19 AEROSPACE AND SATELLITE MODELLING TOOL – ASO (OPTIONAL FEATURE)

ASO enables PROPSIM to emulate high velocity radio channels from ground-to-air and air-to-air links typical in space and satellite applications, aircraft communication, missile control, and radars.

The option contains the Aerospace Model Editor Application and a specific channel emulation engine with the capability to run models with longer path delays than in normal emulation mode and extremely high velocities. The editor application is used to generate, edit, and visualize ASO models.

Key features:

- Capability to emulate radio channels with long delay and high velocity. These emulations have very high accuracy and realistic Doppler effects in both frequency and in code domain.
- Open text-based file format for importing customer specific radio channel and location parameters
- Functionality to validate the model format and data values
- Visualization of user defined dynamic channel models
- Geometric graphs showing channel parameter curves for three-dimensional movement
- Playback functionality to see model evolution in time
- Wizard for generating and editing models

To open the Aerospace Model Editor, select **Utilities > ASO Editor** in the navigation bar.

The Aerospace Model Editor allows user to:

- Validate user created models
- Observe parameters like distance, speed, range rate and gain at different time and object positions
- Visualize geometry of coordinate-based model movement in three dimensions
- Observe Doppler, delay, and range rate curves
- Make partial selections from the most interested part of the model
- Preview time evolution of the models

In addition, the Aerospace Model Editor includes a model generation wizard for creating simple models to different types of ASO file formats. For more information of the ASO file formats, see chapter 21.5.

The following features of PROPSIM are not supported with ASO emulations:

- Butler/calibration bypass functionality
- Shadowing offset and enable/disable in run-time
- Other interferences than AWGN
- AWGN SNR adjustment modes
- Interference profiles
- Emulation step and goto
- Emulation wrapping (wrapping/looping is controlled by individual ASO models in the emulation)

19.1 Aerospace Model Editor

The application window is divided into graphical views and regions with different functions as shown in Figure 270 below.

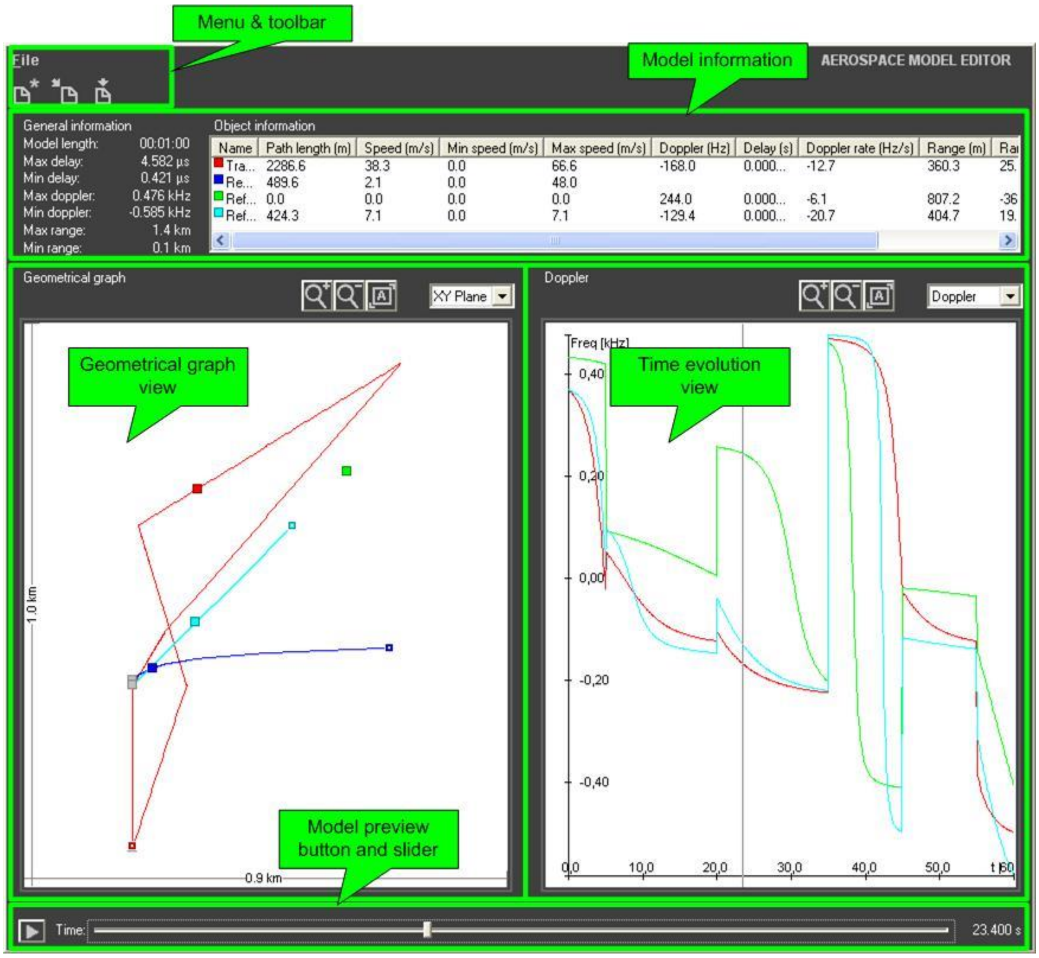


Figure 270. Aerospace Model Editor

19.1.1 File menu

The Aerospace Model Editor file menu contains the following items:

Table 18. File menu of Aerospace Model Editor

Main menu	Submenu	Shortcut	Description
File	Create New ASO model...	Ctrl-N	Opens New Model Generation Wizard
	Open ASO model...	Ctrl-O	Opens existing ASO model
	Edit ASO model...	Ctrl-E	Opens model editing window
	Partial model...	Ctrl-P	Opens Partial model dialog
	Save	Ctrl-S	Saves model
	Save As...		Saves model with new name
	Exit		Closes Aerospace Model Editor

19.1.2 Model Information

General information field shows some main properties of the model.

Table 19. General information of model

General Information	Description
Model length	Model length in seconds
Max delay	Maximum propagation delay of the model
Min delay	Minimum propagation delay of the model. Note: if the minimum delay of the model is smaller than PROPSIM insertion delay, fixed delay offset is added to the model at emulation generation phase.
Max Doppler	Maximum Doppler frequency over the model
Min Doppler	Minimum Doppler frequency over the model
Max range	Maximum range i.e. maximum radio path distance, over the model
Min range	Minimum range (radio path distance) over the model

Object information field shows the status of objects in the model. Object names and information columns shown are dependent on what type of model is selected.

Most of the parameters in this field are time dependent. Current time can be varied by using the model preview slider (see 19.1.4). By clicking the object name with left mouse button, the associated curves in graphs will be highlighted.

Table 20. Object information of model

Object information	Description
Name	Object name: Transmitter, Receiver, Reflector or Link
Path length [m]	Static parameter showing length of the object movement over the model. Information available only for coordinate based (N-type) ASO models
Speed [m/s]	Speed of the object movement. Information available only for coordinate based (N-type) ASO models
Min speed [m/s]	Static parameter showing the minimum speed of the object movement over the model. Information available only for coordinate based (N-type) ASO models
Max speed [m/s]	Static parameter showing the maximum speed of the object movement over the model. Information available only for coordinate based (N-type) ASO models
Doppler [Hz]	Doppler frequency of the link (*
Delay [s]	Delay of the link (*
Doppler rate [Hz/s]	Change rate of the link Doppler (*
Range [m]	Range of the link (e.g. radial distance) (*
Range rate [m/s]	Link range changing rate due to movement of objects (*
Gain [dB]	Actual gain of the propagation link (*
Period [s]	Static parameter showing the time of the periodic cycle used in function-based models. Information available only for Function based ASO models.

**) Note: Propagation links are calculated from other objects to receiver e.g. from transmitter to receiver, directly or via reflections. Therefore, certain link parameters (Doppler, Delay, Doppler rate, Range, Gain) are not shown on the receiver object line.*

19.1.3 Graphical Views

Model time evolution can be observed in the graph on the right-hand side. Doppler / Delay / Range rate curves are shown as a function of time. If there are multiple objects in the model, their graphs are shown in different colours.

In Geometrical graph, on the left side of the Aerospace editor view, object paths can be observed in selected plane (XY, YZ or XZ). The object movement and location geometries are shown only for Coordinate based ASO models.

Both graphical views can be zoomed in and out or reset via corresponding buttons above graphs. Views can be zoomed also by using the scroll wheel of the mouse. Pressing the Ctrl key simultaneously zooms faster when using the scroll wheel. Both views can be panned by holding down the left mouse button and moving mouse around.

19.1.4 Model preview

User can use preview functionality to see how the objects will move over the time and how different values of various parameters change during the actual emulation run. The preview can be started by pressing the start/pause animation button on the left side of the time slider. User can also move the time slider manually by using mouse or use keyboard arrow buttons when the slider is selected.

Values in the Object information box reflect the object states at current time position. Current time is shown at the right end of the slider. If partial model is selected (see 19.1.7) the preview slider will play only the selected active region.

19.1.5 New Model Generation Wizard

Create new ASO model command opens following dialog box (Figure 271) where the user is requested to select what kind of ASO-model will be created. The selected type determines the actual ASO file type. Details about the ASO file formats can be found in chapter 21.5.

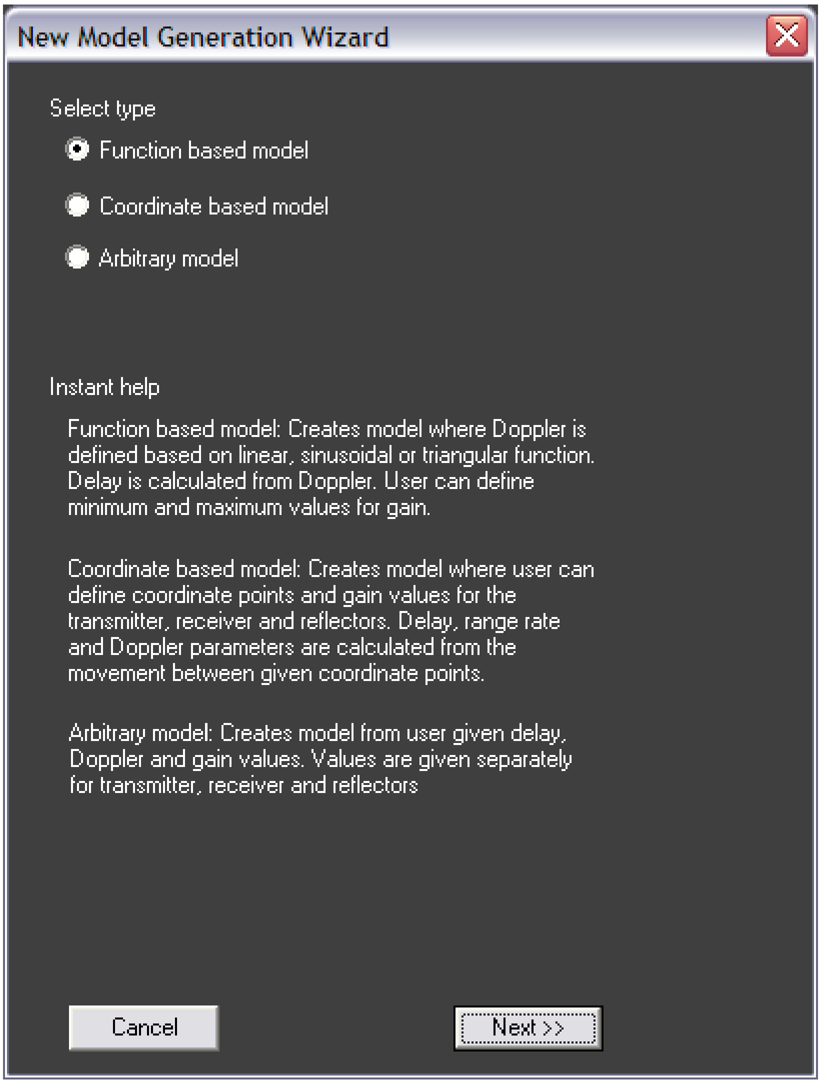


Figure 271. New Model Generation Wizard model type selection dialog

Emulation center frequency allows user to define different frequencies for the model and for the emulation. Doppler is always calculated by using emulation center frequency value. RF center frequency defines the actual frequency used in connections to and from the PROPSIM emulator.

This feature is useful if the user application frequency band is outside PROPSIM frequency range. In this case, the RF connection to PROPSIM can be done, for example, at the customer application IF frequency or by selecting testing frequency so that it matches with the emulator frequency range.

19.1.5.1 Function based model

Function based model creates a model where Doppler is defined based on linear, sinusoidal, or triangular function. Delay is calculated from Doppler. User can define minimum and maximum values for gain. When function-based model is selected, the following dialog opens (Figure 272).

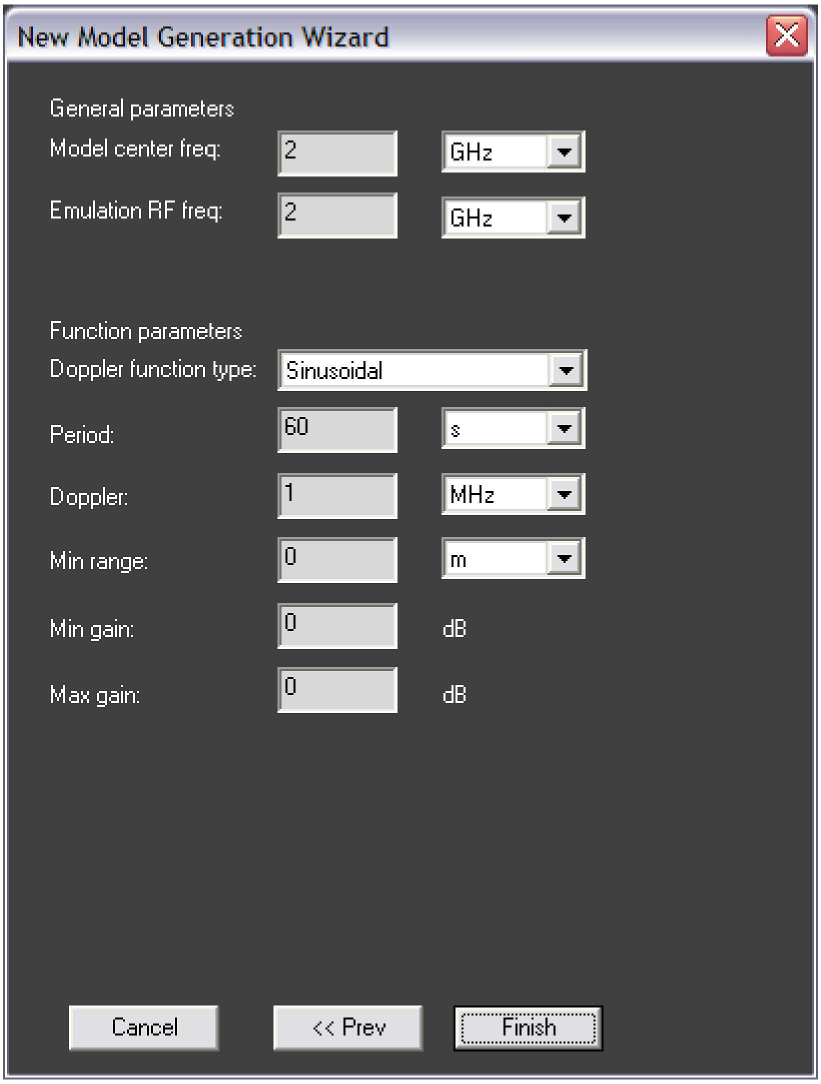


Figure 272. Function based model parameter dialog

19.1.5.2 Coordinate based model

Coordinate based model creates model where user can define coordinate points and gain values for the transmitter, receiver, and reflectors. Delay, range rate and Doppler parameters are calculated from the movement between given coordinate points. Coordinate based model parameter dialog is shown in Figure 273.

Values in table fields can be written by using scientific expressions.

Examples:

- 100km can be written 100e3 [m] and
- 15 microseconds 15e-6 [s].

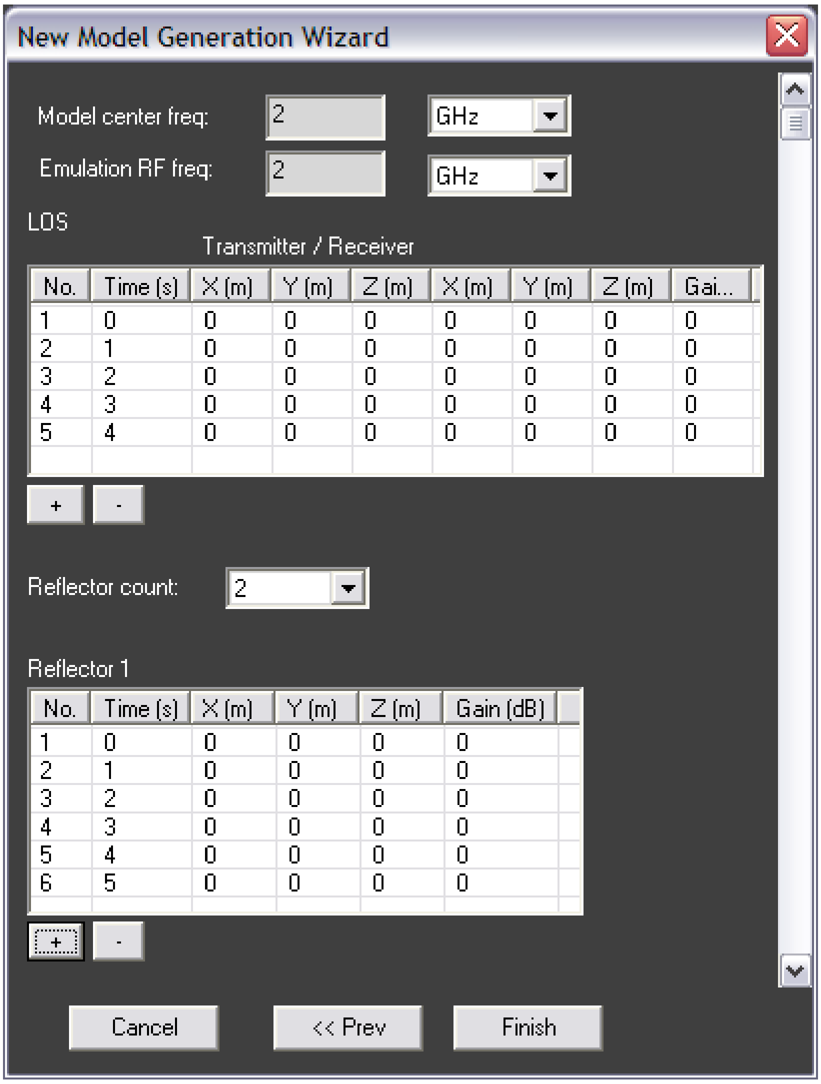


Figure 273. Coordinate based model parameter dialog

19.1.5.3 Arbitrary model

Arbitrary model creates model from user given delay, Doppler, and gain values. Values are given separately for transmitter, receiver, and reflectors. Arbitrary model parameter dialog is shown in Figure 274.

Values in table fields can be written by using scientific expressions.

Example: 15 microseconds can be written 15e-6 [s].

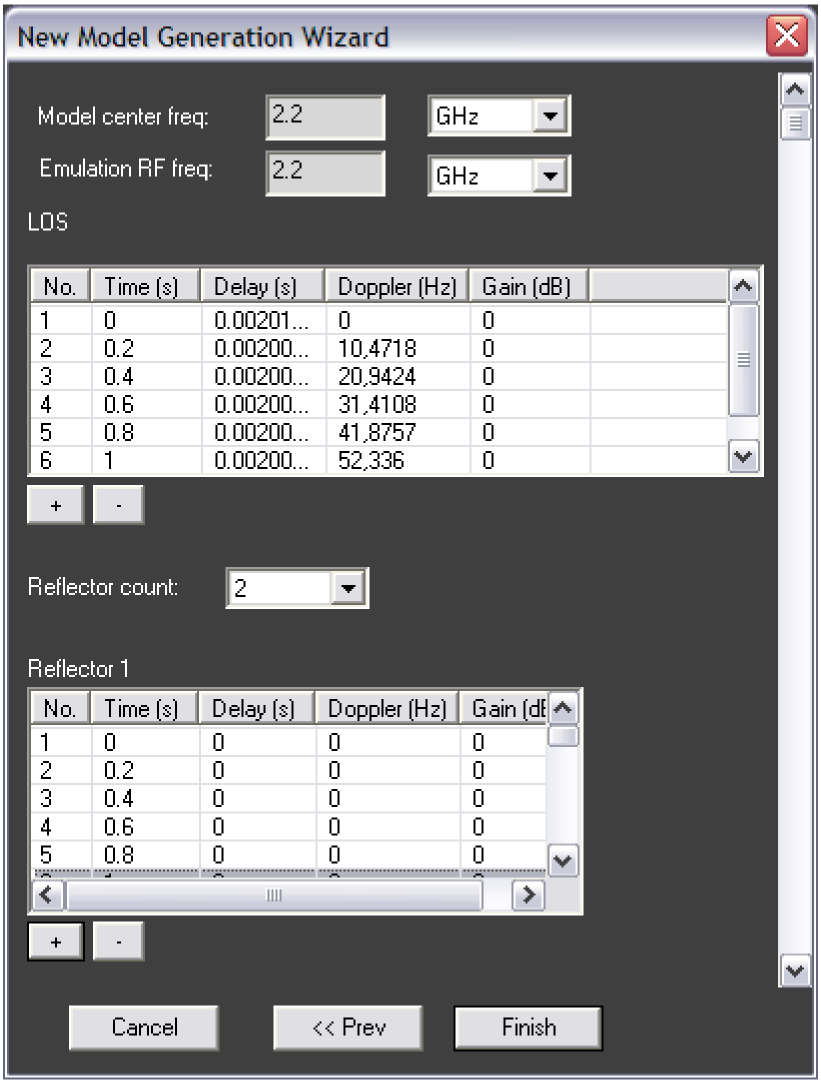


Figure 274. Arbitrary model parameter dialog

19.1.6 Editing existing Aerospace models

The ASO files generated by the Aerospace Model Editor can be further modified by simply editing the file content with any text editing tool. It is recommended that the models are always opened in Aerospace model editor before loading the model into emulation in Scenario Wizard. The Aerospace model editor validates that the model parameters are inside specified range and that the model file syntax is correctly written. It is also useful to see if the model looks as it was intended to be before compiling the emulation file.

19.1.7 Partial models

This functionality enables user to select a smaller part of the model. Partial modelling functionality is useful, when only a specific portion of the longer model needs to be used in the final emulation. Partial modelling functionality can also be us ed for creating several shorter models from different time instances from the longer model range.

Select **File** > **Partial model**. A new dialog box appears. See Figure 275.

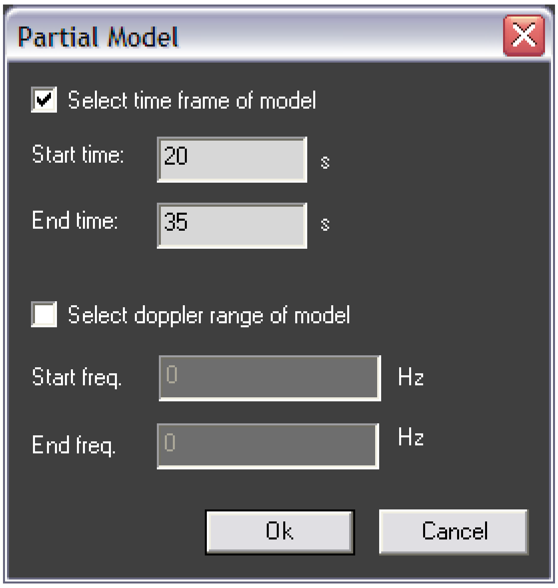


Figure 275. Selecting partial model range

To define the model range, enter the start and end times or Doppler frequency limits of the range. Alternatively, use the left mouse button to drag range markers on the Doppler or delay view. see Figure 276. To see range markers, check the **Select time frame of model** or **Select Doppler range of model** in dialog.

Note: The Doppler range selection is an alternative way of defining start and stop times for the model. The selection is always done in time axis. Doppler frequency selection is available only for function based ASO models.

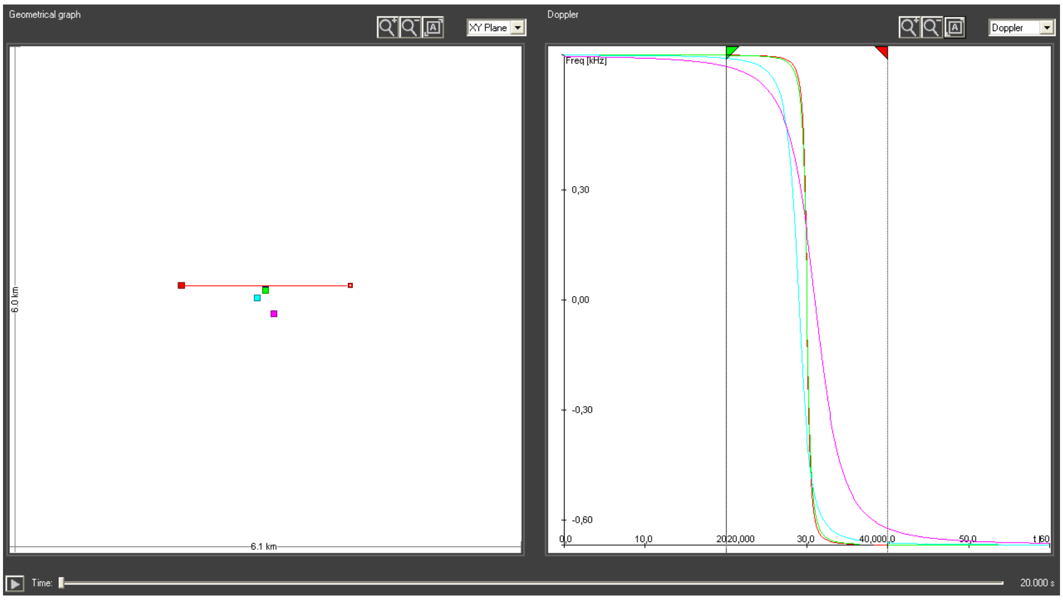


Figure 276 Partial modelling

19.2 Creating and Running Emulation

This chapter describes how the ASO models are used for creating Aerospace emulation and how to run it in real time. Generating and running Aerospace emulation with PROPSIM requires generally following steps:

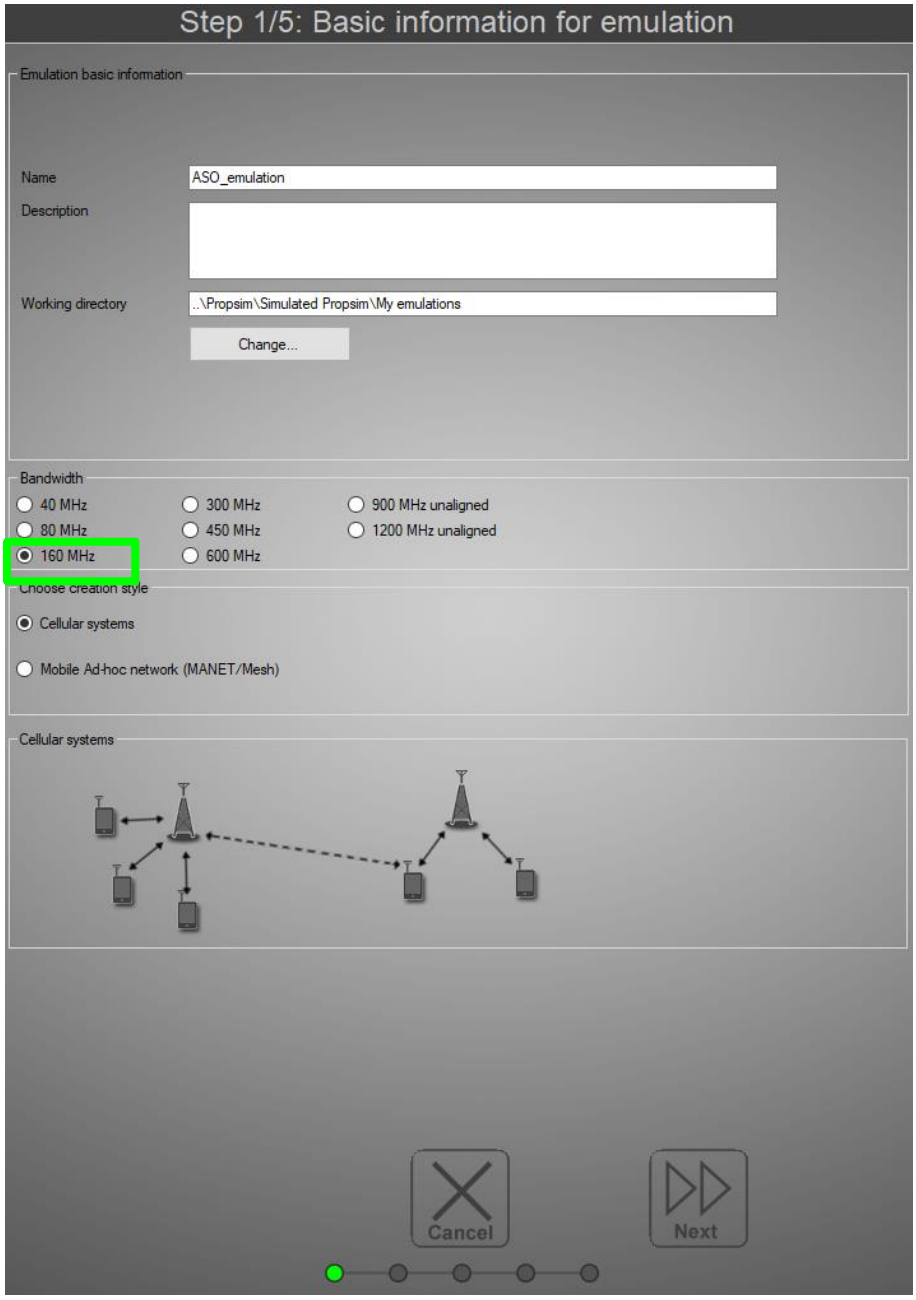
- Creating ASO channel model
- Creating and compiling emulation in Scenario Wizard
- Loading and controlling generated emulation in Emulation Control View

Note: The focus in this section is in ASO specific operation. Refer to chapters 3 and 4 for more information about Scenario Wizard and Emulation Control View.

19.2.1 Creating Aerospace emulation in Scenario Wizard

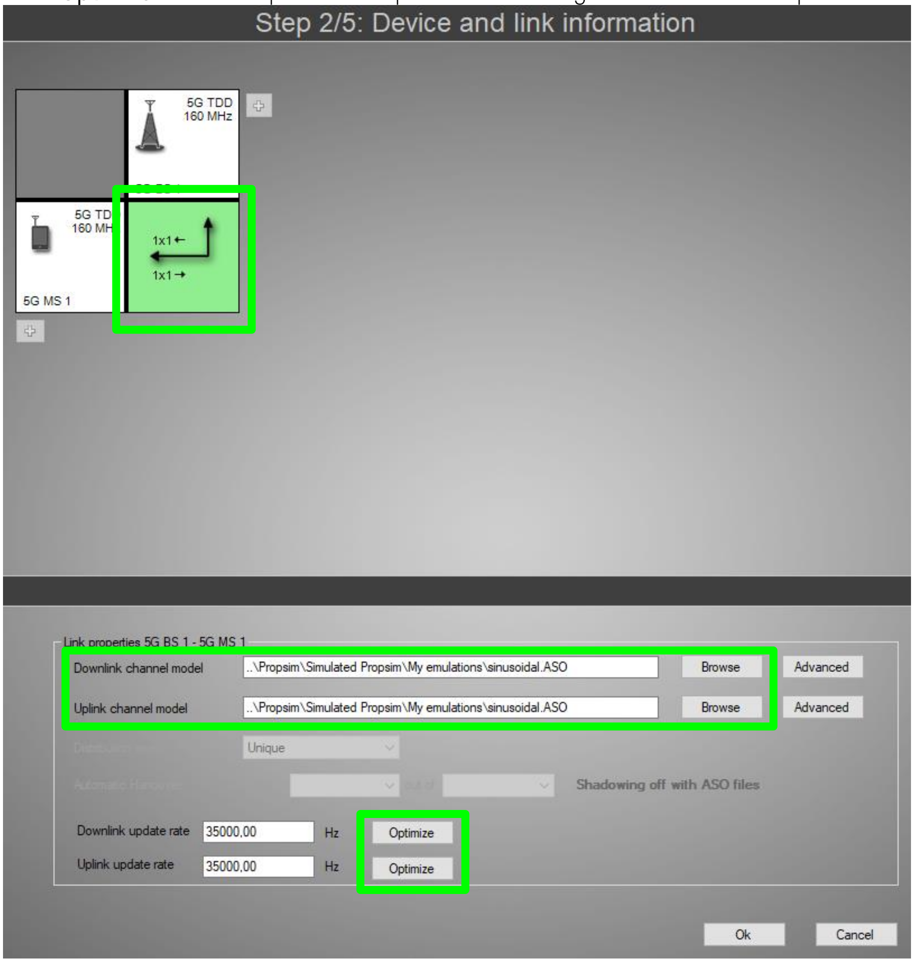
Scenario Wizard is used for creating emulations in PROPSIM. Workflow in creating an Aerospace emulation is similar to workflow when creating normal emulations.

1. Open the Scenario Wizard.
2. In Step 1/5:
   a. Fill in the basic information.
   b. Under **Bandwidth**, select **160 MHz**. (200 MHz in F8800B and F8820B)
   c. Click **Next**.



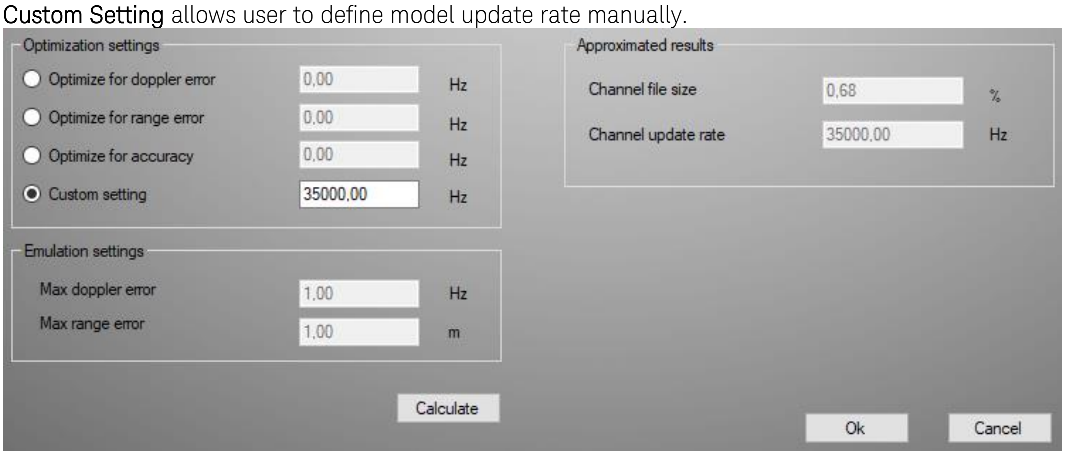
3. In Step 2/5:
   a. Add the base stations and mobile stations.
   b. Create the uplinks and downlinks.
   c. For each link, define which .ASO file to use as the channel model.

d. Click Optimize to define update rate optimization settings for downlink and uplink.



e. Select the optimization option. Depending on the ASO model type, you have 1...3 optional methods to optimize the channel update rate. In addition, you can adjust the update rate parameter manually.

- Optimize for Doppler error method uses user defined Max Doppler error value and calculates the minimum update rate that fulfils the criterion.
- Optimize for Range error method uses user defined Max Range error value and calculates the minimum update rate that fulfils the criterion.
- Optimize for Accuracy method calculates the highest model update rate that allows the emulation to fit into the emulator memory.
- Custom Setting allows user to define model update rate manually.

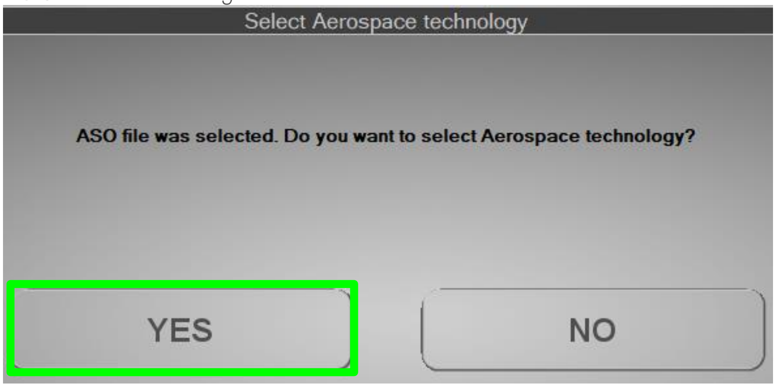


f. To see the resulting update rate and estimated size of the model, click Calculate. Calculating Doppler and range errors for long models might take some time to complete.

g. To save the update rate settings, click OK.

h. To save the channel model settings, click OK .

4. In the **Select Aerospace technology** confirmation dialog, click **YES** to change the technology to ASO.
Note: Other technologies cannot be used in the same emulation with ASO.



The technology of the devices is changed to "Aerospace", and the names of the devices are changed to "ASO device [running number]".

5. Define the rest of the emulation as a regular emulation. For instructions, see the Scenario Wizard instructions in chapter 3.

19.2.2 Running Aerospace emulation in Emulation Control View

Emulation Control View (see Figure 277) is used for controlling and running emulations in PROPSIM.

To open an emulation in the Emulation Control View, click the **Edit or Run emulation** button in the Home view, or select **Emulation > Open** in the navigation bar. In the **Open Emulation** dialog, select the emulation you want to run and click **Run**.

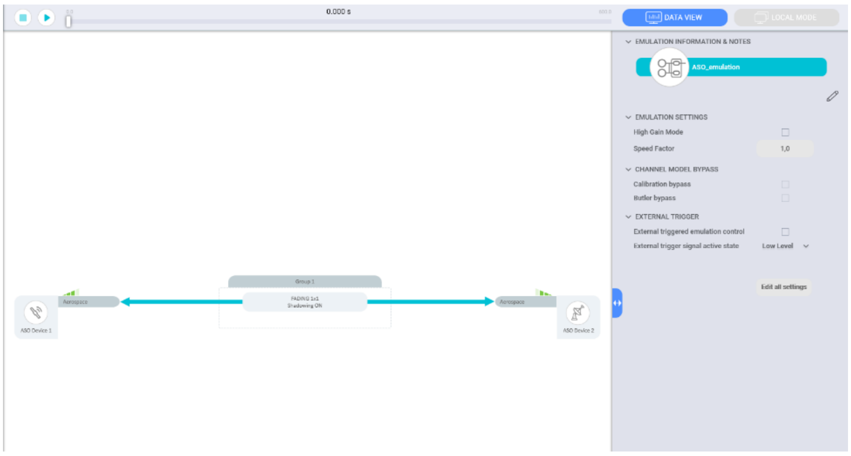


Figure 277 ASO emulation open in the Emulation Control View

You can control the emulation run using the tools in the emulation timeline:

- To start the emulation that is open, click the (Start) button.
- To pause the emulation, click the (**Pause**) button. The emulation pauses in the current time on the timeline.
- To stop the emulation and go back to start, click the (Stop) button. The emulation jumps back to the beginning of the timeline.

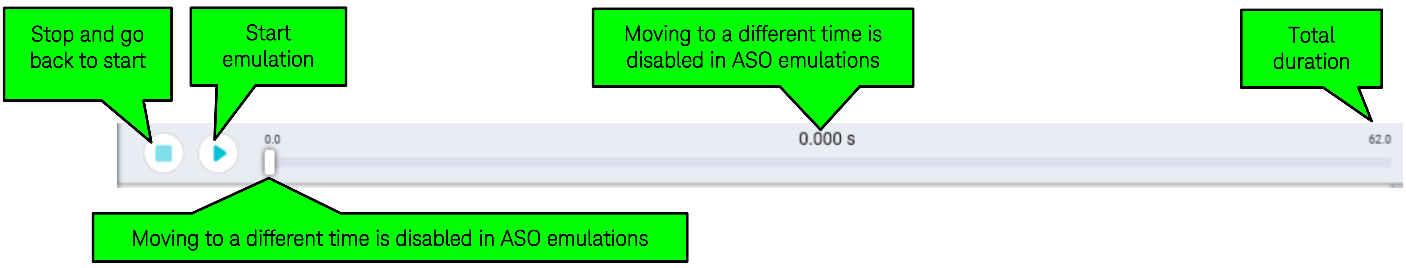


Figure 278 Emulation timeline (moving to a different time is disabled in ASO emulations)

20 STANDARD TOOLS REMOTE CONTROL INTERFACE AND FUNCTIONS

ATE (Automatic Test Equipment) services of PROPSIM allow you to control the emulator remotely via LAN interface. ATE commands are explained in this document.

When PROPSIM emulator boots up, ATE LAN Server is started. As ATE commands are received, they are displayed in the ATE LAN server console, which also shows possible responses and error messages.

ATE provides to a large extent the functionality of the Emulation Control view of the graphical user interface and can be seen as equivalent to it.

20.1 Local and remote modes

PROPSIM can be used in both local and remote modes, and it is possible to switch between the modes easily in normal use even when an emulation is open in the Emulation Control View. The local mode is used for operating PROPSIM on the PROPSIM GUI. The remote mode is used for operating PROPSIM remotely with ATE services.

When you open a remote connection to PROPSIM and issue an ATE command, PROPSIM goes automatically into remote mode. In remote mode, the background of PROPSIM GUI shows the text **"Remote mode"**, some menu items are disabled in the navigation bar, the **INT REF/EXT REF**, **LAB** and **ALIGN** buttons are disabled in the status bar, and the **Local Mode** button in the top right corner button is activated (turns blue). In remote mode, you cannot use all the features of the PROPSIM GUI locally.

To return to local mode, click the **Local Mode** button in the top right corner of the GUI.

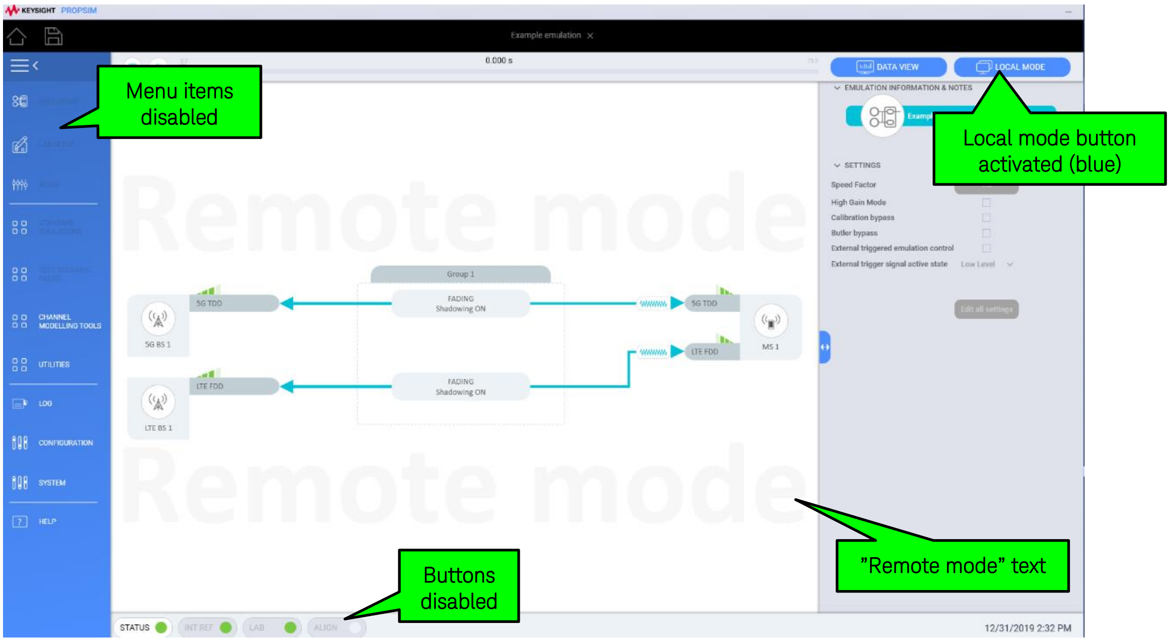


Figure 279 PROPSIM GUI in remote mode

All the issued ATE commands are shown in the ATE LAN monitor view. For instructions on opening the ATE LAN monitor view, see section 5.6.

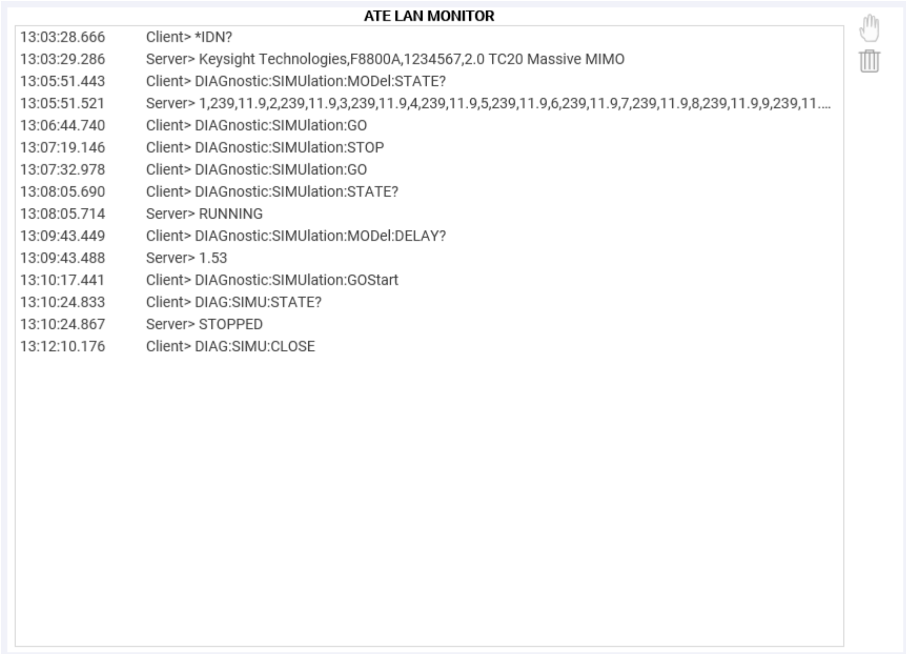


Figure 280 ATE LAN monitor view

20.2 LAN

The LAN interface can be accessed with the following LAN parameters:

Table 21. Access settings for ATE LAN connection

Item	User setting
LAN IP address	IP address of the emulator.
Port	3334
EOS	'\n' (line feed)

20.2.1 LAN Example

Below you can find a short code example that opens connection to the emulator by using ATE LAN interface, requests identification, and resets it. The example is written in MatLab (registered trademark of The MathWorks, Inc.). Please refer to the documentation of test environment you are using for details on how to open and use LAN connections.

```
% Open communication to PROPSIM

% Replace 192.168.0.1 with IP address of the PROPSIM
fid = tcpip(‘192.168.0.1’, 3334);
fopen (fid);

% Read instrument’s ID

fprintf (fid, “*IDN?\n’);
reply = fscanf(fid);
disp([ ‘Connected: ‘ reply(1:length(reply)-1) ] );

% Reset device

fprintf (fid, ‘*RST\n’);

% Close connection
```

```
fclose (fid);
```

20.2.2 LAN Example using Microsoft Visual Studio

The following example demonstrates how PROPSIM can be controlled by using Microsoft Visual Studio .NET 2003, using C++ language and .NET framework.

```
// This is simple example program that communicates with PROPSIM
// emulator using ATE LAN interface.
// Written in C++, using Microsoft Visual Studio 2003 .NET
// and .NET framework

#include "stdafx.h"
#using <mscorlib.dll>
using namespace System;
using namespace System::Text;
using namespace System::IO;
using namespace System::Net;
using namespace System::Net::Sockets;
using namespace System::Collections;

 // Connect to a remote socket by using network API.
Socket * ConnectSocket(String* server, int port)
{
  Socket* s = 0;
  IPHostEntry* hostEntry = 0;

   // Get host related information.
  hostEntry = Dns::Resolve(server);

  IEnumerator* myEnum = hostEntry->AddressList->GetEnumerator();
  while (myEnum->MoveNext())
    {
      IPAddress* address = __try_cast<IPAddress*>(myEnum->Current);
      IPEndPoint* endPoint = new IPEndPoint(address, port);
      Socket* tmpS = new Socket(endPoint->AddressFamily,
                          SocketType::Stream, ProtocolType::Tcp);
      tmpS->Connect(endPoint);
      if (tmpS->Connected)
        { s = tmpS;
          break;
        }
      else
        { continue;
        }
    }
  return s;
}

 // Send an ATE command to PROPSIM. Line termination is added to end of
string,
// and does not need to be included in parameter string.
void sendCommand(Socket *s, String *cmd)
{
  String* request = String::Concat(cmd,S"\n");
  Byte bytesSent[] = Encoding::ASCII->GetBytes(request);
s->Send(bytesSent, bytesSent->Length,
        static_cast<SocketFlags>(0));
}

 // Read last response from socket. Note that if there is no
 // response to be read, this will block until program is
 // terminated. Be sure to only call this after a request has
 // been sent to ATE server.
String* readResponse(Socket *s)
{
```

```
  Byte bytesReceived[] = new Byte[2];
  String* response;

   // Receive message one character at a time; ineffient, but does
   // not require additional buffering.
  do {
int bytes = s->Receive(bytesReceived, 1,
                       static_cast<SocketFlags>(0));
 response = String::Concat(response,
                   Encoding::ASCII->GetString(bytesReceived,
                   0, bytes) );
    } while (response->IndexOf('\n') == -1);

  return response;
}

 // Main program
int _tmain()
{
   // Open connection to PROPSIM. Change "localhost" to PROPSIM's
   // IP address to connect remotely.
  Socket* s = ConnectSocket("localhost", 3334);

   // Send IDN query.
  sendCommand(s, "*IDN?");

   // Read response to previous IDN query.
  String *response = readResponse(s);

   // Write response to console
  Console::WriteLine(response);
  return 0;
}
```

20.3 ATE command syntax

20.3.1 Common syntax

ATE commands have common syntax as follows:

<COMMAND> <PARAMETERS>

or

<COMMAND>

where

- <COMMAND> includes a string of characters with no spaces in between. If the last character is '?' then <COMMAND> is a query.
- <PARAMETERS> include a list of parameters separated with commas (no spaces). Note that there is a space between <COMMAND> and <PARAMETERS>.

General notes:

- Strings do not include surrounding hyphens ("…").
- There can be space characters in the middle of the string (e.g. when defining model name).
- Decimal numbers include at their midst one dot ('.').
- Integers should not include other characters than digits.
- Successive commands sent at the same time should be separated with semicolons (';').
- ATEL LAN commands should have '\n' as an end character.

20.4 ATE command interface

The PROPSIM ATE commands described are divided into groups by their basic functionality and part of emulation they control:

- Common commands, as specified by IEEE 488.2 specification (chapter 20.4.1).
- System commands for general system control (chapter 20.4.2)
- Emulation commands (chapter 20.4.3), for opening, closing and running the emulation.
- Input commands (chapter 20.4.4) for controlling the input parameters of signal, such as input level and crest factor.
- Output commands (chapter 20.4.5) for controlling the output parameters of signal, such as output gain.
- Channel commands (chapters 20.4.6) for controlling the fading channel parameters, such as mobile speed. These also include commands that provide more information of channels parameters.
- Group information commands (chapter 20.4.7) for getting emulation topology information.
- Shadowing commands (chapter 20.4.8) for getting information and controlling emulation shadowing profiles.
- Internal Interference Generator option commands (chapter 20.4.9).
- Channel model information (chapter 20.4.10) and Signal routing commands (chapter 20.4.11) provide information of
    - channel model
    - control of reference signal
    - usage of front panel signal connectors of the emulator
- Commands for controlling external trigger (chapter 20.4.12).
- Multiple emulation loading commands (chapter 20.4.14).

The ATE commands are presented in form of (for example) “SYSTem:ERRor?”. Commands are not case sensitive; capital letters denote the short version of command parts PROPSIM accepts. That is, for above example commands, all forms “system:error?”, “syst:err?”, “system:err?” and “syst:error?” are acceptable alternatives.

Examples presented here contain comments for user (lines beginning with double slashes, “//”), the command and optional reply from the emulator.

Most PROPSIM ATE commands are only available when emulation has been opened. Thus, the normal usage of command follows the basic sequence below:

1. Open emulation (*calculate:filter:file* –command, chapter 20.4.3.1)
2. Adjust emulation parameters
3. Run emulation, do measurements
4. Close emulation

20.4.1 Common commands

20.4.1.1 *CLS

Clear status

Syntax:

*CLS

Description:

This command clears/reset to default partly the status data structures. The following registers are cleared or reset:

- Error/Event Queue
- Operation Status Event Register
- Questionable Status Event Register
- Service Request Status Event Register

- Standard Event Status Register
- Status Byte Event Register
- Status Byte Condition Register: MAV bit
- Status Byte Condition Register: Error/Event Queue bit
- Device goes to operation command idle state.
- Device goes to operation complete query idle state.

Example:

*cls

20.4.1.2 *ESE

Set Standard Event Status Enable Register

Syntax:

*ESE <register value>

Description:

This query allows the user to determine the contents of the Standard Event Status Enable Register.

Example:

// Set Standard Event Status Enable Register so that

// query error is not shown in status or put

// to error/event queue.

*ese 253

20.4.1.3 *ESE?

Get Standard Event Status Enable Register

Syntax:

*ESE?

Description:

Query returns the contents of the Standard Event Status Enable Register and clears it.

Example:

*ese?

0

20.4.1.4 *ESR?

Get Standard Event Status Register

Syntax:

*ESR?

Description:

Query returns the contents of the Standard Event Status Register and clears it.

Example:

*esr?

0

20.4.1.5 *IDN?

Get identification

Syntax:

*IDN?

Description:

Query returns the identification of the ATE device.

Example:

*idn?

Company Name,Device Name,Serial Number,Firmware Version Number

20.4.1.6 *OPC

Operation complete

Syntax:

*OPC

Description:

This command causes the device to set the Operation Complete bit in the Standard Event Status Event Register to 1 after there are no more pending operations.

Example:

*opc

20.4.1.7 *OPC?

Get operation complete

Syntax:

*OPC?

Description:

This query causes the device to return ASCII “1” after there are no more pending operations.

Example:

*opc?

1

20.4.1.8 *RST

Reset command

Syntax:

*RST

Description:

This command performs device reset:

- Disconnects local user, if connected (from Release 1.2.1 onwards)
- Closes the emulation.
- Device goes to operation command idle state.
- Device goes to operation complete query idle state.

Example:

*rst

20.4.1.9 *SRE

Set Service Request Enable Register

Syntax:

*SRE <register value>

Description:

This command sets the contents of the Service Request Enable Register.

Example:

// Set Service Request Enable Register so that neither standard

// events or errors cause service request for the ATE client

*sre 219

20.4.1.10 *SRE?

Get Service Request Enable Register

Syntax:

*SRE?

Description:

This query returns the contents of the Service Request Enable Register.

Example:

*sre?

191

20.4.1.11 *STB?

Get Status Byte Condition Register

Syntax:

*STB?

Description:

This query returns the contents of the Status Byte Condition Register.

Example:

*stb?

0

20.4.1.12 *TST?

Get self-test results

Syntax:

*TST?

Description:

This query returns the self-test results of the device.

Possible results:

0, Self test completed with no errors

1, Self test not completed

2, Self test completed with errors

Example:

*tst?

0

20.4.1.13 *WAI

Wait to continue

Syntax:

*WAI

Description:

This command prevents the device from executing any further commands or queries until the no-operation-pending flag is TRUE.

Example:

*wai

20.4.2 System commands

20.4.2.1 SYSTem:ERRor?

Get first error from queue

Syntax:

SYSTem:ERRor?

Description:

This query returns the oldest error/event message from the error/event queue. Errors and events are described in chapter 20.5.

Example:

syst:err?

0,"No error"

20.4.2.2 SYSTem:VERSion?

Get SCPI version number

Syntax:

SYSTem:VERSion?

Description:

With this command the SCPI version number can be queried. Used version number in the PROPSIM should be 1999.0.

Example:

syst:vers?

1999.0

20.4.2.3 SYSTem:RESet

System reset

Syntax:

SYSTem:RESet

Description:

This command performs system reset:

Closes the emulation.

Example:

syst:res

20.4.2.4 SYSTem:INFO?

System info request

Syntax:

SYSTem:INFO?

Description:

Query returns the basic system info and licenses of the PROPSIM.

<Device Name>,<Number of channels>,<Interface>,<Device HW version>,<Number of Internal RFLOs>,<Band#1>,...,<Band#N>,<License#1>,...,<License#N>

Example:

syst:info?

PROPSIM F64,64,RF,v1.0,16,Band: 450MHz - 3000MHz,Main license,Bandwidth:160.000MHz,Usable connectors:All,Shadowing,Channel specific shadowing, Fading channels per unit:512

20.4.2.5 SYSTem:STATus?

System status request

Syntax:

SYSTem:STATus?

Description:

This query returns some of the warning/caution statuses of the device.

Supported warning/caution sources:

Input cut-off, Digital Clipping, Reference status, Unstable level settings

Possible results:

1 // None of the supported warning sources active

0, <active warning/caution source#1>,...,<active warning/caution source#N>

Example:

syst:stat?

1 // Status ok

syst:stat?

0,Warning: External Reference missing

20.4.2.6 SYSTem:TRANSmitter:OFF

Turn off all propsim transmitters

Syntax:

SYSTem:TRANSmitter:OFF

Description:

Turns off all PROPSIM RF transmitting sources including PROPSIM controlled external RF sources.

Example:

syst:trans:off

20.4.2.7 SYSTem:MAXOUTGain:SET

Set the current maximum output gain limit

Syntax:

SYSTem:MAXOUTGain:SET <maximum output gain limit value>

Description:

This command sets maximum output gain limit.

Example:

//set maximum output gain limit to 10 dB

syst:maxoutg:set 10

20.4.2.8 SYSTem:MAXOUTGain:GET

Get the current maximum output gain limit

Syntax:

SYSTem:MAXOUTGain:GET?

Description:

This command retrieves maximum output gain limit.

Example:

//retrieve maximum output gain limit

syst:maxoutg:get?

10

20.4.2.9 SYSTem:MAXOUTGain:LIMits?

Get the limits of the maximum output gain limit

Syntax:

SYSTem:MAXOUTGain:LIMits?

Description:

This command retrieves limits of the maximum output gain limits.

Example:

//retrieve limits of the maximum output gain limit

syst:maxoutg:lim?

0,16

20.4.2.10 SYSTem:CALIBration:SET

Set used calibration

Syntax:

SYSTem:CALIBration:SET <calibration name>

Description:

This command takes calibration defined in <calibration name> into use.

To remove calibration from use, set 'No calibration' as calibration name.

Example:

// take LTETestSetup calibration in to use

syst:calib:set LTETestSetup

// remove calibration from use

syst:calib:set No calibration

20.4.2.11 SYSTem:CALIBration:GET?

Get used calibration

Syntax:

SYSTem:CALIBration:GET?

Description:

This command retrieves the current calibration in use

Example:

syst:calib:get?

LTETestSetup

20.4.2.12 SYSTem:CALIBration:LIST?

List valid calibrations

Syntax:

SYSTem:CALIBration:LIST?

Description:

This command retrieves a list of valid calibrations.

Example:

//retrieve list of valid calibrations

syst:calib:list?

LTETestSetup,WCDMATestSetup,ACTestSetup

20.4.2.13 SYSTem:CALIBration:VALid?

Retrieves status of the calibration

Syntax:

SYSTem:CALIBration:VALid?

Description:

Status of the current calibration. Format of the reply is as follows:

[calibration in use], [calibration valid for all connectors and frequency]

0, No

1, Yes

Commands inp:calib:valid (20.4.4.43) and outp:calib:valid (20.4.5.23) can be used to query the calibration validity of individual connectors.

Example:

//retrieve status of the calibration

syst:calib:valid?

1,1

20.4.2.14 SYSTem:CALIBration:CONFig:LIST?

List valid calibration configurations

Syntax:

SYSTem:CALIBration:CONFig:LIST?

Description:

This command lists all valid calibration configurations.

Example:

//retrieve list of calibration configs

syst:calib:conf:list?

LTETestConfig,WCDMATestConfig,ACTestConfig

20.4.2.15 SYSTem:CALIBration:START

Start calibration

Syntax:

SYSTem:CALIBration:START <Name of calibration config>,<Name of calibration>

Description:

This command starts calibration with given configuration. Command is supported only with Auto Calibration Unit configurations. After commands has been executed and reply to *OPC? has been received, user should verify the status of operation with SYST:ERR? command. If error has occurred, sending SYST:CALIB:NEXT retries the last calibration step. SYST:CALIB:CANCEL can be used to cancel the whole calibration.

For more details, refer to PROPSIM Application Notes: Auto Calibration TDD-LTE Test Setup and Auto Calibration WLAN Test Setup.

Example:

// starts calibration using config LTETestConfig. Name of the calibration is LTETestSetup

syst:calib:start LTETestConfig,LTETestSetup

20.4.2.16 SYSTem:CALIBration:NEXT

Continue paused calibration

Syntax:

SYSTem:CALIBration:NEXT

Description:

This command continues calibration. Command can be issued to retry calibration in case of an error.

Example:

//continue calibration

syst:calib:next

20.4.2.17 SYSTem:CALIBration:CANCEL

Cancel ongoing calibration

Syntax:

SYSTem:CALIBration:CANCEL

Description:

This command cancels ongoing calibration. This command is parallel and can be issued even if there are other ATE-commands (i.e. SYST:CALIB:START) pending.

Example:

//cancel calibration

syst:calib:cancel

20.4.2.18 SYSTem:CALIBration:USER:SET

Enable or disable the user alignment

Syntax:

SYSTem:CALIBration:USER:SET <enable 1/0>,<name>

Description:

This command can be used to enable or disable the user alignment.

Enable (1/0):

- Whether a user alignment is to be enabled (1) or the current alignment to be disabled (0)

Name:

- Name of the alignment to be enabled (existing or new)

Example:

// Create new user alignment with name "user1"

SYSTem:CALIBration:USER:SET 1,user1

// Unselect user alignment

SYSTem:CALIBration:USER:SET 0

20.4.2.19 SYSTem:CALIBration:USER:GET?

Query the status of user alignment

Syntax:

SYSTem:CALIBration:USER:GET?

Description:

This command can be used to query the status of the user alignment.

Returns:

- Returns the current alignment name, empty string if no user alignment is enabled.

20.4.2.20 SYSTem:CALIBration:USER:INFO

Set additional info to existing/enabled user alignment

Syntax:

SYSTem:CALIBration:USER:INFO <info>

Description:

This command can be used to set the additional info to a user alignment (for example details of the FW/SW, timestamp of the used calibration).

Info:

- A string containing the additional info

Example:

SYSTem:CALIBration:FREQuency:INFO FI1234567, 29.01.2024, External Response Calibration Tool 3.0, PROPSIM FW 10.0

20.4.2.21 SYSTem:CALIBration:USER:INFO?

Request additional info of existing/enabled user alignment.

Syntax:

SYSTem:CALIBration:USER:INFO?

Description:

This command can be used to fetch the additional info of a user alignment (for example details of the FW/SW, timestamp of the used alignment).

Returns:

- A string containing the additional info

Example:

SYSTem:CALIBration:USER:INFO?

FI1234567, 29.01.2024, External Response Calibration Tool 3.0, PROPSIM FW 10.0

20.4.2.22 SYSTem:LABsetup:SET

Set used lab setup

Syntax:

SYSTem:LABsetup:SET <lab setup name>

Description:

This command takes lab setup defined in <lab setup name> into use.

To remove lab setup from use, set 'No lab setup' as <lab setup name>.

Example:

// take LTETestSetup lab setup in to use

syst:lab:set LTETestSetup

// remove lab setup from use

syst:lab:set No lab setup

20.4.2.23 SYSTem:LABsetup:GET?

Get used lab setup

Syntax:

SYSTem:LABsetup:GET?

Description:

This command retrieves the current lab setup in use.

Example:

syst:lab:get?

LTETestSetup

20.4.2.24 SYSTem:LABsetup:LIST?

List of lab setups

Syntax:

SYSTem:LABsetup:LIST?

Description:

This command retrieves a list of lab setups.

Example:

//retrieve list of lab setups

syst:lab:list?

LTETestSetup,WCDMATestSetup,ACTestSetup

20.4.2.25 SYSTem:MEASurements:TARget:SET

Enables/disables emulation data sending and sets the UDP target port and address. See chapter 8 for more details.

Syntax:

SYSTem:MEASurements:TARget:SET <enabled info>,<target port>,<target IP address>

Description:

enabled info:

0 – Emulation data sending disabled

1 – Emulation data sending enabled

target port:

Target port number (optional)

target IP address:

Target IP address (optional)

If port number is given, also IP address is required.

Example:

// Enables data sending, sets input port to 3800 and IP address to 192.168.1.30

syst:meas:tar:set 1,3800,192.168.1.30

// Disables data sending, no target port or IP address updates

syst:meas:tar:set 0

20.4.2.26 SYSTem:MEASurements:TARget:GET?

Gets information about emulation data sending state and target. See chapter 8 for more details.

Syntax:

SYSTem:MEASurements:TARget:GET?

Description:

This command retrieves the state of emulation data sending (0 disabled, 1 enabled) and the target port number and IP address.

Example:

syst:meas:tar:get?

// Emulation data sending are enabled (1), used port is 3800 and IP address is 192.168.1.30

1,3800,192.168.1.30

20.4.2.27 SYSTem:MEASurements:ELEment:SET

Enables or disables sending of individual element type and its report interval for emulation data sending. See chapter 8 for more details.

Syntax:

SYSTem:MEASurements:ELEment:SET <element type>,<element report enabled>,<report interval>

Description:

element type:

1 – Emulation event

3 – Emulation time

101 – Input power

201 – Output power (calculated from input power)

401 – Link Doppler

402 – Link output RSRP

403 - Link AoA angle

404 - Link AoD angle

element report enabled:

0 – disabled

1 – enabled

report interval:

Report interval value is defined in milliseconds (optional). This parameter is ignored if element type is 1 (emulation event). Emulation events are always sent immediately.

Example:

//Element 101 (input power measurement) data reporting is set to enabled (1) and report interval is set to 100 ms

syst:meas:ele:set 101,1,100

//Element 201 (output power measurement) data reporting is set to disabled (0), no report interval changes

syst:meas:ele:set 201,0

20.4.2.28 SYSTem:MEASurements:ELEment:GET?

Get the state of individual data element and its report interval for emulation data sending. See chapter 8 for more details.

Syntax:

SYSTem:MEASurements:ELEment:GET? <element type>

Description:

This command retrieves the enabled state of the element (disabled 0, enabled 1) and its report interval in milliseconds.

element type:

1 – Emulation event

3 – Emulation time

101 – Input power

201 – Output power (calculated from input power)

401 – Link Doppler

402 – Link output RSRP

403 - Link AoA angle

404 - Link AoD angle

Example:

syst:meas:ele:get? 101

// Element 101 (input power) data reporting is enabled (1) and report interval is 0.1 s (100 ms)

1,100

20.4.2.29 SYSTem:MEASurements:PROFiledata:SEND

Sends interference, shadowing or speed profile data to defined UDP measurement data target. See chapter 8 for more details.

Syntax:

SYSTem:MEASurements:PROFiledata:SEND <profile type>,<element number>,<time step>

Description:

Profile type:

1 – Link speed profile

2 – Link shadowing profile

3 – Link interference profile

4 – Link AoA angle profile

5 – Link AoD angle profile

11 – Channel speed profile

12 – Channel shadowing profile

13 – Interference profile

14 – Channel AoA angle profile

15 – Channel AoD angle profile

Element number:

- Link number in case of profile types 1, 2, 3, 4 and 5.
- Channel number in case of profile types 11, 12, 14 and 15
- Interference identification string in case of 13

Profile data format is otherwise the same as other UDP data, but value field (V=) contains "subfields": timeunit, timestep, length, dataunit and data. Data format is:

T=[time]#D=[datatype]#E=[element id]N=[element name]#V=timeunit:<unit, always s >;timestep:<timestep in seconds without unit>;length:<profile length in seconds without unit>;dataunit:<dB (shadowing or interference profile) or Hz (speed profile)>data:<comma separated list of data>\r\n

Datatype D in profile data is:

801=Link speed profile data

802=Link shadowing profile data

803=Link interference profile data

804=Link AoA angle profile data

805=Link AoD angle profile data

811=Channel Speed profile data

812=Channel Shadowing profile data

813=Interference profile data

814=Channel AoA angle profile data

815=Channel AoD angle profile data

Shadowing profile data is reported as attenuation values (-1.0 * gain) in dB, including shadowing offset setting.

Interference profile data is reported as attenuation values (-1.0 * gain) in dB.

Example:

Link number 1 shadowing, curve length 3 s requested with 0.5 s timestep.

syst:meas:prof:send 2,1,0.5

T=2017-04-01T08:45:02.012#D=802#E=2#N=LTE 1 Uplink#V=timeunit:s;timestep:0.5;
length:3;dataunit:dB;data:0.0,3.5,7.0,10.5,7.0,3.5,0.0\r\n

If the requested channel, link or interference does not contain profile data "empty curve" is sent:

T=2016-04-12T08:45:02.012#D=802#E=2#N=LTE 1 Uplink#V=length:0.0\r\n

Note:

Changing certain emulation parameters as shadowing offset, center frequency, speed multiplier, etc. can change the values of shadowing, speed, or interference profile data. In these cases, profile data must be requested again with final emulation parameter settings.

If the requested time step is smaller than the time step used in the original profile, the reported data uses the timestep from original profile.

20.4.2.30 SYSTem:FREQuency:CONVersion:ENable

Enable/disable frequency conversion

Syntax:

SYSTem:FREQuency:CONVersion:ENABLE <0/1 Disable/Enable>

Description:

Enables or disables the frequency conversion feature in PROPSIM. Possible set values are:

- 0, Frequency conversion disabled
- 1, Frequency conversion enabled

Example:

Syst:freq:conv:en 1

20.4.2.31 SYSTem:FREQuency:CONVersion:ENable?

Requests the state of frequency conversion (enable/disable)

Syntax:

SYSTem:FREQuency:CONVersion:ENABLE <0/1 Disable/Enable>

Description:

Queries the state of frequency conversion feature. Possible return values are:

- 0, Frequency conversion disabled
- 1, Frequency conversion enabled

Example:

Syst:freq:conv:en?

1

20.4.2.32 SYSTem:EXTernal:UNIT:LIST?

Retrieves list of external units.

Syntax:

SYSTem:EXTernal:UNIT:LIST? <0/1 Scan devices>

Description:

Retrieves list of external units and could run external unit scan.

External unit serial (control connector)

Example:

syst:ext:unit:list?

ACU 12345 (C5),ACU 67890 (C6)

20.4.2.33 SYSTem:EXTernal:UNIT:STATUS?

Retrieves status of external unit.

Syntax:

SYSTem:EXTernal:UNIT:STATUS? <unit id>

Description:

This command retrieves status of external unit. Status is returned as follows:

<FW version>,<calibration valid (0/1)>,<calibration expiration date (dd.mm.yyyy)>

Example:

syst:ext:unit:status? 12345

1.4.1.2,1,25.12.2015

20.4.2.34 SYSTem:EXTernal:UNIT:SCAN?

Launches external unit scan.

Syntax:

SYSTem:EXTernal:UNIT:SCAN?

Description:

This command scans external units. Scan status is returned as follows:

<Result (0/1/2)>

0, Failed

1, Succeeded devices found

2, No external Hw license

Example:

syst:ext:unit:scan?

1

20.4.2.35 SYSTem:EXTernal:UNIT:ENable

Enable/disable external unit usage. This setting is applied on the next boot.

Syntax:

SYSTem:EXTernal:UNIT: ENable <0/1 Disable/Enable>

Description:

This command enables/disables external unit usage. Possible set values are:

- 0, External units disabled
- 1, External units enabled

Example:

syst:ext:unit:en 1

20.4.2.36 SYSTem:EXTernal:UNIT:ENable?

Requests the state of external unit usage.

Syntax:

SYSTem:EXTernal:UNIT:ENable?

Description:

Queries the state of external unit usage. Possible return values are (x,y):

- X=0, External units disabled
- X=1, External units enabled
- Y=0, Usage mode unchanged

- Y=1, Usage mode changed, boot pending

Example:

syst:ext:unit:en?

1,0

20.4.2.37 SYSTem:POSitioner:ANGLE:SET

Sets positioner angle.

Syntax:

SYSTem:POSitioner:ANGLE:SET <Positioner Role 0/1 (BS/MS)>,<Positioner index>,<Angle 0/1 (azimuth/elevation)>, <angle>

Example:

syst:pos:angle:set 0,0,0,23.5

20.4.2.38 SYSTem:POSitioner:ANGLE:GET?

Get positioner angle.

Syntax:

SYSTem:POSitioner:ANGLE:GET? <Positioner Role 0/1 (BS/MS)>,<Positioner index>,<Angle 0/1 (azimuth/elevation)>

Example:

syst:pos:angle:get? 0,0,0

23.5

20.4.2.39 SYSTem:POSitioner:ANGLE:LIMits?

Get positioner angle limits.

Syntax:

SYSTem:POSitioner:ANGLE:LIMits? <Positioner Role 0/1 (BS/MS)>,<Positioner index>,<Angle 0/1 (azimuth/elevation)>

Example:

syst:pos:angle:lim? 0,0,0

-60.0,60.0

20.4.2.40 SYSTem:CHAMBer:LOCK

Lock/unlock all chambers

Syntax:

SYSTem:CHAMBer:LOCK <0/1 Unlock/Lock>

Description:

Locks or unlocks chambers connected to PROPSIM.

Example:

Syst:chamb:lock 1

20.4.2.41 SYSTem:CHAMBer:STATus?

Requests chamber status

Syntax:

SYSTem:CHAMBer:STATus? <Chamber Role 0/1 (BS/MS)>,<Chamber index>,<Status index>

Description:

Status indexes:

- 0, Door 0/1 (open/locked)
- 1, Temperature in Celsius

Response: <status response> or status not available error

Example:

Syst:chamb:stat? 0,0,0

1

20.4.3 Emulation Control

20.4.3.1 CALCulate:FILTer:FILE

Open emulation

Syntax:

CALCulate:FILTer:FILE <filename>

Description:

This command opens emulation defined in file <filename>. Some communication interface tools require the folder name separators to be duplicated. For example: e:\\test\\my emulation.smu.

Example:

// Open emulation e:\1to1 class.smu.

calc:filt:file e:\1to1 class.smu

20.4.3.2 CALCulate:FILTer:EDIT

Open emulation for editing

Syntax:

CALCulate:FILTer:EDIT <filename>

Description:

This command opens emulation defined in file <filename> to editing mode, allowing user to change emulation settings before loading emulation to hardware. Loading to hardware is done with command CALCulate:FILTer:CONNECT. Typical use case is to load emulation to editing mode, change the center frequencies to match the test setup and then load the emulation to hardware. This allows links to DUT through PROPSIM to remain established, while loading the emulation saved with arbitrary center frequencies.

Example:

// Open emulation e:\1to1 class.smu to editing mode

calc:filt:edit e:\1to1 class.smu

// Change center frequency to 1000 MHz

calc:filt:cent:ch 1,1000

// Load the emulation to hardware and wait operation to complete

calc:filt:connect

*opc?

1

20.4.3.3 CALCulate:FILTer:CONNECT

Loads to hardware the emulation currently in editing mode

Syntax:

CALCulate:FILTer:CONNECT

Description:

This command loads to hardware the emulation opened with CALCulate:FILTer:EDIT.

Example:

// Open emulation e:\1to1 class.smu to editing mode

calc:filt:edit e:\1to1 class.smu

// Load the emulation to hardware

calc:filt:connect

20.4.3.4 DIAGnostic:SIMUlation:MODel:CONTinuous?

Check if emulation is continuous

Syntax:

DIAGnostic:SIMUlation:MODel:CONTinuous?

Description:

This request checks whether emulation is continuous, i.e. all the channel models in emulation are continuous. It should be noted that even though emulation is not continuous, currently it is run as if it would be. In other words: Emulation is not automatically stopped at the end of the emulation, but rather continued from the start after that. Possible return values are:

0 Emulation is not continuous

1 Emulation is continuous

Example:

// Check if emulation is continuous.

diag:simu:mod:cont?

1

20.4.3.5 DIAGnostic:SIMUlation:MODel:STATE?

Get emulation state information

Syntax:

DIAGnostic:SIMUlation:MODel:STATE?

Description:

This request returns information on currently run emulation. The format of the answer is as follows:

< i >,<cir number of channel i+1>,<current emulation time>,

< i+1 >,<cir number of channel i+1>,<current emulation time>, ...

..., < n >,<cir number of channel n>,<current emulation time>

Where:

- i is number of channel
- N is last channel

Example:

// Get emulation information. Actual response from PROPSIM

// does not contain line breaks; added here for clarity.

diag:simu:model:state?

1,345,2.3,

2,345,2.3,

3,345,2.3,

4,345,2.3,

5,99,2.3,

6,99,2.3,

7,99,2.3

20.4.3.6 DIAGnostic:SIMUlation:MODel:INFO?

Get emulation model information

Syntax:

DIAGnostic:SIMUlation:MODel:INFO?

Description:

This request returns information on currently run emulation. The format of the answer is as follows:

<number of inputs>,<number of channels>,<number of outputs>

Example:

// Get emulation information (2 inputs, 4 channels, 2 outputs)

diag:simu:model:info?

2,4,2

20.4.3.7 DIAGnostic:SIMUlation:MODEL:DELAY?

Get insertion delay of model

Syntax:

DIAGnostic:SIMUlation:MODel:DELAY?

Description:

This request returns insertion delay of model in microseconds.

Example:

// Get emulation insertion delay

diag:simu:model:delay?

3.5

20.4.3.8 DIAGnostic:SIMUlation:GO

Run emulation

Syntax:

DIAGnostic:SIMUlation:GO

Description:

This command runs emulation.

Example:

// Run emulation.

diag:simu:go

20.4.3.9 DIAGnostic:SIMUlation:GOTO

Goto a CIR on the specific channel

Syntax:

DIAGnostic:SIMUlation:GOTO <channel number>,<cir>

or

DIAGnostic:SIMUlation:GOTO <time> <unit description>

Description:

This command accepts two different sets of parameters. First version moves the emulation running point to a given CIR on the specific channel. Possible other channels are also run during the time. In other words, the synchronization between channels is kept. Second version moves the emulation running point to a specified time. Only accepted unit description in second s.

Notes:

- *The emulation must be stopped or paused to carry out this operation.*
- *Goto operation to very far in the emulation (i.e. millions of CIRs) may take long time.*

Example:

// Goto cir 99 on channel 1.

diag:simu:goto 1,99

// Goto time 2 seconds

diag:simu:goto 2 s

20.4.3.10 DIAGnostic:SIMUlation:STOP

Pause emulation

Syntax:

DIAGnostic:SIMUlation:STOP

Description:

This command stops emulation without rewinding to the start of the emulation, i.e. performs pause operation.

Example:

// Pause emulation.

diag:simu:stop

20.4.3.11 DIAGnostic:SIMUlation:GOStart

Stop emulation, rewinding to start

Syntax:

DIAGnostic:SIMUlation:GOStart

Description:

This command stops the emulation and rewinds to the start of the emulation, i.e. performs stop operation.

Example:

// Stop emulation.

diag:simu:gos

20.4.3.12 DIAGnostic:SIMUlation:CONTinue

Continue paused emulation

Syntax:

DIAGnostic:SIMUlation:CONTinue

Description:

This command continues paused (see 20.4.3.10) emulation. Behaviour of this command is identical to diag:simu:go (see 20.4.3.8) and has been preserved for backward compatibility.

Example:

// Continue emulation.

diag:simu:cont

20.4.3.13 DIAGnostic:SIMUlation:STEP

Step emulation into next cir

Syntax:

DIAGnostic:SIMUlation:STEP

Description:

This command steps emulation (which is not running) to next channel impulse response change on any channel of the emulation.

Example:

// Step emulation.

diag:simu:step

20.4.3.14 DIAGnostic:SIMUlation:STATE?

Get emulation running state

Syntax:

DIAGnostic:SIMUlation:STATE?

Description:

This request returns the running state of the emulation. Emulation is being run after calling go or continue on emulation. Otherwise it is not being run. Possible return values are:

CLOSED	Emulation has not been loaded
OPENING	Emulation is loading (simultaneous use of ATE and GUI)
STOPPING	Emulation is stopping (simultaneous use of ATE and GUI)
STOPPED	Emulation is not being run
RUNNING	Emulation is being run
EDITING	Emulation is being edited
CLOSING	Emulation is closing (simultaneous use of ATE and GUI)

Example:

//Get emulation running state.

diag:simu:state?

RUNNING

20.4.3.15 DIAGnostic:SIMUlation:DESCription?

Get description of emulation

Syntax:

DIAGnostic:SIMUlation:DESCription?

DIAGnostic:SIMUlation:DESCription? <filename>

Description:

This request returns the description of the emulation. If emulation is opened, request can be given without parameter.

Example:

// get description of emulation

diag:simu:desc? e:\1to1 class.smu

20.4.3.16 DIAGnostic:SIMUlation:MODEL:KEEPPERiod

Keep emulation period

Syntax:

DIAGnostic:SIMUlation:MODEL:KEEPPERiod <keep state>

Description:

This setting will attempt to keep emulation period same when the center frequency of the channel changes, effectively changing the mobile speed instead of CIR update rate. Valid input values are 1 (to enable keeping period) and 0 (to disable setting). This setting is not saved when emulation is closed.

Example:

// Keep emulation period same on center frequency change

diag:simu:model:keepper 1

20.4.3.17 DIAGnostic:SIMUlation:MODEL:KEEPPERiod?

Request keep state

Syntax:

DIAGnostic:SIMUlation:MODEL:KEEPPERiod?

Description:

Returns whether system attempts to keep emulation period the same when center frequency is changed. Returns 1 if keeping period is enabled, and 0 if not. This setting is not saved when emulation is closed.

Example:

// Request whether emulation's period is kept or not

diag:simu:model:keepper?

1

20.4.3.18 DIAGnostic:SIMUlation:CLOSE

Close emulation

Syntax:

DIAGnostic:SIMUlation:CLOSE

Description:

This command closes open emulation.

Example:

// Close emulation.

diag:simu:close

20.4.3.19 DIAGnostic:SIMUlation:HIGHGAIN:SET

Set high gain mode. For more information, see chapter 4.3.2 (Emulation settings).

Syntax:

DIAGnostic:SIMUlation:HIGHGAIN:SET <set value>

Description:

This command sets the high gain mode of the emulation. Possible set values are:

0 set high gain mode off

1 set high gain mode on

Example:

// Set high gain mode on

diag:simu:highgain:set 1

20.4.3.20 DIAGnostic:SIMUlation:HIGHGAIN:GET?

Get high gain mode

Syntax:

DIAGnostic:SIMUlation:HIGHGAIN:GET?

Description:

This command retrieves the high gain mode of the emulation. Possible return values are:

0 high gain mode is off

1 high gain mode is on

Example:

// Get high gain mode

diag:simu:highgain:get?

1

20.4.3.21 DIAGnostic:SIMUlation:INTERPolation:SET

Set the interpolation mode.

Syntax:

DIAG:SIMU:INTERP:SET

Description:

This command sets the interpolation mode when connecting the emulation.

Possible parameter values for interpolatation mode are:

0, No interpolation

1, Coeff interpolation

Example:

// Set Coeff interpolation mode

DIAG:SIMU:INTERP:SET 1

20.4.3.22 DIAGnostic:SIMUlation:INTERPolation:GET?

Get the interpolation mode.

Syntax:

DIAG:SIMU:INTERP:GET?

Description:

This request retrieves the interpolation mode used in the emulation.

Return values are:

0, No interpolation

1, Coeff interpolation

Example:

// Get interpolation mode

DIAG:SIMU:INTERP:GET?

0

20.4.3.23 DIAGnostic:SIMUlation:WRAPmode:SET

Emulation wrapping mode set

Syntax:

DIAGnostic:SIMUlation:WRAPmode:SET <Wrap mode>, <Start time>, <End time>

Description:

This setting will set emulation wrapping mode, Parameters are:

Parameter 1(Wrap mode):

1, Continuous

2, Single run

3, Stop and re-run

Parameter 2 (Wrap modes 2 and 3):

Emulation start position in seconds

Parameter 3 (Wrap modes 2 and 3):

Emulation end position in seconds

Example:

//Set emulation wrap mode to continuous

Diag:simu:wrap:set 1

Example 2:

//Set emulation wrap mode to single run (Single run from 2.5s to 72.423s) diag:simu:wrap:set 2,2.5,72.423

20.4.3.24 DIAGnostic:SIMUlation:WRAPmode:GET?

Emulation wrapping mode get

Syntax:

DIAGnostic:SIMUlation:WRAPmode:GET?

Description:

This setting will get emulation wrapping mode parameters. Possible return values are:

Parameter 1:

1, Continuous
2, Single run
3, Stop and re-run

Parameter 2:

Emulation start position in seconds

Parameter 3:

Emulation end position in seconds

Example 1:

// get emulation wrap mode (continuous)
diag:simu:wrap:get?
1

Example 2:

// get emulation wrap mode (Single run, from 2.5s to 72.423s)
diag:simu:wrap:get?
2,2.5,72.423

20.4.3.25 GROup:BYPass:STate:CH

Set the current channel model bypass state for the channel group where the specified channel belongs.

Syntax:

GROup:BYPass:STate:CH <channel number>,<bypass>

Description:

This command sets channel model bypass state of the channel group where the specified channel belongs.

Possible parameter values for bypass are:

0, bypass off (channel model in use)

1, Butler bypass enabled

Example:

// Set butler bypass for channel group containing channel 5

GROup:BYPass:STate:CH 5,1

20.4.3.26 GROup:BYPass:STate:CH?

Get the current channel model bypass state for the channel group where the specified channel belongs.

Syntax:

GROup:BYPass:STate:CH? <channel number>

Description:

This command gets the current channel model bypass state for the channel group where the specified channel belongs.

Return values are:

-1, Links inside the group have different channel model bypass states

0,	bypass off (channel model in use)

1,	Butler bypass enabled

Example:

// Get bypass state for the channel group containing channel 5, Butler bypass is enabled

GROup:BYPass:STate:CH? 5

1

20.4.3.27 LINK:BYPass:STate:CH

Set the current channel model bypass state for the link where the specified channel belongs.

Syntax:

LINK:BYPass:STate:CH <channel number>,<bypass>

Description:

This command sets channel model bypass state of the link where the specified channel belongs.

Possible parameter values for bypass are:

0,	bypass off (channel model in use)

1,	Butler bypass enabled

Example:

// Set butler bypass for link containing channel 5

LINK:BYPass:STate:CH 5,1

20.4.3.28 LINK:BYPass:STate:CH?

Get the current channel model bypass state for the link where the specified channel belongs.

Syntax:

LINK:BYPass:STate:CH? <channel number>

Description:

This command gets the current channel model bypass state for the link where the specified channel belongs.

Return values are:

0,	bypass off (channel model in use)

1,	Butler bypass enabled

Example:

// Get bypass state for the link containing channel 5, Butler bypass is enabled

LINK:BYPass:STate:CH? 5

1

20.4.4 Channel Input Settings

20.4.4.1 INPut:ENable

Set channel input state (enable / disable)

Syntax:

INPut:ENable <input number>,<set value>

Description:

This command enables or disables the channel input. Possible set values are:

0	Disable input

1	Enable input

Example:

// Disable the input of channel 2.

inp:en 2,0

// Enable the input of channel 2.

inp:en 2,1

20.4.4.2 INPut:ENable?

Get channel input state (enable / disable)

Syntax:

INPut:Enable? <input number>

Description:

This command queries the state of the channel input (enabled or disabled). Possible return values are:

0 Input is disabled

1 Input is enabled

Example:

// Query the input state of channel 2.

inp:en? 2

1

// Return status is 1 (input is enabled)

20.4.4.3 INPut:LEVel:AMPlitude:CH

Set average input level

Syntax:

INPut:LEVel:AMPlitude:CH <input number>,<amplitude value>

Description:

This command sets the average input level of the specific channel input in dBm.

Example:

// Set average input level of channel input 2 to −18.2 dBm.

inp:lev:amp:ch 2,-18.2

20.4.4.4 INPut:LEVel:AMPlitude:CH?

Get average input level

Syntax:

INPut:LEVel:AMPlitude:CH? <input number>

Description:

This command retrieves the average input level of the specific channel input in dBm.

Example:

// Get average input level of channel input 8.

inp:lev:amp:ch? 8

-15

20.4.4.5 INPut:LEVel:AMPlitude:LIMits?

Get average input level limits

Syntax:

INPut:LEVel:AMPlitude:LIMits? <input number>

Description:

This command retrieves the average input level limit values of the specific channel input in dBm. Level cannot be set outside the limits.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get average input level limits of channel input 1.

inp:lev:amp:lim? 1

-23,0

20.4.4.6 INPut:LEVel:MEASure?

Get average input level and crest factor

Syntax:

INPut:LEVel:MEASure? <input number>,<measurement time>

Description:

This command measures and returns the average input level (in dBm) and crest factor (in dB) of the specific channel input. A failed measurement (no input signal or too strong output signal) will produce a device specific error. Measurement time can be 0.5, 1, 3, 5, or 10 seconds.

Example:

// Get average input level and crest factor of input 1 using 3 seconds measurement time.

inp:lev:meas? 1,3

-21.4,4

20.4.4.7 INPut:LEVel:AUTOSET

Set average input level and crest factor values

Syntax:

INPut:LEVel:AUTOSET <input number>,<measurement time>

Description:

This command measures average input level and crest factor and sets the parameters of the input accordingly. Measurement time can be 0.5, 1, 3, 5, or 10 seconds. A failed measurement (no input signal or too strong output signal) does not change previous settings but produces device-specific error. If input number is set to 0, all inputs will be autoset simultaneously.

Example:

// Set average input level and crest factor of input 1 using 3 seconds measurement time.

inp:lev:autoset 1, 3

20.4.4.8 INPut:LEVel:AUTOSETCANCEL

Cancel average input level and crest factor measurements

Syntax:

INPut:LEVel:AUTOSETCANCEL

Description:

This command cancels all ongoing autoset measurements. The command is parallel, so it is executed even if there are other ATE-commands pending. When a measurement is cancelled, input settings are not changed.

Example:

// cancel all ongoing autoset measurements

inp:lev:autosetcancel

20.4.4.9 INPut:CREst:SET

Set crest factor for channel input

Syntax:

INPut:CREst:SET <input number>,<crest factor value>

Description:

This command sets the crest factor for channel input in dB.

Example:

// Set crest factor of channel input 1 to 4 dB.

inp:cre:set 1,4

20.4.4.10 INPut:CREst:GET?

Get crest factor of channel input

Syntax:

INPut:CREst:GET? <input number>

Description:

This command retrieves the crest factor for channel input in dB.

Example:

// Get crest factor of channel input 1.

inp:cre:get? 1

4

20.4.4.11 INPut:CREst:LIMits?

Get crest factor limits of channel input

Syntax:

INPut:CREst:LIMits? <input number>

Description:

This command retrieves the crest factor limit values of the specific channel input in dB. Crest factor cannot be set outside the limits. If attempted, factor is automatically set to closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get crest factor limits of channel input 6:

inp:cre:lim? 6

0,23

20.4.4.12 INPut:IF:TYPE?

Get interface type of channel input

Syntax:

INPut:IF:TYPE? <input number>

Description:

This command retrieves the interface type of channel input. Possible return types are:

RF

Example:

// Get interface type of channel input 1.

inp:if:type? 1

RF

20.4.4.13 INPut:PHAse:CH

Set channel input phase register value

Syntax:

INPut:PHAse:CH <input number>,<phase register value>

Description:

This command sets the phase of the specific channel input by defining the affecting register value. The adjustment range is 1200 ... 3200. Each step represents 0.2 degrees.

Example:

// Set channel input 2 phase register value to 2200.

inp:pha:ch 2,2200

20.4.4.14 INPut:PHAse:CH?

Get channel input phase register value

Syntax:

INPut:PHAse:CH? <input number>

Description:

This request returns the phase register value of the specific channel input.

Example:

// Get channel input 1 phase register value.

inp:pha:ch? 1

1800

20.4.4.15 INPut:PHAse:LIMits?

Get channel input phase register value limits

Syntax:

INPut:PHAse:LIMits? <input number>

Description:

This command retrieves the phase register limit values of the specific channel input. Register value cannot be set outside the limits. If attempted, value is automatically set to closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get phase register value limits of channel input 6:

inp:pha:lim? 6

1200,3200

20.4.4.16 INPut:PHAse:DEGrees:CH

Set channel input phase in degrees

Syntax:

INPut:PHAse:DEG:CH <input number>,<phase value>

Description:

This command sets the phase of the specific channel input. The adjustment range is -200 ... 200 degrees.

Example:

// Set channel input 2 phase to 20 degrees

inp:pha:deg:ch 2,20

20.4.4.17 INPut:PHAse:DEGrees:CH?

Get channel input phase

Syntax:

INPut:PHAse:DEG:CH? <input number>

Description:

This request returns the phase of the specific channel input in degrees.

Example:

// Get channel input 1 phase

inp:pha:deg:ch? 1

30

20.4.4.18 INPut:PHAse:DEGrees:LIMits?

Get channel input phase limits in degrees

Syntax:

INPut:PHAse:DEG:LIMits? <input number>

Description:

This command retrieves the phase limit values of the specific channel input. Phase value cannot be set outside the limits.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get phase limits of channel input 6:

inp:pha:deg:lim? 6

-200,200

20.4.4.19 INPut:PHAse:DEGrees:DELTA:CH

Change channel input phase in degrees

Syntax:

INPut:PHAse:DEG:DELTA:CH <input number>,<phase delta>

Description:

This command adds the given phase delta to the current phase of the specific channel input.

Example:

// Change channel input 2 phase by 10 degrees

inp:pha:deg:ch? 2

20

inp:pha:deg:delta:ch 2, 10

inp:pha:deg:ch? 2

30

20.4.4.20 INPut:LOSS:SET

Set the input loss.

Syntax:

INPut:LOSS:SET <input number>,<loss>

Description:

This command sets the input loss in dB.

Example:

// Set input 1 loss to 2.5 dB

inp:loss:set 1,2.5

20.4.4.21 INPut:LOSS:GET?

Get the input loss

Syntax:

INPut:LOSS:GET? <input number>

Description:

This command gets the input loss in dB.

Example:

// Get input 1 loss

inp:loss:get? 1

2.5

20.4.4.22 INPut:LOSS:LIMits?

Get the input loss limits

Syntax:

INPut:LOSS:LIMits? <input number>

Description:

This command gets the input loss limits in dB.

Example:

// Get input 1 loss limits

inp:loss:lim? 1

-100,100

20.4.4.23 INPut:MEASure:MODE:SET

Set the measurement mode of an input

Syntax:

INPut:MEASure:MODE:SET <input number>,<measurement mode>

Description:

Measurement mode is a numeric value. Available measurement modes are:

0 DISABLED

1 BASIC

2 CONTINUOUS

3 BURST

Example:

// Set measurement mode to continuous on input 2

inp:meas:mode:set 2,2

20.4.4.24 INPut:MEASure:MODE:GET?

Get the measurement mode of an input

Syntax:

INPut:MEASure:MODE:GET? <input number>

Description:

The measurement modes are:

0 DISABLED

1 BASIC

2 CONTINUOUS

3 BURST

Example:

// Get measurement mode of input 2. Is in continuous mode.

inp:meas:mode:get? 2

2

20.4.4.25 INPut:MEASure:FREEZE

Freeze the measurement result

Syntax:

INPut:MEASure:FREEZE <input number>,<mode>

Description:

The measurement taken from an input can be “freezed” as the last measured value or the measurements can be continued.

0 MEASURE

1 FREEZE

Example:

// Set the measurement mode of input 1 to frozen

inp:meas:freeze 1,1

20.4.4.26 INPut:MEASure:FREEZE?

Query the measuring on an input

Syntax:

INPut:MEASure:FREEZE? <input number>

Description:

This command retrieves status information of the input measurements i.e. whether an input is “frozen” as the last measured value or the input is continuously measured.

0 MEASURE

1 FREEZE

Example:

// Query the mode of input 1. It is measuring.

inp:meas:freeze? 1

0

20.4.4.27 INPut:MEASure:BURST:TRIGger:SET

Set the input trigger absolute level in burst measurement mode.

Syntax:

INPut:MEASure:BURST:TRIGger:SET <input number>,<trigger value in dBm>

Description:

Set the absolute trigger level of an input in dBm. This command is available only in burst measurement mode.

Example:

// Set the absolute trigger level of input 1 to -10.1 dBm.

inp:meas:burst:trig:set 1,-10.1

20.4.4.28 INPut:MEASure:BURST:TRIGger:RELative:SET

Set the input trigger relative level in burst measurement mode.

Syntax:

INPut:MEASure:BURST:TRIGger:RELative:SET <input number>,<trigger value in dB>

Description:

Set the trigger level of an input in dB, relative to current input level setting. This command is available only in burst measurement mode.

Example:

// Set the relative trigger level of input 1 to -30.1 dBm (30.1 dB below the average input level setting).

inp:meas:burst:trig:rel:set 1,-30.1

20.4.4.29 INPut:MEASure:BURST:TRIGger:GET?

Get the input trigger absolute level in burst measurement mode.

Syntax:

INPut:MEASure:BURST:TRIGger:GET? <input number>

Description:

Get the current absolute trigger level of an input in dBm. This command is available only in burst measurement mode.

Example:

// Query the trigger level setting of input 1.

inp:meas:burst:trig:get? 1

-10.1

20.4.4.30 INPut:MEASure:BURST:TRIGger:RELative:GET?

Get the input trigger relative level in burst measurement mode.

Syntax:

INPut:MEASure:BURST:TRIGger:GET? <input number>

Description:

Get the current relative trigger level of an input in dB, relative to current input level setting. This command is available only in burst measurement mode.

Example:

// Query the relative trigger level setting of input 1.

inp:meas:burst:trig:rel:get? 1

-30.1

20.4.4.31 INPut:MEASure:BURST:TRIGger:LIMits?

Get the input trigger absolute level limits in burst measurement mode.

Syntax:

INPut:MEASure:BURST:TRIGger:LIMits? <input number>

Description:

Query the absolute trigger level limits of an input in dBm. This command is available only in burst mode measurements.

Example:

// Query the absolute trigger level limits of input 1. Limits are -50 and 0 dBm.

inp:meas:burst:trig:lim? 1

-50,0

20.4.4.32 INPut:MEASure:BURST:TRIGger:RELative:LIMits?

Get the input trigger relative level limits in burst measurement mode.

Syntax:

INPut:MEASure:BURST:TRIGger:RELative:LIMits? <input number>

Description:

Query the trigger level limits of an input, relative to input level setting. This command is available only in burst mode measurements.

Example:

// Query the trigger level limits of input 1. They are -48 and -12 dB.

inp:meas:burst:trig:lim? 1

-48,-12

20.4.4.33 INPut:MEASure:BURST:AVERages:SET

Set the burst measurement length in samples

Syntax:

INPut:MEASure:BURST:AVERages:SET <input number>,<sample count>

Description:

Set the burst measurement length in samples. This command is available only in burst measurement mode.

Example:

// Set the burst measurement length of input 1 to 234567 samples.

inp:meas:burst:aver:set 1,234567

20.4.4.34 INPut:MEASure:BURST:AVERages:GET?

Get the burst measurement length in samples.

Syntax:

INPut:MEASure:BURST:AVERages:GET? <input number>

Description:

Get the current burst measurement length of an input. This command is available only in burst measurement mode.

Example:

// Query the measurement length of input 1. It is 234567 samples.

inp:meas:burst:aver:get? 1

234567

20.4.4.35 INPut:MEASure:BURST:AVERages:LIMits?

Get the burst measurement length limits in samples.

Syntax:

INPut:MEASure:BURST:AVERages:LIMits? <input number>

Description:

Get the measurement length limits in samples. This command is available only in burst measurement mode.

Example:

// Query the measurement length limits of input 1. The limits are 4 and 134217728.

inp:meas:burst:aver:lim? 1

4,134217728

20.4.4.36 INPut:MEASure:RESult:GET?

Returns the latest measurement of the input.

Syntax:

INPut:MEASure:RESult:GET? <input number>

Description:

Get the latest measurement result of an input.

Example:

// Query the last measured value of input 1. It is -10.1 dBm.

inp:meas:res:get? 1

-10.1

20.4.4.37 INPut:MEASure:STATus:GET?

Returns the measurement status of the input.

Syntax:

INPut:MEASure:STATus:GET? <input number>

Description:

Get the current measurement status of an input. Returned values:

0	Idle
1	No Result
2	Disabled
3	Freezed
4	Measuring
5	New Result
6	Result Not Ready

Example:

// Query the status of input 1. It is “New result”.

inp:meas:stat:get? 1

5

20.4.4.38 INPut:MEASure:RESult:OFFSET:SET

Set the offset for input measurements

Syntax:

INPut:MEASure:RESult:OFFSET:SET <input number>,<offset>

Description:

This control sets the offset to an input. The resolution for offset is one decimal place.

Example:

// Set 0.7 dB offset to measurements on input 1

inp:meas:res:offset:set 1,0.7

20.4.4.39 INPut:MEASure:RESult:OFFSET:GET?

Query the offset value of measurements on an input

Syntax:

INPut:MEASure:RESult:OFFSET:GET? <input number>

Description:

Get the measurement offset of an input. The resolution for offset is one decimal place.

Example:

// Set an offset to measurements on input 1. It is 0.7 dB.

inp:meas:res:offset:get? 1

0.7

20.4.4.40 INPut:MEASure:RESult:OFFSET:LIMits?

Get the offset limits for input measurements

Syntax:

INPut:MEASure:RESult:OFFSET:LIMits? <input number>

Description:

Get the offset limits of an input. The resolution for offset is one decimal place.

Example:

// Get the offset limits of input 1. They are -3.0 and +3.0 dB.

inp:meas:res:offset:lim? 1

-3.0,3.0

20.4.4.41 INPut:ATTenuator:ENable

Set channel input attenuator state (enable / disable)

Syntax:

INPut:ATTenuator:ENable <input number>,<set value>

Description:

This command enables or disables the channel input attenuator. Possible set values are:

0 Disable attenuator

1 Enable attenuator

Example:

// Disable the input 2 attenuator

inp:att:en 2,0

// Enable the input 2 attenuator

inp:att:en 2,1

20.4.4.42 INPut:ATTenuator:ENable?

Get channel input attenuator state (enable / disable)

Syntax:

INPut:ATTenuator Enable? <input number>

Description:

This command queries the state of the channel input attenuator (enabled or disabled). Possible return values are:

0 Input attenuator is disabled

1 Input attenuator is enabled

Example:

// Query the input state of channel 2.

inp:att:en? 2

1

// Return status is 1 (input attenuator is enabled)

20.4.4.43 INPut:CALIBration:VALid?

Get channel input calibration status (phase and gain calibration of user's test setup)

Syntax:

INPut:CALIBration:VALid? <input number>

Description:

This command queries the status of calibration for an input.

Possible return values are:

0 Input calibration is not valid

1 Input calibration is valid

Example:

// Query the calibration status for input 1

inp:calib:valid? 1

1

20.4.4.44 INPut:CALIBration:GET?

Get channel input gain and phase calibration data.

Syntax:

INPut:CALIBration:GET? <input number>

Description:

This command queries the gain (dB) and phase (degrees) calibration for an input. The calibration values are returned as follows:

<gain calibration>,<phase calibration>

Example:

// get input calibration data

inp:calib:get? 1

0.7,2.7

20.4.4.45 INPut:NAME?

Get name of the channel input

Syntax:

INPut:NAME? <input number>

Description:

This command retrieves the name of the channel input.

Example:

// Get name of input 1

inp:name? 1

BS1-TX1

20.4.4.46 INPut:CONnector:SET

Set connector of the channel input

Syntax:

INPut:CONnector:SET <input number>,<emulator number>,<unit number>,<connector position in unit>

Description:

This command sets the connector of the channel input. Command requires emulation to be opened to edit-mode, see CALC:FILT:EDIT command in chapter 20.4.3.2.

Example:

// Set connector of input 8 to emulator 1, fourth input in unit 2

inp:con:set 8,1,2,4

20.4.4.47 INPut:CONnector:GET?

Get connector of the channel input

Syntax:

INPut:CONnector:GET? <input number>

Description:

This command retrieves the connector of the channel input. Connector is returned in format:

<emulator number>,<unit number>,<input connector position in unit>

Example:

// Get connector of input 8 (emulator 1, fourth input in unit 2)

inp:con:get? 8

1,2,4

20.4.4.48 INPut:LEVel:AUTO:ENAble

Set automatic input level control (AILC) state (enable / disable). For more information about AILC, see section 4.3.4 Base station/mobile station settings.

Syntax:

INPut:LEVel:AUTO:ENAble <input number>,<set value>

Description:

This command enables or disables the automatic input level control. Possible set values are:

0 Disable automatic input level control

1 Enable automatic input level control

Example:

// Set automatic input level control enabled for input 1

input:level:auto:enable? 1,1

20.4.4.49 INPut:LEVel:AUTO:ENAble?

Get automatic input level control state (enable / disable)

Syntax:

INPut:LEVel:AUTO:ENAble? <input number>

Description:

This command retrieves the automatic input level control state. Possible return values are:

0 Automatic input level control is disabled

1 Automatic input level control is enabled

Example:

// Get automatic input level control state for input 1 (enabled)

input:level:auto:enable? 1

1

20.4.4.50 INPut:LEVel:AUTO:MODE

Set automatic input level control (AILC) mode.

Syntax:

INPut:LEVel:AUTO:MODE <input number>,<mode>

Description:

This command sets the automatic input level control mode. Possible set values are:

1 Prevent cut-off

2 AGC

3 AGC keep path loss

Example:

// Set automatic control mode to AGC and enabled it

input:level:auto:mode 1,2

input:level:auto:ena 1,1

20.4.4.51 INPut:LEVel:AUTO:MODE?

Get automatic input level control (AILC) mode.

Syntax:

INPut:LEVel:AUTO:MODE? <input number>

Description:

This command gets the automatic input level control mode. Possible return values are:

1 Prevent cut-off

2 AGC

3 AGC keep path loss

Example:

// Get automatic input level control mode for input 1. Mode is AGC (2).

input:level:auto:mode? 1

2

20.4.4.52 INPut:LEVel:AUTO:STATus?

Get status of the automatic input level control

Syntax:

INPut:LEVel:AUTO:STATus? <input number>

Description:

This command retrieves the automatic input level control status.

Possible return values for mode 1 (prevent cut-off) are:

0 Path loss is not maintained, input adjusted due to too high input level

1 Path loss maintained, no adjustment done

Possible return values for mode 2 (AGC) are:

0 Input level out of adjustment range

1 Input level in adjustment range or signal under the input burst power meter trigger level

Example:

// Get automatic input level control status for input 1

input:level:auto:stat? 1

1

20.4.4.53 INPut:RSRP:MEAS?

Performs RSRP measurement in the PROPSIM inputs and reports the result in dBm.

Syntax:

INPut:RSRP:MEAS? <Number of inputs (N)>,<input id#1>,...,<input id#N>,<Signal technology>,<Technology Specific parameter #1>,...,<Technology Specific parameter #M>

Description:

This command performs RSRP measurement for a given list of PROPSIM inputs. Input list must contain all the inputs connected to the same user device (ie. single LTE base station) and its primary antenna must be the first. Possible parameter values are:

Signal technology:

LTE, Signal technology LTE downlink
5G, Signal technology 5G downlink

LTE technology specific parameters:

Parameter#1: Signal bandwidth in MHz (3, 5, 10, 15 and 20MHz are supported)

Parameter#2: Cell Id (integer number)

Parameter#3: Signal center frequency in MHz

Note: *Measurement typically takes 10-60 seconds.*

5G technology specific parameters:

Parameter#1: Signal bandwidth in MHz (20, 50 and 100MHz are supported)
Parameter#2: Cell Id (integer number)
Parameter#3: Signal center frequency in MHz
Parameter#4: (optional): Sub carrier spacing (SCS) in kHz (15, 30, 60, 120 kHz are supported)

Example 1:

// Measure RSRP of the 4 antenna LTE base station connected to PROPSIM inputs 5,6,7 and 8 (input
// 5 is the primary BS antenna). Signal bandwidth is 10 MHz, Cell id is 54 and center frequency is
// 2620 MHz

inp:rsrp:meas? 4,5,6,7,8,LTE,10,54,2620

-86.5

Example 2:

// Measure RSRP of the 4 antenna 5G base station connected to PROPSIM inputs 5,6,7 and 8 (input
// 5 is the primary BS antenna). Signal bandwidth is 20 MHz, Cell id is 54, center frequency is
// 2620 MHz and subcarrier spacing is 15kHz

inp:rsrp:meas? 4,5,6,7,8,5G,20,54,2620,15

-86.5

20.4.4.54 INPut:RSRP:GET?

Gets the previously measured RSRP value in dBm.

Syntax:

INPut:RSRP:GET? <input number>

Description:

This command retrieves the previously measured RSRP value entering given the PROPSIM input in dBm. Return value “not measured” indicates that RSRP measurement is not performed after the emulation has been loaded.

Example:

// Get the previous RSRP measurement result of input 5

inp:rsrp:get? 5

-86.5

20.4.4.55 INPut:SUBBand:COUNT?

Get input sub-band count (emulation bandwidth > 200 MHz)

Syntax:

INPut:SUBBand:COUNT? <input number>

Description:

This command retrieves the input sub-band count.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get number of sub-bands on input 1

inp:subband:count? 1

2

20.4.4.56 INPut:SUBBand:OFFSet:GAIN:GET?

Get input sub-band gain offset (emulation bandwidth > 200 MHz)

Syntax:

INPut:SUBBand:OFFSet:GAIN:GET? <input number>,<sub-band number>

Description:

This command retrieves the input sub-band gain offset in dB.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get sub-band 1 gain offset on input 1

inp:subband:offset:gain:get? 1,1

-0.1

20.4.4.57 INPut:SUBBand:OFFSet:GAIN:SET

Set input sub-band gain offset (emulation bandwidth > 200 MHz)

Syntax:

INPut:SUBBand:OFFSet:GAIN:SET <input number>,<sub-band number>,<gain>

Description:

This command sets the input sub-band gain offset in dB.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// set sub-band 1 gain offset on input 1 to -0.1dB

inp:subband:offset:gain:set 1,1,-0.1

20.4.4.58 INPut:SUBBand:OFFSet:GAIN:LIMits?

Get input sub-band gain offset limits (emulation bandwidth > 160 MHz)

Syntax:

INPut:SUBBand:OFFSet:GAIN:LIMits? <input number>,<sub-band number>

Description:

This command retrieves the input sub-band gain offset limits in dB. The returned values are as follows:

<lower limit>,<higher limit>

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get sub-band 1 gain offset limits on input 1

inp:subband:offset:gain:limits? 1,1

-3,0

20.4.4.59 INPut:SUBBand:OFFSet:PHASE:GET?

Get input sub-band phase offset (emulation bandwidth > 200 MHz)

Syntax:

INPut:SUBBand:OFFSet:PHASE:GET? <input number>,<sub-band number>

Description:

This command retrieves the input sub-band phase offset in degrees.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get sub-band 1 phase offset on input 1

inp:subband:offset:phase:get? 1,1

53.2

20.4.4.60 INPut:SUBBand:OFFSet:PHASE:SET

Set input sub-band phase offset (emulation bandwidth > 200 MHz)

Syntax:

INPut:SUBBand:OFFSet:PHASE:SET <input number>,<sub-band number>,<phase>

Description:

This command sets the input sub-band phase offset in degrees.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// set sub-band 1 phase offset on input 1 to 53.2 deg

inp:subband:offset:phase:set 1,1,53.2

20.4.4.61 INPut:SUBBand:OFFSet:PHASE:LIMits?

Get input sub-band phase offset limits (emulation bandwidth > 200 MHz)

Syntax:

INPut:SUBBand:OFFSet:PHASE:LIMits? <input number>,<sub-band number>

Description:

This command retrieves the input sub-band phase offset limits in degrees. The returned values are as follows:

<lower limit>,<higher limit>

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get sub-band 1 phase offset limits on input 1

inp:subband:offset:phase:limits? 1,1

-200,200

20.4.4.62 INPut:SUBBand:OFFSet:DELAY:GET?

Get input sub-band delay offset (emulation bandwidth > 200 MHz)

Syntax:

INPut:SUBBand:OFFSet:DELAY:GET? <input number>,<sub-band number>

Description:

This command retrieves the input sub-band delay offset in nano seconds.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get sub-band 1 delay offset on input 1

inp:subband:offset:delay:get? 1,1

0.55

20.4.4.63 INPut:SUBBand:OFFSet:DELAY:SET

Set input sub-band delay offset (emulation bandwidth > 200 MHz)

Syntax:

INPut:SUBBand:OFFSet:DELAY:SET <input number>,<sub-band number>,<delay>

Description:

This command sets the input sub-band delay offset in nano seconds.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// set sub-band 1 delay offset on input 1 to 0.55 ns

inp:subband:offset:delay:set 1,1,0.55

20.4.4.64 INPut:SUBBand:OFFSet:DELAY:LIMits?

Get input sub-band delay offset limits (emulation bandwidth > 200 MHz)

Syntax:

INPut:SUBBand:OFFSet:DELAY:LIMits? <input number>,<sub-band number>

Description:

This command retrieves the input sub-band delay offset limits in nano seconds. The returned values are as follows:

<lower limit>,<higher limit>

Note: The sub-band indexing parameter starts from ‘0’. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get sub-band 1 delay offset limits on input 1

inp:subband:offset:delay:limits? 1,1

-1,1

20.4.4.65 INPut:FREQuency:SET

Set input frequency (frequency conversion purposes).

Syntax:

INPut:FREQuency:SET <input number>,<frequency in MHz>

Description:

This command enables frequency conversion and sets input frequency. Frequency conversion setting must be enabled to use this command.

Example:

// Set input 2 frequency to 1000 MHz.

inp:freq:set 2,1000

20.4.4.66 INPut:FREQuency:GET?

Get input frequency.

Syntax:

INPut:FREQuency:GET? <input number>

Description:

This command retrieves the input frequency.

Example:

// Get input 2 frequency. 1000 MHz.

inp:freq:get? 2
1000

20.4.4.67 INPut:FREQuency:LIMits?

Get input frequency limits.

Syntax:

INPut:FREQuency:LIMits? <input number>

Description:

This command retrieves the input frequency limits.

Example:

// Get input 2 frequency limits. 450 MHz – 6000 MHz.

inp:freq:lim? 2
450,6000

20.4.4.68 INPut:DELAY:CH

Set channel input delay offset

Syntax:

INPut:DELAY:CH <input number>,<delay offset in ns>

Description:

This command sets the delay offset of the specific channel input.

Example:

// Set channel input 2 delay offset to 200ps.

inp:delay:ch 2,0.200

20.4.4.69 INPut:DELAY:CH?

Get channel input delay offset

Syntax:

INPut:DELAY:CH? <input number>

Description:

This request returns the delay offset of the specific channel input.

Example:

// Get channel input 1 delay offset.

inp:delay:ch? 1

1.1

20.4.4.70 INPut:DELAY:LIMits?

Get channel input delay offset limits

Syntax:

INPut:DELAY:LIMits? <input number>

Description:

This command retrieves the delay offset limit values of the specific channel input. Delay offset cannot be set outside the limits. If attempted, value is automatically set to closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get delay offset value limits of channel input 6:

inp:delay:lim? 6

-10,50

20.4.5 Channel Output Settings

20.4.5.1 OUTPut:ENable

Set channel output state (enable / disable)

Syntax:

OUTPut:ENable <output number>,<set value>

Description:

This command enables or disables the channel output. Possible set values are:

0, Disable output

1, Enable output

Example:

// Disable the output of channel 2.

outp:en 2,0

// Enable the output of channel 2.

outp:en 2,1

20.4.5.2 OUTPut:ENable?

Get channel output state (enable / disable)

Syntax:

OUTPut:Enable? <output number>

Description:

This command queries the state of the channel output state (enable or disable). Possible return values are:

0, Output is disabled

1, Output is enabled

Example:

// Query the output state of channel 2.

outp:en? 2

1

// return status is 1 (output is enabled)

20.4.5.3 OUTPut:LEVel:AMPlitude:CH

Set average output level

Syntax:

OUTPut:LEVel:AMPlitude:CH <output number>,<amplitude value>

Description:

This command sets the average output level of the specific channel output in dBm.

Example:

// Set average output level of channel output 2 to –40 dBm.

outp:lev:amp:ch 2,-40

20.4.5.4 OUTPut:LEVel:AMPlitude:CH?

Get average output level

Syntax:

OUTPut:LEVel:AMPlitude:CH? <output number>

Description:

This command retrieves the average output level of the specific channel output in dBm.

Example:

// Get average output level of channel output 7.

outp:lev:amp:ch? 7

-40

20.4.5.5 OUTPut:LEVel:AMPlitude:LIMits?

Get average output level limits

Syntax:

OUTPut:LEVel:AMPlitude:LIMits? <output number>

Description:

This command retrieves the average output level limit values of the specific channel output in dBm. Level cannot be set outside the limits.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get average output level limits of channel output 1.

outp:lev:amp:lim? 1

-68.8401,-23.8401

20.4.5.6 OUTPut:GAIN:CH

Set output gain

Syntax:

OUTPut:GAIN:CH <output number>,<gain value>

Description:

This command sets the gain of the specific channel output in dB.

Example:

// Set gain of channel output 2 to –5 dB.

outp:gain:ch 2,-5

20.4.5.7 OUTPut:GAIN:CH?

Get output gain

Syntax:

OUTPut:GAIN:CH? <output number>

Description:

This command retrieves the gain of the specific channel output in dB.

Example:

// Get gain of channel output 2.

outp:gain:ch? 2

-5

20.4.5.8 OUTPut:GAIN:LIMits?

Get output gain limits

Syntax:

OUTPut:GAIN:LIMits? <output number>

Description:

This command retrieves the gain limit values of the specific channel output in dB. Gain can not be set outside the limits. If attempted, gain is automatically set to closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get gain limits of channel output 1.

outp:gain:lim? 1

-45,0

20.4.5.9 OUTPut:PHAse:CH

Set channel output phase register value

Syntax:

OUTPut:PHAse:CH <output number>,<phase register value>

Description:

This command sets the phase of the specific channel output by defining the affecting register value. The adjustment range is 1200 ... 3200. Each step represents 0.2 degrees.

Example:

// Set channel output 6 phase register value to 2200.

outp:pha:ch 6,2200

20.4.5.10 OUTPut:PHAse:CH?

Get channel output phase register value

Syntax:

OUTPut:PHAse:CH? <output number>

Description:

This request returns the phase register value of the specific channel output. The adjustment range is 1200 ... 3200. Each step represents 0.2 degrees.

Example:

// Get channel output 1 phase register value.

outp:pha:ch? 1

1800

20.4.5.11 OUTPut:PHAse:LIMits?

Get channel output phase register value limits

Syntax:

OUTPut:PHAse:LIMits? <output number>

Description:

This command retrieves the phase register limit values of the specific channel output. Register value cannot be set outside the limits. If attempted, value is automatically set to closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get phase register value limits of channel output 2.

outp:pha:lim? 2

1200,3200

20.4.5.12 OUTPut:PHAse:DEGrees:CH

Set channel output phase in degrees

Syntax:

OUTPut:PHAse:DEG:CH <output number>,<phase value>

Description:

This command sets the phase of the specific channel output. The adjustment range is -200 ... 200 degrees.

Example:

// Set channel output 2 phase value to 20 degrees

outp:pha:deg:ch 2,20

20.4.5.13 OUTPut:PHAse:DEGrees:CH?

Get channel output phase

Syntax:

OUTPut:PHAse:DEG:CH? <output number>

Description:

This request returns the phase of the specific channel output in degrees.

Example:

// Get channel output 1 phase

outp:pha:deg:ch? 1

30

20.4.5.14 OUTPut:PHAse:DEGrees:LIMits?

Get channel output phase limits

Syntax:

OUTPut:PHAse:DEG:LIMits? <output number>

Description:

This command retrieves the phase limit values of the specific channel output. Phase value cannot be set outside the limits.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get phase limits of channel output 6:

outp:pha:deg:lim? 6

-200,200

20.4.5.15 OUTPut:PHAse:DEGrees:DELTA:CH

Change channel output phase in degrees

Syntax:

OUTPut:PHAse:DEG:DELTA:CH <output number>,<phase delta>

Description:

This command adds the given phase delta to the current phase of the specific channel output.

Example:

// Change channel output 2 phase by 10 degrees

outp:pha:deg:ch? 2

20

outp:pha:deg:delta:ch 2, 10

outp:pha:deg:ch? 2

30

20.4.5.16 OUTPut:SIGnal:Status

Set the status of signal path to output

Syntax:

OUTPut:SIGnal:STatus <output number>,<signal status>

Description:

This command sets the status of signal path to specific channel output. Possible settings are:

0, Signal disabled

1, Signal enabled

Example:

// Disable signal to channel output 1.

outp:sig:st 1,0

20.4.5.17 OUTPut:SIGnal:STatus?

Get the status of signal path to output

Syntax:

OUTPut:SIGnal:STatus? <output number>

Description:

This command retrieves the status of signal path to specific channel output. Possible return values are:

0, Signal disabled

1, Signal enabled

Example:

// Get the status of signal to channel output 1.

outp:sig:st? 1

0

20.4.5.18 OUTPut:IF:TYPE?

Get interface type of channel output

Syntax:

OUTPut:IF:TYPE? <output number>

Description:

This command retrieves the interface type of channel output. Possible return types are:

RF

Example:

// Get interface type of channel output 1.

outp:if:type? 1

RF

20.4.5.19 OUTPut:LOSS:SET

Set the output loss.

Syntax:

OUTPut:LOSS:SET <output number>,<loss>

Backward compatible syntax:

OUTPut:CABLELoss:SET <output number>,<loss>

Description:

This command sets the output loss in dB.

Example:

// Set output 1 loss to 2.5 dB

outp:loss:set 1,2.5

20.4.5.20 OUTPut:LOSS:GET?

Get the output loss

Syntax:

OUTPut:LOSS:GET? <output number>

Backward compatible syntax:

OUTPut:CABLELoss:GET? <output number>

Description:

This command gets the output loss in dB.

Example:

// Get output 1 loss

outp:loss:get? 1

2.5

20.4.5.21 OUTPut:LOSS:LIMits?

Get the output loss limits

Syntax:

OUTPut:LOSS:LIMits? <output number>

Backward compatible syntax:

OUTPut:CABLELoss:LIMits? <output number>

Description:

This command gets the output loss limits in dB.

Example:

// Get output 1 loss limits

outp:loss:lim? 1

-30,80

20.4.5.22 OUTPut:MEASure:RESult:GET?

Returns the latest measurement of the output power.

Syntax:

OUTPut:MEASure:RESult:GET? <output number>,[measurement option]

Description:

Get the latest power measurement result of an output, including shadowing attenuation and out loss. If parameter "measurement option" is not defined measurement option 0 is used (legacy mode).

Possible measurement options are:

0, Signal power at output is calculated based on input power measurements and average channel model gain(s). The power of internal interference generator(s) at output is excluded.

1, Total power is measured at output including the power of internal interference generator(s), measurement averaging time is fixed 0.5 seconds.

If the measurement result is not ready, "not ready" is returned.

Example:

// Query the last measured value of output 1 based on input measurements (legacy mode). Power is -56.1 dBm.

outp:meas:res:get? 1

-56.1

// Query the last measured value of output 3, measured at output connector. Power is -58.3 dBm.

outp:meas:res:get? 3,1

-58.3

20.4.5.23 OUTPut:CALIBration:VALid?

Get channel output calibration status (phase and gain calibration of user's test setup)

Syntax:

OUTPut:CALIBration:VALid? <output number>

Description:

This command queries the status of calibration.

Possible return values are:

0 Output calibration is not valid

1 Output calibration is valid

Example:

// Query the calibration status for output 2

outp:calib:valid? 2

1

20.4.5.24 OUTPut:CALIBration:GET?

Get channel output gain and phase calibration data.

Syntax:

OUTPut:CALIBration:GET? <output number>

Description:

This command queries the gain (dB) and phase (degrees) calibration for an output. The calibration values are returned as follows:

<gain calibration>,<phase calibration>

Example:

// get output calibration data

outp:calib:get? 1

0.2,3.1

20.4.5.25 OUTPut:NAME?

Get name of the channel output

Syntax:

OUTPut:NAME? <output number>

Description:

This command retrieves the name of the channel output

Example:

// Get name of output 1

outp:name? 1

MS1-RX1

20.4.5.26 OUTPut:CONnector:SET

Set connector of the channel output

Syntax:

OUTPut:CONnector:SET <output number>,<emulator number>,<unit number>,<connector position in unit>

Description:

This command sets the connector of the channel output. Command requires emulation to be opened to edit-mode, see CALC:FILT:EDIT command in chapter 20.4.3.2.

Example:

// Set connector of output 8 to emulator 1, fourth input in unit 2

outp:con:set 8,1,2,4

20.4.5.27 OUTPut:CONnector:GET?

Get connector of the channel output

Syntax:

OUTPut:CONnector:GET? <output number>

Description:

This command retrieves the connector of the channel output. Connector is returned in format:

<emulator number>,<unit number>,<output connector position in unit>

Example:

// Get connector of output 8 (emulator 1,fourth output in unit 2)

outp:con:get? 8

1,2,4

20.4.5.28 OUTPut:SUBBand:COUNT?

Get input sub-band count (emulation bandwidth > 160 MHz)

Syntax:

OUTPut:SUBBand:COUNT? <output number>

Description:

This command retrieves the output sub-band count.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get number of sub-bands on output 1

outp:subband:count? 1

2

20.4.5.29 OUTPut:SUBBand:OFFSet:GAIN:GET?

Get output sub-band gain offset (emulation bandwidth > 200 MHz)

Syntax:

OUTPut:SUBBand:OFFSet:GAIN:GET? <output number>,<sub-band number>

Description:

This command retrieves the output sub-band gain offset in dB.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get sub-band 1 gain offset on output 1

outp:subband:offset:gain:get? 1,1

-0.2

20.4.5.30 OUTPut:SUBBand:OFFSet:GAIN:SET

Set output sub-band gain offset (emulation bandwidth > 200 MHz)

Syntax:

OUTPut:SUBBand:OFFSet:GAIN:SET <output number>,<sub-band number>,<gain>

Description:

This command sets the output sub-band gain offset in dB.

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// set sub-band 1 gain offset on output 1 to -0.2dB outp:subband:offset:gain:set 1,1,-0.2

20.4.5.31 OUTPut:SUBBand:OFFSet:GAIN:LIMits?

Get output sub-band gain offset limits (emulation bandwidth > 200 MHz)

Syntax:

OUTPut:SUBBand:OFFSet:GAIN:LIMits? <output number>,<sub-band number>

Description:

This command retrieves the output sub-band gain offset limits in dB. The returned values are as follows:

<lower limit>,<higher limit>

Note: *The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).*

Example:

// get sub-band 1 gain offset limits on output 1

outp:subband:offset:gain:limits? 1,1

-3,0

20.4.5.32 OUTPut:SUBBand:OFFSet:PHASE:GET?

Get output sub-band phase offset (emulation bandwidth > 200 MHz)

Syntax:

OUTPut:SUBBand:OFFSet:PHASE:GET? <output number>,<sub-band number>

Description:

This command retrieves the output sub-band phase offset in degrees.

Note: *The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).*

Example:

// get sub-band 1 phase offset on output 1

outp:subband:offset:phase:get? 1,1

12.5

20.4.5.33 OUTPut:SUBBand:OFFSet:PHASE:SET

Set output sub-band phase offset (emulation bandwidth > 200 MHz)

Syntax:

OUTPut:SUBBand:OFFSet:PHASE:SET <output number>,<sub-band number>,<phase>

Description:

This command sets the output sub-band phase offset in degrees.

Note: *The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).*

Example:

// set sub-band 1 phase offset on output 1 to 12.5 deg

outp:subband:offset:phase:set 1,1,12.5

20.4.5.34 OUTPut:SUBBand:OFFSet:PHASE:LIMits?

Get output sub-band phase offset limits (emulation bandwidth > 200 MHz)

Syntax:

OUTPut:SUBBand:OFFSet:PHASE:LIMits? <output number>,<sub-band number>

Description:

This command retrieves the output sub-band phase offset limits in degrees. The returned values are as follows:

<lower limit>,<higher limit>

Note: *The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).*

Example:

// get sub-band 1 phase offset limits on output 1

outp:subband:offset:phase:limits? 1,1

-200,200

20.4.5.35 OUTPut:SUBBand:OFFSet:DELAY:GET?

Get output sub-band delay offset (emulation bandwidth > 200 MHz)

Syntax:

OUTPut:SUBBand:OFFSet:DELAY:GET? <output number>,<sub-band number>

Description:

This command retrieves the output sub-band delay offset in nano seconds.

Note: *The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).*

Example:

// get sub-band 1 delay offset on output 1

outp:subband:offset:delay:get? 1,1

-0.45

20.4.5.36 OUTPut:SUBBand:OFFSet:DELAY:SET

Set output sub-band delay offset (emulation bandwidth > 200 MHz)

Syntax:

OUTPut:SUBBand:OFFSet:DELAY:SET <output number>,<sub-band number>,<delay>

Description:

This command sets the output sub-band delay offset in nano seconds.

Note: *The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).*

Example:

// set sub-band 1 delay offset on output 1 to -0.45 ns

outp:subband:offset:delay:set 1,1,-0.45

20.4.5.37 OUTPut:SUBBand:OFFSet:DELAY:LIMits?

Get output sub-band delay offset limits (emulation bandwidth > 200 MHz)

Syntax:

OUTPut:SUBBand:OFFSet:DELAY:LIMits? <output number>,<sub-band number>

Description:

This command retrieves the output sub-band delay offset limits in nano seconds. The returned values are as follows:

<lower limit>,<higher limit>

Note: The sub-band indexing parameter starts from '0'. This means the sub-band indexing parameter range is 0 ... (number of sub-bands – 1).

Example:

// get sub-band 1 delay offset limits on output 1

outp:subband:offset:delay:limits? 1,1

-1,1

20.4.5.38 OUTPut:FREQuency:SET

Set output frequency (frequency conversion purposes).

Syntax:

OUTPut:FREQuency:SET <output number>,<frequency in MHz>

Description:

This command enables frequency conversion and sets output frequency. Frequency conversion setting must be enabled to use this command.

Example:

// Set output 2 frequency to 1000 MHz

outp:freq:set 2,1000

20.4.5.39 OUTPut:FREQuency:GET?

Get output frequency.

Syntax:

OUTPut:FREQuency:GET? <output number>

Description:

This command retrieves the output frequency.

Example:

// Get output 2 frequency. 1000 MHz.

outp:freq:get? 2
1000

20.4.5.40 OUTPut:FREQuency:LIMits?

Get output frequency limits.

Syntax:

OUTPut:FREQuency:LIMits? <output number>

Description:

This command retrieves the output frequency limits.

Example:

// Get output 2 frequency limits. 450 MHz – 6000 MHz.

outp:freq:lim? 2
450,6000

20.4.5.41 OUTPut:DELAY:CH

Set channel output delay offset

Syntax:

OUTPut:DELAY:CH <output number>,<delay offset in ns>

Description:

This command sets the delay offset of the specific channel output.

Example:

// Set channel output 2 delay offset to 200ps.

inp:delay:ch 2,0.200

20.4.5.42 OUTPut:DELAY:CH?

Get channel output delay offset

Syntax:

OUTPut:DELAY:CH? <output number>

Description:

This request returns the delay offset of the specific channel output.

Example:

// Get channel output 1 delay offset.

inp:delay:ch? 1

1.1

20.4.5.43 OUTPut:DELAY:LIMits?

Get channel output delay offset limits

Syntax:

OUTPut:DELAY:LIMits? <output number>

Description:

This command retrieves the delay offset limit values of the specific channel output. Delay offset cannot be set outside the limits. If attempted, value is automatically set to closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get delay offset value limits of channel output 6:

inp:delay:lim? 6

-10,50

20.4.6 Channel Settings

20.4.6.1 CALCulate:FILTer:CENTer:CH

Set center frequency for channel group

Syntax:

CALCulate:FILTer:CENTer:CH <channel number>,<frequency>

Description:

This command sets the center frequency of the specific channel group in MHz. It should be noted that frequency is set according to the given channel number. Frequency is set for given channel and for all the other channels belonging to the same group. Channels belong to same group if at least one of the following is true:

- They have same input
- They have same output

Example:

// Set center frequency of channel 1 (and all the channels

// belonging to same channel group) to 900 MHz.

calc:filt:cent:ch 1,900

20.4.6.2 CALCulate:FILTer:CENTer:CH?

Get center frequency of channel group

Syntax:

CALCulate:FILTer:CENTer:CH? <channel number>

Description:

This request retrieves the center frequency of the specific channel group in MHz. It should be noted that channel group is specified according to the given channel number. Frequency is retrieved from the channel group the channel belongs to. Channels belong to the same group if at least one of the following is true:

- They have the same input
- They have the same output

Example:

// Get center frequency of the channel group of channel 1

calc:filt:cent:ch? 1

1800

20.4.6.3 CALCulate:FILTer:CENTer:LOCKed?

Get locking status of center frequency of channel group

Syntax:

CALCulate:FILTer:CENTer:LOCKed? <channel number>

Description:

This request retrieves locking status of the center frequency of the specific channel group. It should be noted that channel group is specified according to the given channel number. Result can be 1 (locked) or 0 (not locked).

Example:

// Get locking status of center frequency of the channel group of channel 1

calc:filt:cent:lock? 1

0

20.4.6.4 CALCulate:FILTer:CENTer:VIRTual:CH

Set virtual center frequency for channel group

Syntax:

CALCulate:FILTer:CENTer:VIRTual:CH <channel number>,<frequency>

Description:

This command sets the virtual center frequency of the specific channel group in MHz. It should be noted that frequency is set according to the given channel number. Frequency is set for given channel and for all the other channels belonging to the same group. Channels belong to same group if at least one of the following is true:

- They have the same input
- They have the same output

Example:

// Set virtual center frequency of channel 1 (and all the channels

// belonging to same channel group) to 900 MHz.

calc:filt:cent:virt:ch 1,900

20.4.6.5 CALCulate:FILTer:CENTer:VIRTual:CH?

Get virtual center frequency of channel group

Syntax:

CALCulate:FILTer:CENTer:VIRTual:CH? <channel number>

Description:

This request retrieves the virtual center frequency of the specific channel group in MHz. It should be noted that channel group is specified according to the given channel number. Frequency is retrieved from the channel group the channel belongs to. Channels belong to the same group if at least one of the following is true:

- They have the same input
- They have the same output

Example:

// Get virtual center frequency of the channel group of channel 1

calc:filt:cent:virt:ch? 1

1800

20.4.6.6 CALCulate:FILTer:CENTer:VIRTual:ENable

Enable/disable virtual center frequency for channel group.

Syntax:

CALCulate:FILTer:CENTer:VIRTual:ENable <channel number>, <0/1 Disable/Enable>

Description:

Enables or disables the virtual center frequency of the specific channel group.

- 0, Virtual center frequency disabled
- 1, Virtual center frequency enabled

It should be noted that channel group is specified according to the given channel number. State is set for the channel group the channel belongs to. Channels belong to the same group if at least one of the following is true:

- They have the same input
- They have the same output

Example:

// Enable virtual center frequency of channel 1 (and all the channels belonging to same channel group).

calc:filt:cent:virt:en 1,1

20.4.6.7 CALCulate:FILTer:CENTer:VIRTual:ENable?

Requests the state of virtual center frequency of channel group.

Syntax:

CALCulate:FILTer:CENTer:VIRTual:ENable? <channel number>

Description:

Queries the state of virtual center frequency of the specific channel group. Possible return values are:

- 0, Virtual center frequency disabled
- 1, Virtual center frequency enabled

It should be noted that channel group is specified according to the given channel number. State is retrieved from the channel group the channel belongs to. Channels belong to the same group if at least one of the following is true:

- They have the same input
- They have the same output

Example:

// See if virtual center frequency enabled for the channel group of channel 1 calc:filt:cent:virt:en? 1

1

20.4.6.8 CALCulate:FILTer:CENTer:LIMits?

Get center frequency limits for group

Syntax:

CALCulate:FILTer:CENTer:LIMits? <channel number>

Description:

This retrieves the center frequency limits of the specific channel group in MHz. Frequency can not be set outside the limits. If tried, frequency is automatically set to closest acceptable value.

It should be noted that channel group is specified according to the given channel number. Limits are retrieved from the channel group that the channel belongs to. Channels belong to the same group if at least one of the following is true:

- They have the same input
- They have the same output

Limit values are returned as follows:

<lower limit>,<higher limit>

Note: If the channel group has multiple center frequency ranges, the limits of each range are separated with a semicolon: <lower limit>,<higher limit>;<lower limit>,<higher limit>;...

Example:

// Get center frequency limits of the channel group of channel 1

calc:filt:cent:lim? 1

350,6000

20.4.6.9 DIAGnostic:SIMUlation:FIRUPDatespeed:MANual:CH

Set cir update rate

Syntax:

DIAGnostic:SIMUlation:FIRUPDatespeed:MANual:CH <channel number>,<cir update rate>

Description:

This command sets the channel impulse response update rate of the specific channel in 1/s.

Example:

// Set cir update rate of channel 1 to 1000.

diag:simu:firupd:man:ch 1,1000

20.4.6.10 DIAGnostic:SIMUlation:FIRUPDatespeed:CH?

Get cir update rate

Syntax:

DIAGnostic:SIMUlation:FIRUPDatespeed:CH? <channel number>

Description:

This request retrieves the channel impulse response update rate of the specific channel in 1/s.

Example:

// Get cir update rate of channel 8.

diag:simu:firupd:ch? 8

135000

20.4.6.11 DIAGnostic:SIMUlation:FIRUPDatespeed:LIMits?

Get cir update rate limits

Syntax:

DIAGnostic:SIMUlation:FIRUPDatespeed:LIMits? <channel number>

Description:

This request retrieves the channel impulse response update rate limit values of the specific channel in 1/s. Update rate cannot be set outside the limits. If attempted, rate is automatically set to closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get cir update rate limits of channel 1.

diag:simu:firupd:lim? 1

0.01,3.90533e+006

20.4.6.12 DIAGnostic:SIMUlation:FIRUPDatespeed:LOCKed?

Query whether cir update rate is locked

Syntax:

DIAGnostic:SIMUlation:FIRUPDatespeed:LOCKed? <channel number>

Description:

This request checks whether the channel impulse response update rate is locked (cannot be changed) in the specific channel. Possible return values are:

0, Cir update rate not locked

1, Cir update rate locked

Example:

// Check if cir update rate is locked in channel 1.

diag:simu:firupd:lock? 1

0

20.4.6.13 DIAGnostic:SIMUlation:MOBilespeed:MANual:CH

Set mobile speed

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:MANual:CH <channel number>,<mobile speed> <possible unit description>

Description:

This command sets the mobile speed of the specific channel. If there is no unit description after the mobile speed setting value the speed is set as km/h. It is also possible to define the speed as m/s or alternatively as maximum Doppler (Hz) by using following unit descriptions. Note that unit descriptions are case sensitive.

MS

M/S

m/s

Hz

Example:

// Set mobile speed of channel 1 to 80 m/s.

diag:simu:mob:man:ch 1,80 m/s

20.4.6.14 DIAGnostic:SIMUlation:MOBilespeed:MANual:CHG

Set mobile speed for a channel group

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:MANual:CHG <channel number>,<mobile speed> <possible unit description>

Description:

This command sets the mobile speed for a channel group containing the given channel. A channel group consists of channels which share common components. For example two channels with a same output create one channel group.

If there is no unit description after the mobile speed setting value the speed is set as km/h. It is also possible to define the speed as m/s or alternatively as maximum Doppler (Hz) by using following unit descriptions. Note that unit descriptions are case sensitive.

MS

M/S

m/s

Hz

Example:

// Set mobile speed 80 m/s for a channel group containing channel 1.

diag:simu:mob:man:chg 1,80 m/s

20.4.6.15 DIAGnostic:SIMUlation:MOBilespeed:CH?

Get mobile speed in km/h

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:CH? <channel number>

Description:

This request retrieves the mobile speed of the specific channel in km/h.

Example:

// Get mobile speed of channel 8.

diag:simu:mob:ch? 8

200

20.4.6.16 DIAGnostic:SIMUlation:MOBilespeed:LIMits?

Get mobile speed limits in km/h

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:LIMits? <channel number>

Description:

This request retrieves the mobile speed limit values of the specific channel in km/h. Mobile speed cannot be set outside the limits. If tried, speed is automatically set to closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get mobile speed limits of channel 1.

diag:simu:mob:lim? 1

0.00229954,14967.4

20.4.6.17 DIAGnostic:SIMUlation:MOBilespeed:LOCKed?

Is mobile speed locked?

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:LOCKed? <channel number>

Description:

This request checks whether the mobile speed is locked (i.e. cannot be changed) in the specific channel. Possible return values are:

0, Mobile speed not locked

1, Mobile speed locked

Example:

// Check if mobile speed is locked in channel 1.

diag:simu:mob:lock? 1

0

20.4.6.18 DIAGnostic:SIMUlation:MOBilespeed:VALid?

Is mobile speed valid?

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:VALid? <channel number>

Description:

This request checks whether the mobile speed is valid in the specific channel i.e. sample density of the model is not 0. Possible return values are:

0, Mobile speed not valid

1, Mobile speed is valid

Example:

// Check if mobile speed is valid in channel 1.

diag:simu:mob:val? 1

1

20.4.6.19 DIAGnostic:SIMUlation:MOBilespeed:MPS:CH?

Get mobile speed in m/s

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:MPS:CH? <channel number>

Description:

This request retrieves the mobile speed of the specific channel in m/s.

Example:

//Get mobile speed of channel 1.

diag:simu:mob:mps:ch? 1

158.094

20.4.6.20 DIAGnostic:SIMUlation:MOBilespeed:MPS:LIMits?

Get mobile speed limits in m/s

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:MPS:LIMits? <channel number>

Description:

This request retrieves the mobile speed limit values of the specific channel in m/s. Mobile speed cannot be set outside the limits. If tried, speed is automatically set to closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get mobile speed limits of channel 5.

diag:simu:mob:mps:lim? 5

0.000638762,4573.39

20.4.6.21 DIAGnostic:SIMUlation:MOBilespeed:FILE:SOURCE?

Get mobile speed profile file name

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:FILE:SOURCE? <channel number>

Description:

This command retrieves the mobile speed profile file name of the specific channel.

Example:

// Get the profile file name of channel 2

diag:simu:mob:file:source? 2

e:\SpeedProfile_2.spf

20.4.6.22 DIAGnostic:SIMUlation:MOBilespeed:FACTOR:SET

Set mobile speed factor

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:FACTOR:SET <speed factor>

Description:

This command sets the mobile speed factor for the emulation.

Example:

// Set the speed factor

diag:simu:mob:factor:set 1.5

20.4.6.23 DIAGnostic:SIMUlation:MOBilespeed:FACTOR:GET?

Get mobile speed factor

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:FACTOR:GET?

Description:

This command retrieves the mobile speed factor for the emulation.

Example:

// Get the emulation mobile speed factor

diag:simu:mob:factor:get?

1.5

20.4.6.24 DIAGnostic:SIMUlation:MOBilespeed:FACTOR:LIMits?

Get mobile speed factor limits

Syntax:

DIAGnostic:SIMUlation:MOBilespeed:FACTOR:LIMITS?

Description:

This command retrieves the mobile speed factor limits for the emulation.

Example:

// Get the speed factor limits

diag:simu:mob:factor:lim?

1,10

20.4.6.25 DIAGnostic:SIMUlation:MODEL:STATIC

Set static model state (Disabled / Channel model bypass / Butler bypass / Calibration bypass). When static model is enabled, emulation is paused and all the channel models in the emulation are replaced with 1 path constant model (bypass). Attenuation, phase and delay depend on the selected bypass mode. Modes and details are listed below. Please refer to chapter 4.3.2 for more details about bypass functionality.

Syntax:

DIAGnostic:SIMUlation:MODEL:STATIC <state>

Description:

This request sets the static state of the emulation, i.e. channel model bypass, specified by <state>. Possible states are:

0, Bypass disabled

1, Channel model bypass enabled

2, Butler bypass enabled

3, Calibration bypass enabled

Bypass disabled

Bypass is disabled and fading from channel model is applied. Emulation continues to run, if it was running before entering bypass (static state).

Channel model bypass:

Channel attenuation is the average attenuation of the fading model.
Channel delay is the minimum path delay of the fading model on current model position.
Channel phase is zero.

Butler bypass:

Channel attenuation is the average attenuation of the fading model.
Channel delay is the minimum path delay of the fading model on current model position.
Channel phase is calculated with Butler Matrix and depends on the channel topology (MISO/SIMO/MIMO)

Calibration bypass:

Channel attenuation is identical for all channels in the emulation.
Channel delay is identical for all channels in the emulation.
Channel phase is zero.

Example:

// Set static state to Butler bypass

diag:simu:model:static 2

20.4.6.26 DIAGnostic:SIMUlation:MODEL:STATIC?

Get current static model state (Disabled / Bypass / Butler bypass / Calibration bypass)

Syntax:

DIAGnostic:SIMUlation:MODEL:STATIC?

Description:

This request retrieves the static state of the emulation, i.e. channel model bypass mode. Return values are:

0, Bypass disabled (channel model applied)

1, Channel model bypass enabled

2, Butler bypass enabled

3, Calibration bypass enabled

Example:

// Query static state

diag:simu:model:static?

1

20.4.7 Channel Group Information

20.4.7.1 GROup:GET?

Get number of channel groups in emulation.

Syntax:

GROup:GET?

Description:

This command retrieves the number of channel groups in emulation.

Example:

// Get the number of channel groups

group:get?

2

20.4.7.2 GROup:INputs:GET?

Get inputs of a channel group.

Syntax:

GROup:INPUTS:GET? <group number>

Description:

This command retrieves the list of inputs in a channel group.

Example:

// Get inputs of group 1

group:inputs:get? 1

1,2

20.4.7.3 GROup:OUTputs:GET?

Get outputs of a channel group.

Syntax:

GROup:OUTputs:GET? <group number>

Description:

This command retrieves the list of outputs is a channel group.

Example:

// Get outputs of group 1

group:outputs:get? 1

1,2

20.4.7.4 GROup:CHannels:GET?

Get channels of a channel group.

Syntax:

GROup:CHannels:GET? <group number>

Description:

This command retrieves the list of channels in a channel group.

Example:

// Get channels of group 1

group:channels:get? 1

1,2,3,4

20.4.7.5 GROup:NAME?

Get name of the channel group.

Syntax:

GROup:NAME? <group number>

Description:

This command retrieves the name of a channel group.

Example:

// Get name of group 1

group:name? 1

downlink

20.4.7.6 GROup:CLIpping:GET?

Get amount of digital clipping in channel group. For more details, see chapter 4.3.3.3 (Digital clipping).

Syntax:

GROup:CLIpping:GET? <group number>,<reset>

Description:

This command retrieves the average amount of digital clipping in per mill in channel group. Possible reset parameter values are:

0, Clipping value average calculation is not reset

1, Clipping value average calculation is reset

Example:

// Get clipping per mill value in group 1, do not reset average calculation

group:clipping:get? 1

28.75

20.4.7.7 GROup:BANDWidth:GET?

Get bandwidth of the channel group in MHz.

Syntax:

GROup:BANDWidth:GET? <group number>

Description:

This command retrieves the group bandwidth in MHz.

Example:

// Get bandwidth of group 1 (example here 80 MHz)

group:bandw:get? 1

80

20.4.7.8 GROup:BANDWidth:SET

Set bandwidth of the channel group in MHz.

Syntax:

GROup:BANDWidth:SET <group number>,<bandwidth>

Description:

This command set the group bandwidth in MHz.

Example:

// Set bandwidth of group 1 (example here 80 MHz)

group:bandw:set 1,80

20.4.8 Shadowing settings and information

20.4.8.1 GROup:SHADowing:ENable:CH

Set the current shadowing state for the channel group where the specified channel belongs.

Syntax:

GROup:SHADowing:ENable:CH <channel number>,<enable>

Description:

This command enables/disables the shadowing of the channel group where the specified channel belongs.

Possible parameter values for enable are:

0, Shadowing disabled

1, Shadowing enabled

Example:

// Set shadowing off for channel group containing channel 5

GROup:SHADowing:ENable:CH 5,0

20.4.8.2 GROup:SHADowing:ENable:CH?

Get the current shadowing state for the channel group where the specified channel belongs.

Syntax:

GROup:SHADowing:ENable:CH? <channel number>

Description:

This command gets the current shadowing state for the channel group where the specified channel belongs.

Return values are:

0, Shadowing disabled

1, Shadowing enabled

2, Links inside the group have different shadowing states

Example:

// Get shadowing state for the channel group containing channel 5, shadowing is enabled

GROup:SHADowing:ENable:CH? 5

1

20.4.8.3 GROup:SHADowing:OFFSET:CH

Set the shadowing offset for the channel group where the specified channel belongs to.

Syntax:

GROup:SHADowing:OFFSET:CH <channel number>,<offset>

Description:

This command sets the shadowing offset for the channel group where the specified channel belongs to.

This command can be used only when output shadowing is in use.

Example:

// Set shadowing offset for a channel group containing channel 5

GROup:SHADowing:OFFSET:CH 5,5.0

20.4.8.4 GROup:SHADowing:OFFSET:CH?

Get shadowing offset for the channel group where the specified channel belongs to.

Syntax:

GROup:SHADowing:OFFSET:CH? <channel number>

Description:

This command gets the shadowing offset (in dB) for the channel group where the specified channel belongs to.

Shadowing offset value is returned if it is the same for all channels in the channel group. If values vary between channels, return value is 'diff'.

Example:

// Get shadowing offset for the channel group containing channel 5

GROup:SHADowing:OFFSET:CH? 5

5.0

20.4.8.5 LINK:SHADowing:ENable:CH

Set the current shadowing state for the link where the specified channel belongs.

Syntax:

LINK:SHADowing:ENable:CH <channel number>,<enable>

Description:

This command enables/disables the shadowing of the link where the specified channel belongs.

Possible parameter values for enable are:

0, Shadowing disabled

1, Shadowing enabled

Example:

// Set shadowing off for the link containing channel 5

LINK:SHADowing:ENable:CH 5,0

20.4.8.6 LINK:SHADowing:ENable:CH?

Get the current shadowing state for the link where the specified channel belongs.

Syntax:

This command gets the current shadowing state for the link where the specified channel belongs.

LINK:SHADowing:ENable:CH? <channel number>

Description:

Return values are:

0, Shadowing disabled

1, Shadowing enabled

Example:

// Get shadowing enable state for the link containing channel 5, shadowing is on

LINK:SHADowing:ENable:CH? 5

1

20.4.8.7 LINK:SHADowing:OFFSET:CH

Set the shadowing offset for the link where the specified channel belongs to.

Syntax:

LINK:SHADowing:OFFSET:CH <channel number>,<offset>

Description:

This command sets the shadowing offset for the link where the specified channel belongs to.

This command can be used only when output shadowing is in use.

Example:

// Set shadowing offset for a link containing channel 5

LINK:SHADowing:OFFSET:CH 5,5.0

20.4.8.8 LINK:SHADowing:OFFSET:CH?

Get the shadowing offset for the link where the specified channel belongs to.

Syntax:

LINK:SHADowing:OFFSET:CH? <channel number>

Description:

This command gets the shadowing offset (in dB) for the link where the specified channel belongs to.

Shadowing offset value is returned if it is the same for all channels in the link. If values vary between channels, return value is 'diff'.

Example:

// Get shadowing offset for the link containing channel 5

LINK:SHADowing:OFFSET:CH? 5

5.0

20.4.8.9 OUTPut:SHADowing:GAIN:GET?

Get the current shadowing gain on an output.

Syntax:

OUTPut:SHADowing:GAIN:GET? <output number>

Description:

Get the current shadowing gain on an output in dB.

Example:

// Query the last shadowing gain on output 1. It is -5.0 dB.

outp:shad:gain:get? 1

-5.0

20.4.8.10 OUTPut:SHADowing:OFFSET:SET

Set the current shadowing offset on an output.

Syntax:

OUTPut:SHADowing:OFFSET:SET <output number>, <offset>

Description:

Set the current shadowing offset on an output in dB.

Example:

// Set shadowing offset on output 1 to 5.0 dB.

outp:shad:offset:set 1,5.0

20.4.8.11 OUTPut:SHADowing:OFFSET:GET?

Get the current shadowing offset on an output.

Syntax:

OUTPut:SHADowing:OFFSET:GET? <output number>

Description:

Get the current shadowing offset on an output in dB.

Example:

// Query the last shadowing offset on output 1. It is 5.1 dB.

outp:shad:offset:get? 1

5.1

20.4.8.12 CHannel:SHADowing:GAIN:GET?

Get the shadowing gain on a channel.

Syntax:

CHannel:SHADowing:GAIN:GET? <channel number>,<emulation time>

Description:

Get the shadowing gain on a channel at any emulation time in dB. If “emulation time” parameter is not defined, shadowing gain from current emulation position is returned.

Example:

// Query the current shadowing gain on channel 1. It is -5.0 dB.

ch:shad:gain:get? 1

-5.0

// Query the shadowing gain on channel 2 in emulation time 23.5 s. Gain is -23.0 dB.

ch:shad:gain:get? 2,23.5

-23.0

20.4.8.13 CHannel:SHADowing:POSition:GET?

Get the current shadowing position on a channel.

Syntax:

CHannel:SHADowing:POSition:GET? <channel number>

Description:

Get the current shadowing position on a channel. The reply format is as follows

position,iterations,length

“position” shadowing position in current iteration (seconds)

“iterations” number of times shadowing profile has wrapped around

“length” shadowing profile length (seconds)

Example:

// Query the current shadowing position on channel 1. (2.4 seconds,four iterations,profile length 10 seconds)

ch:shad:pos:get? 1

2.4,4,10

20.4.8.14 CHannel:SHADowing:POSition:MAXGAIN:GET?

Get the shadowing position on a channel where the shadowing gain is at maximum.

Syntax:

CHannel:SHADowing:POSition:MAXGAIN:GET? <channel number>

Description:

Get the shadowing position on a channel as seconds, where the shadowing gain is at maximum. If the specified channel does not have shadowing “NO SHADOWING” is returned.

Example:

// Query the shadowing position where the shadowing gain is on it’s maximum on channel 3. (4.34 seconds) ch:shad:pos:maxgain:get? 3

4.34

20.4.8.15 CHannel:SHADowing:OFFSET:GET?

Get the current shadowing offset on a channel.

Syntax:

CHannel:SHADowing:OFFSET:GET? <channel number>

Description:

Get the current shadowing offset on a channel.

Example:

// Query the current shadowing offset on channel 1. (6.5 dB)

ch:shad:offset:get? 1

6.5

20.4.9 Internal Interference Generator control (Optional)

Internal Interference Generator commands are available only if PROPSIM has optional interference sources installed. AWGN and CW are available.

20.4.9.1 OUTPut:INTERFerence:ADD

Add an interference source to the channel output.

Syntax:

OUTPut:INTERFerence:ADD <output number>,<interference identification>,<interference type>

Description:

This command adds an interference source to specific channel output. Interference identification is user-given identifier to the interference; either a plain number (012, leading 0 is discarded) or a string (such as TX2_INTERF). Strings are case-sensitive (i.e. tx1_noise and TX1_NOISE are not same identifiers). Each identifier is unique, identifiers already in use cannot be given. Possible settings for the interference type are:

1, AWGN

2, Carrier Wave (CW)

Example:

// Add an AWGN noise source to output 2.

outp:interf:add 2,tx2_interf,1

20.4.9.2 OUTPut:INTERFerence:REMove

Remove the interference source from output.

Syntax:

OUTPut:INTERFerence:REMove <interference identification>

Description:

This command removes the interference source from the channel output. The removed interference is defined by the interference identification (see 20.4.9.1).

Example:

// Remove the AWGN noise source with identification tx2_interf.

outp:interf:rem tx2_interf

20.4.9.3 OUTPut:INTERFerence:STatus

Set the status of the specified interference source.

Syntax:

OUTPut:INTERFerence:STatus <interference identification>,<interference status>

Description:

This command sets the status of the interference source. The interference, which status is being set is defined by the interference identification (see 20.4.9.1). Possible status settings are:

0, Interference disabled

1, Interference enabled

Example:

// Disable the AWGN noise source with identification tx2_noise.

outp:interf:st tx2_noise,0

20.4.9.4 OUTPut:INTERFerence:STatus?

Get the status of the specified interference source.

Syntax:

OUTPut:INTERFerence:STatus? <interference identification>

Description:

This command retrieves the status of the interference source defined by the interference identification. Possible return values are:

0, Interference disabled

1, Interference enabled

Example:

// Get the status of the AWGN noise source with identification tx2_noise.

outp:interf:st? tx2_noise

1

20.4.9.5 OUTPut:INTERFerence:GET?

List all the interferers in emulation.

Syntax:

OUTPut:INTERFerence:GET?

Description:

This command gets and lists all the interference sources. The returned list is in the following form "output number, interference identification, type of the interferer, output number, interference identification, type of the interference..." where "interference identification" is the user defined interference identification given by the user when the interference has been added (see chapter 20.4.9.1) and "type of the interference" tells the type of the interferer in numeric. Possible values for the interference type are:

1, AWGN

2, Carrier Wave (CW)

Zero is returned if there are no interference sources in use.

Example:

// Get the status of the interference sources. Output 1 has AWGN type of interference with identification TX1_AWGN and output 2 CW type or interference with identification TX2_CW.

outp:interf:get?

1, TX1_AWGN, 7, 2, TX2_CW, 2

20.4.9.6 OUTPut:INTERFerence:STRATegy:SET

Set interference adjustment strategy for specified interference.

Syntax:

OUTPut:INTERFerence:STRATegy:SET <interference identification>, <adjustment strategy>

Description:

This command sets the adjustment strategy of the specified interference source. Possible settings are:

0, Constant carrier-to-interference ratio (C/I)

1, Constant interference power

2, Constant C/I where the interference level is also user-given and fixed (AWGN only).

Example:

```
// Choose constant carrier-to-interference ratio to
// interference signal with identification tx1_cw.
outp:interf:strat:set tx1_cw,0
```

20.4.9.7 OUTPut:INTERFerence:STRATegy:GET?

Get the interference adjustment strategy of the specified interference.

Syntax:

OUTPut:INTERFerence:STRATegy:GET? <interference identification>

Description:

This request retrieves the adjustment strategy of the specified interference source. Possible return values are:

0, Constant carrier-to-interference ratio (C/I)

1, Constant interference power

2, Constant C/I where the interference level is also user-given and fixed (AWGN only)

Example:

```
// Get the chosen interference adjustmentstrategy of the
// interference with identification tx1_cw
outp:interf:strat:get? tx1_cw
1
```

20.4.9.8 OUTPut:INTERFerence:POWer:SET

Set the power level of a specific interference signal.

Syntax:

OUTPut:INTERFerence:POWer:SET <interference identification>,<power level>

Description:

This command sets the output power level of a specified interference signal in dBm.

Example:

```
// Set power level of interference with identification
// tx1_cw to -30 dBm
outp:interf:pow:set tx1_cw,-30
```

20.4.9.9 OUTPut:INTERFerence:POWer:GET?

Get the power level of the specified interference signal.

Syntax:

OUTPut:INTERFerence:POWer:GET? <interference identification>

Description:

This request retrieves the output power level of the specified interference signal in dBm.

Example:

```
// Get the power level of the interference signal with
// identification tx1_cw.
outp:interf:pow:get? tx1_cw
```

-30

20.4.9.10 OUTPut:INTERFerence:POWer:LIMits?

Get the limits of the specified interference signal power at output.

Syntax:

OUTPut:INTERFerence:POWer:LIMits? <interference identification>

Description:

This command retrieves power level limits of the specified interference signal in dBm. The power level cannot be set outside these limits. If this is attempted the value is automatically set to the closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get power level limits of the interference signal

// with identification tx1_cw.

outp:interf:pow:lim? tx1_cw

-50,-15

20.4.9.11 OUTPut:INTERFerence:BANDWidth:SET

Set the bandwidth of the specified interference (AWGN) at output.

Syntax:

OUTPut:INTERFerence:BANDWidth:SET <interference identification>,<bandwidth>

Description:

This command sets the AWGN interference bandwidth at channel output in MHz.

Example:

// Set noise bandwidth to 30 MHz for the interference with

// identification tx1_awgn

outp:interf:bandw:set tx1_awgn,30

20.4.9.12 OUTPut:INTERFerence:BANDWidth:GET?

Get the bandwidth of the interferer (AWGN).

Syntax:

OUTPut:INTERFerence:BANDWidth:GET? <interference identification>

Description:

This request retrieves the AWGN bandwidth setting of the specified interference signal in MHz.

Example:

// Get the bandwidth of the interference signal with

// identification tx1_awgn.

outp:interf:band:get? tx1_awgn

3.84

20.4.9.13 OUTPut:INTERFerence:BANDWidth:LIMits?

Get the bandwidth limits of the specified interference.

Syntax:

OUTPut:INTERFerence:BANDWidth:LIMits? <interference identification>

Description:

This command retrieves the AWGN bandwidth limit values of the specified interference signal in MHz. The bandwidth value cannot be set outside the se limits. If this is attempted the value is automatically set to the closest acceptable value.

The limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get the bandwidth limits of the interference signal

// with identification tx1_awgn

outp:interf:bandw:lim? tx1_awgn

0.02,70

20.4.9.14 OUTPut:INTERFerence:BANDWidth:GENerated:SET

Set the generated bandwidth of the specified interference (AWGN) at output.

Syntax:

OUTPut:INTERFerence:BANDWidth:GENerated:SET <interference identification>,<generated bandwidth>

Description:

This command sets the AWGN interference generated bandwidth at channel output in MHz.

Example:

// Set generated noise bandwidth to 5 MHz for the interference with

// identification tx1_adj_awgn

outp:interf:bandw:gen:set tx1_adj_awgn,5

20.4.9.15 OUTPut:INTERFerence:BANDWidth:GENerated:GET?

Get the generated bandwidth of the interferer (AWGN).

Syntax:

OUTPut:INTERFerence:BANDWidth:GENerated:GET? <interference identification>

Description:

This request retrieves the generated AWGN bandwidth setting of the specified interference signal in MHz.

Example:

// Get the bandwidth of the interference signal with identification tx1_adj_awgn.

outp:interf:band:gen:get? tx1_adj_awgn

3.84

20.4.9.16 OUTPut:INTERFerence:BANDWidth:GENerated:LIMits?

Get the generated bandwidth limits of the specified interference.

Syntax:

OUTPut:INTERFerence:BANDWidth:GENerated:LIMits? <interference identification>

Description:

This command retrieves the generated AWGN bandwidth limit values of the specified interference signal in MHz. The bandwidth value cannot be set outside the se limits. If this is attempted the value is automatically set to the closest acceptable value.

The limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get the generated bandwidth limits of the interference signal

// with identification tx1_adj_awgn

outp:interf:bandw:gen:lim? tx1_adj_awgn

0.015,20

20.4.9.17 OUTPut:INTERFerence:LEVel:SET

Set output level of a specific interferer (AWGN).

Syntax:

OUTPut:INTERFerence:LEVel:SET <interference identification>,<level>

Description:

This command sets the output level of the specified (AWGN) interference in dBm/Hz.

Example:

// Set the noise level of interferer tx1_awgn to -100dBm/Hz

outp:interf:lev:set tx1_awgn,-100

20.4.9.18 OUTPut:INTERFerence:LEVel:GET?

Get output level of a specific interferer (AWGN).

Syntax:

OUTPut:INTERFerence:LEVel:GET? <interference identification>

Description:

This request retrieves the output level of the specified (AWGN) interference in dBm/Hz.

Example:

// Get the noise level of interferer tx1_awgn

outp:interf:lev:get? tx1_awgn

-100

20.4.9.19 OUTPut:INTERFerence:LEVel:LIMits?

Get limits of the of a specific interferer (AWGN).

Syntax:

OUTPut:INTERFerence:LEVel:LIMits? <interference identification>

Description:

This command retrieves the limits of the specified interference (AWGN) in dBm/Hz. The power level value cannot be set outside these limits. If this is attempted the value is automatically set to the closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get the power level limits of the interferer

// tx1_awgn

outp:interf:level:lim? tx1_awgn

-150,-100

20.4.9.20 OUTPut:INTERFerence:DATARate:SET

Set the data rate in kbps of the information signal for Eb/N0 setting (AWGN)

Syntax:

OUTPut:INTERFerence:DATARate:SET <interference identification>,<data rate>

Description:

This command sets the data rate of the information signal in kbps to which the identified AWGN noise is summed. This is needed if the signal-to-noise ratio is defined as Eb/N0.

Example:

// Set the data rate of the information signal to

// 200kbps. Noise level is adjusted to meet the user's

// Eb/N0 setting.

outp:interf:datar:set tx1_awgn,200

20.4.9.21 OUTPut:INTERFerence:DATARate:GET?

Get the data rate in kbps of the information signal set by the user

Syntax:

OUTPut:INTERFerence:DATARate:GET? <interference identification>

Description:

This command retrieves the data rate of the information signal in kbps to which the identified AWGN noise is summed. This is needed if the signal-to-noise ratio is defined as Eb/N0.

Example:

// Get the user set data rate of the information

// signal. Noise level is adjusted to meet the user's

// Eb/N0 setting.

outp:interf:datar:get? tx1_awgn

200

20.4.9.22 OUTPut:INTERFerence:DATARate:LIMits?

Get the data rate in kbps limits of the information signal

Syntax:

OUTPut:INTERFerence:DATARate:LIMits? <interference identification>

Description:

This command retrieves the limits of the specified interferer in kbps. This is needed if the signal-to-noise ratio is defined as Eb/N0. The data rate value cannot be set outside these limits. If this is attempted the value is automatically set to the closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get the data rate limits of the information signal.

// Noise level is adjusted to meet the Eb/N0 setting.

outp:interf:datar:lim? tx1_awgn

1,5000

20.4.9.23 OUTPut:INTERFerence:EBN0:SET

Set the Eb/N0 value.

Syntax:

OUTPut:INTERFerence:EBN0:SET <interference identification>,<Eb/N0>

Description:

With this command the user is able to define the needed Eb/N0 value. The interference has to be AWGN noise.

Example:

// Set the Eb/N0 value to 5.4dB. The AWGN noise has been

// given identification tx1_awgn

outp:interf:ebn0:set tx1_awgn,5.4

20.4.9.24 OUTPut:INTERFerence:EBN0:GET?

Get the Eb/N0 value for interferer (AWGN).

Syntax:

OUTPut:INTERFerence:EBN0:GET? <interference identification>

Description:

This command gets the Eb/N0 value set to the specified AWGN noise.

Example:

// Get the Eb/N0 value related to AWGN noise with

// identification tx1_awgn

outp:interf:ebn0:get? tx1_awgn

5.4

20.4.9.25 OUTPut:INTERFerence:EBN0:LIMits?

Get the Eb/N0 limits for interferer (AWGN).

Syntax:

OUTPut:INTERFerence:EBN0:LIMits? <interference identification>

Description:

This command retrieves the limits of the Eb/N0 value for the specified AWGN noise. The Eb/N0 value cannot be set outside these limits. If this is attempted the value is automatically set to the closest acceptable value.

The limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get the Eb/N0 value limits related to AWGN noise with

// identification tx1_awgn.

outp:interf:ebn0:lim? tx1_awgn

0,47

20.4.9.26 OUTPut:INTERFerence:FREQuency:SET

Set the center frequency of the specific interferer.

Syntax:

OUTPut:INTERFerence:FREQuency:SET <interference identification>,<center frequency>

Description:

With this command the user is able to set the center frequency of the specified interference in MHz.

Example:

// Set the center frequency of interference tx1_cw

// to 1877.5 MHz

outp:interf:freq:set tx1_cw,1877.5

20.4.9.27 OUTPut:INTERFerence:FREQuency:GET?

Get the center frequency of the specific interferer.

Syntax:

OUTPut:INTERFerence:FREQuency:GET? <interference identification>

Description:

This request retrieves the center frequency of the specified interference signal in MHz.

Example:

// Get the center frequency of the interference

// tx1_cw.

outp:interf:freq:get? tx1_cw

2000

20.4.9.28 OUTPut:INTERFerence:FREQuency:LIMits?

Get the center frequency limits of the interferer.

Syntax:

OUTPut:INTERFerence:FREQuency:LIMits? <interference identification>

Description:

This command retrieves the center frequency limit values of the specified interference signal. The center frequency value cannot be set outside these limits. If this is attempted the value is automatically set to the closest acceptable value.

The limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get the center frequency limit values of the

// tx1_awgn interferer.

outp:interf:freq:lim? tx1_awgn

2170,2230

20.4.9.29 OUTPut:INTERFerence:FREQuency:OFFSet:SET

Set the frequency offset of the specific interferer.

Syntax:

OUTPut:INTERFerence:FREQuency:OFFSet:SET <interference identification>,< frequency offset>

Description:

With this command the user is able to set the frequency offset of the specified interference in MHz.

Example:

// Set the frequency offset of interference tx1_cw

// to 13.5 MHz

outp:interf:freq:offs:set tx1_cw,13.5

20.4.9.30 OUTPut:INTERFerence:FREQuency:OFFSet:GET?

Get the frequency offset of the specific interferer.

Syntax:

OUTPut:INTERFerence:FREQuency:OFFSet:GET? <interference identification>

Description:

This request retrieves the frequency offset of the specified interference signal in MHz.

Example:

// Get the frequency offset of the interference

// tx1_cw.

outp:interf:freq:offs:get? tx1_cw

10

20.4.9.31 OUTPut:INTERFerence:FREQuency:OFFSet:LIMits?

Get the frequency offset limits of the interferer.

Syntax:

OUTPut:INTERFerence:FREQuency:OFFSet:LIMits? <interference identification>

Description:

This command retrieves the frequency offset limit values of the specified interference signal. The frequency offset value cannot be set outside these limits. If this is attempted the value is automatically set to the closest acceptable value.

The limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get the frequency offset limit values of the

// tx1_adj_awgn interferer.

outp:interf:freq:offs:lim? tx1_adj_awgn

-30,30

20.4.9.32 OUTPut:INTERFerence:RATio:SET

Set the output carrier-to-interference ratio

Syntax:

OUTPut:INTERFerence:RATio:SET <interference identification>,<carrier-to-interference ratio>

Description:

This command sets the output carrier-to-interference or signal-to-interference ratio related to the specified interferer in dB.

Example:

// Set the carrier-to-interference ratio of the

// interferer tx1_awgn to -10dB

outp:interf:rat:set tx1_awgn,-10

20.4.9.33 OUTPut:INTERFerence:RATio:GET?

Get the output carrier-to-interference ratio

Syntax:

OUTPut:INTERFerence:RATio:GET? <interference identification>

Description:

This command gets the output carrier-to-interference or signal-to-interference ratio related to the specified interferer in dB.

Example:

// Get the carrier-to-interference ratio of the

// interferer tx1_awgn.

outp:interf:rat:get? tx1_awgn

-10

20.4.9.34 OUTPut:INTERFerence:RATio:LIMits?

Get the output carrier-to-interference ratio limits

Syntax:

OUTPut:INTERFerence:RATio:LIMits? <interference identification>

Description:

This command gets the output carrier-to-interference ratio limits related to the specified interferer in dB. The output carrier-to-interference ratio cannot be set outside these limits. If this is attempted the value is automatically set to the closest acceptable value.

Limit values are returned as follows:

<lower limit>,<higher limit>

Example:

// Get the carrier-to-interference ratio limits of the

// interferer tx1_awgn.

outp:interf:rat:lim? tx1_awgn

-20,20

20.4.9.35 OUTPut:INTERFerence:RATio:MODE:SET

Set measurement mode of interferer

Syntax:

OUTPut:INTERFerence:RATio:MODE:SET <interference identification>,<mode>

Description:

Set measurement mode of interferer for C/I ratio adjustment. Mode is:

0, Measure all input signals connected to output and adjust C/I ratio based on combined power of signals. Note that high correlation between input signals may distort measurement.

1, Measure only input signal of the first channel and adjust C/I ratio based on that

Example:

// Set measurement mode so that all input signals connected to

// output are measured

outp:interf:rat:mode:set tx1_awgn,0

20.4.9.36 OUTPut:INTERFerence:RATio:MODE:GET?

Get measurement mode of interferer

Syntax:

OUTPut:INTERFerence:RATio:MODE:GET? <interference identification>

Description:

Query returns measurement mode of specified interferer:

0, Measure all input signals connected to output and adjust C/I ratio based on combined power of signals. Note that high correlation between input signals may distort measurement.

1, Measure only input signal of the first channel and adjust C/I ratio based on that

Example:

// Get measurement mode

outp:interf:rat:mode:get? tx1_awgn

1

20.4.9.37 OUTPut:INTERFerence:FILE:PROFile?

Get interference profile file name

Syntax:

OUTPut:INTERFerence:FILE:PROFile? <interference identification>

Description:

This command retrieves the interference profile file name.

Example:

// Get the profile file name of the interference tx1_awgn

outp:interf:file:prof? tx1_awgn

e:\InterferenceProfile.shd

20.4.9.38 OUTPut:INTERFerence:POWer:PROFile?

Get the profile gain of the specified interference.

Syntax:

OUTPut:INTERFerence:POWer:PROFile? <interference identification>,<emulation time>

Description:

This request retrieves the interference profile gain at any emulation time in dB. If “emulation time” parameter is not defined, gain from current emulation position is returned.

Example:

// Get the current profile gain of the interference with

// identification tx1_cw.

outp:interf:pow:prof? tx1_cw

-30

20.4.9.39 OUTPut:INTERFerence:POWer:TOTal?

Get the current total power level of the specified interference.

Syntax:

OUTPut:INTERFerence:POWer:TOTal? <interference identification>

Description:

This request retrieves the current total power level of the specified interference in dBm.

Example:

// Get the total power level of the interference with

// identification tx1_cw.

outp:interf:pow:tot? tx1_cw

-30

20.4.9.40 OUTPut:INTERFerence:LEVel:PROFile?

Get current profile gain of a specific interferer (AWGN).

Syntax:

OUTPut:INTERFerence:LEVel:PROFile? <interference identification>,<emulation time>

Description:

This request retrieves the interference profile gain at any emulation time in dB. If “emulation time” parameter is not defined, gain from current emulation position is returned.

Example:

// Get the current profile gain of interferer tx1_awgn

outp:interf:lev:prof? tx1_awgn

-30

20.4.9.41 OUTPut:INTERFerence:LEVel:TOTal?

Get current total level of a specific interferer (AWGN).

Syntax:

OUTPut:INTERFerence:LEVel:TOTal? <interference identification>

Description:

This request retrieves the current total level of the specified (AWGN) interference in dBm/Hz.

Example:

// Get the current total level of interferer tx1_awgn

outp:interf:lev:tot? tx1_awgn

-100

20.4.10 Channel Model Information

20.4.10.1 CHannel:MODel:GAIN:MODel?

Get channel model gain of channel

Syntax:

CHannel:MODel:GAIN:MODel? <channel number>

Description:

This command retrieves the channel model gain of the specific channel in dB.

Example:

//Get channel model gain of channel 1.

ch:mod:gain:mod? 1

-19.8401

20.4.10.2 CHannel:MODel:GAIN:TOTal?

Get total gain of channel

Syntax:

CHannel:MODel:GAIN:TOTal? <channel number>

Description:

This command retrieves the total gain of the specific channel in dB. Total gain consists of channel model gain and input and output gain/level settings.

Example:

// Get total channel gain of channel 1.

ch:mod:gain:tot? 1

-15.8401

20.4.10.3 CHannel:MODel:GAIN:ADJust:SET

Adjust gain imbalance for channel

Syntax:

CHannel:MODel:GAIN:ADJust:SET <channel number>,<adjustment value>

Description:

This command can be used to adjust gain imbalance of the specific channel in dB.

Example:

// Set gain imbalance offset of channel 1 to -15 dB.

ch:mod:gain:adj:set 1,-15.0

20.4.10.4 CHannel:MODel:GAIN:ADJust:GET?

Get gain imbalance adjustment setting for channel

Syntax:

CHannel:MODel:GAIN:ADJust:GET? <channel number>

Description:

This command retrieves gain imbalance adjustment setting value of the specific channel in dB.

Example:

// Get imbalance adjustment setting of channel 1.

ch:mod:gain:adj:get? 1

-10

20.4.10.5 CHannel:MODel:GAIN:ADJust:LIMits?

Get gain imbalance adjustment setting limits for channel

Syntax:

CHannel:MODel:GAIN:ADJust:LIMits? <channel number>

Description:

This command retrieves gain imbalance adjustment setting limits of the specific channel in dB.

Example:

// Get imbalance adjustment setting limits of channel 1.

ch:mod:gain:adj:lim? 1

-100,5

20.4.10.6 CHannel:MODel:GAIN:ANAlog?

Get analog part of channel model gain.

Syntax:

CHannel:MODel:GAIN:ANAlog? <channel number>

Description:

This command retrieves the analog part of the channel model gain of the specific channel in dB. This command is useful only with models processed with "MultiEmulatorScaler" –tool or models created with "MIMO OTA" -tool. Other models return the value 0 dB.

Example:

// Get analog channel model gain of channel 1.

ch:mod:gain:ana? 1

-15.8401

20.4.10.7 CHannel:MODel:ENABLE

Enable or disable channel

Syntax:

CHannel:MODel:ENABLE <channel number>,<state>

Description:

This command either enables or disables of the specific channel.

1 channel enabled

0 channel disabled

Example:

// Disable channel 1.

ch:mod:enable 1,0

20.4.10.8 CHannel:MODel:ENABLE?

Query whether channel is enabled or disabled

Syntax:

CHannel:MODel:ENABLE? <channel number>

Description:

This command retrieves the state of specific channel.

1 channel enabled

0 channel disabled

Example:

// Get status of channel 1 (disabled)

ch:mod:enable? 1

0

20.4.10.9 CHannel:MODel:PHASE:ADJust:SET

Adjust phase imbalance for channel

Syntax:

CHannel:MODel:PHASE:ADJust:SET <channel number>,<adjustment value>

Description:

This command can be used to adjust phase imbalance of the specific channel in degrees.

Example:

// Adjust phase imbalance of channel 1.

ch:mod:phase:adj:set 1,25.0

20.4.10.10 CHannel:MODel:PHASE:ADJust:GET?

Get phase imbalance adjustment setting for channel

Syntax:

CHannel:MODel:PHASE:ADJust:GET? <channel number>

Description:

This command retrieves phase imbalance adjustment setting value of the specific channel in degrees.

Example:

// Get phase imbalance adjustment setting of channel 1.

ch:mod:phase:adj:get? 1

25.0

20.4.10.11 CHannel:MODel:PHASE:ADJust:LIMits?

Get phase imbalance adjustment setting limits for channel

Syntax:

CHannel:MODel:PHASE:ADJust:LIMits? <channel number>

Description:

This command retrieves phase imbalance adjustment setting limits of the specific channel in degrees.

Example:

// Get phase imbalance adjustment setting limits of channel 1.

ch:mod:gain:adj:lim? 1

-180,180

20.4.10.12 CHannel:MODel:FILE:CIR?

Get channel model control file name

Syntax:

CHannel:MODel:FILE:CIR? <channel number>

Description:

This command retrieves the channel model control file name (*.sim).

Example:

// Get the control file name of the channel model of channel 2.

ch:mod:file:cir? 2

e:\class_1_S4_T4.sim

20.4.10.13 CHannel:MODel:FILE:SOURCE?

Get channel model source file name

Syntax:

CHannel:MODel:FILE:SOURCE? <channel number>

Description:

This command retrieves the channel model source file name.

Example:

// Get the source file name of the channel model of channel 2.

ch:mod:file:source? 2

e:\class.tap

20.4.10.14 CHannel:MODel:CIR?

Get number of impulse responses of channel model

Syntax:

CHannel:MODel:CIR? <channel number>

Description:

This command retrieves the amount of impulse responses on the specific channel model.

Example:

// Get the number of impulse responses on the channel model

// of channel 2.

ch:mod:cir? 2

128000

20.4.10.15 CHannel:MODel:SD?

Get sample density of channel model

Syntax:

CHannel:MODel:SD? <channel number>

Description:

This command retrieves the sample density on the specific channel model.

Example:

//Get the sample density of the channel model of channel 2.

ch:mod:sd? 2

64

20.4.10.16 CHannel:MODel:TIME?

Get emulation time of channel model

Syntax:

CHannel:MODel:TIME? <channel number>

Description:

This command retrieves the emulation time (length) of the specific channel model in seconds. Note that the length of the channel model is valid for settings defined in channel model design.

Example:

// Get the emulation time of the channel model of channel 2.

ch:mod:time? 2

0.948148

20.4.10.17 CHannel:MODel:CORRelating?

Check if channel model is correlating

Syntax:

CHannel:MODel:CORRelating? <channel number>

Description:

This command checks if the specific channel model is correlating. Possible return values are:

0, Channel model is not correlating

1, Channel model is correlating

Example:

// Check if the channel model of channel 1 is correlating.

ch:mod:corr? 1

0

20.4.10.18 CHannel:MODel:DELay:ABSolute:MINimum:GET?

Get the current channel absolute minimum delay in microseconds.

Syntax:

CHannel:MODel:DELay:ABSolute:MINimum:GET? <channel number>

Description:

This request retrieves the minimum delay of the specific channel in microseconds.

Example:

// Get minimum delay of channel 5

ch:mod:del:abs:min:get? 5

20.4.10.19 CHannel:MODel:CMI:CH?

Get concurrent channel model index.

Syntax:

CHannel:MODel:CMI:CH? <channel number>

Description:

This command retrieves the current concurrent channel model index of the link to which the specified channel belongs.

Example:

// In this example three channel model sets defined for the link to which channel 1 belongs (possible concurrent channel model indexes are then 0,1,2). In this example concurrent index 2 has been activated earlier. Get concurrent index for channel 1

ch:mod:cmi:ch? 1

2

20.4.10.20 CHannel:MODel:CMI:CH

Set concurrent channel model index.

Syntax:

CHannel:MODel:CMI:CH <channel number>,<index>

Description:

This command sets a channel model index of the link to which the specified channel belongs. All channels of the link get the same concurrent channel model index.

Example:

// Set the concurrent channel model index 2 for all channels of the link to which the specified channel (1) belongs.

ch:mod:cmi:ch 1,2

20.4.10.21 CHannel:MODel:FILE:CONCurrent?

Get the list of concurrent model names in a channel.

Syntax:

CHannel:MODel:FILE:CONCurrent? <channel number>

Description:

This command returns a comma separated list of concurrent channel model files in the selected channel.

Example:

// Get concurrent model names of the channel number 2

ch:mod:file:conc? 2

D:\User Emulations\Emulation.wiz\model1.tap,D:\User Emulations\Emulation.wiz\model2.tap,D:\User Emulations\Emulation.wiz\model3.tap

20.4.11 Signal routing

20.4.11.1 ROUTe:PATH:REFerence

Set reference clock source

Syntax:

ROUTe:PATH:REFerence <source>

Description:

This command sets the source of the reference clock. Possible sources are:

EXT, External reference clock

INT, Internal reference clock

Example1:

//Set external reference clock

rout:path:ref EXT

Example2:

//Set internal reference clock

rout:path:ref INT

20.4.11.2 ROUTe:PATH:REFerence?

Get current reference clock source

Syntax:

ROUTe:PATH:REFerence?

Description:

This commands gets the source of the reference clock. Possible sources are:

EXT External reference clock

INT Internal reference clock

Example1:

//Get reference clock source

rout:path:ref?

EXT

20.4.11.3 ROUTe:PATH:CONNector?

Get channel's physical connectors

Syntax:

ROUTe:PATH:CONNector? <channel id>

Description:

This query retrieves information of input, output and RF local connectors that are allocated to specified channel. PROPSIM will reply with following information:

sim,in,out,inlo,outlo

"sim" specifies emulator; 1 being commander, 2 second emulator (first follower) and so on.

"in" specifies type and physical input number for channel.

"out" specifies type and physical output number for channel.

"inlo" specifies channel's RFLO connector for input signal.

"outlo" specifies channel's RFLO connector for output signal.

Note that in most cases "inlo" and "outlo" specify the same connector. If channel is baseband channel or internal local oscillator is used, local oscillator connectors are returned as "-" (dash).

Example:

ROUT:PATH:CONN? 1

1,RF-1,RF-1,1,1

20.4.11.4 ROUTe:PATH:REFLOCKED?

Get current reference locking status

Syntax:

ROUTe:PATH:REFLOCKED?

Description:

This command gets the locking status of reference lock. Possible return values are:

0, System is not locked to reference

1, System is locked to reference

Example:

//Ask reference status

rout:path:reflocked?

1

20.4.11.5 ROUTe:PATH:ID?

Get channels input and output id

Syntax:

ROUTe:PATH:ID? <channel id>

Description:

This query retrieves the input and output numbers where the given channel is connected.

Example:

// Get input and output of the channel 3 (from input 1 to output 2)

ROUT:PATH:ID? 3

1,2

20.4.12 External Trigger

20.4.12.1 DIAGnostic:SIMUlation:TRIG:SET

Enable or disable external trigger

Syntax:

DIAGnostic:SIMUlation:TRIG:SET <ON/OFF>

Description:

This command enables or disables external trigger.

Off, Disable external trigger

On, Enable external trigger

Example:

// Enable external trigger.

diag:simu:trig:set on

20.4.12.2 DIAGnostic:SIMUlation:TRIG:SET?

Query if external trigger is enabled

Syntax:

DIAGnostic:SIMUlation:TRIG:SET?

Description:

This request checks whether the trigger is enabled or disabled

Example:

//trigger enabled.

diag:simu:trig:set?

External trigger is on

20.4.12.3 DIAGnostic:SIMUlation:TRIG:CONFiguration

Set triggering configuration

Syntax:

DIAGnostic:SIMUlation:TRIG:CONF <triggering mode>,<mode parameter>

Description:

This command sets triggering configuration. Possible source values are:

Triggering Mode:

LEVEL, Level triggering
EDGE, Edge triggering

Mode Parameter:

LOW, Low level triggering
HIGH, High Level triggering
RISING, Trigger at rising edge
FALLING, Trigger at falling edge

Example1:

// Set triggering on low level.

diag:simu:trig:conf level,low

Example2:

// Set triggering on high level.

diag:simu:trig:conf level,high

Example3:

// Set triggering on falling edge.

diag:simu:trig:conf edge,falling

Example4:

// Set triggering on rising edge.

diag:simu:trig:conf edge,rising

20.4.12.4 DIAGnostic:SIMUlation:TRIG:CONFiguration?

Get current triggering configuration

Syntax:

DIAGnostic:SIMUlation:TRIG:CONF?

Description:

This command gets triggering configuration.

Look at 20.4.12.3 for possible return values.

Example:

// triggering on low level

diag:simu:trig:conf?

Trigger at LEVEL,LOW

20.4.12.5 DIAGnostic:SIMUlation:TRIG:FORCEOFF

Force triggering off

Syntax:

DIAGnostic:SIMUlation:TRIG:FORCEOFF

Description:

This command disables trigger. Command is parallel, so command is executed even if there are other ATE-commands pending.

Force off command is useful in situations where the emulator is waiting for external trigger and for some reason could not get trigger signal. Executing force off command disable external trigger, external trigger signal waiting stops and ATE command execution continues normally.

Example:

// Force trigger off

diag:simu:trig:forceoff

20.4.13 Mass storage commands (file transfer)

20.4.13.1 MMEMory:CATalog?

Get contents of current directory

Syntax:

MMEM:CAT?

Description:

This command gets contents of currently active directory. The response contains disk usage information (drive capacity, free space available in bytes) followed by contents of directory as entries with <file_name>,<file_type>,<file_size>. Directory names are surrounded with brackets [].

Example:

mmem:cat?

1022821920768,606657343488,EmulationData.csv,,0,[n261_4AoA_n5_varying_1dB_shadowing_small.wiz],,

20.4.13.2 MMEMory:CDIRectory

Change current directory

Syntax:

MMEM:CDIR <directory>

MMEM:CDIR

Description:

This command changes active directory. This directory acts only as default path with other MMEMory - commands.

Example1:

// go to default directory (d:\User playbacks)

mmem:cdir

Example2:

// go to d:\User emulations

mmem:cdir "d:\User emulations"

20.4.13.3 MMEMory:CDIRectory?

Get current directory

Syntax:

MMEM:CDIR?

Description:

This command returns currently active directory path.

Example:

mmem:cdir?

d:\User emulations

20.4.13.4 MMEMory:DATA

Copy file to location

Syntax:

MMEM:DATA <filename>,<data>

Description:

This command copies a remote file into currently active directory or given path. <data> is given in 488.2 block format.

Example1:

// copy file to current directory

mmem:data “IQ_Data",#210<10 bytes of data><nl>

“IQ_Data”	Filename
#	This character indicates the beginning of the data block
2	Number of digits in the byte count (always a single digit here - 123456789)
10	Byte count
<10 bytes of data>	Data
<nl>	Data should always end with a newline \n , this is not included at data byte count

Example2:

// copy file to d:\User playbacks

mmem:data "d:\User playbacks\playback1.wfm",<data>

20.4.13.5 MMEMory:DATA?

Get a file from location

Syntax:

MMEM:DATA? <filename>

Description:

This command copies a given file to calling process' location. The response is the associated data in 488.2 block format.

Example:

// copy d:\User playbacks\playback1.wfm to remote location

mmem:data? "d:\User playbacks\playback1.wfm"

<data>

20.4.13.6 MMEMory:DELete

Delete a file

Syntax:

MMEM:DEL <filename>

Description:

This command deletes the given file.

Example1:

// delete capture1.sdf from currently active directory

mmem:del "capture1.sdf"

Example2:

// delete capture1.sdf from d:\My captures

mmem:del "d:\User captures\capture1.sdf"

20.4.13.7 MMEMory:MDIRectory

Make a directory

Syntax:

MMEM:MDIR <directory path>

Description:

This command creates a directory (with subdirectories) to location.

Example1:

// make directory 'subdir' to currently active directory

mmem:mdir subdir

Example2:

// make whole directory path d:\User captures\subdirectory\subdir

mmem:mdir "d:\User captures\subdirectory\subdir"

20.4.14 Multiple emulations loading

By using multiple emulations loading user can proactively load emulations to emulator's memory.

20.4.14.1 MMEM:LOAD

Proactive emulation loading

Syntax:

MMEM:LOAD <filename>

Description:

This command loads emulation (defined in file <filename>) to emulator's memory. Some communication interface tools require the folder name separators to be duplicated. For example: e:\\test\\my emulation.smu.

Example:

// Load e:\1to1 class.smu emulation to emulator

mmem:load e:\1to1 class.smu

20.4.14.2 MEM:ALL?

List proactively loaded emulations in emulator

Syntax:

MEM:ALL?

Description:

This request returns all proactively loaded emulations in emulator that are up to date.

Example:

// Request all proactive loaded emulations in emulator

mem:all?

e:\1to1 class.smu

// Only e:\1to1 class.smu emulation was loaded to emulator

20.4.14.3 MEM:CURRENT?

Get the current emulation's name

Syntax:

MEM:CURRENT?

Description:

This request returns the current emulation name

Example:

// Query current emulation name

mem:current?

e:\1to1 class.smu

20.4.14.4 MEM:DEL:ALL

Remove proactively loaded emulations from memory

Syntax:

MEM:DEL:ALL

Description:

This command deletes all proactively loaded emulations in emulator. Command can be used when all emulations are closed.

Example:

// Removes all emulations from emulator

mem:del:all

20.4.14.5 MEM:DEL:NAME

Remove single loaded emulation from emulator's memory

Syntax:

MEM:DEL:NAME <filename>

Description:

This command deletes proactively loaded emulation (defined in file <filename>) in emulator. Some communication interface tools require the folder name separators (backslashes) to be duplicated. For example: e:\\test\\my emulation.smu. Command can be used when all emulations are closed.

Example:

// Remove e:\1to1 class.smu emulation from emulator

mem:del:name e:\1to1 class.smu

20.4.14.6 MEM:FREE?

Returns emulators' free space

Syntax:

MEM:FREE?

Description:

This request returns known emulators' names and free space (in megabytes).

Example:

// Retrieves free space in emulator

mem:free?

PROPSIM-202452:2103

20.4.14.7 MEM:FILE:SIZE?

Returns emulation's size

Syntax:

MEM:FILE:SIZE? <filename>

Description:

This request returns emulation's size (in megabytes). Some communication interface tools require the folder name separators (backslashes) to be duplicated. For example: e:\\test\\my emulation.smu.

Example:

//request e:\1to1 class.smu emulation's size

mem:file:size? e:\1to1 class.smu

19

20.4.15 Multi-emulator synchronization

20.4.15.1 SYSTem:MSIMulator:CONFig

Syntax:

SYSTem:MSIMulator:CONFig <emulator position>,<emulator total amount>

Description:

This command defines the position of an emulator in multi-emulator configuration. Accepted values for emulator total amount: 1-36

Accepted values for emulator position: 1- emulator total amount.

Note: max. synchronization of max 2 devices supported in current release.

Example:

// Example to set the device to be number 1 in chain of 2 emulators.

SYSTem:MSIMulator:CONFig 1,2

20.4.15.2 SYSTem:MSIMulator:CONFig?

Syntax:

SYSTem:MSIMulator:CONFig?

Description:

This command can be used to query position of emulator in multi-emulator configuration.

Configuration information is returned as follows:

<emulator position>,<emulator total amount>

Example:

// In the example, the device is number 2 in chain of 3 emulators

SYSTem:MSIMulator:CONFig?

2,3

20.4.15.3 SYSTem:MSIMulator:CABle

Syntax:

SYSTem:MSIMulator:CABle <cable length>

Description:

This command can be used to define the synchronization cable length used in multi-emulator configuration. Possible length definitions are: 2.0 m, 4.0 m and 6.0 m.

Example:

// Example to set the synchronization cable length to 4.0 m

SYSTem:MSIMulator:CABle 4.0 m

20.4.15.4 SYSTem:MSIMulator:CABle?

Syntax:

SYSTem:MSIMulator:CABle?

Description:

This command can be used to query the synchronization cable length used in multi-emulator configuration. Possible lengths are: 2.0 m, 4.0 m and 6.0 m.

Example:

// Example to query the synchronization cable length (4.0 m).

SYSTem:MSIMulator:CABle?

2.0 m

20.4.16 Signal capture commands

20.4.16.1 SIGNal:CAPTure:START

Start or cancel predefined or given capture setup

Syntax:

SIGNal:CAPTure:START <signal capture id>,<start (0/1)>

Description:

This command starts or cancels given capture setup. Possible operation modes are:

0 Cancel capture

1 Start capture

Example:

// Start capture for setup id 1

SIGN:CAPT:START 1,1

20.4.16.2 SIGNal:CAPTure:STATus?

Query status of given capture setup

Syntax:

SIGNal:CAPTure:STATus? < signal capture id>

Description:

This request returns the current status of the individual capture setup. Possible return values are:

ERROR	Error in previous capture and new capture can be started
IDLE	Previous capture was cancelled and new capture can be started
PENDING	Capture is started and ongoing
TRIGGER ARMED	Capture is started and waiting for trigger
COPYING	Capture is ongoing and data is copied from the memory
READY	Previous capture is finished and new capture can be started

Example:

// Query status for capture setup id 1

SIGNal:CAPTure:STATus? 1

20.4.17 RF impairment commands

20.4.17.1 EMUlation:RFIMPAIRment:GET?

List all RF impairments IDs, types and directions

Syntax:

EMUlation:RFIMPAIRment:GET?

Description:

This command returns all the RF impairments present in the emulation. Specified in the return value is each RF impairment's ID, it's type and direction. Possible return values for each are:

Types:

1, Phase noise

Example:

// Get a list of RF impairments

emu:rfimpair:get?

1,1,RX,2,1,TX

20.4.17.2 EMUlation:RFIMPAIRment:ENable

Enable or disable certain RF impairment

Syntax:

EMUlation:RFIMPAIRment:ENable <id>, <enable/disable>

Description:

Sets the given RF impairment to enabled or disabled. Possible argument values are:

1, enable

0, disable

Example:

// Enable the RF impairment with ID 2

emu:rfimpair:enable 2, 1

20.4.17.3 EMUlation:RFIMPAIRment:ENable?

Get the status of enable for certain RF impairment

Syntax:

EMUlation:RFIMPAIRment:ENable? <id>

Description:

Fetches the information about whether the given RF impairment is enabled or disabled. Possible return values are:

1, enabled

0, disabled

Example:

// Get the status of enabled for the RF impairment with ID 2

emu:rfimpair:enable? 2

1

20.4.17.4 EMUlation:RFIMPAIRment:INPuts:GET?

Fetch a list of inputs for a certain RF impairment

Syntax:

EMUlation:RFIMPAIRment:INPuts:GET? <id>

Description:

Fetches a list of inputs for the given RF impairment.

Example:

// Fetch a list of inputs for the RF impairment with the ID 2

emu:rfimpair:inp:get? 2

1,2

20.4.17.5 EMUlation:RFIMPAIRment:OUTPuts:GET?

Fetch a list of outputs for a certain RF impairment

Syntax:

EMUlation:RFIMPAIRment:OUTPuts:GET? <id>

Description:

Fetches a list of outputs for the given RF impairment.

Example:

// Fetch a list of outputs for the RF impairment with the ID 1

emu:rfimpair:outp:get? 1

3,4

20.4.18 Signal waveform playback commands

20.4.18.1 SIGNal:PLAYBack:ENable

Set playback state

Syntax:

SIGNal:PLAYBack:ENable <playback id>,<state>

Description:

This command sets the playback state.

0 Disable playback

1 Enable playback

Example:

// Set playback state enabled

SIGNal:PLAYBack:ENable 1,1

20.4.18.2 SIGNal:PLAYBack:ENable?

Get playback state

Syntax:

SIGNal:PLAYBack:ENable? <playback id>

Description:

This command queries the playback state.

0 Playback disabled

1 Playback enabled

Example:

// Get playback state

SIGNal:PLAYBack:ENable? 1

1

20.4.18.3 SIGNal:PLAYBack:STATus?

Request playback status

Syntax:

SIGNal:PLAYBack:STATus? <playback id>

Description:

This command requests the playback status.

0 Playback stopped

1 Playback running

2 Playback waiting trigger

Example:

// Request playback status

SIGNal:PLAYBack:STATus? 1

1

20.4.18.4 SIGNal:PLAYBack:POWer:SET

Sets playback level

Syntax:

SIGNal:PLAYBack:POWer:SET <playback id>,<Level>

Description:

This command sets the playback level.

Example:

// Set playback level

SIGNal:PLAYBack:POWer:SET 1,-20.0

20.4.18.5 SIGNal:PLAYBack:POWer:GET?

Request playback level

Syntax:

SIGNal:PLAYBack:POWer:GET? <playback id>

Description:

This command requests the playback level.

Example:

// Get playback level

SIGNal:PLAYBack:POWer:GET? 1

-20.0

20.4.18.6 SIGNal:PLAYBack:POWer:LIMits?

Request playback level limits

Syntax:

SIGNal:PLAYBack:POWer:LIMits? <playback id>

Description:

This command requests the playback level limits.

Example:

// Get playback level limits

SIGNal:PLAYBack:POWer:LIMits? 1

-20.0,-100.0

20.4.18.7 SIGNal:PLAYBack:POWer:PEAK?

Request playback peak level

Syntax:

SIGNal:PLAYBack:POWer:PEAK? <playback id>

Description:

This command requests the playback peak level.

Example:

// Get playback peak level

SIGNal:PLAYBack:POWer:PEAK? 1

-12.8

20.4.18.8 SIGNal:PLAYBack:STREAM:COUNT?

Request playback stream count

Syntax:

SIGNal:PLAYBack:STREAM:COUNT? <playback id>

Description:

This command requests the playback stream count.

Example:

// Request playback stream count

SIGNal:PLAYBack:STREAM:COUNT? 1

4

20.4.18.9 SIGNal:PLAYBack:DATA:FILE?

Request playback data file(s)

Syntax:

SIGNal:PLAYBack:DATA:FILE? <playback id>,<stream id optional>

Description:

This command requests the playback data file(s).

Example:

// Get playback data file

SIGNal:PLAYBack:DATA:FILE? 1

5G_FR1_TDD_ANT_1, 5G_FR1_TDD_ANT_2, 5G_FR1_TDD_ANT_3

20.4.18.10 SIGNal:PLAYBack:DATA:SAMPLES?

Request playback data sample count

Syntax:

SIGNal:PLAYBack:DATA:SAMPLES? <playback id>

Description:

This command requests the playback samples.

Example:

// Get playback data file

SIGNal:PLAYBack:DATA:SAMPLES? 1

100000

20.4.18.11 SIGNal:PLAYBack:DATA:LENGTH?

Request playback data length

Syntax:

SIGNal:PLAYBack:DATA:LENGTH? <playback id>

Description:

This command requests the playback length in seconds.

Example:

// Get playback data file

SIGNal:PLAYBack:DATA:LENGTH? 1

0.01

20.5 Errors and events

There is a number of error / event messages that are inserted to the error / event queue (if enabled) and can be read by the user. The error event messages have the following syntax:

<error code>,<error string>;<device specific error string>

The error codes and related error strings are listed in the following chapters.

20.5.1 Command error

Error code:

-100

Reason:

Received user command is not supported by ATE server:

- Command itself is not supported or is misspelled
- Command contains parameter data when it should not, or vice versa

Example:

//Command is misspelled

dag:simu:go

-100,"Command error;ATE command not supported"

20.5.2 Execution error

Error code:

-200

Reason:

Command execution failed:

- Command parameter data did not contain required parameters
- Command could not and would not be executed due to the device internal state
- Command execution returned failure

Example:

// Emulation is tried to be run even though no

// Emulation has been opened.

diag:simu:go

-200,"Execution error;Wrong device state for command"

20.5.3 Device-specific error

Error code:

-300

Reason:

Unexpected device-specific error or event occurred.

Example:

//Temperature warning.

-300,"Device-specific error;HW temperature over limit"

20.5.4 Error/Event queue overflow

Error code:

-350

Reason:

Error/event queue overflow occurred. Queue can contain up to 100 error messages. If the user does not read the messages or clear the queue, it is possible that overflow occurs.

Example:

//Error/event queue overflow.

-350,"Queue overflow"

20.5.5 Communication error

Error code:

-360

Reason:

Communication error occurred. Possible reasons:

- LAN receive buffer got full
- LAN receive was interrupted by remote host
- Another user interface interrupted the connection

It should be noted that this error occurs also while closing the remote LAN connection at the end of emulation run, which is not necessarily an error condition. See the following example.

Example:

// User closes TCP/IP connection after emulation was

// closed

-360,"Communication error;LAN receive error"

20.5.6 Query error

Error code:

-400

Reason:

This error occurs typically while the user has not read the response to a query before issuing a next command.

Example:

// User sends a command without reading the response to

// the previous query.

*idn?

calc:filt:file e:\1to1 class.smu

-400,"Query error"

20.6 Troubleshooting

Reliability of ATE communication is dependent of the used interface. ATE LAN interface (TCP/IP) includes higher level protocol in itself, taking care of the possible data communication failures. Even if the test system has been set up with care, communication failures tend to occur. The responsibility to take care of these failure occasions lies on ATE client application, which is the controller-in-charge of the test system.

PROPSIM ATE interface provides several supporting functions for the client, which should be used in the client-side design. The purpose of this chapter is not to list all the provided functionality, but rather to describe the most typical problem situations, offer help on noticing those cases and give some suggestions on how to recover. The failure handling and recovery of the test system depends heavily on the test system itself i.e. what is being done. Please refer to IEEE 488.2 standard for additional information.

20.6.1 Failure handling

The failure handling support of PROPSIM ATE interfaces include the following main areas:

- Emulation status
- Operation state
- ATE device status

20.6.1.1 Emulation status

It is possible to check the status of the settings by using query -commands related to the setting command. This method is valuable in order to detect the cases where data communication error has not caused an error in the device, e.g. if the mobile speed setting command misses the last digit. Queries can also be used for retrieving information on those settings that are set as default and have not been changed by the user. This can be further on be used for documentation purposes.

Example:

```
// Opening and running of ready-made emulation
calc:filt:file d:\My Emulations\SN\emulation.smu
diag:simu:go

// Emulation is predefined, but we still need to make
// measurements with different output gain settings

// Set gain of channel output 1 to -5 dB
outp:gain:ch 1,-5
```

```
// Get gain of channel output 1
outp:gain:ch? 1
-5

// Set gain of channel output 1 to –10 dB
outp:gain:ch 1,-10

// Get gain of channel output 1
outp:gain:ch? 1
-10

// Set gain of channel output 1 to –15 dB
outp:gain:ch 1,-15

// Get gain of channel output 1
outp:gain:ch? 1
-1

// Error! Digit 5 did not go through! Do something!
// Let’ s send again
outp:gain:ch 1,-15

// Get gain of channel output 1
outp:gain:ch? 1
-15
```

20.6.1.2 Operation state

PROPSIM ATE interfaces execute commands sequentially. This means that there is no need to follow operation execution status before sending the next command (for instance using *wai command). Instead, operation state query (*opc?) can be used for synchronization purposes in test system. Note however that the result of the *opc? query indicates only that previous operation has been executed, not that there has not been any errors during the execution.

Example:

```
// We wish to load emulation, run it and measure output
// level when emulation is running

calc:filt:file d:\My Emulations\SN\Emulation.smu
diag:simu:go
*opc?
1

// We received “1” from emulator. Emulation is
// running. Now we can measure with our measurement
// device
abor
calc:mark:y?
```

20.6.1.3 Device status

ATE interfaces provide status information on different kind of errors, which might occur during the test session. These errors include command errors, execution errors, device specific errors and interface errors. Refer to chapter 20.5 of this document.

It is feasible to check for these errors after every ATE command. Most of the error situations can be detected by this way. One way to do this is to check for status byte of the ATE device by using *stb? query. Depending on SCPI register settings, status byte indicates any existing error conditions. Furthermore, the status byte can give indication on readable data, which is typically a result of a query command.

Example:

```
// Initialize SCPI registers

// Set Standard Event Status register to indicate all
// errors/events
// By default this is 0: SESR information is not enabled
*ese 255

// Check Service Request Enable Register
// By default this is 191. SESR summed output,
```

```
// error/event queue status and message available bit as
// are shown in status byte. RQS is always summed from
// other status byte conditions and therefore not
// editable
*sre?
191

// Send identification query
*idn?

// Send again. This causes an error because response
// data from a query has to be always read
*idn?

// Read status byte
*stb?
100

// The result indicates:
// Error/event queue is not empty
// SESR indicates an error
// RQS is set
// Note that due to error, MAV –bit is not set
```

Normally it is not required to use SCPI register information to perform failure handling on device state. By default, all the errors are added to the error/event queue and all the errors in the queue can be read by using syst:err? query. Query responds with text string indicating latest error. Note that there might be multiple errors in the queue, so it is recommended to read all the errors. Errors are destroyed from the queue as they are read. Note that error can also occur on the error checking query itself.

Example:

```
// Set center frequency of channel 1 (and all the
// channels belonging to same channel group) to 2200 MHz
calc:filt:cent:ch 1,2200

// Check error
// Whole message does not go through
sy

// Read return value
// read operation causes timeout because
// emulator will not respond

//Resend
syst:err?

// Read return value
-100,"Command error;ATE command not supported"

// That was caused by failure on error check
// Check for possible previous error
syst:err?

// Read return value
0,"No Error."

// Initial message went through correctly
```

21 FILE FORMATS

This section describes the file formats supported by PROPSIM for exporting and importing channel impulse response and other emulation related data.

21.1 .ASC File

An ASC-file is a simple text file format for representing channel impulse response data. PROPSIM software supports conversions between textual ASC-files and binary IR-files. An ASC-file can be created, for example, in Microsoft Excel spreadsheet application by saving the document as tab delimited text file.

A short ASC-file is presented below. It, like all ASC-files, consists of two main parts: header and tap data. Each line in the tap data corresponds to a single CIR. Entries on each line are separated by tabulator characters.

```
*****	Header	*****
1008	CIRs
2	Taps/CIR
2200000000	Carrier_Frequency
Route_Closed
CIRUpdateRate_Unlocked
CarrierFrequency_Unlocked
5	Delay_Resolution
2	Sample_Density
26092.1		CIR_Update_Rate
*****	Tap data *****
Delay	Re	Im	Delay	Re	Im
0.00000		-0.27080	-0.92316	50.00000	0.86202		-0.12606
0.00000		-0.29589	-0.93519	50.00000	0.78361		-0.05152
0.00000		-0.32334	-0.95466	50.00000	0.62736		0.08362
0.00000		-0.34575	-0.95903	50.00000	0.55293
 0.1627272
```

21.1.1 Header

The header contains the following information

- Number of CIRs (usually in range of 4 - 1,000,000)
- Number of taps per CIR (1 to 48, though system limitations with tap placement apply)
- Carrier frequency (in Hz)
- Model continuity
  - “Route_Closed” for continuous models
  - “Route_Open” for non-continuous models
- Lock / Unlock of CIR update rate
  - In case CIRUpdateRate_Unlocked, values can be changed in run-time
  - In case CIRUpdateRate_Locked, values cannot be changed run-time
- Lock / Unlock carrier frequency
  - In case CarrierFrequency_Unlocked, values can be changed in run-time
  - In case CarrierFrequency_Locked, values cannot be changed run-time
- Delay resolution (nanoseconds, use always 5ns)
- Sample density i.e. the number of CIRs calculated for one wavelength
- CIR update rate

21.1.2 Tap data

The Tap data contains information about complex taps. Tap data for each tap data contains three values ("triplet"):

- Delay value (measured in nanoseconds)
- Real value
- Imaginary value

For each CIR, equal number of these triplets must exist on the row as specified in the header. For example, if the header specifies 4 taps/CIR, there must be 4 triples, or 12 values (delay, real, imaginary) per row.

21.2 .IR File

PROPSIM IR-files are backwards compatible with the IR-files of earlier PROPSIM products.

The .IR-file consists of the header and impulse response data (CIR). Header contains ASCII type description about measurement and essential parameters like carrier wave center frequency, impulse response sample density and delay resolution. IR-data is a table of impulse responses. Each IR consists of many taps. Taps are described as their delay and complex strength. .IR-data is handled as a table.

Table 1. Data types used in IR-file syntax description

Data type	Explanation
uint8	8-bit unsigned two's complement integer
uint16	16-bit little-endian unsigned two's complement integer
uint32	32-bit little-endian unsigned two's complement integer
float32	32-bit little-endian IEEE floating point number
float64	64-bit little-endian IEEE floating point number
String	String using 8-bit wide characters and usually ending in character '\0' (it is possible that when string length is known otherwise, tailing zero may be absent.) Used character set may be for example ISO-8859-1, SJIS or similar. Use of anything else than ASCII characters is not recommended if files are used in different locales.

21.2.1 Syntax

```
IR_FILE {
INDEX_TABLE
HEADER_DATA
TAP_DATA
}

INDEX_TABLE {
HEADER_DATA_INDEX   : uint32     = offset of HEADER_DATA
TAP_DATA_INDEX      : uint32     = offset of TAP_DATA
RESERVED_INDEX      : uint32     = 0xFFFFFFFF
RESERVED_INDEX      : uint32     = 0xFFFFFFFF
RESERVED_INDEX      : uint32     = 0xFFFFFFFF
NULL_INDEX    : uint32     = 0x00000000
}

HEADER_DATA {
COMMON_HEADER_DATA
CREATOR_SPECIFIC_HEADER_DATA
NULL_BLOCK
}

NULL_BLOCK : block {
fields : uint16     = 0
}

COMMON_HEADER_DATA : block {
fields : uint16     = 8 + 3
```

```
TITLE  : field {
size   : uint16     = size of data
data   : string     = user comment of current file
}
SOURCE_TIME_STAMP   : field {
size   : uint16     = size of data
data   : string     = date & time from source file. a free-form string.
}
SOURCE_FILE_NAME    : field {
size   : uint16     = size of data
data   : string     = the source file name
}
CARRIER_FRQ  : field {
size   : uint16     = size of data
data   : uint32     = carrier frequency in Hz or
  float64    = carrier frequency in Hz
}
IR_SAMPLE_DENSITY   : field {
size   : uint16     = size of data
data   : float32    = samples/half-wave or
float64      = samples/half-wave. Used for mobile speed calculation. If SD
== 0, mobile speed is not meaningful.
}
DELAY_RESOLUTION    : field {
size   : uint16     = size of data
data   : uint16     = nanoseconds or
  float64    = seconds. If 0, not available.
}
CLOSED_ROUTE  : field {
size   : uint16     = size of data
data   : uint8      = 0 = open, 1 = closed
}
INTERPOLATION : field {
size   : uint16     = size of data
data   : uint8      = 0 – round to nearest
                    1 – default interpolation
                    4 – four tap interpolator
                    14 – fourteen tap interpolator
}
CIR_UPDATE_RATE     : field {
size   : uint16     = size of data
data   : float64    = CIR update rate in Hz.
}
CIR_UPDATE_RATE_LOCKED     : field {
size   : uint16     = size of data
data   : uint8      = 0 : unlocked
               1 : locked
}
CARRIER_FRQ_LOCKED  : field {
size   : uint16     = size of data
data   : uint8      = 0 : unlocked
               1 : locked
}

}

CREATOR_SPECIFIC_HEADER_DATA : block {
fields : uint16     = field count
CREATOR      : field {
size   : uint16     = size of data
data   : string     = name of creator (no spaces allowed)
}
}

PROPSOFT_HEADER_DATA : CREATOR_SPECIFIC_HEADER_DATA {
fields : uint16     = 3
CREATOR      : field {
size   : uint16     = size of data
data   : string     = "PropSoft"
}
CELL_TYPE    : field {
```

```
size   : uint16     = size of data
data   : string     = CELL type
}
SEED   : field {
size   : uint16     = size of data
data   : float32    = seed for random generator
}
}

GEN_IR_GEN_HEADER_DATA : CREATOR_SPECIFIC_HEADER_DATA {
fields : uint16     = 4
CREATOR       : field {
size   : uint16     = size of data
data   : string     = "General-IR-Generator"
}
CHANNEL_TYPE  : field {
size   : uint16     = size of data
data   : uint8      = 0 = channel 1, 1 = channel 2, 2 = correlative channel
2
}
CORRELATION_FACTOR  : field {
size   : uint16     = size of data
data   : float32    = correlation factor (0, when not correlative channel)
}
OTHER_CHN_FILE_NAME : field {
size   : uint16     = size of data
data   : string     = other channel file name (NUL when one channel only)
}
}
TOOLBOX_HEADER_DATA : CREATOR_SPECIFIC_HEADER_DATA {
fields : uint16     = amount of fields
CREATOR       : field {
size   : uint16     = size of data
data   : string     = "ToolBox"
}
}

TAP_DATA {
IRS    : uint32     = number of impulse responses
TAPS   : uint16     = amount of TAP on each IR. If TAPS = 0, amount of taps
is specified separately at the beginning of each CIR.
IR : table of IRS members {
OPTIONAL_IR_TAPS    : uint16     = amount of taps in this impulse response
(present only if TAPS=0)
TAP : table of TAPS or OPTIONAL_IR_TAPS members {
DELAY  : float32    = delay value in nanoseconds
RE     : float32    = real part
IM     : float32    = imaginary part[1]
}
}
```

[1] If IRs have different number of taps and TAPs is not set to 0, the number of taps per IR is selected according to longest one. These "shorter" IRs must then be filled with dummy taps having `Re` and `Im` values zero. Note that delays are required to be in increasing order.

With non zero taps, `Re` and `Im` may have any values because values are re-scaled when converted to .SIM emulator format.

.IR to .SIM conversion keeps relative gains between channels and therefore each .IR file used at the same emulation must be equally normalised.

21.3 .MAT File

MAT file is a MathWorks Inc's MatLab® internal file format, usually generated with MatLab. Importing .MAT files are supported as either directly in emulation, or via conversion to .IR file format. Imported .MAT file must have four variables and may additionally have nine more. Any variables not explicitly mentioned here are discarded during conversion. All variable names are case sensitive.

Please note that .mat file must be compatible with Matlab version 6. If you are using newer version of Matlab, use the save-command from Matlab command line with parameter –V6 to create version 6 compatible mat-file.

For example (typed to Matlab shell to save .mat –file):

save 'c:\my_model.mat' –V6

Reading .mat files uses Matlab Component Runtime (MCR), provided by MATLAB. 1984 – 2008, The MathWorks Inc. It is delivered under licensing terms defined by *"The Mathworks, inc. Software license agreement - Deployment Addendum"*.

21.3.1 Required variables

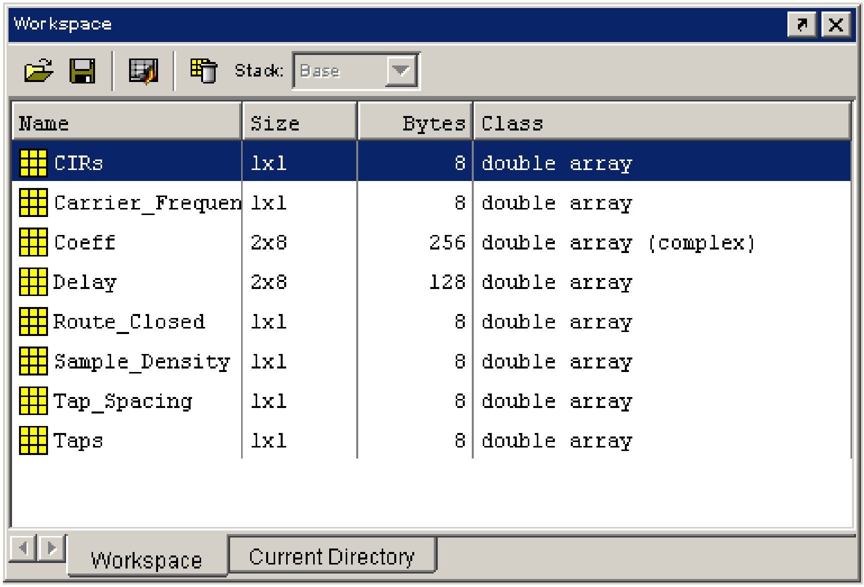
Name	Explanation
CIRs	Number of channel impulse responses in model, min. 1000
Taps	Number of taps (paths) in model
Coeff	Matrix of complex impulse response coefficients, with "Taps" rows and "CIRs" columns. Each coefficient must have a delay value in delay matrix.
Delay	Matrix of coefficient delay values, with "Taps" rows and "CIRs" columns. Delays are expressed in nanoseconds.

21.3.2 Optional variables

These variables may be omitted from .MAT file. If any of these is not found, the default value is used.

Name	Default	Explanation
CIRUpdateRate	10000 Hz	Update rate of model, in Hertz. Valid range is 0.01 to 1000000 Hz.
Carrier_Frequency	2.2 GHz	Carrier center frequency. Valid range is 30MHz to 6 GHz.
Tap_Spacing	5 ns	Tap spacing
Route_Closed	1	Is route closed i.e. is model continuous. 0 = Non-continuous 1 = Continuous This parameter is only informative.
Sample_Density	64	Samples per half-wave; 2 to 1000. **Note: it is recommended to define this variable and use sample density 2.**
Hardware_Usage	0	For backwards compatibility, value is ignored
Description	"Mat2Ir generated file"	Description of model.
CIRUpdateRateLocked	0	Is CIR update rate locked or changeable run-time 0 = Not locked (changeable) 1 = Locked
CarrierFrequencyLocked	0	Is carrier frequency of model locked or changeable run-time 0 = Not locked (changeable) 1 = Locked

21.3.3 MAT file example



Name	Size	Bytes	Class
CIRs	1x1	8	double array
Carrier_Frequen	1x1	8	double array
Coeff	2x8	256	double array (complex)
Delay	2x8	128	double array
Route_Closed	1x1	8	double array
Sample_Density	1x1	8	double array
Tap_Spacing	1x1	8	double array
Taps	1x1	8	double array

Figure 281 Example of workspace (as shown in MatLab) stored in .MAT file

Variables listed have following values;

CIRs 8

Taps 2

Coeff [1.0 0.8 0.6 0.4 0.2 0.4 0.6 0.8;

0.2i 0.4i 0.6i 0.8i 1.0i 0.8i 0.6i 0.4i]

Delay [0 0 0 0 0 0 0 0;

50 50 50 50 50 50 50 50]

Carrier_Frequency 2.1e+9

Route_Closed 1

Sample_Density 8

Tap_Spacing 5

21.4 Multi Emulator Scaler calibration file

The file format for calibration file is described in the example below.

```
<?xml version="1.0" ?>
<Calibration>
<Date>19.05.2010 (10:17:27)</Date>
<VNA_ID>Network analyzer device ID</VNA_ID>
<Fader_ID>PROPSIM device ID</Fader_ID>
<Frequency_MHz>2450</Frequency_MHz>
<Power_dBm>-15</Power_dBm>
<Output>
<CH1>
    <!--gain and phase measurement for output 1-->
                    <Gain_dB>-2.10</Gain_dB>
                    <Phase_val>1175</Phase_val>
       </CH1>
<CH2>
```

```
    <!--gain and phase measurement for output 2-->
                    <Gain_dB>-3.15</Gain_dB>
                    <Phase_val>1175</Phase_val>
       </CH2>
<CH3>
    <!--gain and phase measurement for output 3-->
                    <Gain_dB>-1.05</Gain_dB>
                    <Phase_val>1175</Phase_val>
       </CH3>
<CH4>
    <!--gain and phase measurement for output 4-->
                    <Gain_dB>-1.05</Gain_dB>
                    <Phase_val>1175</Phase_val>
</CH4>
<CH5>
    <!--gain and phase measurement for output 5-->
                    <Gain_dB>0.00</Gain_dB>
                    <Phase_val>1175</Phase_val>
       </CH5>
<CH6>
    <!--gain and phase measurement for output 6-->
                    <Gain_dB>-3.15</Gain_dB>
                    <Phase_val>1175</Phase_val>
       </CH6>
<CH7>
    <!--gain and phase measurement for output 7-->
                    <Gain_dB>-4.20</Gain_dB>
                    <Phase_val>1175</Phase_val>
       </CH7>
<CH8>
    <!--gain and phase measurement for output 8-->
                    <Gain_dB>-2.10</Gain_dB>
             <Phase_val>1175</Phase_val>
       </CH8>
</Output>
<Input>
<CH1>
    <!--level and phase measurement for input 1-->
                    <Level_dBm>-19.95</Level_dBm>
                    <Phase_val>1919</Phase_val>
       </CH1>
<CH2>
    <!--level and phase measurement for input 2-->
                    <Level_dBm>-19.45</Level_dBm>
                    <Phase_val>0</Phase_val>
       </CH2>
</Input>
</Calibration>
```

21.5 .ASO File

An ASO-file is a simple text file format for representing channel impulse response data for use with Aerospace and Satellite channel modelling Option. PROPSIM software supports conversions between textual ASO-files and binary ASO SIM-files.

The ASO file format describes either periodical Function based, Coordinate based (N-type) or Arbitrary model (A-type) parameters. Only one model type can be specified in a file at same time.

21.5.1 Function based .ASO file

Function based model creates model where Doppler is defined based on linear, sinusoidal, or triangular function. Delay is calculated from Doppler. User can define minimum and maximum values for gain.

Example of function-based model

```
; Propsim Aerospace Model file, version 1.0
[Model]
SimulationCenterFrequency=27000000000 Hz
RFCenterFrequency=370000000 Hz
```

```
[Sinusoidal 1]
Period = 2.0 s
MinDoppler = -1.0 MHz
MaxDoppler = 1.0 MHz
MinDelay = 100 us
MinGain = 0 dB
MaxGain = -6 dB
```

21.5.2 Coordinate based .ASO file

Coordinate based model contains a point-to-point link between three-dimensionally moving transmitter and receiver. Link is assumed to be line-of-sight, with possibility to have up to three reflectors somewhere in the area causing multi-path effects. Reflectors will cause reflected signal to arrive to receiver delayed and attenuated by user-given attenuation factor.

In coordinate-based model trajectories of the objects are defined by their coordinates in 3D space. The coordinates are then used to calculate delay and Doppler values.

Example of coordinate-based model

```
; Propsim Aerospace Model file, version 1.0
[Model]
SimulationCenterFrequency=27000000000 Hz
RFCenterFrequency=370000000 Hz

[LOS]
N=0.0, 10.0,0.0,0.0, 1.0,0.0,0.0, 0.0
N=1.0, 10.0,0.0,0.0, 2.0,0.0,0.0, -1.0
N=2.0, 10.0,0.0,0.0, 3.0,0.0,0.0, -2.0
[Reflector 1]
N=0.0, 3.14,2.71,0, -5.0
```

21.5.3 Arbitrary .ASO file

Arbitrary model contains a point-to-point link between moving transmitter and receiver. Link is assumed to be line-of-sight, with possibility to have up to three reflectors causing multi-path effects. Delay, Doppler and gain values for transmitter, receiver and reflectors are defined directly by the user.

Example Arbitrary model

```
; Propsim Aerospace Model file, version 1.0

[Model]
SimulationCenterFrequency = 27000000000 Hz
RFCenterFrequency = 370123400 Hz

[LOS]
A = 0, 0.005, 0, 0
A = 10, 0.002, 100, -10

[Reflector 1]
A = 0, 0.001, 100, -10
A = 10, 0.002, 200, -20
```

21.5.4 File syntax

.ASO file itself is a plain text file. It consists of comment header and multiple sections of data. Lines beginning with semicolon (;) are comment lines and ignored, with exception of header command line, which is required.

Section's name is enclosed in "[ ]" characters. The data itself is in "keyword=value" lines, each keyword-value pair being on its own line. The type of all numeric variables is double.

ASO file can contain either N- or A-keywords. Any mixing of A- and N-keywords in a single file is not allowed.

Comment	Propsim Aerospace Model file, version 1.0
Description	Normal comment header in .ASO file. Required field.

21.5.4.1 [Model] section

Section	Model
Description	General model parameters.

Key	SimulationCenterFrequency
Value	*Frequency* Hz
Description	Center frequency of model, in Hz. Doppler and channel update rates are calculated by using this value

Key	RFCenterFrequency
Value	*Frequency* Hz
Description	Propsim center frequency, given in Hz's.

Key	RangeStart
Value	Time
Description	Optional field describing start of the range. Range is used to crop model at the time of emulation creation. If field is missing, range is not enabled.

Key	RangeEnd
Value	Time
Description	Optional field describing end of the range. Range is used to crop model at the time of emulation creation. If field is missing, range is not enabled.

Key	AutomaticStop
Value	*True/False*
Description	Optional field. If set to true, compiler adds CIR's for 0.5 seconds at the end of the emulation. Compiler also sets delta delay to zero and Doppler to last Doppler value in the model. If the field is missing, default value is true. If model is continuous (function-based model), no extra CIR's are added.

21.5.4.2 [LOS] section

Section	LOS
Description	Line-of-sight component definition.

Key	N
Value	*Time,X1,Y1,Z1,X2,Y2,Z2,Gain*
Description	Positions of devices and link gain on given time index. Time specifies time index, in seconds. Xn,Yn, Zn specifies device position in 3D space in meters. Gain specifies link gain, in dB. Whitespace characters are allowed between values and commas. Note that using "N" keys in file disallows usage of "A" keys in LOS or in Reflector sections.

Key	A
Value	*Time,delay,Doppler,gain*
Description	Delay, Doppler and gain of path on given time. Time specifies time index, in seconds. Delay specifies current LOS path delay, in seconds. Doppler specifies current Doppler of link, in Hz. Gain specifies link gain, in dB. Whitespace characters are allowed between values and commas. Note that using "A" keys in file disallows usage of "N" keys in LOS or in Reflector sections.

21.5.4.3 [Reflector X] section

Section	Reflector *x*
Description	Data for reflector number *x* (1, 2 or 3)

Key	N
Value	*Time,X,Y,Z,Gain*
Description	Position of reflector and link gain on given time index. Time specifies time index, in seconds. X,Y,Z specify reflector position in 3D space, in meters. Gain specifies link gain in dB. Whitespace characters are allowed between values and commas. Note that using “N” keys in file disallows usage of “A” keys in LOS or in Reflector sections.

Key	A
Value	*Time,delay,Doppler,gain*
Description	Delay, Doppler and gain of path on given time. Time specifies time index, in seconds. Delay specifies current LOS path delay, in seconds. Doppler specifies current Doppler of link, in Hz. Gain specifies link gain, in dB. Whitespace characters are allowed between values and commas. Note that using “A” keys in file disallows usage of “N” keys in LOS or in Reflector sections.

21.5.4.4 [Sinusoidal X] section

Section	Sinusoidal *x*
Description	Data for sinusoidal function number *x* (1)

Key	Period
Value	*Time* s
Description	Period of the function in seconds.

Key	MinDoppler
Value	*Frequency* Hz
Description	Minimum Doppler value. Usually equals to MaxDoppler with difference of the sign.

Key	MaxDoppler
Value	*Frequency* Hz
Description	Maximum Doppler value. Usually equals to MinDoppler with difference of the sign.

Key	MinDelay
Value	*Time* s
Description	Minimum delay for model. Used to calculate position offset for fictional moving object in model.

Key	MinGain
Value	*Gain* dB
Description	Minimum gain value for the model. Gain is following used function.

Key	MaxGain
Value	*Gain* dB
Description	Maximum gain value for the model. Gain is following used function.

21.5.4.5 [Triangle X] section

Section	Triangle *x*

Description	Data for triangle function number *x* (1)

Key	Period
Value	*Time* s
Description	Period of the function in seconds.

Key	MinDoppler
Value	*Frequency* Hz
Description	Minimum Doppler value. Usually equals to MaxDoppler with difference of the sign.

Key	MaxDoppler
Value	*Frequency* Hz
Description	Maximum Doppler value. Usually equals to MinDoppler with difference of the sign.

Key	MinDelay
Value	*Time* s
Description	Minimum delay for model. Used to calculate position offset for fictional moving object in model.

Key	MinGain
Value	*Gain* dB
Description	Minimum gain value for the model. Gain is following used function.

Key	MaxGain
Value	*Gain* dB
Description	Maximum gain value for the model. Gain is following used function.

21.5.4.6 [LinearDoppler X] section

Section	LinearDoppler *x*
Description	Data for Linear Doppler function number *x* (1)

Key	Period
Value	*Time* s
Description	Period of the function in seconds.

Key	MinDoppler
Value	*Frequency* Hz
Description	Minimum Doppler value. Usually equals to MaxDoppler with difference of the sign.

Key	MaxDoppler
Value	*Frequency* Hz
Description	Maximum Doppler value. Usually equals to MinDoppler with difference of the sign.

Key	MinDelay
Value	*Time* s
Description	Minimum delay for model. Used to calculate position offset for fictional moving object in model.

Key	MinGain
Value	*Gain* dB
Description	Minimum gain value for the model. Gain is following used function.

Key	MaxGain
Value	*Gain* dB
Description	Maximum gain value for the model. Gain is following used function.

22 MAINTENANCE

PROPSIM is designed to require minimum maintenance. The main points of maintenance are discussed below.

22.1 Calibration

PROPSIM factory calibration period is 12 months. Calibration expiration date for PROPSIM and connected ACU devices is visible in the **Device information** dialog, see section 9.1.

22.2 Connectors and cables

To maintain the high performance of the emulator, proper care must be taken when connecting cables to the emulator.

The connectors shouldn't be touched, especially not the center conductor to prevent dust and dirt going into the connector. Also, possible damages caused by static discharges are avoided.

22.3 Cleaning

The instrument front and rear panels should be cleaned using an anti-static soft cloth. PROPSIM should be kept in a dust-free environment. To prevent electrical shock, disconnect PROPSIM from mains before cleaning. Do not attempt to clean internally.

22.4 Over current protector

The main power switch of the emulator doubles as an over current protector. The emulator has no fuse that the user can change. If over current occurs, the emulator turns itself off and the switch returns to the OFF position.

If the over current protection turns the emulator off repeatedly, it is an indication of some fault in the emulator. The device should be sent to a facility designated by Keysight for appropriate service.

Note: *The power switch must not be forced to the ON position if the over current protection shuts down the emulator. It will not help keep the emulator operational.*

23 TROUBLESHOOTING

This document explains the error messages that the PROPSIM emulator may produce and advises the proper troubleshooting actions.

The error indicators of PROPSIM consist of LEDs and error messages in the Graphical User Interface (GUI).

Most of the typical problems can be analyzed by using the basic one path constant channel model and a test setup with signal generators and a spectrum analyzer.

23.1 Typical problems

This chapter lists the typical errors that may be encountered and the corresponding procedures to solve the problems. If the problem still appears after the actions have been taken, please contact Keysight NES Wireless Solutions Help Desk (http://www.keysight.com/my) or customersupport.di@keysight.com

23.1.1 Emulator does not start

- Check that both power cords are connected to the rear panel of the emulator.
- Check that both power switches are turned on.
- Check that the standby switch is alighted when the standby switch in front panel is pressed.
- Check that the fans are running.

23.1.2 Self-test fails

- Shut down the emulator using the standby switch in the front panel of the emulator.
- Turn off the power using the power switches in the rear panel of the emulator.
- Wait at least 15 seconds and turn the switch back on.
- If the reboot doesn't help, contact Keysight.

23.1.3 No signal at the emulator output

- Check the frequency and the level of the input signal.
- Check that the test setup connections are according to the Active connectors view (in Data views window).

23.1.4 Status LED is red

Red color of the control unit status LED indicates a warning due to

- overheating,
- over voltage or
- self-test failure.

The reason of the warning is reported in the **System log** dialog and the **STATUS** button after an emulation is loaded.

23.1.5 Power LED is red

- Red color of the power LED indicates power failure.
- Note: during initialization the LED is blinking red

23.1.6 RF channel status LED is red

Red color of the RF channel status LED indicates a warning due to

- overheating,
- over voltage,

- input cut-off,
- self test failure,
- HW failure.

The reason of the warning is reported in the System log dialog and the STATUS button after emulation is loaded.

23.1.7 Incorrect signal level

- Connect the emulator to a test setup with a signal generator and a spectrum analyzer.
- Load and run the constant one tap channel model
- Adjust the levels on the generators and the spectrum analyzer according to the figures in the GUI emulation settings.
- Measure the output signal level. The level should be the same that can be read as the expected output level in the GUI.

23.1.8 Incorrect test results

Incorrect DUT performance results arise usually from incorrect signal levels either at the input or output of the emulator. This can be avoided by going through the following steps:

- Open one channel static emulation in Emulation Control View from D:\Standard Emulations\Emulation Examples\Static Emulation\1to1 Static.smu
- Check the PROPSIM Input RF level setting (BS settings pane) in the Emulation Control View. It should be 0 dBm.
- Average level of input signal must be the same i.e. 0 dBm.
- Enter the input signal Crest factor in the GUI.
- Check with the spectrum analyzer that the signal level in the emulator input is 0 dBm.
- Connect test signal generator to PROPSIM RF 1 connector.
- Start emulation by clicking the (Start) button in the Emulation Control View.
- Check the PROPSIM output RF level (MS settings pane) in the Emulation Control View. It should be -31 dB.
- Check with the spectrum analyzer that the output signal level value corresponds to the expected average level of the output in the GUI. Note the attenuation caused by the cable.

23.2 Error and warning messages

In this chapter, the error and warning messages of PROPSIM are listed and their meaning explained. The actions to be taken are also described for each error case.

23.2.1 Running view errors

The warning and errors of Running view are reported in Status window at the bottom of the Running view along with other status information. The Status window can be enabled or disabled from Window > Show Status Window.

Error Message	Description / Actions
Cannot find emulation file	Check that the emulation is generated and re-build if needed.
Cannot open emulation file	Check that the emulation is generated and re-build if needed.
Channel allocation failed	Emulation requires more HW resources than is available (path resources or signal routing capability). It is necessary to simplify the channel models (reduce number of taps) or reduce the number of channels in emulation.
Connection to emulators failed	Connection to emulator failed. Check that there is no open emulation through ATE interface.
Connection to emulator failed	Connection to emulator failed. Check that there is no open emulation through ATE interface.

Error Message	Description / Actions
Could not open <filename>	Check that file exists and is readable.
Calibration data corrupted in slot <number> unit	Contact Keysight.
Current warning	Contact Keysight.
Emulation file copy failed. The destination cannot be reached	Re-build the emulation.
Emulation file missing, rebuild emulation	Re-build the emulation.
Emulation open failed	Re-build the emulation.
Emulation run failed	Re-build the emulation.
Emulator cannot be reached	Reboot the device. If the problem still occurs, contact Keysight.
Emulator control file missing, rebuild emulation	Re-build the emulation.
Emulator is out of extended delay area license	Emulation cannot be opened because of missing license.
Emulator is out of interference license	Emulation cannot be opened because of missing license.
Emulator is out of MIMO license	Emulation cannot be opened because of missing license.
Emulator is out of noise features	Emulation cannot be opened because of missing license.
Emulator is out of noise licenses	Emulation cannot be opened because of missing license.
Error changing state	Reboot the device. If the problem still occurs, contact Keysight.
External reference clock not present	Check the external reference clock connection.
Failed to configure emulator	Reboot the device. If the problem still occurs, contact Keysight.
Go error	Reboot the device. If the problem still occurs, contact Keysight.
HW delay compensation failed	Emulator internal delay compensation failed. Reboot the device. If the problem still occurs, contact Keysight.
HW failure <number>	Contact Keysight.
Input cut off warning	Peak input level is too high. Check Average Input Level and Crest Factor settings.
Internal RF local level too high (RF <number>)	Contact Keysight.
Missing emulation file	Check that emulation is build.
Network client closed error	Reboot the device. If the problem still occurs, contact Keysight.
Network connection error	Reboot the device. If the problem still occurs, contact Keysight.
Network receive error	Reboot the device. If the problem still occurs, contact Keysight.
Network send error	Reboot the device. If the problem still occurs, contact Keysight.
No AWGN option license available	Emulation cannot be opened because of missing license.
Not enough emulators	Emulation is too big for one emulator, reduce channel count.
No shadowing license available	Emulation cannot be opened because of missing license.
No shadowing option license	Emulation cannot be opened because of missing license.
Not enough noise sources available	Emulation cannot be opened because of missing license.
Parity error	Contact Keysight.

Error Message	Description / Actions
Self test failed	Contact Keysight.
Setting of RF local sources failed	Reboot the device. If the problem still occurs, contact Keysight.
Shadowing profile file not found	Check that shadowing file (.SHD) exists.
SMU file closing failed	Reboot the device. If the problem still occurs, contact Keysight.
Source file missing, check emulation	Check that channel model file exists (.TAP/.IR/.MAT/.ASC)
Stopping emulation failed	Reboot the device. If the problem still occurs, contact Keysight.
System clock unlocked. Check External 10MHz reference clock frequency	External reference clock frequency is not in specified range (10 MHz ±5 Hz) or signal level is too low (< 0 dBm).
System clock unlocked (oscillator warming up).	After changing from external reference clock to internal reference clock, internal oscillator warming up time is up to 10 minutes before it set up to proper frequency. If the message persists, contact Keysight.
Unknown network client error	Reboot the device. If the problem still occurs, contact Keysight.
Update rate file missing, rebuild emulation	Re-build the emulation.
Voltage warning	Emulator has an internal voltage level error. If restart does not help, the device requires service. Please contact Keysight.
Warning: High device temperature. Ensure free airflow to prevent device shutdown.	Internal temperature is too high. Turn the emulator off and make sure that there is enough airflow through the emulator. When CPU temperature is over limit, the CPU clock frequency is decreased.

23.2.2 Channel model view errors

Error Message	Description / Actions
ERROR: .cor file channel count not matching!	Number of selected channels in the New Model Generation Wizard must match to number of channels selected in the Correlation Editor.

23.2.3 Correlation editor errors

Error Message	Description / Actions
ERROR: Invalid matrix declared.	Check the matrix status. The message appears if the status of the matrix is not valid.

23.2.4 ATE Command specific warnings

Error Message	Description / Actions
ATE command not supported	Invalid command. Check the command syntax. If the LAN server does not accept commands even though the syntax seem to be correct, check that the End Of String mark character is '\n' (line feed). When using for example TELNET the user must use Ctrl-ENTER instead of plain ENTER.

23.3 Gathering info when contacting customer support

After going through the steps below before contacting us we can help you much faster and better at the Keysight NES Wireless Devices Help Desk (http://www.keysight.com/my).

23.3.1 Get the serial number

Select Configuration > Device information and a dialog opens showing the serial number. Serial number can also be found from a sticker on the rear panel of the device.

23.3.2 Get the log files

The log files often provide invaluable information on what has happened and if there are any HW failures.

Selecting “Help -> Technical Support -> Export system log files” will open a dialog for exporting the most important system logs to a zipped folder, which can be shared to the customer support.

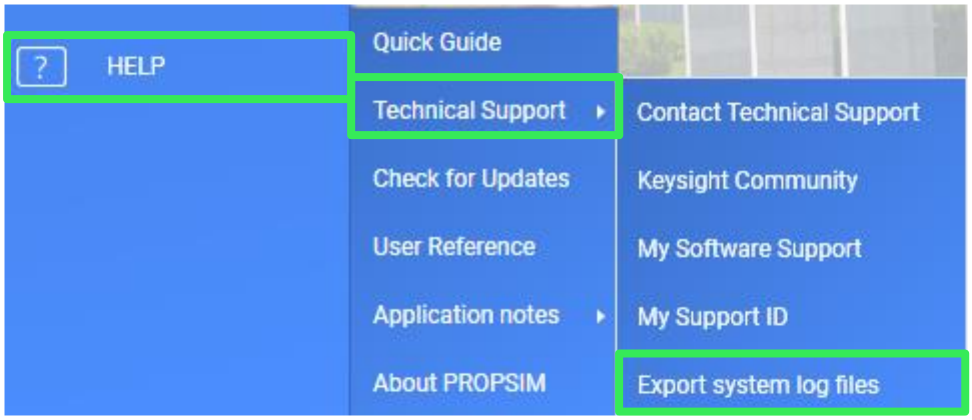


Figure 282 Exporting system log files

The most important log files are:

- "SimulatorServer.log” and
- "SimulatorServer.old.log"

both of which are located in "C:\PROPSIM\".

These files contain information only from the current and previous boot of the device and that is why it is essential to save them on the same boot after the issue has occurred. These files also contain the device's serial number and therefore when sending these files, it is not necessary to send the serial number separately. In the same folder there are also other log files with ".log" extension that also provide valuable information in some cases.

If the issue has occurred when using PROPSIM remotely via ATE LAN these files should be included:

ATE Logs:

- "ATE.log",
- "ATE.old.log",
- "ATELan.log" and
- "ATELan.old.log"

all of which are located in "C:\PROPSIM\".

23.3.3 Get the emulation files

Including the emulation file(s) (*.smu) and the channels models used when contacting us helps us in both excluding common mistakes concerning emulation creation and reproducing the issue.

Emulation files are especially important when the issue is reproducible with only a certain emulation.

23.3.4 Describe the test setup

PROPSIM is always part of a larger test setup and therefore excluding external devices, components and cabling is essential. A brief written description of the test setup is good, a diagram with an explanation and aim of the test is better.

23.3.5 Describe the aim of the test and the encountered issue

A comprehensive description of the issue preventing to achieve the aim is the most important thing to include when contacting us. It can be anything from a few sentences accompanied with a screenshot to a slideshow or even a video.