Keysight M8070A System Software for M8000 Series of BER Test Solutions

User Guide

KEYSIGHT TECHNOLOGIES

Notices

Trademarks

PCI Express® and PCIe® are registered trademarks of PCI-SIG.

Manual Part Number

M8020-91030

Edition

Edition 5.1, March 2016

Keysight Technologies Deutschland GmbH

Herrenberger Strasse 130,

71034 Böblingen, Germany

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.

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.

Safety Notices

CAUTION

A CAUTION notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in damage to the product or loss of important data. Do not proceed beyond a CAUTION notice until the indicated conditions are fully understood and met.

WARNING

A WARNING notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in personal injury or death. Do not proceed beyond a WARNING notice until the indicated conditions are fully understood and met.

Safety Summary

The following general safety precautions must be observed during all phases of operation of this instrument. Failure to comply with these precautions or with specific warnings or operating instructions in the product manuals violates safety standards of design, manufacture, and intended use of the instrument. Keysight Technologies assumes no liability for the customer's failure to comply with these requirements. Product manuals are provided with your instrument on CD-ROM and/or in printed form. Printed manuals are an option for many products. Manuals may also be available on the Web. Go to www.keysight.com and type in your product number in the Search field at the top of the page.

General

This product is a Safety Class 1 instrument (provided with a protective earth terminal). The protective features of this product may be impaired if it is used in a manner not specified in the operation instructions.

All Light Emitting Diodes (LEDs) used in this product are Class 1 LEDs as per IEC 60825-1.

Environment Conditions

This instrument is intended for indoor use in an installation category II, pollution degree 2 environment. It is designed to operate at a maximum relative humidity of 95% and at altitudes of up to 2000 meters.

Refer to the specifications tables for the ac mains voltage requirements and ambient operating temperature range.

Before Applying Power

Verify that all safety precautions are taken. The power cable inlet of the instrument serves as a device to disconnect from the mains in case of hazard. The instrument must be positioned so that the operator can easily access the power cable inlet. When the instrument is rack mounted the rack must be provided with an easily accessible mains switch.

Ground the Instrument

To minimize shock hazard, the instrument chassis and cover must be connected to an electrical protective earth ground. The instrument must be connected to the ac power mains through a grounded power cable, with the ground wire firmly connected to an electrical ground (safety ground) at the power outlet. Any interruption of the protective (grounding) conductor or disconnection of the protective earth terminal will cause a potential shock hazard that could result in personal injury.

Do Not Operate in an Explosive Atmosphere

Do not operate the instrument in the presence of flammable gases or fumes.

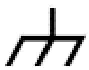
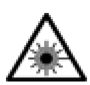
Do Not Remove the Instrument Cover

Operating personnel must not remove instrument covers. Component replacement and internal adjustments must be made only by qualified personnel.

Instruments that appear damaged or defective should be made inoperative and secured against unintended operation until they can be repaired by qualified service personnel.

Safety Symbols

Table 1 Safety Symbol

Symbol	Description
	Indicates warning or caution. If you see this symbol on a product, you must refer to the manuals for specific Warning or Caution information to avoid personal injury or damage to the product.
	Frame or chassis ground terminal. Typically connects to the equipment's metal frame.
	Indicates hazardous voltages and potential for electrical shock.
	Indicates that antistatic precautions should be taken.
	Indicates hot surface. Please do not touch.
	Indicates laser radiation turned on.
	CSA is the Canadian certification mark to demonstrate compliance with the Safety requirements.
	CE compliance marking to the EU Safety and EMC Directives. ISM GRP-1A classification according to the international EMC standard. ICES/NMB-001 compliance marking to the Canadian EMC standard.

Compliance and Environmental Information

Table 2 Compliance and Environmental Information

Safety Symbol	Description
	This product complies with WEEE Directive (2002/96/EC) marking requirements. The affixed label indicates that you must not discard this electrical/electronic product in domestic household waste. Product Category: With reference to the equipment types in WEEE Directive Annex I, this product is classed as a “Monitoring and Control instrumentation” product. Do not dispose in domestic household waste. To return unwanted products, contact your local Keysight office, or see http://about.keysight.com/en/companyinfo/environment/takeback.shtml for more information.

About This Guide

Here is how the information in this document is organized.

Introduction

This chapter provides an overview of this manual.

Know Your Hardware

This chapter provides an information on the various modules of M8020A/M8030A, their setup and the provided accessories.

Exploring M8070A User Interface

This chapter describes the M8070A user interface and the functionality provided by its common GUI elements.

Configuring Your System

This chapter describes how to configure the M8020A/M8030A system using the Module View, Group View and System View.

Setting up Generator

This chapter provides information on settings provided by the M8020A/M8030A Generator.

Setting up Analyzer

This chapter provides information on settings provided by the M8020A/M8030A Analyzer.

Setting up Patterns

This chapter describes the functionality provided by the M8070A Pattern Editor and Sequence Editor.

Working with Measurements

This chapter describes the setup, execution, monitoring and results of the measurements supported by M8070A system software.

Utilities

This chapter describes the utilities provided by the M8070A system software.

Licenses

This chapter provides information on the M8020A/M8030A and M8070A licenses and their installation procedure.

Troubleshooting

This chapter provides information that can help you to troubleshoot M8020A/M8030A in case any problem occurs.

Contents

10 Licenses

1 Introduction

Overview

J-BERT M8020A high-performance BERT

The Keysight J-BERT M8020A high-performance BERT enables fast, accurate receiver characterization of single-and multi-lane devices running up to 16 or 32 Gb/s.

With today's highest level of integration, the M8020A streamlines your test setup. In addition, automated in situ calibration of signal conditions ensures accurate and repeatable measurements.

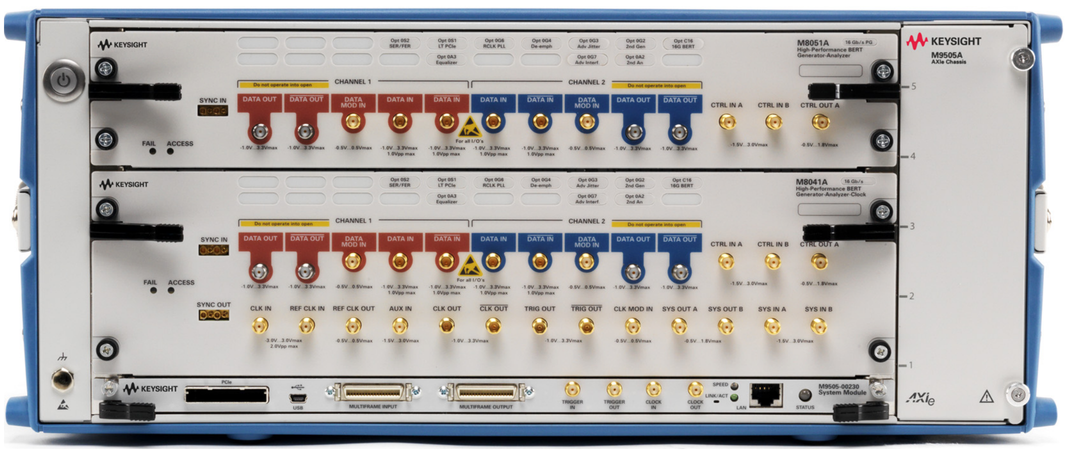


Figure 1 Typical base M8020A instrument configuration

Key Features

- Data rates up to 8.5 and 16 Gb/s expandable to 32 Gb/s
- 1 to 4 BERT channels in a 5-slot AXIe chassis
- Integrated and calibrated jitter injection: RJ, PJ1, PJ2, SJ, BUJ, sinusoidal level interference (common-mode and differential-mode), SSC (triangular and arbitrary, residual) and Clock/2
- 8 tap de-emphasis, positive and negative
- Integrated and adjustable Intersymbol Interference
- Interactive link training for PCI Express
- Built-in clock recovery and equalization
- All options and modules are upgradeable

M8020A Applications

The J-BERT M8020A is designed for R&D and test engineers who characterize and verify compliance of chips, devices, boards and systems with serial I/O ports up to 16 Gb/s and 32 Gb/s in the consumer, computer, mobile computing, data center and communications industry.

The J-BERT M8020A can be used to test popular serial bus standards, such as PCI Express®, SATA/SAS, DisplayPort, USB Super Speed, MIPI M-PHY, SD UHS-II, Fibre Channel, QPI, memory buses , backplanes, repeaters, active optical cables, Thunderbolt, 10/40 GbE/SFP+/QSFP, 100 GbE/CFP2.

M8030A Multi-Channel BERT

The M8030A Multi-Channel BERT extents the J-BERT M8020A platform to a real multi-channel BERT solution supporting up to 10 pattern generators and 10 error detectors (EDs) to allow multi-channel application tests like, PCIe and PON. Wherever multi-channel measurements are required in order to speed up throughput or test under real application conditions, the M8030A is a perfect solution.

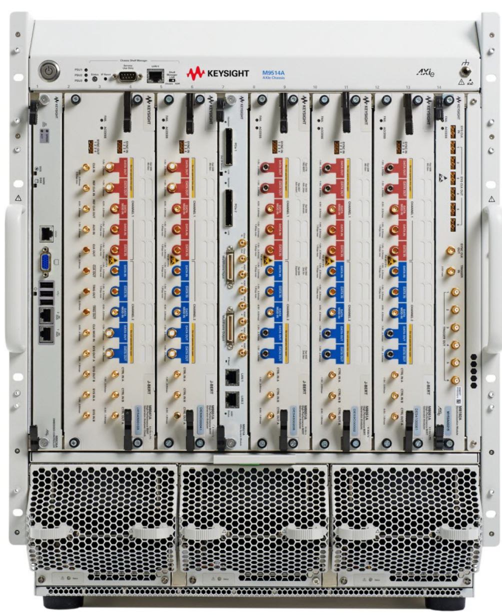


Figure 2 Typical base M8030A instrument configuration

Key Features

- Data rates up to 8.5 and 16 Gb/s
- 1 to 10 BERT pattern generator and analyzer and channels in a 14-slot AXIe chassis
- Clock synchronization between all modules
- 8 tap de-emphasis, positive and negative
- Integrated and adjustable ISI
- Built-in clock recovery and equalization

M8030A Applications

The M8030A multi-channel BERT is designed for R&D and test engineers who characterize and verify compliance for electronic chips, devices, boards, systems with multiple I/O ports (up to 16 Gb/s) in various industry segments dealing with basic consumer goods, computer devices, communication equipments, etc. Typical applications are:

- PCIe multi-channel test
- PON applications
- XAUI and GAUI multi-lane test

Document History

Table 3 Document History

Edition	Description
Edition 1.0, June 2014	Edition 1.0 of the M8070A user guide is in accordance to the M8070A version 1.0.
Edition 2.0, October 2014	Edition 2.0 of the M8070A user guide is in accordance to the M8070A version 1.5.
Edition 3.0, March 2015	Edition 3.0 of the M8070A user guide is in accordance to the M8070A version 2.0.
Edition 4.0, August 2015	Edition 4.0 of the M8070A user guide is in accordance to the M8070A version 2.5.
Edition 4.1, September 2015	Edition 4.1 of the M8070A user guide is in accordance to the M8070A version 2.6.

Edition	Description
Edition 4.2, January 2016	Edition 4.2 of the M8070A user guide is in accordance to the M8070A version 2.7.
Edition 5.0, February 2016	Edition 5.0 of the M8070A user guide is in accordance to the M8070A version 3.0.
Edition 5.1, March 2016	Edition 5.1 of the M8070A user guide is in accordance to the M8070A version 3.1.

Related Documents

Table 4 Related Documents

Document Part No.	Document Title
M8000-91010	Tips for Preventing Damage to M8020A and M8030A
M8000-91020	M8020A Start Here Document
M8020-91010	M8020A and M8030A Installation Guide
M8020-91020	M8020A and M8030A Getting Started Guide
M8020-91040	M8070A Programming Guide
M8070-91010	Getting Started with Keysight M8070A Plug-ins

Abbreviations used in this Document

Table 5 Abbreviations

Abbreviation	Extended Form
AXIe	AdvancedTCA Extensions for Instrumentation and Test
AWG	Arbitrary Waveform Generator
BER	Bit Error Ratio
CDR	Clock Data Recovery
CTLE	Continuous-Time Linear Equalizer
DUT	Device Under Test
ESD	Electrostatic Discharge
ESM	Embedded System Module

Abbreviation	Extended Form
GB	Gigabyte
GUI	Graphical User Interface
J-BERT	Jitter-Bit Error Ratio Tester
LED	Light-Emitting Diode
MB	Megabyte
MIPI	Mobile Industry Processor Interface
PC	Personal Computer
PCIe	Peripheral Component Interconnect Express
PLL	Phase Locked Loop
PRBS	Pseudorandom Binary Sequence
PXI	PCI eXtensions for Instrumentation
R & D	Research & Development
SAS	Serial Attached SCSI
SATA	Serial Advanced Technology Attachment
SCPI	Standard Commands for Programmable Instruments
SMA	SubMiniature Version A
SSC	Spread Spectrum Clock
TCP	Transmission Control Protocol
USB	Universal Serial Bus
VCO	Voltage Controlled Oscillator

2 Know Your Hardware

KEYSIGHT TECHNOLOGIES

M8020A Overview

The Keysight's J-BERT M8020A High-Performance BERT is a modular instrument which supports the following modules:

- M8041A high-performance BERT generator-analyzer-clock 8/16 Gb/s.
- M8051A high-performance BERT generator-analyzer 8/16 Gb/s.
- M8061A multiplexer 2:1 with de-emphasis 32 Gb/s.
- M8062A 32Gb/s Front-end for J-BERT M8020A High-Performance BERT
- In addition to above J-BERT modules, it also supports M8195A 65 GSa/s Arbitrary Waveform Generator. Details of this module can be found at www.keysight.com/find/m8195a.

M8020A being a modular product includes different sets of modules which are hosted in an AXI chassis. Each module and its features have their own license. You need to install these options in your instrument in order to use the modules or features. For details, refer to the chapter Licenses on page 433. However, if you have ordered M8020A-BU1, no license is required.

The M8041A module must be installed in slots 1 through 3 in the AXIe chassis unless the M9536A AXIe Embedded Controller is installed. The following configurations are possible in an M9505A 5-slot chassis:

- **1 or 2-channel, 16 Gb/s -** (1) M8041A
- **3 or 4-channel, 16 Gb/s -** (1) M8041A + (1) M8051A
- **1-channel, 32 Gb/s (Pattern Generator only) -** (1) M8041A + (1) M8061A
- **1-channel, 32 Gb/s (Pattern Generator only or full BERT) -** (1) M8041A + (1) M8062A
- M8195A module can more or less arbitrarily be added into the 5 slot module

Additionally, the M8061A, M8062A and M8195A modules can be installed and operated in a 2 slot frame.

For details on the features and hardware components of each of the above-mentioned modules, refer to M8020A / M8030A Modules on page 35.

M9505A AXIe Chassis

The M9505A AXIe Chassis is a modular instrument chassis that supports complex and high density testing. The chassis provides slots for installing multiple AXIe based instrument modules such as the M8041A, M8051A, M8061A and M8062A modules. Besides providing a frame for the installation of these instrument modules, the M9505A AXIe Chassis also provides power, a cooling system, a PCIe Gen2 local data bus, a Gigabit LAN interconnect, and a cabled USB (USB option required) and PCIe connection for external host computer connectivity.

The following model of the M9505A AXIe Chassis supports the M8020A modules:

- M9505A - a 5-Slot AXIe chassis

NOTE **The USB connection is recommended when using a laptop or desktop PC as an external controller. The PCIe connection is recommended for use with a desktop PC as an external controller only.**

Refer to the *Keysight M9505A AXIe Chassis Startup Guide* to get detailed information about the AXIe chassis.

M9505-00230 AXIe Embedded System Module (USB ESM)

The bottom slot of the AXIe chassis is reserved for the M9505-00230 Embedded System Module (ESM) which is factory-installed. The ESM has a USB 2.0 interface as well as a PCIe x8, Gen1 and Gen2 compliant interface to connect an external host computer to the chassis. The following figure shows the PCIe Port and USB Port in ESM.

The ESM:

- runs the chassis embedded operating system which manages all internal tasks and communications.
- tracks inserted modules and manages power requirements.
- monitors chassis temperature and controls variable-speed chassis fans.
- monitors module sensors and reports component failures to a system log.
- acts as a Gigabit Ethernet switch; forwards frames along the backplane.
- connects an external host computer to the chassis.
- synchronizes timing across all modules through the Keysight Trigger Bus, using an internal or external clock source.
- LAN connector on AXIe ESM is not used. Only use LAN connection on the host computer.
- Either the PCIe (desktop only) or USB (desktop or laptop) port can be used in this ESM but not both simultaneously. When you use the PCIe port, the USB port is automatically disabled until the PCIe port is no longer in use.

M8030A Overview

The M8030A is a modular instrument which supports the following modules.

- M8041A high-performance BERT generator-analyzer-clock 8/16 Gb/s
- M8051A high-performance BERT generator-analyzer 8/16 Gb/s
- M8092A Multi-channel synchronization module

The modules must be installed in the M9514A AXIe 14-slot chassis in the following way:

Table 6 M8030A Modules Arrangement

Slot Number	Module
# 1	M8030A-BU1 AXIe embedded controller. For M8030A-BU2 this slot is empty and covered with filler front-plane
# 2, 3 & 4	M8041A module
# 5 & 6	M8051A module
# 7	M9521A AXIe system module, always included in M8030A-BU1 or M8030A-BU2, must be in this slot
# 8 & 9	M8051A module
# 10 & 11	M8051A module
# 12 & 13	M8051A module
# 14	M8192A multi-channel synchronization module, always required in this slot

For details on the features and hardware components of each of the above mentioned modules, refer to *M8020A and M8030A Getting Started Guide*.

M9514A AXIe Chassis

The Keysight M9514A AXIe 14-slot chassis (one slot for the AXIe System Module plus 13 instrument module slots) is a modular instrument chassis fully compatible with the AXIe 1.0 Hardware specifications. It allows multiple application-specific instrument modules to share a common chassis frame, power supply, cooling system, PCI Express (PCIe) Gen 2 data bus, Gigabit LAN hub, local bus for module-to-module signaling, and host PC connections.

Multiple chassis may be interconnected for scalability. The chassis provides 13 general purpose peripheral slots that accept 1U AXIe instrument modules. Each module slot has a Gen 2 x4 link (maximum of 2 GB/s data rate per module) to the chassis primary data ‘fabric’ hub—a x8 PCIe switch and data bus.

The chassis requires a full module height AXIe System Module (ASM) such as the Keysight M9521A, to manage chassis functions.

.

NOTE **The USB connection is recommended when using a laptop or desktop PC as an external controller. The PCIe connection is recommended for use with a desktop PC as an external controller only.**

NOTE **PCIe connectivity between the M9514A AXIe Chassis and an external desktop PC controller is recommended when full channel plus large patterns need to be downloaded.**

Refer to the *Keysight M9514A AXIe Chassis Startup Guide* to get detailed information about the AXIe chassis.

M9521A AXIe System Module (ASM)

The M9521A AXIe System Module (ASM) is installed in the system slot of the M9514A (slot 7). It provides the system communication and synchronization functions required in an AXIe chassis including:

- Trigger bus and clock routing.
- Managing clocks, including internal or external reference sources.
- Gigabit LAN switching with front panel RJ45 LAN connections.
- AXIe Fabric 1 switching (Gen 2 x4 lanes to each module slot).

Keysight M9536A AXIe Embedded Controller Module

The M9536A AXIe Embedded Controller is a one slot module that you can install in the M9505A AXIe Chassis like any other instrument module. This module acts as a host computer when installed in the M9505A AXIe Chassis. It may be installed in any slot in the of the M9505A AXIe chassis except for Slot 7 which is reserved for the ASM. However, to eliminate interference with the local bus used for E-Keying (if your AXIe modules use E-Keying), you should install the controller in one of the outside slots; e.g., 1 or 14 first, then 2 or 13, etc.

The following figure displays this module.

The ESM:

- runs the chassis embedded operating system which manages all internal tasks and communications.
- tracks inserted modules and manages power requirements.
- monitors chassis temperature and controls variable- speed chassis fans.
- monitors module sensors and reports component failures to a system log.
- acts as a Gigabit Ethernet switch; forwards frames along the backplane.
- connects an external host computer to the chassis.
- synchronizes timing across all modules through the Keysight Trigger Bus, using an internal or external clock source.

LAN connector on AXIe ESM is not used. Only use LAN connection on the host computer.

Either the PCIe (desktop only) or USB (desktop or laptop) port can be used in this ESM but not both simultaneously. When you use the PCIe port, the USB port is automatically disabled until the PCIe port is no longer in use.

M8020A / M8030A Modules

The M8020A/M8030A modules are recognized by the model number and name located on their front panel.

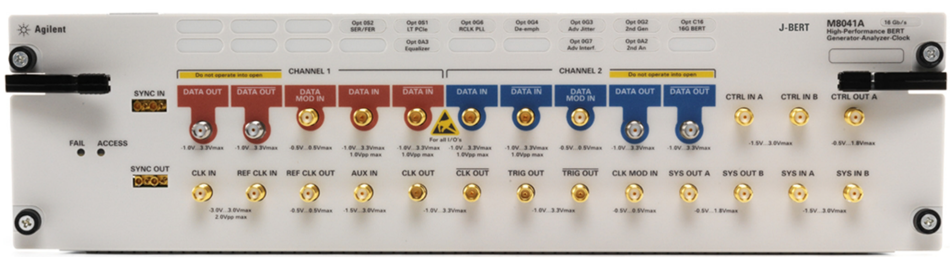
Each BERT module can be configured for 8 Gb/s or 16 Gb/s operation, as generator-only or as full BERT. Some upgraded features/components of a module are licensed and are only available when you purchase a license for that option. The M8062A module supports 32 Gb/s and the M8195A as a BERT module can support higher speeds than 16 Gb/s. The M8195A cannot be used as a full BERT. It has no analyzer.

The following sections describe each of the M8020A/M8030A instrument modules in detail.

J-BERT M8041A Generator-Analyzer-Clock Module

The J-BERT M8041A is a BERT module that can be installed into an Keysight M9505A 5-slot AXIe chassis. This module occupies three slots.

The M8041A is a two channel bit error ratio tester with built-in clock and data generator for performing compliance and characterization measurements. The second channel requires a license.



M8041A Features

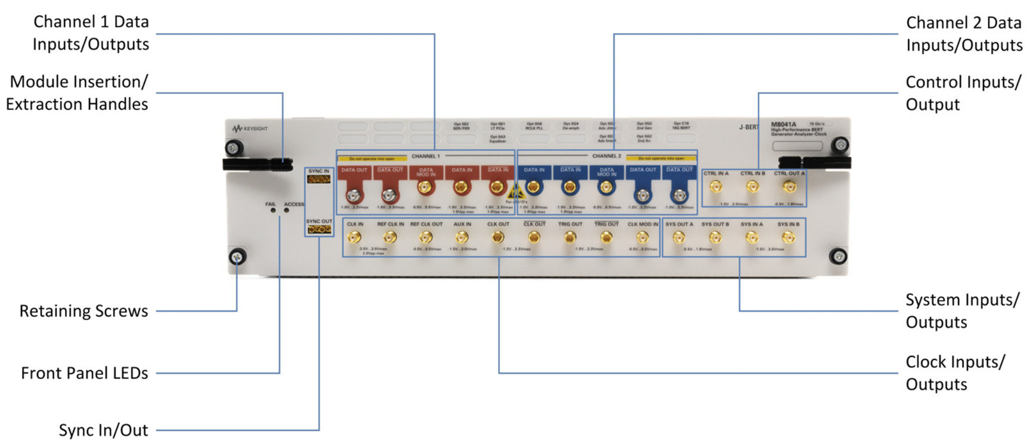
The M8041A module provides the following features:

- Two channel pattern generator (option 0G2) and two channel error detector (option 0A2)
- Data rate from 256 Mb/s to 16.2 Gb/s (option G16 or C16) for pattern generation and error detection
- Built in jitter injection (option 0G3)

- Adjustable ISI offered for M8041A and M8051A (option 0G5), software 2.0 and serial number >= DE55300500
- Built in 8 tap de-emphasis (option 0G4)
- Built in receiver equalization (CTLE, option 0A3)
- Built in reference clock multiplier for pattern generator (option 0G6)
- Simultaneous common mode and differential mode level interference (option 0G7)
- Interactive link training (option 0S1, Software 1.5)
- Four universal control inputs with adjustable threshold
- Three universal control outputs with adjustable levels
- 2 Gb pattern memory per channel (requires software 1.5)

M8041A Module Components

The following figure displays the front panel of the M8041A module with its various components labeled.



The M8041A module has the following components.

Table 7 Insertion/Extraction and Retaining

Component	Description
Retaining screws	The screws on both ends of the module are used to retain the module tightly inside the M9505A AXIe Chassis slot once you have fully placed it inside the chassis. To remove the module, you first need to loosen these screws ensuring that these screws disengage completely.
Module Insertion/Extraction Handles	The handles on both sides of the module to insert or eject the module from the slot of the M9505A AXIe Chassis.

Table 8 Front Panel LEDs

Connector Name	Active when...	Color
Fail	power-up fault condition	red
Access	power-up ready state	green
Data In x	input is overloaded	red
Data Out x	output is overloaded	red
Data Mod In x	input is active	green
Ctrl In A/Ctrl In B	logic level is detected	green
Ctrl Out A	output is active	green
Clk In	signal is detected	green
Ref Clk In	signal is detected	green
Ref Clk Out	output is active	green
Aux In	not used	n/a
Clk Out	output is active	green
Trig Out	output is active	green
Clk Mod In	input is active	green
Sys Out A/Sys Out B	output is active	green
Sys Ctrl In A/Sys Ctrl In B	logic level is detected	green

M8041A Front Panel Connector Inputs/Outputs

CAUTION The inputs of the M8041A module are sensitive to static electricity. Therefore, take necessary anti-static precautions, such as wearing a grounded wrist strap, to minimize the possibility of electrostatic damage.

Table 9 Channel x Data Inputs/Outputs

Component	Description
Data Out and /Data Out	Differential data outputs (3.5 mm, female).
Data In and /Data In	Differential data inputs (3.5 mm, female).
Data Mod In	Accepts an external source for data out delay modulation (SMA, female).

Table 10 Clock Inputs/Outputs

Component	Description
Clk In	External clock input in the range of 8.1 to 16.2 GHz. This input is used as a direct clock for all channels in forwarded clock applications (SMA, female).
Ref Clk In	Reference clock input for applications that provide a host reference clock in the range of 10 MHz to 16 GHz. The clock signal may be SSC modulated and is used as the reference for the system clock of all Tx and Rx channels. A SSC tolerant PLL is used to multiply the reference clock to the system clock (SMA, female).
Ref Clk Out	The reference clock output is used to provide a 10 MHz or 100 MHz reference clock to the DUT or other test equipment (SMA, female).
Clk Out and /Clk Out	Differential clock output (3.5 mm, female).
Trig Out and /Trig Out	This output is used to send a trigger signal to another connected device, such as an oscilloscope (3.5 mm, female). It can also be used as a sub rate clock.
Clk Mod In	Input for delay modulation of the Trig Out and Clk Out channel. Both outputs are always affected (SMA, female).

Table 11 Sync In/Sync Out

Component	Description
Sync In	This input is used to synchronize two or more modules to a common system clock. It is connected to the Sync Out of the other module.
Sync Out	This output is used to synchronize two or more modules to a common system clock. It is connected to the Sync In of the other module.

Table 12 System Inputs/Outputs

Component	Description
Sys Out A/Sys Out B	System level control outputs used to signal events to the DUT or external instruments (SMA, female).
Sys In A/Sys In B	System level control inputs used to generate sequencer events (SMA, female).

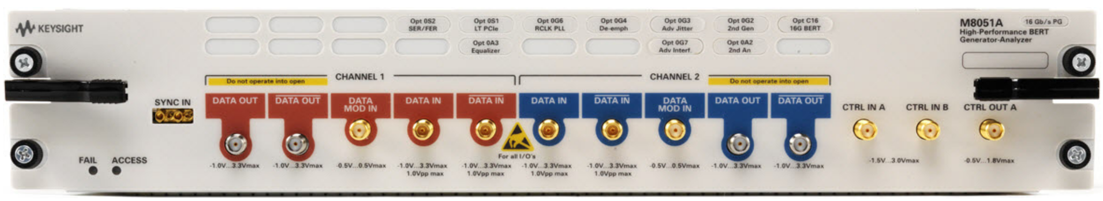
Table 13 Control Inputs/Output

Component	Description
Ctrl In A/Ctrl In B	The module has two control inputs at the font panel each with the following selectable functionality (SMA, female): **Error Add Input** Every rising edge at the input generates a single error in the output data stream by flipping a single bit. The maximum repetition rate is data rate divided by 4 times the vector size. **Output Blanking** If the input level is above the threshold level the pattern generator stops and only 0's are sent on data output. Normal operation resumes when the input level is below the threshold. **Electrical Idle** If the input level is above the threshold level the output amplifier enters electrical idle. Normal operation resumes when the input level is below the threshold. **Gating Input** If a logical high is applied to the gating input the error detector will ignore the incoming bits during a BER measurement. The ignored bit sequence is always a multiple of the vector size. For measuring data in bursts of bits, rather than a continuous stream of bits, a special operating mode is used. This mode is the burst sync mode. In this case, the signal at the gating input controls the synchronization and the error counting for each burst.
Ctrl Out A	The module has one control output at the front panel with the following functionality (SMA, female): **Error Output** This signal can be used to trigger an external instrument to help in error analysis. If an error occurs, a single RZ pulse is generated. Continuous errors will result in a clock signal.

J-BERT M8051A Generator-Analyzer

The J-BERT M8051A is an instrument module that can be installed into the M9505A 5-slot AXIe Chassis. This module occupies two slots and requires the M8041A module for proper operation.

The M8051A is a two channel Generator and two channel Analyzer for performing compliance and characterization measurements.

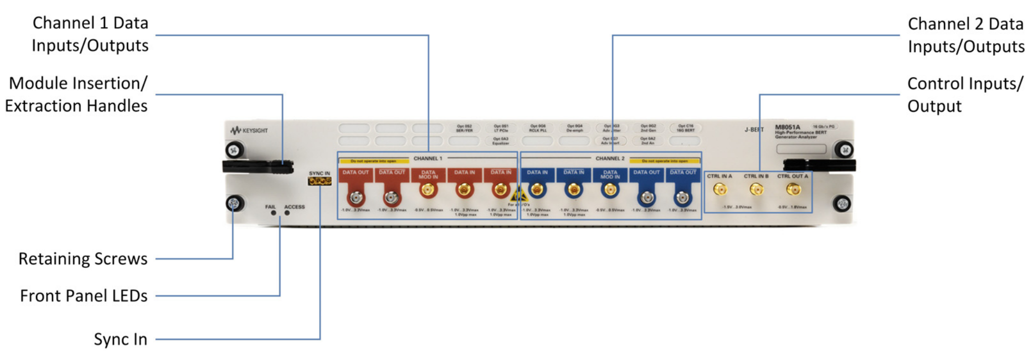


M8051A Features

The main M8051A features are the same as the M8041A features. For details, refer to J-BERT M8041A Generator-Analyzer-Clock Module on page 35.

M8051A Module Components

The following figure displays the front panel of the M8051A module with its various components labeled.



The M8051A module has the following components.

Table 14 Insertion/Extraction and Retaining

Component	Description
Retaining screws	The screws on both ends of the module are used to retain the module tightly inside the M9505A AXIe Chassis slot once you have fully placed it inside the chassis. To remove the module, you first need to loosen these screws ensuring that these screws disengage completely.
Module Insertion/Extraction Handles	The handles on both sides of the module to insert or eject the module from the slot of the M9505A AXIe Chassis.

Table 15 Front Panel LEDs

Connector Name	Active when...	Color
Fail	power-up fault condition	red
Access	power-up ready state	green
Data In x	input is overloaded	red
Data Out x	output is overloaded	red
Data Mod In x	input is active	green
Ctrl In A/Ctrl In B	logic level is detected	green

M8051A Front Panel Connector Inputs/Outputs

CAUTION The inputs of the M8051A module are sensitive to static electricity. Therefore, take necessary anti-static precautions, such as wearing a grounded wrist strap, to minimize the possibility of electrostatic damage.

Table 16 Channel x Data Inputs/Outputs

Component	Description
Data Out and /Data Out	Differential data outputs (3.5 mm, female).
Data In and /Data In	Differential data inputs (3.5 mm, female).
Data Mod In	Accepts an external source for data out delay modulation (SMA, female).

Table 17 Sync In

Component	Description
Sync In	This input is used to synchronize two or more modules to a common system clock. It is connected to the Sync Out of the other module or to the clock distribution module if more than two modules are installed. The sync cable is required if M8051A is connected with M8041A module.

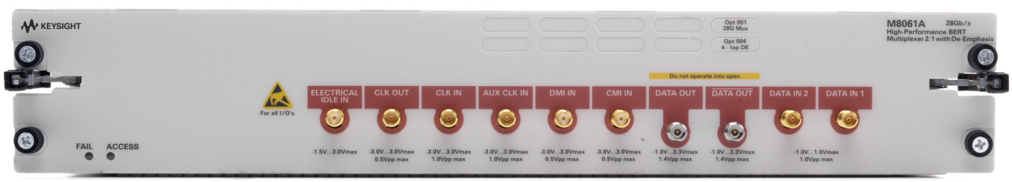
Table 18 Control Inputs/Output

Component	Description
Ctrl In A/Ctrl In B	The module has two control inputs at the font panel each with the following selectable functionality (SMA, female): **Error Add Input** Every rising edge at the input generates a single error in the output data stream by flipping a single bit. The maximum repetition rate is data rate divided by 4 times the vector size. **Output Blanking** If the input level is above the threshold level the pattern generator stops and only 0's are sent on data output. Normal operation resumes when the input level is below the threshold. **Electrical Idle** If the input level is above the threshold level the output amplifier enters electrical idle. Normal operation resumes when the input level is below the threshold. **Gating Input** If a logical high is applied to the gating input the error detector will ignore the incoming bits during a BER measurement. The ignored bit sequence is always a multiple of the vector size. For measuring data in bursts of bits, rather than a continuous stream of bits, a special operating mode is used. This mode is the burst sync mode. In this case, the signal at the gating input controls the synchronization and the error counting for each burst.
Ctrl Out A	The module has one control output at the front panel with the following functionality (SMA, female): **Error Output** This signal can be used to trigger an external instrument to help in error analysis. If an error occurs, a single RZ pulse is generated with the width of half a vector length. Continuous errors will result in a clock signal.

M8061A 32 Gb/s Multiplexer with De-emphasis Module

The M8061A is an instrument module that can be installed into the M9502A 2-slot or M9505A 5-slot AXIe Chassis. This module occupies two slots.

The M8061A is used to characterize serial interfaces of up to 32 Gb/s. The M8061A provides four calibrated de-emphasis taps, which can be extended to eight taps, built-in superposition of level interference, and Clock/2 jitter injection.

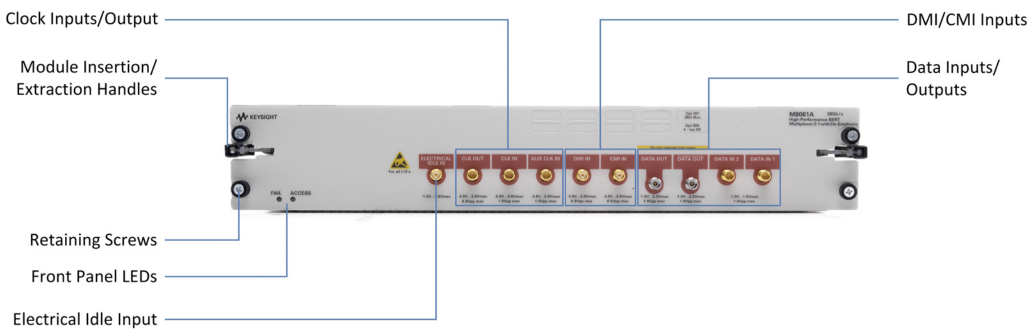


M8061A Features

- Expands data rate of M8041A and M8051A generators up to 32 Gb/s enabling accurate and complete receiver stress testing
- Integrated and calibrated 4-tap de-emphasis, expandable to 8 taps
- Internal superposition of interference for common mode and differential mode
- Transparent to jitter generated by the J-BERT M8020A, Clock/2 jitter can be added
- Supports electrical idle
- Control from J-BERT M8020A user interface via USB.

M8061A Module Components

The following figure displays the front panel of the M8061A module with its various components labeled.



The M8061A module has the following components.

Table 19 Insertion/Extraction and Retaining

Component	Description
Retaining screws	The screws on both ends of the module are used to retain the module tightly inside the M9505A AXIe Chassis slot once you have fully placed it inside the chassis. To remove the module, you first need to loosen these screws ensuring that these screws disengage completely.
Module Insertion/Extraction Handles	The handles on both sides of the module to insert or eject the module from the slot of the M9505A AXIe Chassis.

Table 20 Front Panel LEDs

Connector Name	Active when...	Color
Fail	power-up fault condition	red
Access	power-up ready state	green

M8061A Front Panel Connector Inputs/Outputs

CAUTION The inputs of the M8061A module are sensitive to static electricity. Therefore, take necessary anti-static precautions, such as wearing a grounded wrist strap, to minimize the possibility of electrostatic damage.

Table 21 Electrical Idle Input

Component	Description
Electrical Idle In	This input is used to enable/disable the output signal by an external control signal. If the input level is above the threshold level the module enters electrical idle. Normal operation resumes when the input level is below the threshold (SMA, female).

Table 22 Clock Inputs/Output

Component	Description
Clk Out	Reference clock output used in clean clock mode to provide the synthesizer signal to the J-BERT without external splitter (3.5 mm, female).
Clk In	Reference clock input used in clean clock mode, the synthesizer should be connected to this port (3.5 mm, female).
Aux Clk In	External clock input in the range of 150 MHz to 14.2 GHz. This input is used with the J-BERT's internal clock. In presence of jitter, this provides the same jitter as the data outputs (3.5 mm, female).

Table 23 DMI/CMI Inputs

Component	Description
DMI In	Differential mode interference input independent of ground (SMA, female).
Clk In	Common mode interference input relative to ground (SMA, female).

Table 24 Data Inputs/Outputs

Component	Description
Data Out and /Data Out	Differential or single-ended data output (2.4 mm, female).
Data In x	Single-ended data input (3.5 mm, female).

M8062A 32Gb/s Front-end for J-BERT M8020A High-Performance BERT

The M8062A extends the data rate of the J-BERT M8020A Bit Error Ratio Tester to the speeds required for testing devices with lane rates in the 25-28 Gb/s range. When combined with a two channel M8041A, the system provides data pattern generation and full-rate error analysis for users and systems with lane rates up to 32.4 Gb/s.

M8062A Features

- Extends maximum data rate of J-BERT M8020A up to 32.4 Gb/s
- Seamless control of pattern generator and error analyzer
- Integrated 8-tap de-emphasis
- Built in ISI generator for channel emulation
- Analyzer equalization eliminates errors resulting from closed eyes in loop back path
- Built in CDR for data rates up to 32 Gb/s

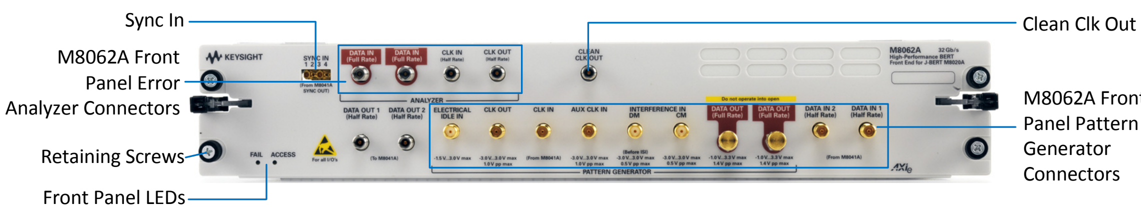
NOTE The CDR license (M8062A-0A4) is required to enable the CDR feature. M8062A modules with serial numbers < MY55400300 may also require a hardware upgrade in order to enable this feature.

Refer to the *Online Help* installed and integrated into the M8070A software to learn about how to use this module.

NOTE Phase-matched cables must be used when connecting the M8041A data and clock outputs to the M8062A data and clock inputs. The provided cable set, Keysight M8062-61643, meets this requirement.

M8062A Module Components

The following figure displays the front panel of the M8062A module with its various components labeled.



The M8062A module has the following components.

Table 25 Insertion/Extraction and Retaining

Component	Description
Retaining screws	The screws on both ends of the module are used to retain the module tightly inside the M9505A AXIe Chassis slot once you have fully placed it inside the chassis. To remove the module, you first need to loosen these screws ensuring that these screws disengage completely.
Module Insertion/Extraction Handles	The handles on both sides of the module to insert or eject the module from the slot of the M9505A AXIe Chassis.

Table 26 Front Panel LEDs

Connector Name	Active when...	Color
Fail	power-up fault condition	red
Access	power-up ready state	green

CAUTION

The inputs and outputs of the M8062A module are sensitive to static electricity. Therefore, take necessary anti-static precautions, such as wearing a grounded wrist strap, to minimize the possibility of electrostatic damage.

Table 27 Sync In/Clean Clk Out

Connector Name	Description
Sync In	This input is used to synchronize two or more modules to a common system clock. It is connected to the Sync Out of the other module. The sync cable is required if M8062A is connected with M8041A module.
Clean Clk Out	Half-rate, or divided, clock output with no applied jitter.

M8062A Front Panel Pattern Generator Connectors

Table 28 Electrical Idle Input

Component	Description
Electrical Idle In	This input is used to enable/disable the output signal by an external control signal. If the input level is above the threshold level the module enters electrical idle. Normal operation resumes when the input level is below the threshold (SMA, female).

Table 29 Pattern Generator Clock Inputs/Output

Component	Description
Clk Out	Half-rate Pattern Generator clock output. Carries the same jitter as the full-rate data output.
Clk In	Pattern Generator clock input (half-rate). Connect to clock output of M8041A.
Aux Clk In	Alternate Pattern Generator clock input (half-rate). Typically unused.

Table 30 DMI/CMI Inputs

Component	Description
DMI In	Differential Mode Interference input. Applies a single-ended, external interference source differentially to the data output (SMA, female).
CMI In	Common Mode Interference input. Applies a single-ended, external interference source to both the normal and complement data output signals (SMA, female).

Table 31 Pattern Generator Data Inputs/Outputs

Component	Description
Data Out and /Data Out	Differential or single-ended, full-rate data output to the device under test. Unused outputs must be terminated into 50 Ω. (2.4 mm, female).
Data In 1 and Data In 2	Single-ended, half-rate data inputs from the M8041A module (3.5 mm, female).

M8062A Front Panel Analyzer Connectors

Table 32 Error Analyzer Data Inputs/Outputs

Component	Description
Data In and /Data In	Differential or single-ended, full-rate data input from the device under test. Unused input should be terminated into 50 Ω. (2.4 mm, female). These ports are AC coupled.
Data Out 1 and Data Out 2	Single-ended, half-rate data outputs to the M8041A module (3.5 mm, female).

Table 33 Error Analyzer Clock Inputs/Output

Component	Description
Clk Out	Half-rate Error Analyzer clock output, synchronous with analyzer sampling.
Clk In	Half-rate, Error Analyzer clock input. Allows external clocking of the Error Analyzer.

M8195A 65 GSa/s Arbitrary Waveform Generator

The In addition to above J-BERT modules, it also supports M8195A 65 GSa/s Arbitrary Waveform Generator (AWG) which can be accessed through the M8070A system software. Currently, it just supports M8195A Revision 1.

The Keysight M8195A Arbitrary Waveform Generator comes with the highest combination of speed, bandwidth and channel density. Flexible signal generation at up to 32 Gbaud. Clean and distorted signal to stress the device to the limits.

High speed AWG with up to 65 GSa/s sample rate and 20 GHz bandwidth on up to 4 channels per module. The M8195A arbitrary waveform generator offers an output amplitude of up to 2 Vpp(diff.) and adjustable DC offset. Multi-channel operation with up to 16 channels per 5-slot AXIe chassis is supported.

In 5 slots per 4 channels the maximum number of channels is 20. This is valid for channels which are not synchronized across module boundaries. When adding an AWG synch module which also occupies a single slot, the AWG modules can be used synchronously in the future but then only 4 modules can be inserted into the frame. But for the non-synchronous startup, 5 modules can be put into a single frame resulting in 20 channels.

Details of M8195A module can be found at www.keysight.com/find/m8195a.

Host Computer

A host computer is used to:

- host all the software components of the instrument modules needed to control, configure, and use the modules.
- communicate with the ESM of the M9505A AXIe Chassis to allow you to monitor and control the chassis.

A host computer can be:

- the M9536A AXIe Embedded Controller module.
- a laptop with a USB port or with PCIe port.
- a desktop PC with a USB port or x8 or wider PCIe slot for the cabled PCIe adapter card.

Keysight M9536A AXIe Embedded Controller Module

The M9536A AXIe Embedded Controller is a one slot module that you can install in the M9505A AXIe Chassis like any other instrument module. This module acts as a host computer when installed in the M9505A AXIe Chassis. It is always installed in slot 1 of the M9505A AXIe Chassis.

The following figure displays this module.

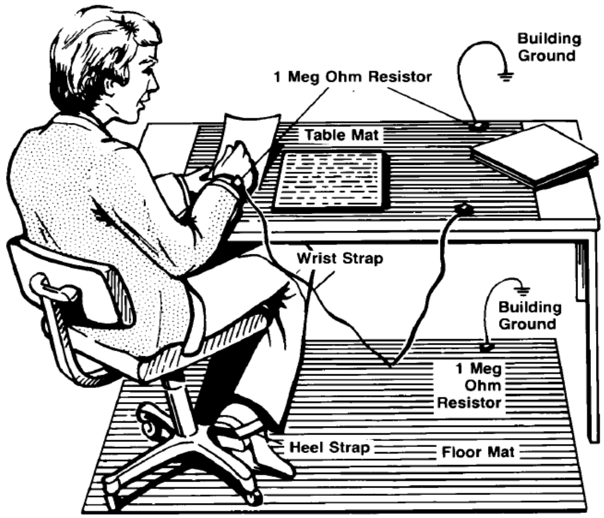
ESD Protection

CAUTION

Electrostatic discharge (ESD) can damage the circuits of the components on M8020A/M8030A modules (M8041A, M8051A, M8061A and M8062A). Avoid applying static discharges to the front-panel connectors. Before connecting any coaxial cable to the connectors, momentarily short the center and outer conductors of the cable together. Avoid touching the front-panel connectors without first touching the frame of the instrument. Be sure the instrument and all connected devices (DUT, etc.) are properly earth-grounded (to a common ground) to prevent buildup of static charge and electrical over-stress. Take necessary anti-static precautions, such as wearing a grounded wrist strap, to minimize the possibility of electrostatic damage.

Electrostatic discharge (ESD) can damage or destroy electronic components. All work on electronic assemblies should be performed at a static-safe work station. The following list and figure shows an example of a static-safe work station using two types of ESD protection. Purchase acceptable ESD accessories from your local supplier.

- Conductive table-mat and wrist-strap combination.
- Conductive floor-mat and heel-strap combination.



Both types, when used together, provide a significant level of ESD protection. Of the two, only the table-mat and wrist-strap combination provides adequate ESD protection when used alone. To ensure user safety, the static-safe accessories must provide at least 1 MW of isolation from ground.

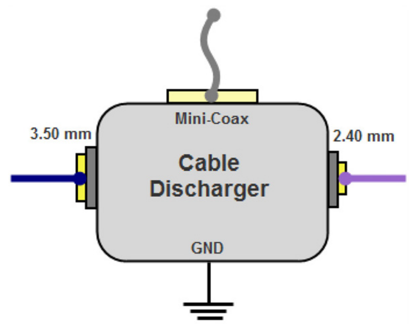
WARNING **These techniques for a static-safe work station should not be used when working on circuitry with a voltage potential greater than 500 volts.**

Discharging Cables

Loose cables are like a capacitor and can hold electrostatic charges. The free end of a cable touching surfaces that have voltage levels will cause product damage. Before connecting any cable to product connector, short the center and outer conductors of the cable together to ground momentarily.

You should use the cable discharger provided with the initial product shipment and shown in the following figures.

While discharging a cable, make sure to ground the box appropriately, via the “GND” connector of the box, to the ground connector of the AXIe chassis as shown in the figure.



That is either directly using the accessories provided with the discharger like the grounding cable, or via an ESD mat, which is connected to the ground connector of the AXIe chassis.

Discharge your cables using the matching connector e.g. 2.40 mm (also for 1.85 mm), 3.50 mm (also for 2.92 mm) and Mini-Coax. You may stick the cable discharger box to your instrument/AXIe chassis e.g. using the fastener tape provided.

Fixture made of plastic can store charges, and probing powered devices can subject inputs to damaging voltage and power levels. Poor AC power supply connected to product or DUT may create AC transients, insufficient grounding, floating neutral lines which cause damaging currents to flow into or out of the instrument.

For more information about electrostatic discharge, contact the Electrostatic Discharge Association www.esda.org.

M8020A Module Setup

M8020A being a modular product includes different sets of modules which are hosted in an AXI chassis. It comprises of exactly one M8041A generator-analyzer-clock module and optionally one additional M8051A generator-analyzer module. The M8041A generator-analyzer-clock module is a true superset of the M8051A generator-analyzer module.

Setting up a Single Channel System

The single channel system is the smallest configuration consisting of one M8041A generator-analyzer-clock module within a 5-slot frame. You can upgrade it to a 2 channel system by adding the two channel options (second channel generator and second channel analyzer).

Setting up a Multi-Channel System

The multi-channel system is comprised of exactly one clock / data module and one or more data channels mounted in an AXIe frame.

The multi-channel system can be:

2-Channel System

A two-channel system consists of one clock/data module.

The following figure illustrates a two-channel system.

4-Channel System

A four-channel system consists of one clock module and one data modules.

The following figure illustrates a four-channel system.

For more details on how to establish connections between the M8020A modules, refer to the *M8020A/M8030A Installation Guide*.

M8030A Module Setup

The M8030A is a modular test solution which can be tailored to your specific needs from two channels with one M8041A to up to 10 channels. The modules must be installed in the M9514A AXIe 14-slot chassis as described in Table 34 on page -58:

Table 34 M8030A Modules Configuration

Slot Number	Module
# 1	M8030A-BU1 AXIe embedded controller.
# 2, 3 & 4	M8041A module
# 5 & 6	M8051A module
# 7	M9521A AXIe system module
# 8 & 9	M8051A module
# 10 & 11	M8051A module
# 12 & 13	M8051A module
# 14	M8192A multi-channel synchronization module

M8030A Modules Arrangement Example

The following figure shows the M8030A modules arrangement in the M9514A AXIe 14-slot chassis:

Slot #1	Slot #2	Slot #3	Slot #4	Slot #5	Slot #6	Slot #7	Slot #8	Slot #9	Slot #10	Slot #11	Slot #12	Slot #13	Slot #14
M9536A AXIe Embed-ded control-ler	M8041A J-BERT High-performance BERT module			M8051A J-BERT High-performance BERT module		M9521A AXIe System Module	M8051A J-BERT High-performance BERT module		M8051A J-BERT High-performance BERT module		M8051A J-BERT High-performance BERT module		M8192A AXIe Clock Sync module

Accessories

Default Accessories

The M8020A/M8030A modules comes with the following default accessories:

Table 35 M8020A/M8030A Default Accessories

M8041A module	Eight 50 Ω terminations, commercial calibration report ("UK6"), certificate of calibration, ESD protection kit.
M8051A module	Four 50 Ω terminations, clock synchronization cable, commercial calibration report ("UK6"), certificate of calibration
M8061A module	See M8061A data sheet.
M8062A module	Two 50 Ω terminations, two individual semi-rigid cables, one set of three phase matched semi-rigid cables, clock synchronization cable, commercial calibration report ("UK6"), certificate of calibration.
M8020A-BU1	M9505A AXIe chassis with embedded controller, USB cable, getting started guide, AXIe filler panel, power cord
M8020A-BU2	M9505A AXIe chassis, USB cable, getting started guide, AXIe filler panel, power cord
M8030A-BU1	M9514A AXIe chassis with embedded controller, USB cable, getting started guide, AXIe filler panel, power cord
M8030A-BU2	M9514A AXIe chassis, USB cable, getting started guide, AXIe filler panel, power cord
M8070A	CD-ROM with M8070A system software

Recommended Accessories

The following are the recommended accessories for M8020A/M8030A modules.

Table 36 Recommended Accessories

Matched cable pair, 2.92 mm (m) to 2.92 mm (m), 0.85 m (recommended for each data output of M8041A/51A. This 2.92 mm cable is compatible with 3.5 mm front panel connectors of M8041A/51A.)	M8041A-801
Bandpass filter 11.4 to 15.6 GHz, SMA (for use with M8061A in clock path to minimize intrinsic RJ of M8061A for data rate of 25.78 Gb/s)	M8061A-802
Bandpass filter 11.1 to 17.5 GHz, SMA (for use with M8061A in clock path to minimize intrinsic RJ of M8061A for data rates from 25.0 to 32.0 Gb/s)	M8061A-803
Cable kit for connecting M8061A with M8020A, 3x 3.5 mm, 0.6 m	M8061A-804
Matched cable pair, 2.4 mm (m) to 2.4 mm (m), 0.85 m (recommended for full-rate data output and input of the M8062A)	N4910A
DC block, 26 GHz, 3.5 mm	N9398C
ISI channels, four short traces	M8048A-001
ISI channels, four long traces	M8048A-002
Short matched cable pair, SMA (m) to SMA (m) for cascading M8048A ISI channels	
Four SMA cables, unmatched	15442A
Rack-mount kit for AXIe 5-slot chassis M9505A	Y1226A

General Characteristics

The general characteristics of the modules included in M8020A/M8030A system are listed in Table 37 on page -62.

Table 37 General Characteristics

	Instrument Models		
	M8041A	M8051A	M8061A / M8062A
Operating temperature	5 °C to 40 °C (41 °F to 104 °F)		
Storage temperature	-40 °C to +70 °C (modules) (-40 °F to + 158 °F)		
Operating humidity	15% to 95% relative humidity at 40°C (non-condensing)		15% to 95% relative humidity at 35°C (non-condensing)
Storage humidity	24% to 90% relative humidity at 65°C (non-condensing)		
Power requirements	350 W	250 W	90 VA
Warm-up time	30 minutes		
Cooling requirements	Slot airflow direction is from right to left. When operating the M8041A/51A/61/62A choose a location that provides at least 80 mm of clearance at rear, and at least 50 mm (2 “) of clearance at each side. See also start-up guide for M9502A chassis.		
Recommended recalibration period	1 year		
Warranty period	3 years return to Keysight		

For general characteristics of M8195A, refer to data sheet. Visit the web-page www.keysight.com/find/m8195a.

Keysight M8070A System Software for M8000 Series of BER Test Solutions

User Guide

3 Quick Tour with M8070A User Interface

KEYSIGHT TECHNOLOGIES

Overview

The M8070A system software for the M8000 Series of BER Test Solutions is required to control M8020A/M8030A modules (M8041A, M8051A, M8061A, M8062A and M8195A Revision 1). It provides a user-friendly experience that can be used with standard or touchscreen enabled computers. It is fully supported by the Microsoft Windows 7 (64 bit) SP1, Windows 8 (64 bit) and Windows 8.1 (64 bit) operating system. The M8070A software also supports an off-line version, that does not require any license to operate.

The M8070A user interface provides an interactive graphical display in multiple windows containing controls, that enable you to perform the tasks. The GUI components include menus, tool bars, dialog boxes, toggle buttons, standard windows buttons, drop-down lists, sliders, and many more which are further discussed in this manual.

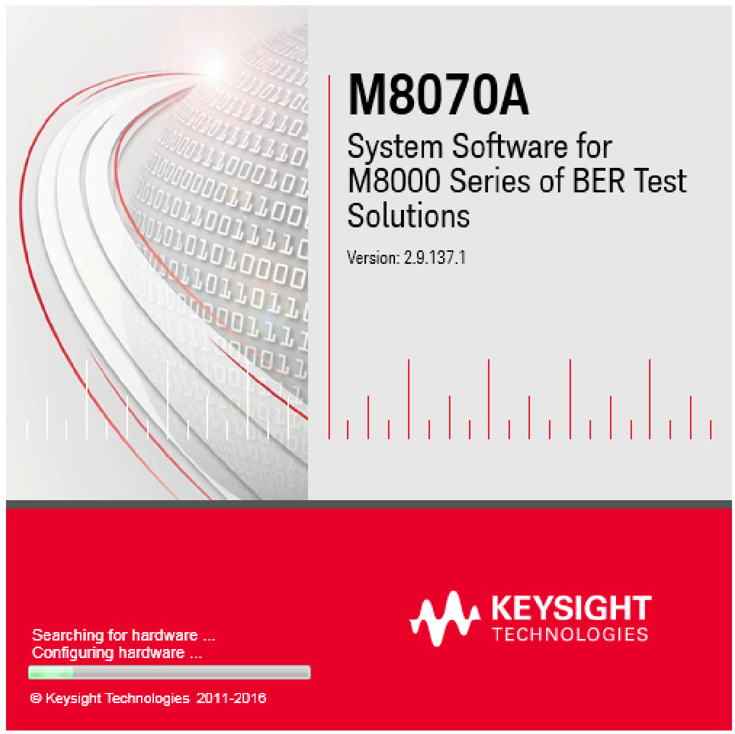
NOTE

Depending upon the M8020A/M8030A bundle with of chassis/controller pre-installation, you may have to install the M8070A software. Refer to *M8020A and M8030A Installation Guide* and follow the given instructions to install the M8070A software.

Launching M8070A User Interface

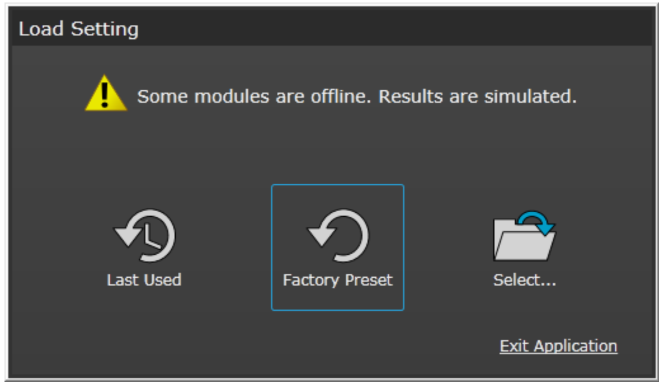
To launch the M8070A user interface, go to **Start** menu and then click **All Programs** > **Keysight M8070A** > **Keysight M8070A**.

The splash screen will be displayed as shown in the following figure:



Load Setting

Before the M8070A software is launched, a **Load Setting** dialog will appear which allows you to load settings.



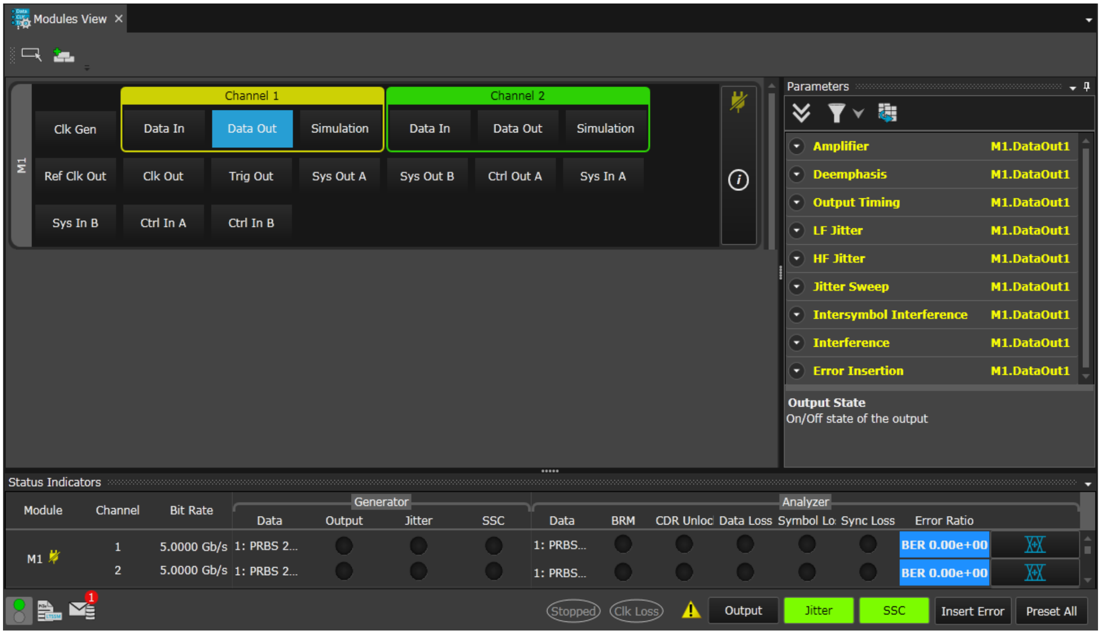
The **Load Setting** dialog provides the following settings:

- **Last Used**: Launches the M8070A user interface with the last used settings.
- **Factory Preset**: Launches the M8070A user interface with factory default settings.
- **Select...**: Opens the **Recall Instrument State** dialog which allows you to load the M8070A user interface with the stored settings. For details, refer to Recall Instrument State on page 103.
- **Exit Application**: Terminates the M8070A application.

NOTE The **Load Setting** dialog will appear each time the M8070A software is launched.

NOTE A warning message will appear on the top of **Load Setting** dialog if the modules are offline. In this case, the results are simulated.

Once the settings are loaded, the M8070A software launches. The following figure shows the M8070A user interface when there is only one module connected to the M8020A/M8030A system.

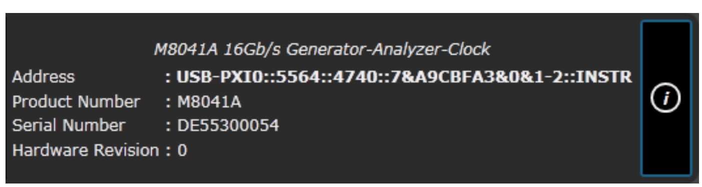


However, if there are two modules connected to the M8020A/M8030A system, the M8070A user interface will appear as shown in the following figure:

Get Module Information

You can get the module information that is connected to

the M8020A/M8030A by clicking the icon present at the right side of each module. If properly connected, the module information will be shown as depicted in the following figure:



It provides the following information about the module:

- **Address** - Address of module, e.g. USB-PXIO::11::0::INSTR
- **Product Number** - Product no. of the module, e.g. M8041A

- **Serial Number** - Serial no. of the module, e.g. DE53C00061
- **Hard ware Revision** - Hardware revision of module, e.g. 0

However, if you unable to get the module information, we suggest you to restart the M8070A software.

In the off-line mode, the icon will appear in the M8070A GUI.

Exploring M8070A User Interface

The M8070A user interface consists of the following GUI elements:

- Title Bar
- Menu Bar
- Main Window
- Status Bar

The detailed information on these GUI elements are described in the following sections.

Title Bar

The title bar contains an application icon, title, a context-sensitive help icon and standard buttons to minimize, maximize or to close the window.

The ? context-sensitive help icon provides information about the M8070A user interface relative to the task a user performs.

The title bar is shown in the following figure.

Menu Bar

The menu bar consists of various drop-down menus which provide access to different functions, and launch interactive GUI controls.

The menu bar includes the following drop-down menus:

- File
- Application
- System
- Clock
- Generator
- Analyzer
- Patterns

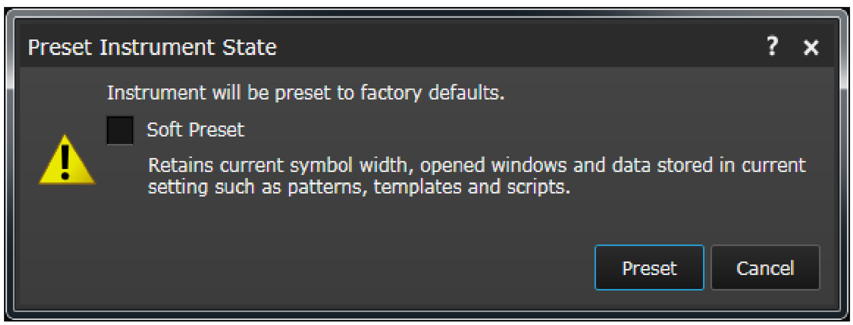
- Measurements
- Utilities
- Window
- Help
- Increase/Decrease Splitter Size

Each drop-down menu and their options are described in the following sections.

File Menu

The File menu provides the following selection:

- **Preset Instrument State...** - Opens the **Preset Instrument State** dialog.



This dialog allows you to:

- **Soft Preset** - For a **Soft Preset**, select the check box and then click **Preset** button. This option resets the instrument state to factory default settings. However, it retains current symbol width, opened windows and data stored in current setting such as patterns, templates and scripts.
- **Preset** - Resets the parameters to their default values.
You can also perform a soft preset/preset to the instrument by sending *RST command in the SCPI panel. For details on *RST command, refer to *M8070A Programming Guide*.

- **Recall Instrument State...** - Opens the **Recall Instrument State** dialog which allows you to retrieve the user-defined settings or the factory settings.
For more details, refer to Recall Instrument State on page 103.

- **Save Instrument State...** - Opens the **Save Instrument State** dialog which allows you to save the current instrument state.
  For more details, refer to Save Instrument State on page 104.
- **Exit** - Closes the M8070A user interface.

Application Menu

The **Application** menu allows you to select application specific features for the following:

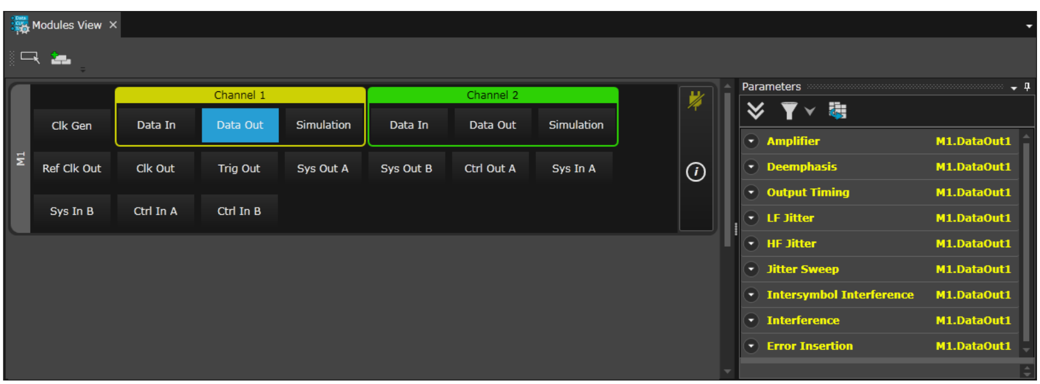
- PCI Express
- USB
- SATA
- C-PHY (Included in separately available MIPI plug-in)
- D-PHY (Included in separately available MIPI plug-in)

On selecting any of these standards from the **Application** menu, a standard specific dialog or plug-in interface will appear.

System Menu

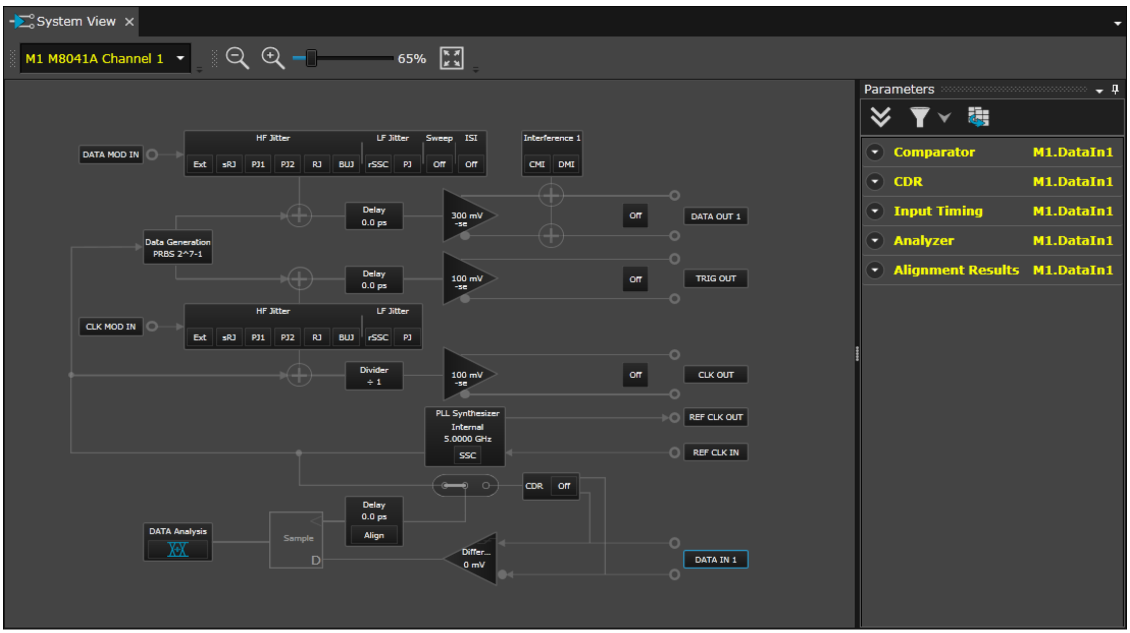
The System menu allows you to launch the various display views provided by M8020A/M8030A. It provides the following selections:

- **Module View** - Opens the **Module View** user interface as shown in the following figure:



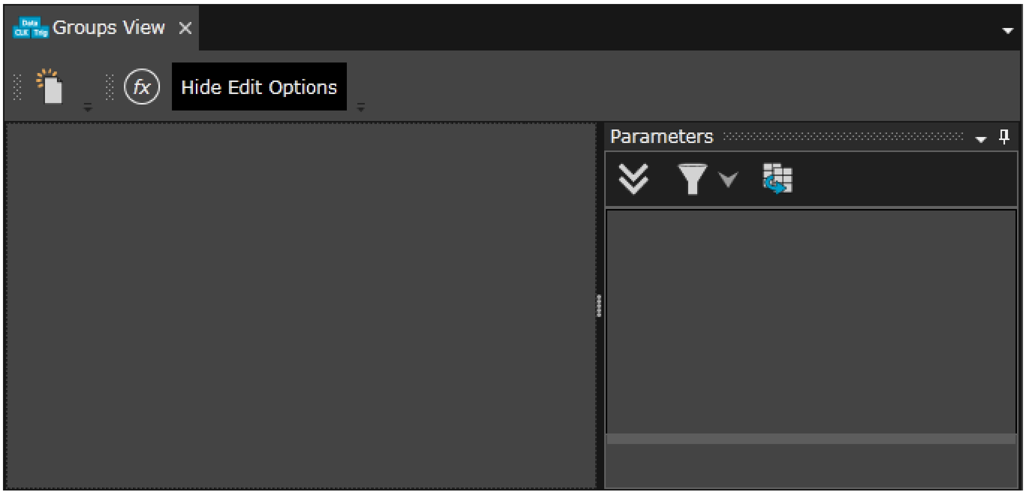
The **Module View** is a graphical representation of the input/output ports that are present on the front panel of the modules, connected to the M8020A/M8030A. You can use the Module View to configure the properties of a single port or a group (combination of multiple ports). For details, refer to Module View on page 107.

- **System View** - Opens the **System View** user interface as shown in the following figure:



The **System View** displays the block representation of the currently selected channel of the M8020A/M8030A. In addition, it also allows you to interactively modify the configuration settings for each channel. For details, refer to System View on page 119.

- **Groups View** - Opens the **Group View** user interface as shown in the following figure:



The **Group View** allows you to create a group of ports and re-program their properties. For details, refer to Group View on page 136.

- **Ctrl In/ Ctrl Out** - Opens the **Module View** with the Ctrl In/Ctrl Out ports selected and the corresponding parameters are reflected in the **Properties** window.
- **Sys In/Out A/B** - Opens the **Module View** with the Sys In/Out ports selected and the corresponding parameters are reflected in the properties window.

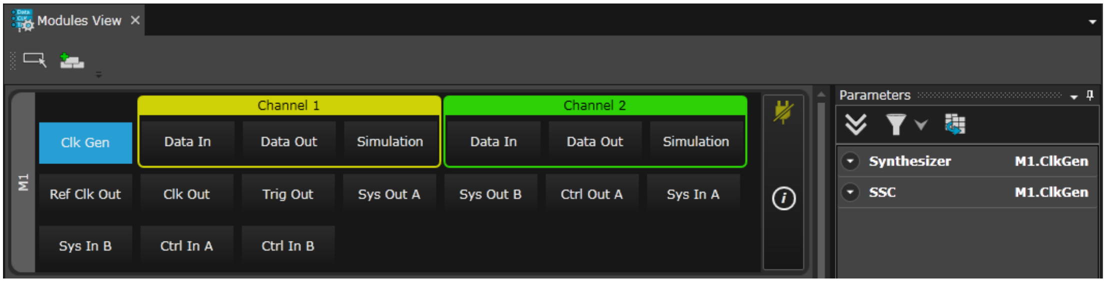
Clock Menu

The **Clock** menu provides the following selections:

- **Clock Generator** - Allows you to configure **Clk Gen** port.
- **Clock Output** - Allows you to configure **Clk Out** port.

Once you make a selection, the **Module View** appears with the **Clk Gen/Clk Out** ports selected and the corresponding parameters are reflected in the **Properties** window.

The following figure shows the **Module View** with the **Clk Gen** port selected:



The generator's output ports are used to supply a clock signal and trigger for another device (for example, analyzer), and an arbitrary data signal for testing your device. For details, refer to Setting up Generator on page 157.

To change bit rate, go to the **Properties** window and then click on the **Synthesizer** function block.

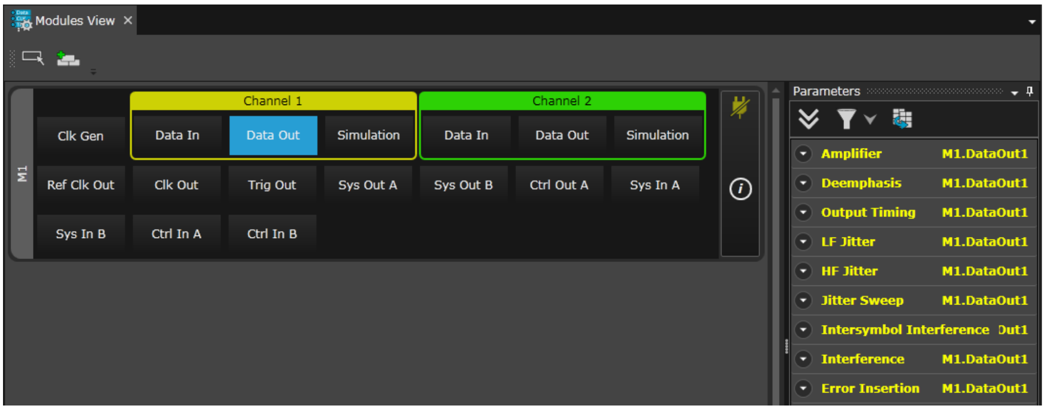
Generator Menu

The **Generator** menu provides the following selections:

- **Data Output** - Allows you to configure **Data Out** port.
- **Trigger Out** - Allows you to configure **Trig Out** port.

Once you make a selection, the **Module View** shows the **Data Out/Trig Out** ports selected and the corresponding parameters are reflected in the **Properties** window.

The following figure shows the **Module View** with the **Data Out** port selected:



The generator's ports are used to set the clock frequency and the output signal with respect to jitter, error insertion and signal output. For details, refer to Setting up Generator on page 157.

To change amplitude/offset, go to **Properties** window and then click the **Amplifier** function block.

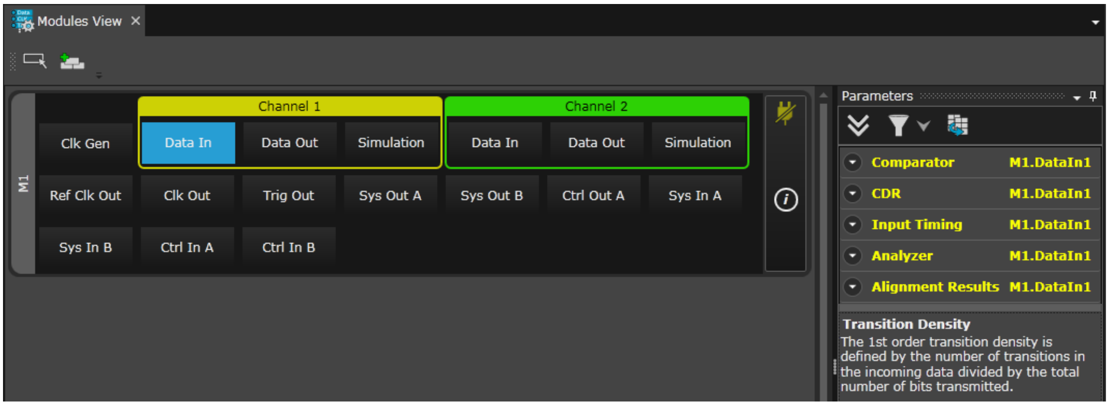
Analyzer Menu

The **Analyzer** menu provides the following selections:

- **Data Input** - Allows you to configure **Data In** port.

Once you make a selection, the **Module View** shows the **Data In** ports selected and the corresponding parameters are reflected in the **Properties** window.

The following figure shows the **Module View** with the **Data In** port selected:

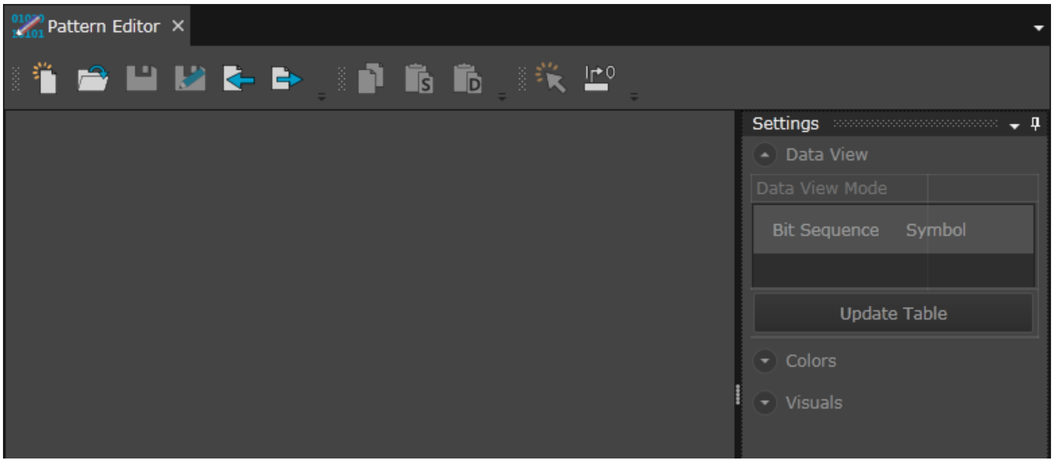


The analyzer's ports are used for running tests and for connecting external equipment. For details, refer to Setting up Analyzer on page 197.

Patterns Menu

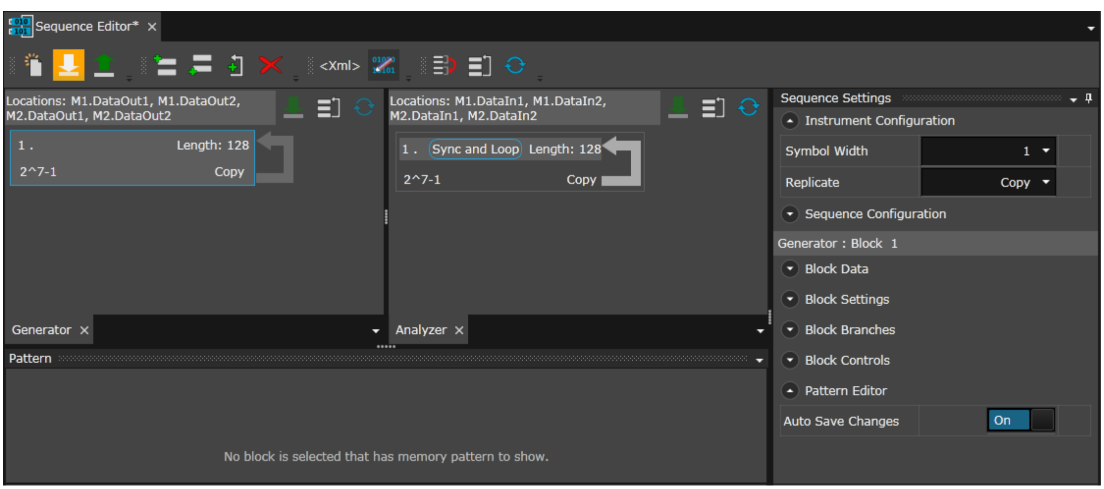
The **Patterns** menu provides the following selections:

- **Select Pattern...** - Opens the **Select Sequence Pattern** dialog. It allows you to override all sequences with a single block loop. For details refer to Pattern Editor on page 250.
- **Pattern Editor** - Opens the **Pattern Editor** user interface as shown in the following figure:



The pattern editor provides an interactive user-interface for creating, editing and importing patterns. For details, refer to Pattern Editor on page 250.

- **Sequence Editor** - Opens the **Sequence Editor** user interface as shown in the following figure:

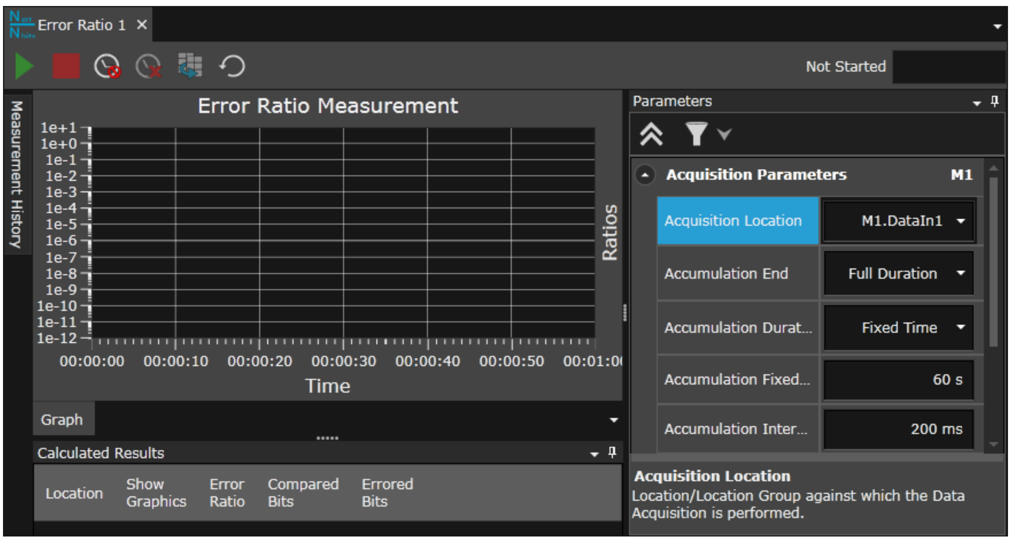


The **Sequence Editor** allows you to create and maintain sequences. In addition to this, it also allows you to edit the memory patterns. For details, refer to Sequence Editor on page 227.

Measurements Menu

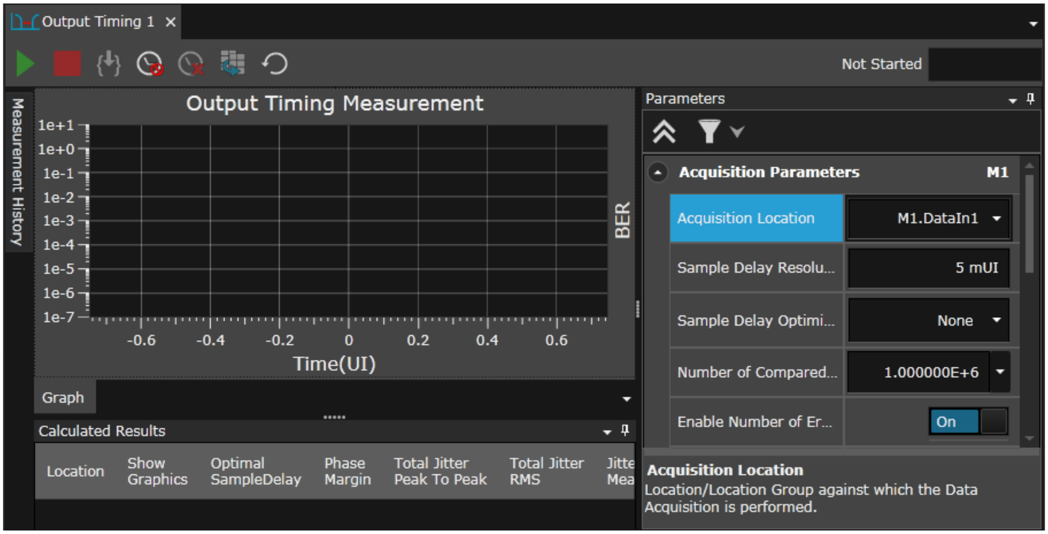
The **Measurements** menu provides the following selections:

- **Error Ratio** - Opens the **Error Ratio** measurement user interface as shown in the following figure:



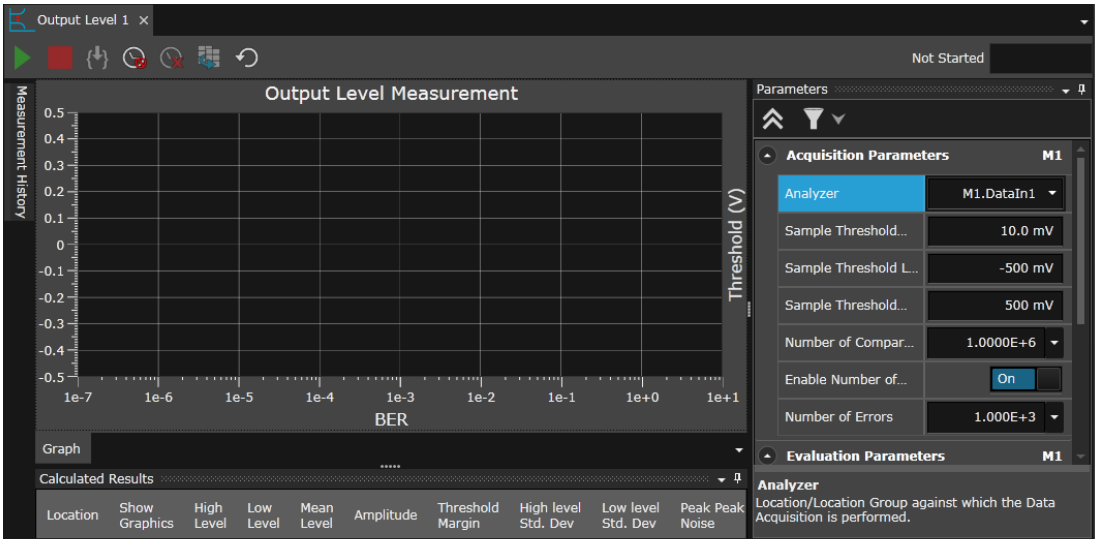
The error ratio measurement allows you to collect measurement data over a specific period. This can be used to create test scenarios that are reproducible and comparable. Also, you can let tests run over long times and then evaluate the results afterwards. For details, refer to Error Ratio Measurement on page 301.

- **Output Timing** - Opens the **Output Timing** measurement user interface as shown in the following figure:



The output timing measurement is used to measure the timing and jitter behavior for a device under test (DUT). It uses a bit error rate (BER) measurement to evaluate the shape of the eye for the output signal of the DUT. It also analyzes the jitter, separates the random jitter and deterministic jitter components, and estimates the total jitter. For details, refer to Output Timing Measurement on page 309.

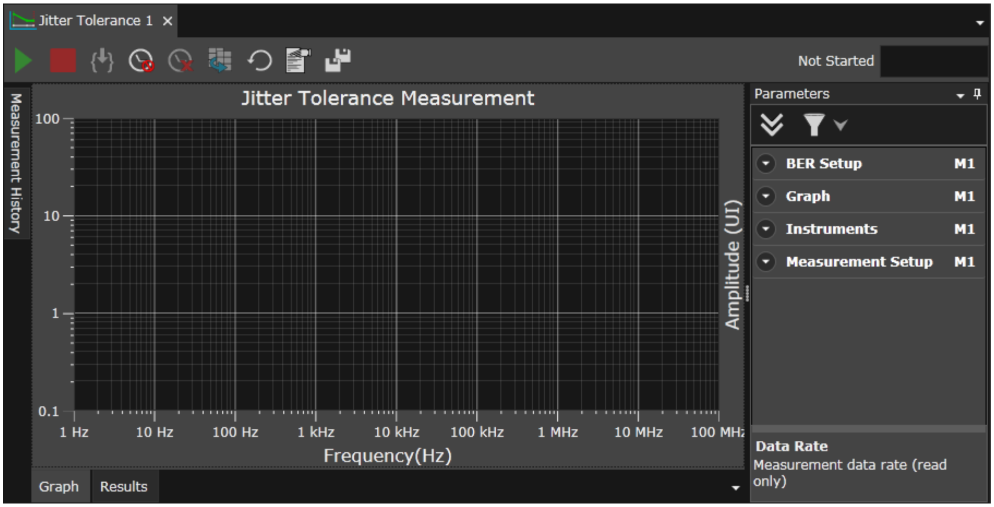
- **Output Level** - Opens the **Output Level** measurement user interface as shown in the following figure



The **Output Levels** measurement allows you to characterize the behavior of the output levels of a device under test (DUT). The sampling delay is fixed. The analyzer's decision threshold is automatically swept within a user-defined range.

For details, refer to Output Level Measurement on page 325.

- **Jitter Tolerance** - Opens the **Jitter Tolerance** measurement user interface as shown in the following figure:

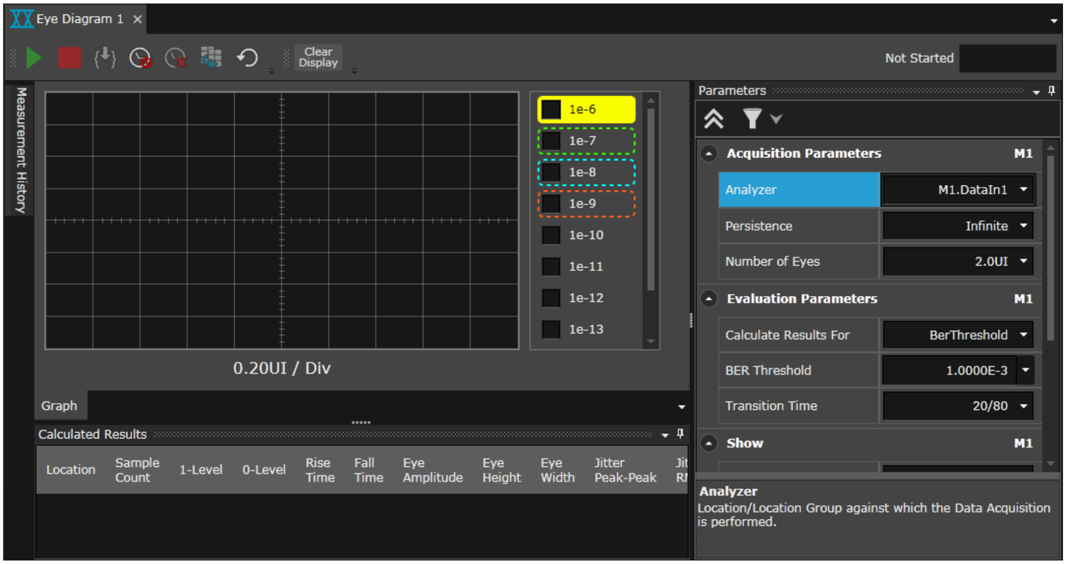


Jitter Tolerance measurement is used to determine the ability of a device or system to maintain communication quality in the presence of jitter. It comes in two varieties:

- **Jitter Tolerance Characterization** determines the jitter levels where the device under test can no longer maintain a desired bit error ratio (BER).
- **Jitter Tolerance Compliance** verifies that the device under test is able to maintain a BER level at pre-defined jitter levels and jitter frequencies, as defined by a standard.

For details, refer to Jitter Tolerance Measurement on page 338.

- **Eye Diagram** Measurement - Opens the **Eye Diagram** measurement user interface as shown in the following figure:



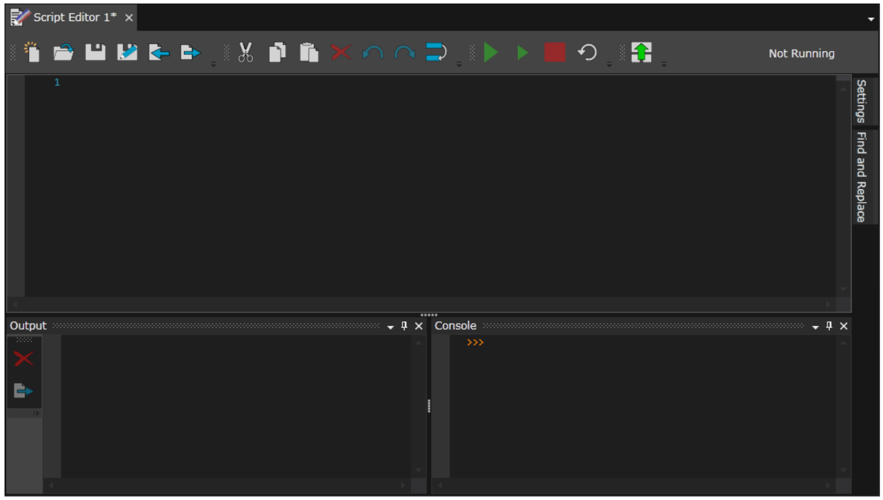
The M8070A System Software provides quick design analysis with the Eye Diagram capability. The Eye Diagram allows a quick check for the DUT's signal output, and determines the signal quality. The eye contour lines display the measured eye at a deeper BER level, for accurate results.

For details, refer to Eye Diagram Measurement on page 355.

Utilities Menu

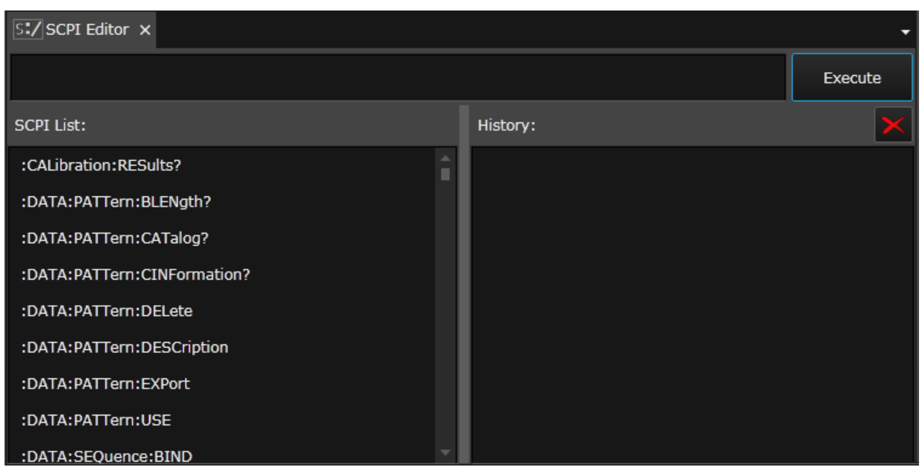
The **Utilities** menu provides the following selections:

- **Script Editor** - Opens the **Script Editor** user interface as shown in the following figure:



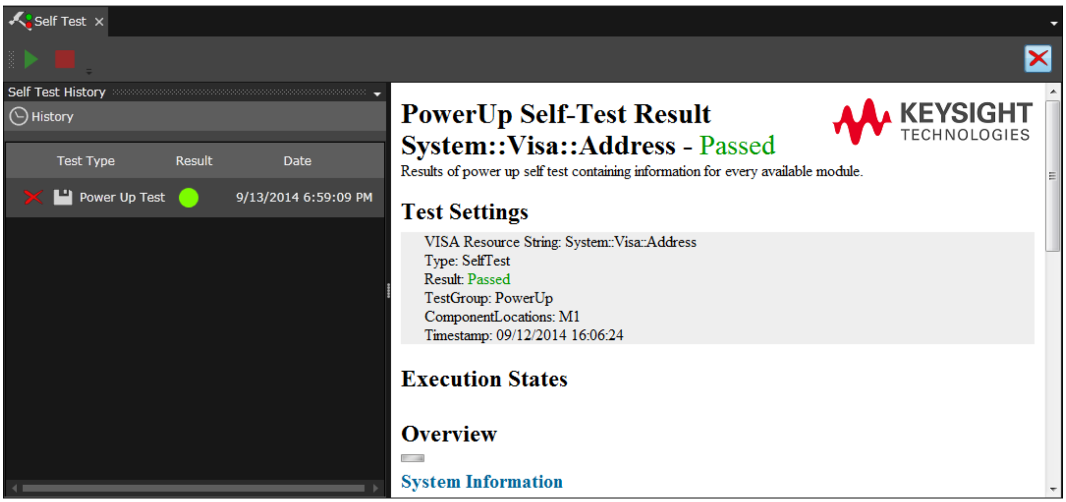
The **Script Editor** provides flexibility to the programmers to automate their plug-ins thus allowing them to do everything they want in the measurements. For details, refer to Script Editor on page 367.

- **SCPI Editor** - Opens the **SCPI Editor** user interface as shown in the following figure:



- The **SCPI Editor** lists all SCPIs that can be used to program M8020A/M8030A and also provides a platform to execute them. For details, refer to SCPI Editor on page 411.

- **Self Test** - Opens the **Self Test** utility user interface as shown in the following figure:

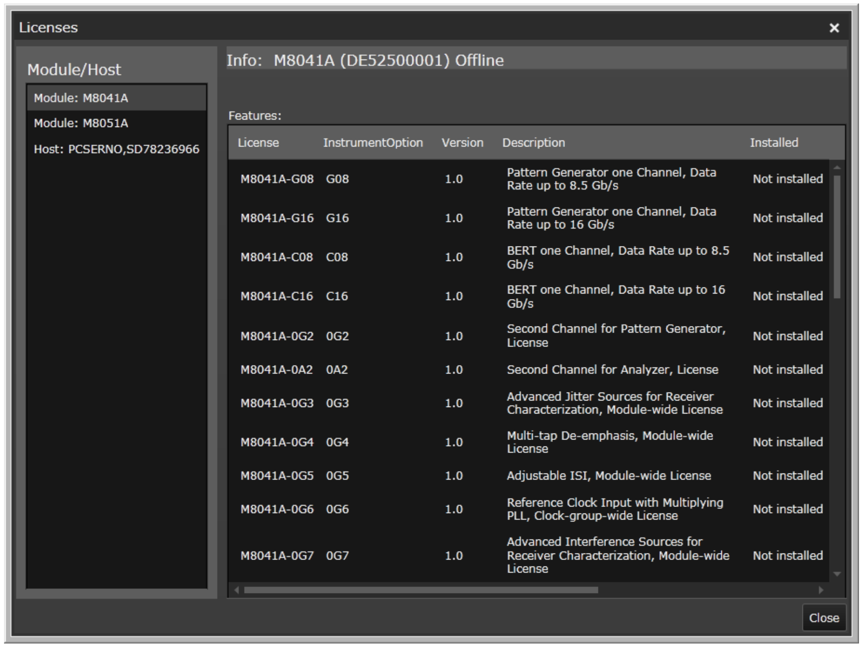


The **Self Test** utility checks the specific system information of the hardware components for basic functionality. On execution, the following results are displayed:

- System related information such as connected modules, serial no. and hardware revision.
- Module related information such as calibration, power supplies and memory controller.

For details, refer to Self Test Utility on page 416.

- **Licenses...** - Opens the **Licenses** window user interface as shown in the following figure:



The **Licenses** window displays the license information currently installed in the modules or host. For details, refer to Licenses Window on page 421.

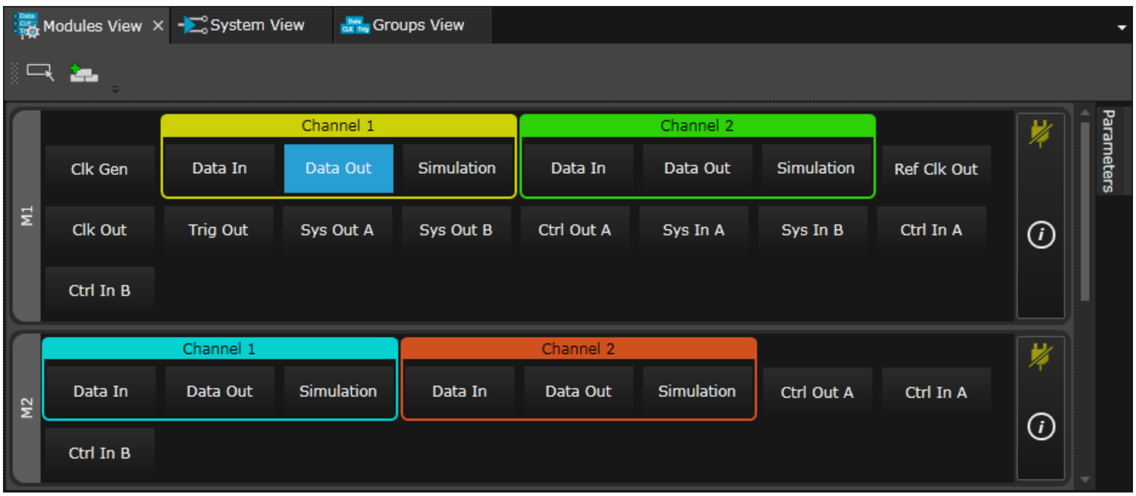
- **Settings...** - Opens the **Settings** window which allows you to set the display and channel settings in the user interface. For details, refer to Settings Window on page 422.
- **Logger** - The Logger window displays errors, warnings and information messages along with their respective descriptions, applications from where they are generated and their time stamps. For details, refer to Logger Window on page 424.
- **Plug-in Manager** - The **Plug-in Manager** simplifies all the tasks related to plug-in management. It displays list of plug-ins that are installed in the software. In addition, the **Plug-in Manager** also allows you to install, uninstall and update the plug-ins. For details, refer to Plug-in Manager on page 426.

Window Menu

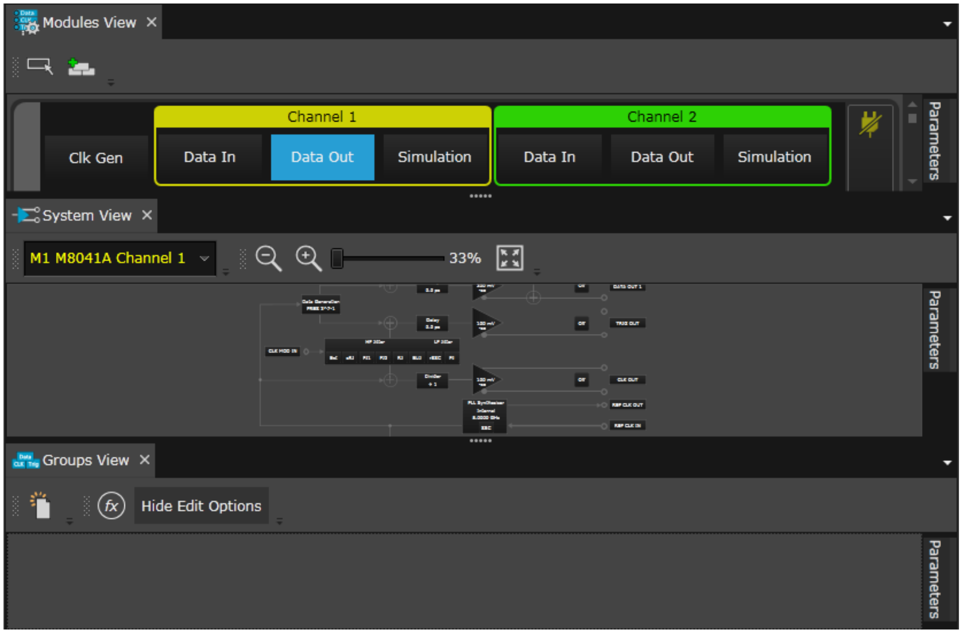
The **Window** menu allows you to change the layout/arrangement of various open windows. It includes the following selections:

- **Cascade** - Arranges the multiple opened windows in a docked view so that you can see all or part of each window; consisting of individual tabs for each window.

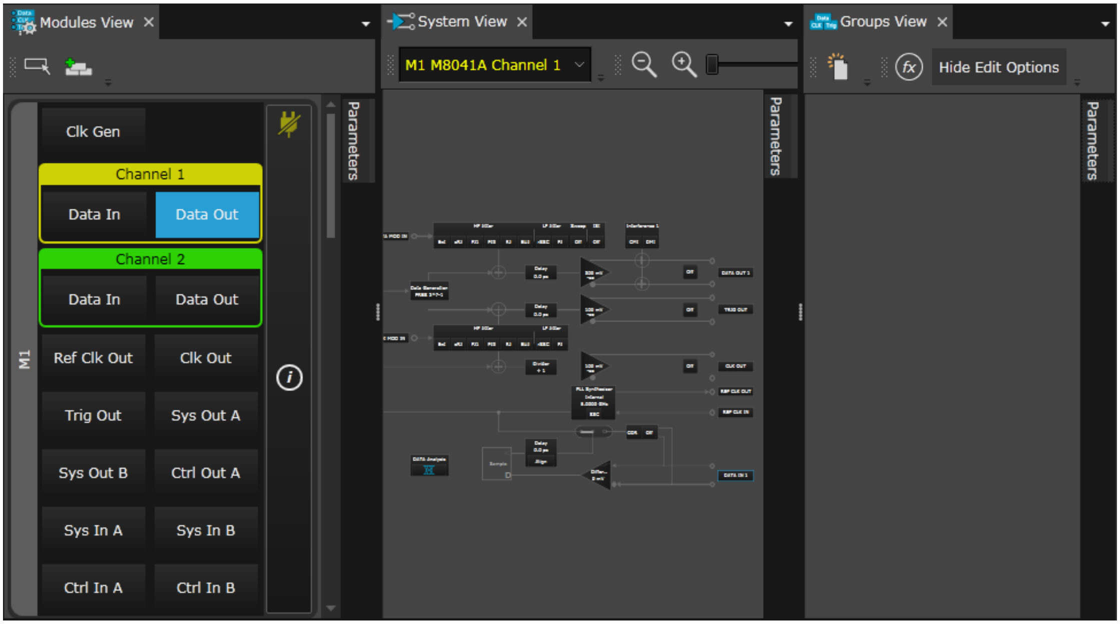
The following figure shows the docked/tabbed view of three different windows.



- **Tile Horizontally** - Aligns the multiple opened windows in a horizontal sequence. The following figure shows the horizontal sequence of three different windows.



- **Tile Vertically** - Aligns open windows in a vertical sequence. The following figure shows the vertical sequence of three different windows.

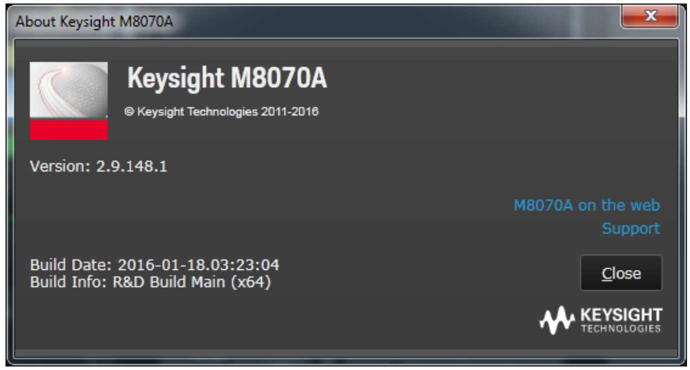


- **Focus Center Content** - Minimizes or hides additional dialog boxes or property sheets of the multiple open windows to focus on the center content.
- **Close All** - Closes all the open windows.

Help Menu

The **Help** menu includes the following selections:

- **View Help** - Opens the online help that provides you with the information you need for working with the Keysight M8070A GUI.
- **About** - Shows product information of M8070A including version number, build date, build info and web links for M8070A information and support.



Increase/Decrease Splitter Size

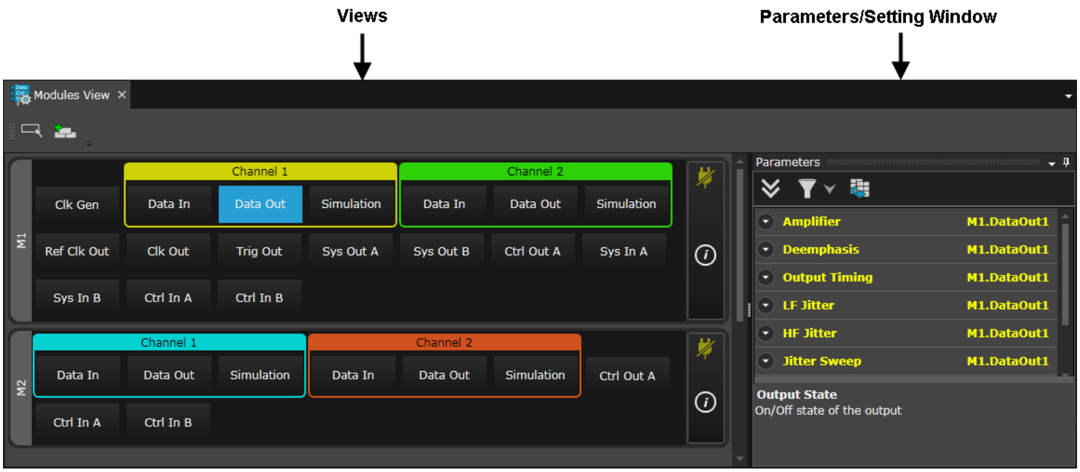
The **Splitter** option allows you to either increase or decrease the divided space between the windows forms used in the user interface. This icon is located on the right side of the menu bar. The splitter functionality provides an easy navigation to the users who are using a touchscreen interface.

Main Window

The main window refers to the middle area of the M8070A user interface that allows you to launch different display views (e.g. **Module View**, **Group View**, **System View**, etc.). You can use the menu bar to launch these views on the main window. Each view can be configured through the **Parameters** window. The detailed description of different views, their parameter settings, controls and dialogs are described in the subsequent sections in this manual.

When you launch the M8070A software, by default, it shows the **Module View** on the main window.

The following figure shows how the main window appears:



Status Bar

The status bar is located at the bottom of the M8070A user interface. The status bar is shown in the following figure:

It provides the following functions:

- Toggle icon to open/close the **Status Indicator** window. For details, refer to Status Indicators Window on page 92.
- Toggle icon to open/close the **Logger** window that displays errors, warnings and information messages which are generated from the M8020A/M8030A system. For details, refer to Logger Window on page 93.
- Toggle icon to open/close the **Show Link Training Log** window. For details, refer to Link Training Log Window on page 94.
- Error indicators for the current GUI state and clock loss. When the current GUI is stopped or there is any clock loss, the respective indicator turns red.
- Toggle buttons to enable/disable the **Output**, **Jitter** and **SSC** state. Once these features are enabled, the buttons turn green.

- **Progress Indicator** to visualize the progression of multiple GUI operations and/or background operations in a single bar. However, if the process takes too long to complete, you have an option to terminate that process by clicking on the **Abort** button which appears on the **Status Bar**.

NOTE

The behavior of progress indicator depends upon the number of process involved in any operation. It displays progress of each individual process in a single bar. Time for each operation may vary and sometimes may be very quick. Therefore, at some instance, it may display a progress to be complete (100%) and then suddenly switch to incomplete (50%).

WARNING

You will see a warning sign on the status bar if the “Global Output State” is off. In this case, all data outputs will be disabled until the “Global Output State” is turned on. Click the “Output” button present on the status bar to turn on the “Global Output State”.

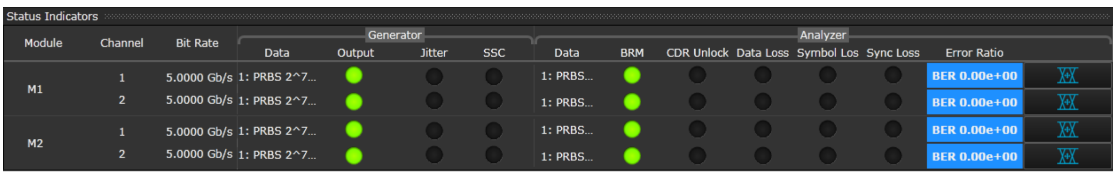
- **Insert Error** button to insert a single bit error on each Data Out location of the connected modules.
- **Preset All** button opens the **Preset Instrument** dialog that allows to reset the instrument state to factory default settings.

Status Indicators Window

The **Status Indicators** window displays the status indicators for the generator and analyzer ports of each channel of the connected modules. This includes:

- Setup information such as **Bit Rate** for each channel of the connected module.
- Generator port information such as:
  - Data information that shows the name/type of the pattern downloaded to that data block and its indicator shows which data block is currently transmitting the sequence.
  - State information of generator such as **Output**, **Jitter** and **SSC** (indicated by green LED).
- Analyzer port information such as:

- Data information that shows the name/type of the pattern downloaded to that data block and its indicator shows which data block is currently transmitted the sequence.
- State information of analyzer such as **BRM** (indicated by green LED).
- Error indicators of analyzer such as CDR Unlock, Data Loss, Symbol Loss and Sync Loss (indicated by red LED).
- Calculated BER
- **Alignment BER Threshold** button to start BER threshold auto alignment.

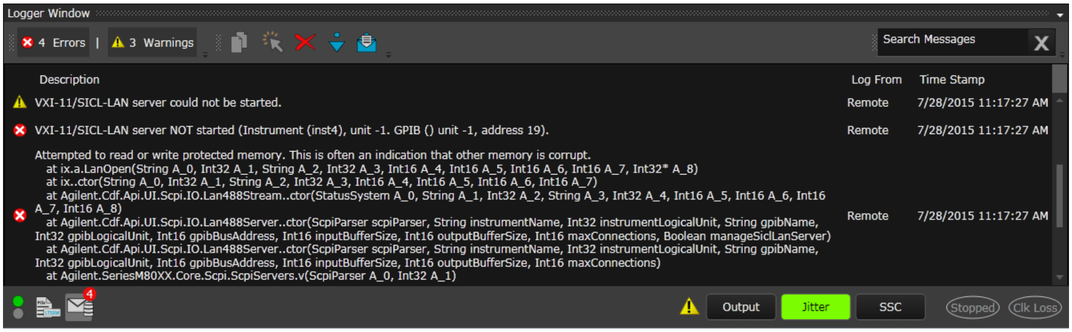


The **Status Indicator** window always appears whenever you launch the M8070A user interface. However, you can open/close this window by clicking on the **Status Indicator** icon, present on the status bar.

Logger Window

The **Logger** window displays errors and warnings messages along with their respective descriptions, applications from where they are generated and their time stamps.

The **Logger** window always appears whenever you launch the M8070A user interface. However, you can open/close this window by clicking on the **Logger** icon, present on the status bar.



The **Logger** window allows you to:

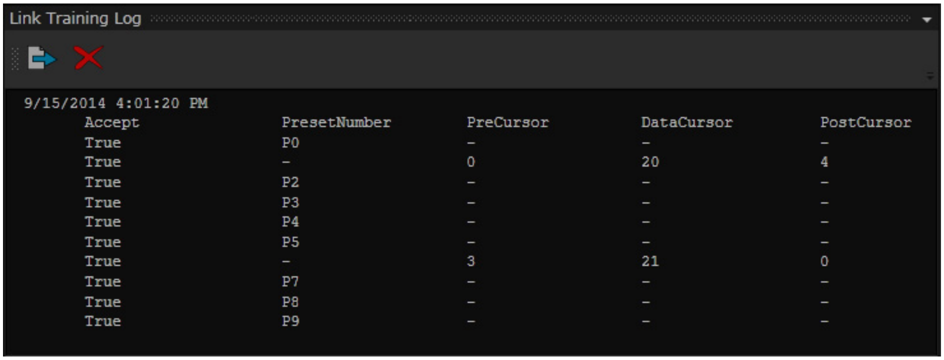
- **Message Selection** - Use this option to choose whether you want to view errors, warnings or information message.
- **Copy** - Use this option to copy a message. You need to select a message in order to enable copy feature.
- **Select All** - Use this option to select all messages. It also enables copying all messages.
- **Clear Messages** - Use this option to delete all messages.
- **Auto Scroll** - Use this option to enable/disable auto scroll option. When the **Auto Scroll** option is enabled, it will automatically scroll you to the new message without using the scroll bar.
- **Open On Message** - Turn this button OFF if you don't want the Logger window to automatically pop-up whenever a message is received.
- **Search Messages** - Use this option to search messages by providing an input in the **Search Messages** search box.

Link Training Log Window

The **Link Training Log** window displays the logs for link training PCIe 3.0.

You can open/close the **Link Training Log** window by clicking on the **Link Training Log** icon present in the status bar. The **Link Training Log** icon with orange background indicates an update or new entry in the link training log.

The following figure shows the **Link Training Log** window:



Accept	PresetNumber	PreCursor	DataCursor	PostCursor
True	P0	–	–	–
True	–	0	20	4
True	P2	–	–	–
True	P3	–	–	–
True	P4	–	–	–
True	P5	–	–	–
True	–	3	21	0
True	P7	–	–	–
True	P8	–	–	–
True	P9	–	–	–

The **Link Training Log** window provides the following options:

- **Export Log** - Click the **Export Log** icon to save the link training log.
- **Clear Logs** - Click the **Clear Logs** icon to delete all link training logs.

For more details on link training, refer to Interactive Link Training on page 285.

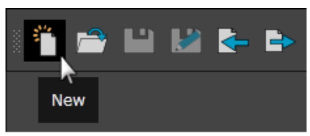
Other GUI Features

Following are the GUI elements that make the M8070A user interface interactive.

Tooltip

The tooltip is a small pop-up window that concisely describes the object being pointed to, such as descriptions of toolbar controls, icons, graphics, links, menu items and taskbar buttons.

The following example shows the tooltip providing a description of toolbar buttons.



Here is another example where the tooltip provides information to the user on the minimum and maximum values the parameter can hold.

Toggle Button

The toggle button allows you to toggle between the two features. For example, in the following figure, we are using the toggle button to either expand or collapse the parameter list.

ON/OFF Switch

The ON/OFF switch enables or disables a given feature.

The following figure shows how a ON/OFF switch is used to turn the state feature ON.

However, at some instances in the GUI, the ON/OFF switch will be enabled when you select the corresponding check box.

Execute Button

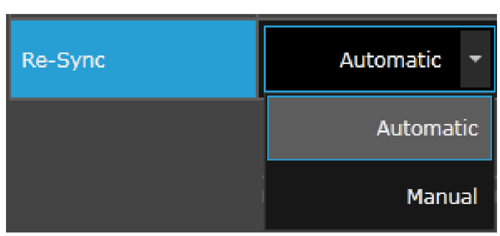
The **Execute** button allows you to perform an activity, once you click on it.

The following figure shows an **Execute** button that is used to perform pattern synchronization.

Drop-Down List

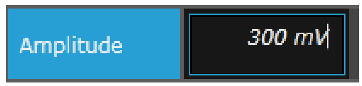
The drop-down list allows you to choose either one or sometimes multiple selections from the provided list.

The following shows the drop-down list to choose the **Re-Sync** mode.



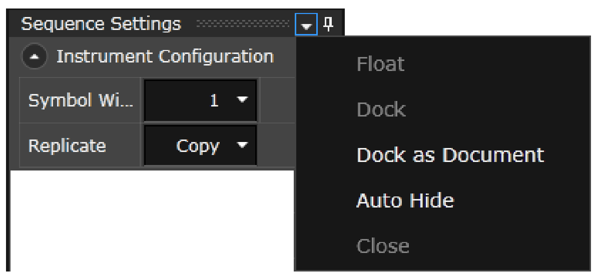
Numeric Entries

Most numeric entries have a pre-defined maximum, minimum, and default value displayed in their corresponding pop-up menu.



Numeric values in numeric entries can be changed using the on-screen numeric keypad. For details, refer On-Screen Numeric Keypad on page 100.

Window Option



The window options allow you to float or dock anywhere in the application window. It provides quick access to logically grouped features from one location. For example, you can select and generate various layouts from the single window.

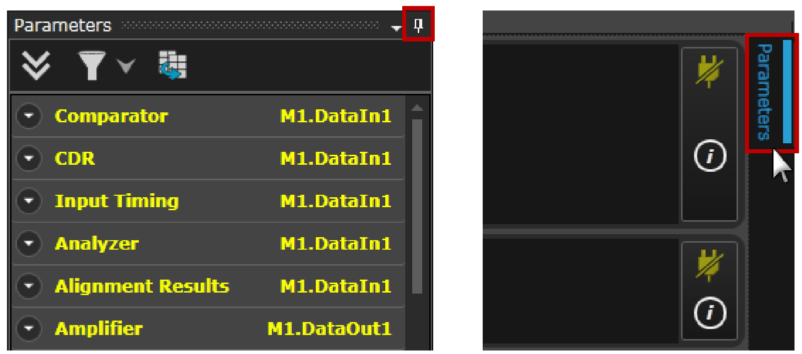
You can move a window anywhere on the screen or to a different monitor. You can also use the auto-hide feature of the windows to show or hide them on the desktop. You can also close the floating windows.

Auto Hide Feature

The automatic hiding functionality gives you the ability to imitate the behavior of the dock windows in the M8070A user interface. When it's enabled for a dock panel, this panel is automatically hidden when the mouse pointer leaves its area. Dock panels are hidden at the nearest form's edge. For example, if a panel is docked to the right edge of the form, it will be hidden at the right edge.

End-users can enable the automatic hiding functionality by clicking the auto hide button displayed within the panel's caption.

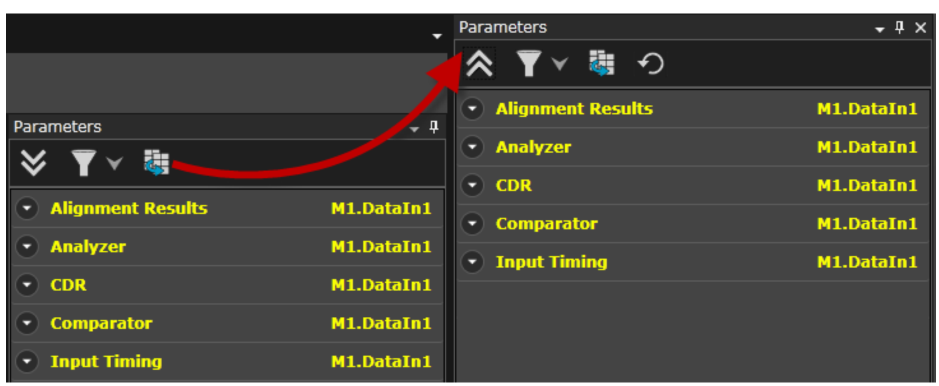
The image below illustrates how the automatic hiding functionality can be enabled and how to show the hidden dock panel.



Copying Parameter Window

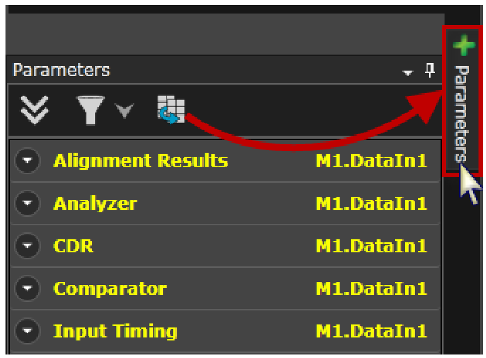
The copying feature creates a replica of parameters window to enhance the usability. It allows you to work on two different instances of the application. The changes you make in one window display immediately in the other window.

The following figure shows the copied parameters window.



Once copied and then enabling the auto hide feature, the cloned parameters window is docked to the right edge of the main user interface. The copied parameters window pops up once you click on it.

The following figure shows the copied parameters window docked to the right edge of the main user interface.



On-Screen Numeric Keypad

The on-screen numeric keypad makes it easier to enter the numbers, units, etc., specially if you are using a touchscreen monitor. The on-screen numeric keypad pops up whenever you tap mouse pointer or touch (in case of touchscreen) in a text field or other area where user inputs are required.

NOTE Remember to unselect the **Disable** Keypad option in the **Settings** window in order to enable the on-screen numeric keypad. For details, refer to Settings Window on page 422.

The following screen shows how to use an on-screen numeric keypad to provide a value to a parameter.

The on-screen numeric keypad contains the following buttons:

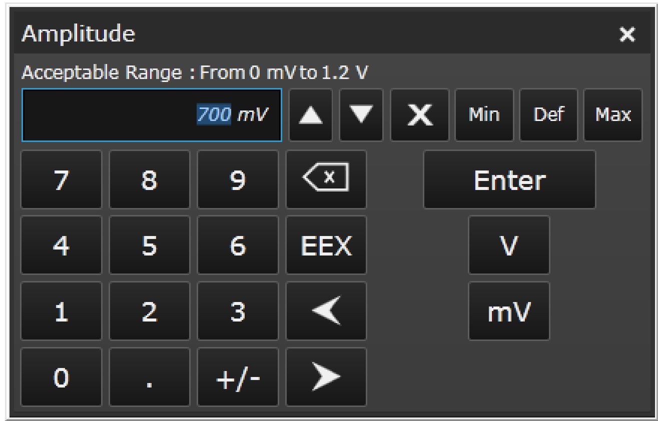


Table 38 On-screen numeric keypad

Button	Name	Description
	Number Buttons	Press these buttons to enter a numeric value. The acceptable range of a parameter is shown at the top of the keypad.
	Decimal Button	Press this button to enter a decimal numeric value.
	Function Button	Press this button to toggle between the addition (+) and subtraction (-) function.
	UP/DOWN Button	Press these buttons to increase or decrease the numeric values, respectively.
	Backspace Button	Press this button to delete any character before the current position of the pointer.

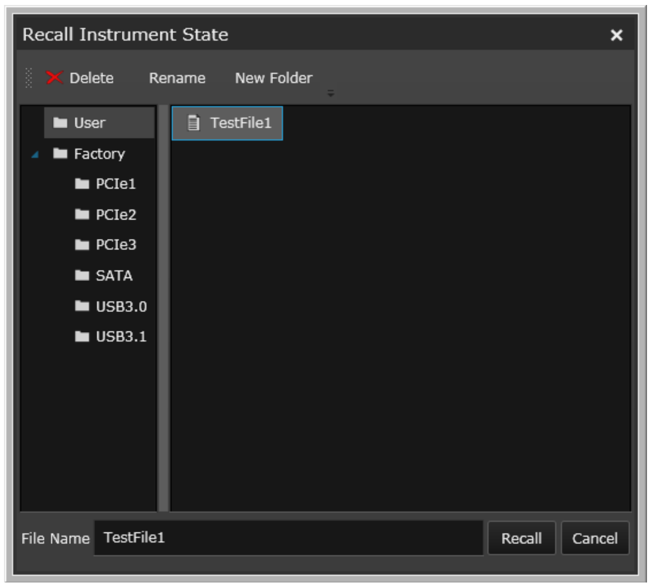
Button	Name	Description
< > >	LEFT/RIGHT Button	Press these buttons to move the pointer one position left or right, respectively.
EEX	Exponent Button	Press this button to enter exponents.
Enter	Enter Button	Press this button to apply the value.
mV	Units Button	Press this button to assign units to the numeric value. Please note that the units may change depending on the parameter.
X	Clear Field Button	Press this button to clear the text field.
Min Def Max	Minimum/ Default/ Maximum Button	Press this button to insert either minimum, or default, or maximum numeric value of the parameter.

Recall/Save Instrument State

Recall Instrument State

To recall an instrument state, do the following:

- Go to **File** menu and then click **Recall Instrument State...**. The **Recall Instrument State** dialog will appear as shown in the following figure.

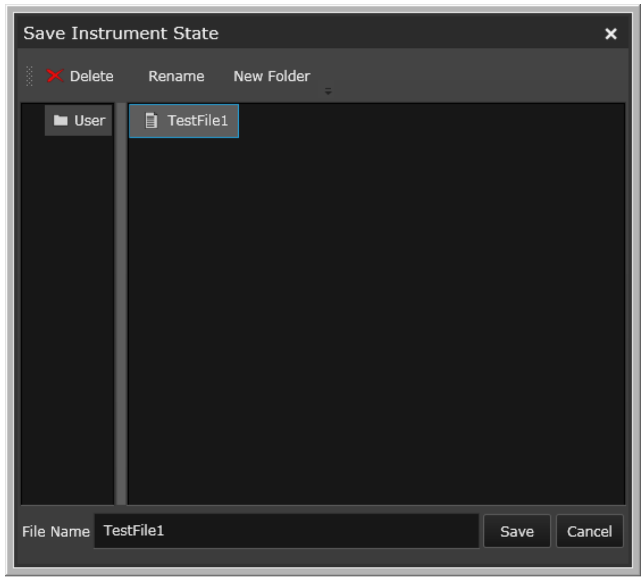


1. Select the folder (User or Factory) to view the files.
2. Select the file which is to be recalled.
3. Click **Recall**.
4. To rename a file, select the file and click Rename. The filename will become editable.
5. To delete a file, select the file and click **Delete**.
6. To add new folder, select location you want to create your folder and then click **New Folder**.
7. To rename a folder, select the folder and click **Rename**. The folder name will become editable.
8. To delete a folder, select the folder and click **Delete**.

Save Instrument State

To save an instrument state, do the following:

1 Go to **File** menu and then click **Save Instrument State...**. The **Save Instrument State** dialog will appear as shown in the following figure.



2 Enter a file name and click **Save**. The current settings will be saved under the filename.

3 To rename a file, select the file and click **Rename**. The filename will become editable.

4 To delete a file, select the file and click **Delete**.

5 To add new folder, select location you want to create your folder and then click **New Folder**.

6 To rename a folder, select the folder and click **Rename**. The folder name will become editable.

7 To delete a folder, select the folder and click **Delete**.

Keysight M8070A System Software for M8000 Series of BER Test Solutions

User Guide

4 User Interface - M8070A Display Views

KEYSIGHT TECHNOLOGIES

Overview

The M8070A system software provides the following types of views:

- Module View
- System View
- Group View
- Adjustable Intersymbol Interference
- Controlling M8195A AWG from M8070A User Interface for Generating PAM-4 Signals

Module View

The **Module View** is a graphical representation of the input/output ports that are present on the front panel of the modules, configured into the M8020A/M8030A. The M8020A/M8030A supports the M8041A (8.5/16G Generator, Analyzer, Clock Module), M8051A (Generator-Analyzer), M8061A (32 Gb/s Multiplexer with De-emphasis) and M8062A modules that are installed into an Keysight M9505A 5-slot AXIe chassis. These modules have a different set of input/output ports depending upon their functionality e.g. generator, analyzer or clock. Each module consists of two channels (channel 1 and channel 2) depending upon the licenses, you have ordered. Each channel contains Data Out (Generator) and Data In (Analyzer) ports.

The modules are identified as M1, M2, M3 so on in the GUI.

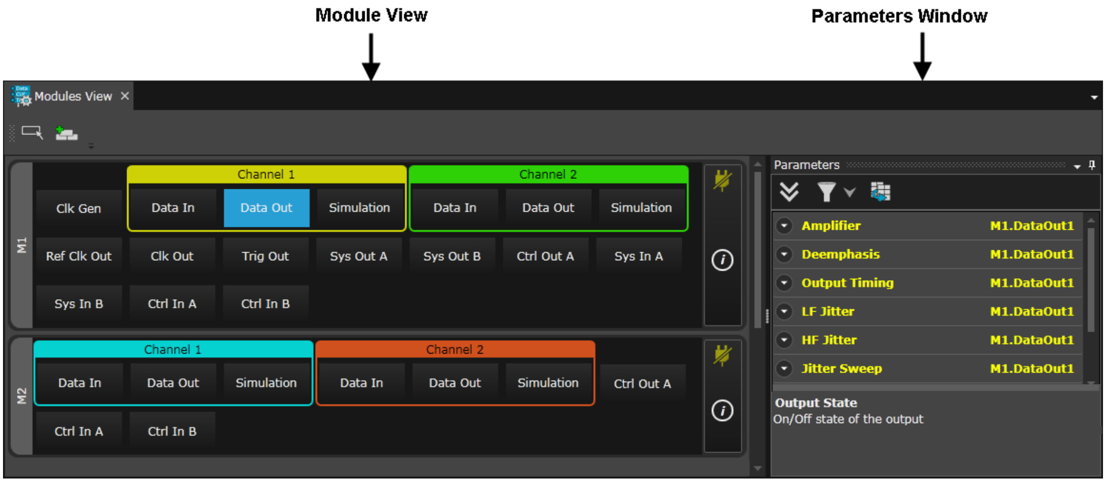
You can use the **Module View** to configure the properties of a single port or a group (combination of multiple ports). For details, refer to System View on page 119.

How to Launch Module View

The **Module View** is launched by-default whenever you launch the M8070A user interface. However, if it is not available or is closed, you still can launch it.
To do so:

- Go to the **Menu Bar** > **System** and then select **Module View**.

The following figure shows an example of **Module View** when M8041A (M1) and M8051A (M2) modules are connected:



The left side shows the connected modules and the right side shows the **Parameters** window. Each module has input and output ports which can be configured through the **Parameters** window. For details on inputs and output ports, refer to Input and Output Ports on page 109. When you click on the port, the respective configurable parameters are displayed in the **Parameters** window. For details, refer to Parameters Window on page 115.

The following figure shows an example of **Module View** when M8041A (M1), M8051A (M2) and M8061A(M3) modules are connected:

Input and Output Ports

M8041A and M8051A Modules

The M8041A (8.5/16G Generator, Analyzer, Clock Module) and M8051A (Generator-Analyzer) modules can have the following input and output ports:

- **Clock Gen**: Clock can be generated from the internal oscillator or an external source.
- **Data Out**: Data Out acts as the output port for the Generator which may be connected to the DUT. The data outputs serve as device stimuli and can be set up so that they are compatible with a variety of logic families.
- **Data In**: Data In acts as the input port for the Analyzer. This port is connected to the data signal which is the output of the DUT. Here the signal received and the signal generated internally is compared for calculating the bit error ratio.

- **Ref Clk Out**: The reference clock output is used to provide a 10 MHz or 100 MHz reference clock to the DUT or other test equipment (SMA, female).
- **Clock Out**: The clock out port serves as frequency (bit rate) references. If we want to operate the external device at the system clock frequency then the device operating frequency can be set up using the clock out.
- **Trig Out**: Trig Out serves as an output port of the Generator. It allows you to connect a trigger for another device which can be used to provide a 10 MHz or
100 MHz reference clock to the DUT or other test equipment.
- **Sys Out A/Sys Out B**: System level control outputs used to signal events to the DUT or external instruments.
- **Ctrl Out A:** The module has one control output at the front panel with the following selectable functionality of Error Output.
- **Sys In A/Sys In B**: System level control inputs used to generate sequencer events.
- **Ctrl In A/Ctrl In B**: The M8041A and M8051A modules has two control inputs at the front panel each with the following selectable functionality of Error Add Input, Output Blanking, Electrical Idle and Gating Input.

M8061A Module

The M8061A (32 Gb/s Multiplexer with De-emphasis) module has the following input and output ports:

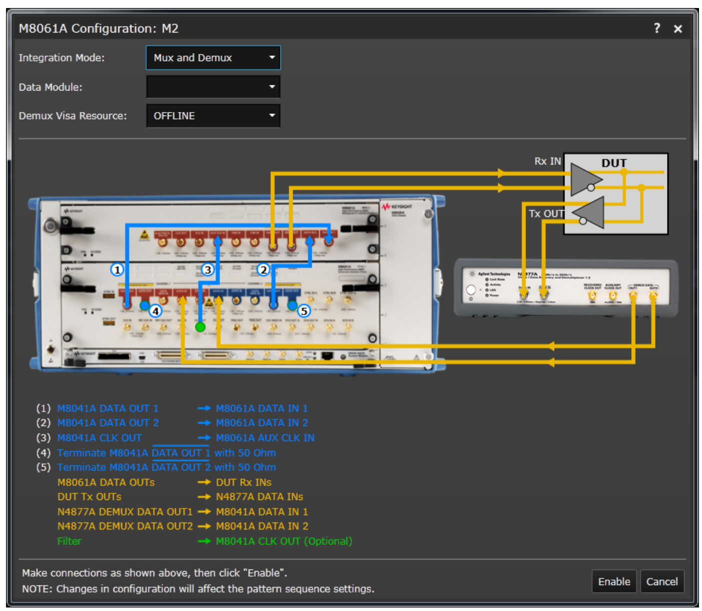
- **Data In** - Data In acts as the input port for the Analyzer. This port is connected to the data signal which is the output of the DUT. Here the signal received and the signal generated internally is compared for calculating the bit error ratio.
- **Data Out**: Data Out acts as the output port for the Generator which may be connected to the DUT. The data outputs serve as device stimuli and can be set up so that they are compatible with a variety of logic families. Unused outputs must be terminated into 50 Ω. (2.4 mm, female).
- **Clk Gen**: Clock can be generated from the internal oscillator or an external source.
- **Elect Idle In**: This port is used to enable/disable the output signal by an external control signal. If the input level is above the threshold level the module enters electrical idle.

M8061A Configuration

The **M8061A Configuration** dialog provides configuration settings for M8061A. To open the **M8061A Configuration** dialog:

- Go to **Menu Bar** > **System** and then click **Module View**.
- Locate the **M8061A** module.
- Click **Configuration...** button.

The **M8061A Configuration** dialog will appear as shown in the following figure:



This dialog provides the following setting options:

- **Integration Mode**: Allows you to select the integration mode. The default mode is **Standalone** Mode. The other modes you can select are **Mux Only** mode, **Demux Only** mode and **Mux and Demux** mode.
- **Data Module**: Allows you to specify the data module to which the M8061A module is connected.

- **Demux Visa Resource**: Allows you to provide the visa resource string for N4877A. This is required to connect N4877A instrument with M8020A.

In addition, this dialog displays the connection diagram and connection instructions for each mode.

- Click **Apply**. Depending upon the configuration settings, the ports which in which settings can be done do not ayou will see the changes in the **Module View**. Some ports that are used by the data module and M8061A modules will be disabled. For the block diagram representation and interactively modify the settings of the currently mode, switch to **System View**. For details, refer System View with M8061A Integration on page 125.

M8062A Module

The M8062A (32Gb/s Front-end for J-BERT M8020A High-Performance BERT) module has the following ports on its front panel:

On Pattern Generator Side

- **Data In 1 and Data In 2 (Half Rate) -** Single-ended, half-rate data inputs from the M8041A module (3.5 mm, female).
- **Data Out and /Data Out -** Differential or single-ended, full-rate data output to the device under test. Unused outputs must be terminated into 50 Ω. (2.4 mm, female).
- **DMI In -** Differential Mode Interference input. Applies a single-ended, external interference source differentially to the data output (SMA, female).
- **CMI In -** Common Mode Interference input. Applies a single-ended, external interference source to both the normal and complement data output signals (SMA, female).
- **Clk Out -** Half-rate Pattern Generator clock output. Carries the same jitter as the full-rate data output.
- **Clk In -** Pattern Generator clock input (half-rate). Connect to clock output of M8041A.
- **Aux Clk In -** Alternate Pattern Generator clock input (half-rate). Typically unused.
- **Electrical Idle In -** This input is used to enable/disable the output signal by an external control signal. If the input level is above the threshold level the module enters electrical idle. Normal operation resumes when the input level is below the threshold (SMA, female).

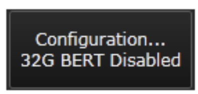
On Analyzer Side

- **Data In and /Data In -** Differential or single-ended, full-rate data input from the device under test. Unused input should be terminated into 50 Ω. (2.4 mm, female). These ports are AC coupled.
- **Data Out 1 and Data Out 2 -** Single-ended, half-rate data outputs to the M8041A module (3.5 mm, female).
- **Clk Out -** Half-rate Error Analyzer clock output, synchronous with analyzer sampling.
- **Clk In -** Half-rate, Error Analyzer clock input. Allows external clocking of the Error Analyzer.

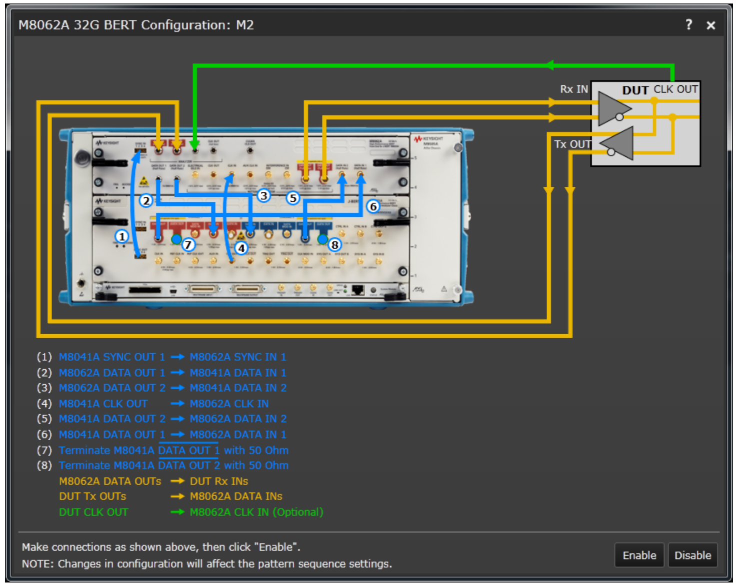
M8062A Configuration

The **M8062A Configuration** dialog provides configuration settings for M8062A. To open the **M8062A Configuration** dialog:

- Go to **Menu Bar** > **System** and then click **Module View**.
- Locate the **M8062A** module.
- Click **Configuration...** button.



The **M8062A Configuration** dialog will appear as shown in the following figure:



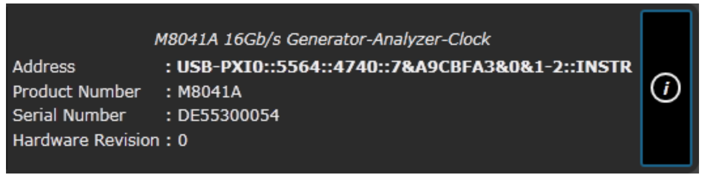
This dialog displays the connection diagram and connection instructions.

- Click **Enable**. In the 32G mode, access to some M8041A user controls are disabled to facilitate software control of this configuration. For the block diagram representation and to interactively modify the settings of the currently 32G mode, switch to **System View**. For details, refer System View with M8062A Integration on page 132.

Show Module Information

You can get the module information that is connected to

the M8020A by clicking the icon present at the right side of each module. The module information will be shown as depicted in the following figure:



It provides the following information about the module:

- **Address** - Address of module, e.g. USB-PXIO::11::0::INSTR
- **Product Number** - Product no. of the module, e.g. M8041A
- **Serial Number** - Serial no. of module, e.g. DE53C00061
- **Hardware Revision** - Hardware revision of module, e.g. 0

Selecting Single/Multiple Ports

You can select either single or multiple ports by pressing the toggle button, available on the top of the main window. It provides the following modes selection:

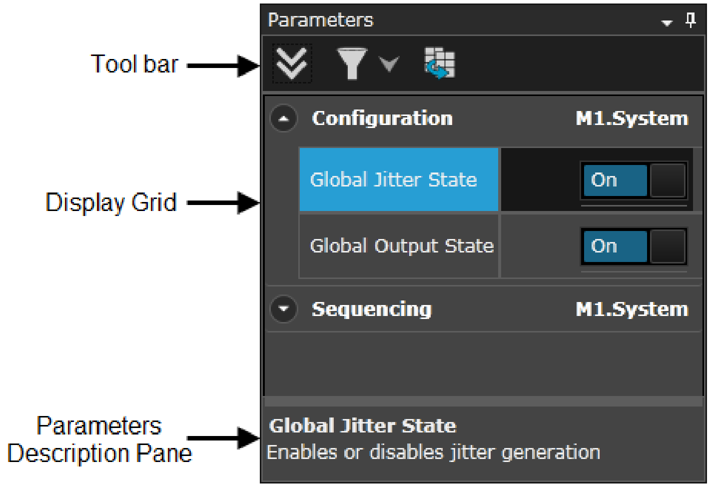
 **Single Selection Mode** - allows you to select only one port at a time from the connected modules.

 **Multiple Selection Mode** - allows you to select multiple ports from the connected modules.

Parameters Window

The **Parameters** window displays the functional blocks of the selected port/port groups. Each functional block has a set of parameters. The **Parameters** window also allows you to set the parameters of the selected port/port groups.

The **Parameters** window is shown in the following figure:



The **Parameters** window has the following sections:

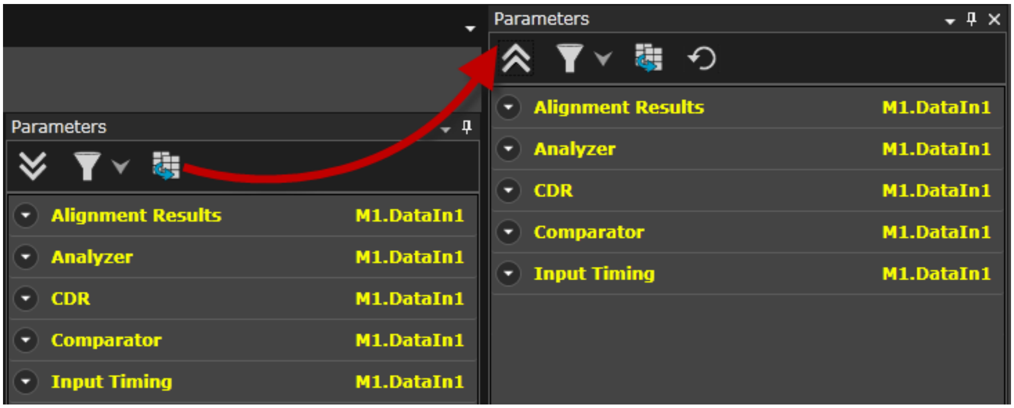
- Tool bar
- Display Grid
- Parameters Description Pane

The **Parameters** window tool bar includes the following icons:

- Expand/Collapse All Group: Click this icon to expand or collapse the functional blocks.

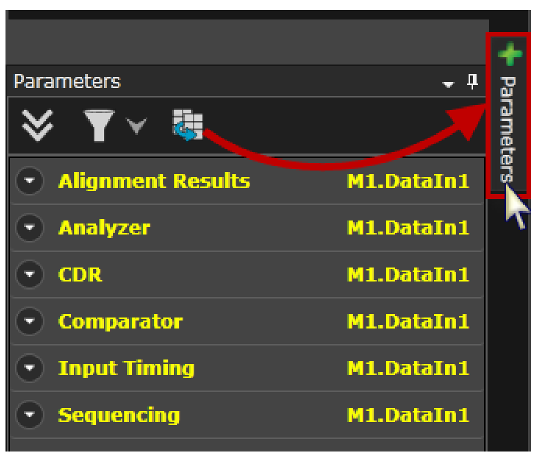
- Show Search Option: Allows you to filter and customize your results by using the following options:
  - By Location
  - By Functional Block
  - By Property

- Copying Parameter Window: Creates a replica of **Parameters** window to enhance the usability. It allows you to work on two different instances of the application. The changes you make in one window display immediately in the other window. The icon present on the copied parameter window loads all properties of the module. The following figure shows the copied **Parameters** window.



Once copied and then enabling the auto hide feature, the copied parameters window is docked to the right edge of the main user interface. The copied parameters window pops up once you click on it.

The following figure shows the copied parameters window docked to the right edge of the main user interface.



- **Display Grid**: Displays the parameters of the selected port/port groups within a grid. The left column contains the parameter names; the right column contains the parameter values.

  In addition, it also allows you to set the parameters of the selected port/port groups.

The naming convention used for the port/port group is explained with the help of following example:

M1.DataIn1

Where,

- M1 stands for Module1
- DataIn stands for DataIn port
- 1 for channel 1

The functional blocks use the different color schemes to represent different channels. You can set the color schemes of each channel from the Setting window. For details, refer to Settings Window on page 422.

- **Parameter Description Pane**: Provides the description of the currently selected port.

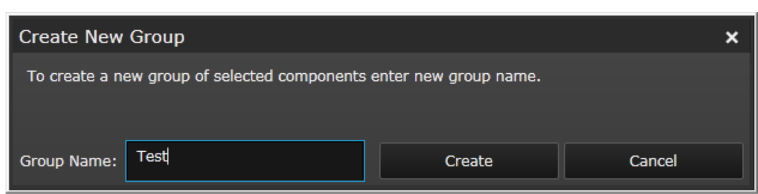
Creating Groups in the Module View

The **Module View** allows you to create a group of available ports and simultaneously allows you to configure their parameters.

To create a group:

- Switch to **Multiple Selection Mode** by pressing **Single Selection Mode** icon.
- From your keyboard, hold the Ctrl key and select the ports from the modules. You need to select at least two ports in order to create a group.
- Right-click on the selected ports and click **Create Group** option or

  alternatively you can select the ports and click **Create Group of Selection** icon. A **Create New Group** dialog will appear as shown in the following figure:



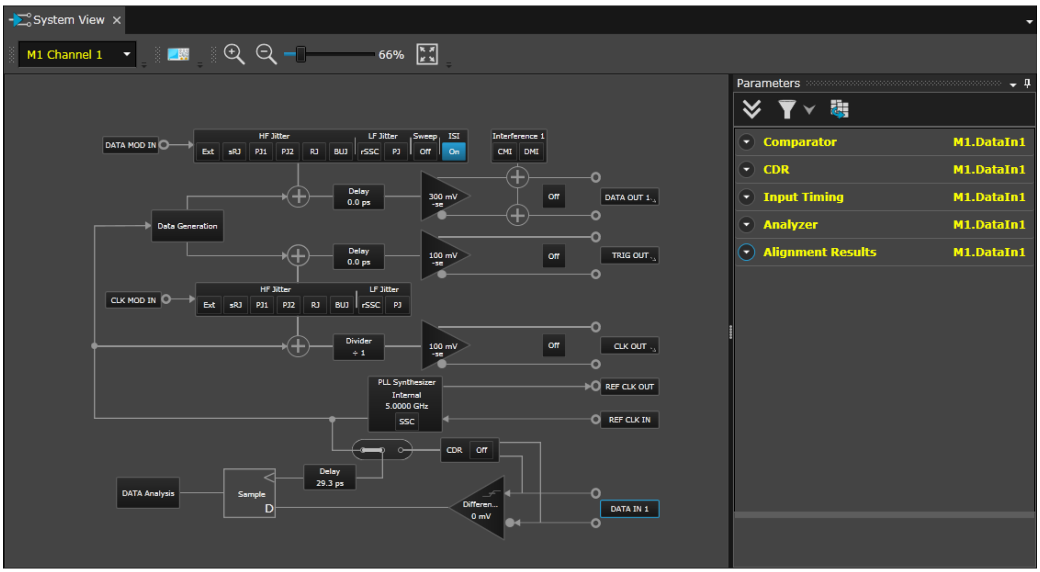
- Provide a group name and press **Create**.

A new group will be created in the **Group View**.

System View

The **System View** displays the block diagram representation of the currently selected channel of the M8020A/M8030A. In addition, it also allows you to interactively modify the settings for each channel.

The **System View** user interface is shown in the following figure.



The principal parts are represented by blocks which are connected by lines/arrows to show the relationship of the blocks. These blocks are highlighted by the blue lines around block on mouse focus and the corresponding parameters are displayed in the **Parameters** window. The block may contain feature elements (for example, LF Jitter contain PJ and rSSC). You can click on the respective button to enable the feature elements. Once enabled, the feature elements are highlighted in blue color.

Channels

Displays the channels that are connected to the module. You can select the channel on which you want to configure channel settings. You can assign a different color to each channel so that they are easily identified. For details, refer to Settings Window on page 422.

Using the Zoom Tool

The following figure shows the options provided by the zoom tool.

The zoom tools provide the following functionality:

- The  **Zoom In** button allows you to enlarge the block diagram to view more details.
- The  **Zoom Out** button allows you to reduce the block diagram.
- The **Zoom Slider** allows you to zoom in or zoom out the block diagram.

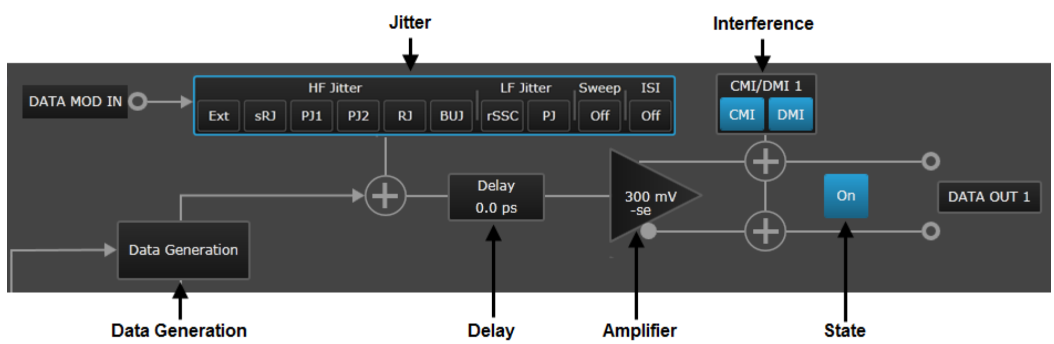
  The mouse wheel also provides a quick alternative to the zoom control. To zoom in and out using the mouse, hold down the [Ctrl] key while you turn the mouse wheel. Each click, up or down, increases or decreases the zoom factor by 25%.
- The  **Fit to View** button fits the width of the block diagram so that the user does not have to scroll the block diagram to the right or to the left.

Understanding the System View

To understand the **System View**, let's divide it into the following sections:

Generator

The following section of the System View represents the generator function:



It allows you to apply settings on the Data Out port or location (data generator).

It includes the following blocks:

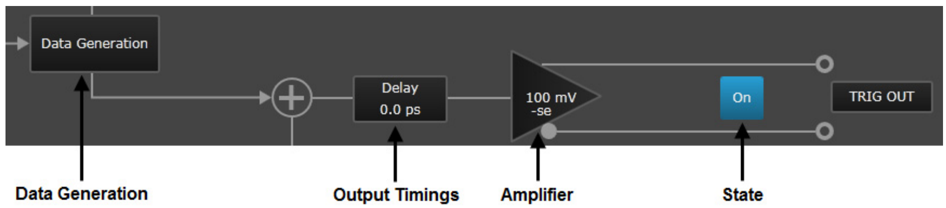
- State - Click on the 

 button to enable the outputs of Data Out for the selected channel. Once the outputs are enabled, the button changes to “ON” 

. If you press the button again, it will turn the state “OFF”.
- Jitter - Use this block to enable the jitter source of High Frequency Jitter, Low Frequency and Sweep. For more details on jitter, refer to Jitter Setup on page 186.
- Intersymbol Interference -
- Interference - Use this block to enable the interference (CMI or DMI). For details refer to Interference on page 184.
- Amplifier - This block represents the current value of Data Out's amplitude parameter. You can change the value by clicking on this block and modify the respective parameter in the Parameters window.
- Delay - Set the delay of the active edge of the data output.
- Data Generation - Acts as the output port for the generator which may be connected to the DUT.

Trigger

The following section of the System View represents the trigger function:



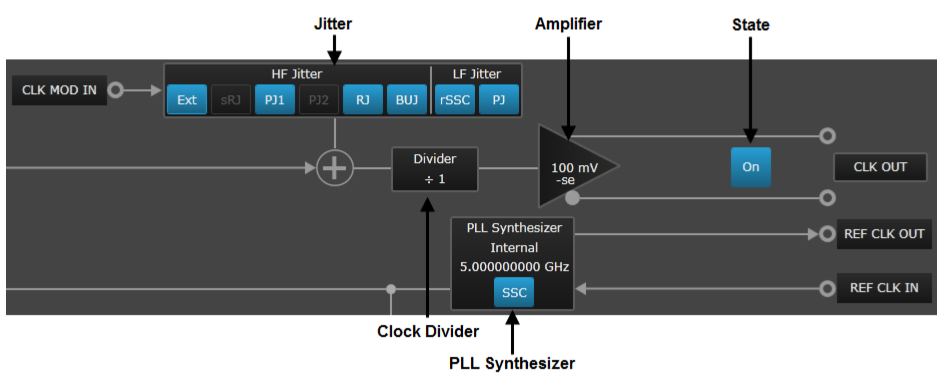
It allows you to apply settings on the Trigger Out port or location.

It includes the following blocks:

- State - Click on the Off button to enable the output of Trigger Output for the channel. Once the output is enabled, the button changes to “ON” On. If you press the button again, it will turn the state “OFF”.
- Amplifier - Use this block to set the parameters related to amplifier of the trigger output.
- Output Timing - Use this block to set the delay of the active edge of the trigger output.

Clock

The following section of the System View represents the clock function:

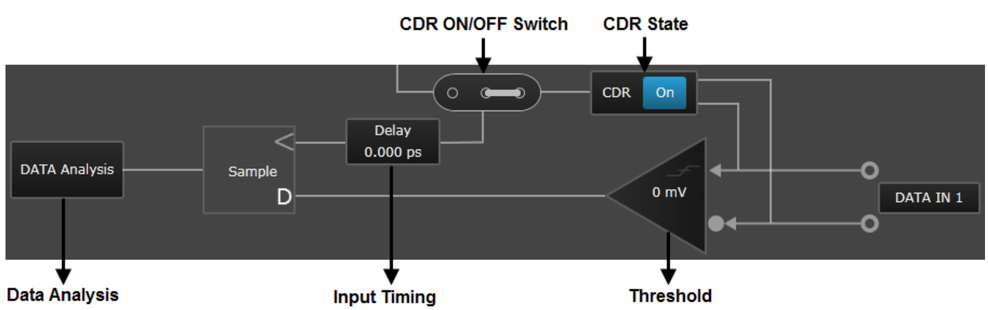


The clock function allows you to apply settings on the Clock Out port or location. It includes the following blocks:

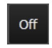
- State - Click on the  button to enable the output of Clock Source. Once the output is enabled, the button changes to “ON” . If you press the button again, it will turn the state “OFF”.
- Jitter - Use this block to enable the elements of High Frequency Jitter, Low Frequency. For more details on jitter, refer to Jitter Setup on page 186.
- Amplifier - Use this block to set the parameters related to amplifier of the clock output.
- Clock Divider - Use this block to set a factor on which the output signal will be divided.
- PLL Synthesizer - This block represents the currently selected clock source. You can change the clock source by clicking on this block and the modify the respective parameter in Parameters pane. To enable the SSC state jitter source, click on the  button. Once the SSC state is enabled, the button changes to “ON” . This is how the bit rate is set. For example, 5 GHz sets the bit rate to 5 Gb/s. If you press the button again, it will turn the state “OFF”.

Analyzer

The following section of the System View represents the analyzer function:



The Analyzer function allows you to apply settings on the Data In port or location (analyzer). It includes the following blocks:

- State - Click on the  button to enable the CDR state. Once the CDR state is enabled, the button changes to “ON”  . For more details, refer to CDR Setup on page 208. If you press the button again, it will turn the state “OFF”.
- Threshold - Use this block to set the threshold of the input comparator.
- Input Timing - Use this block to set input timing delay.
- Data Analysis - Use this block to set input threshold and sampling point delay.

System View with M8061A Integration

This section describes the block diagrams provided by the **System View** when M8061A is integrated with other modules of M8020A.

Channels

Displays the channels that are connected to the M8020A. The following figure shows M1 (2 channels of M8041A), M2 (2 channels of M8051A) and M3 (M8061A) modules connected to the instrument.

When you select the channel, the corresponding block diagram of that channel appears which allows you to interactively configure channel settings. You can also assign a different color to each channel so that they are easily identified. For details, refer to Settings Window on page 422.

Integration Modes

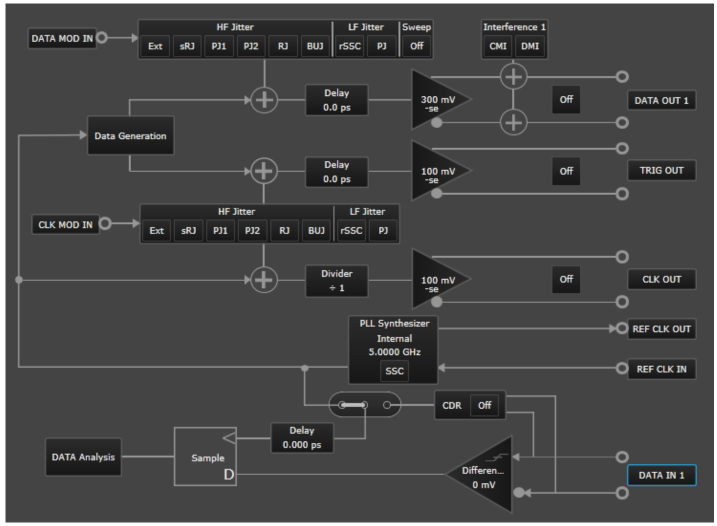
The **System View** with M8061A integration provides the following modes:

- Standalone Mode
- Mux Only Mode
- Demux Only Mode
- Mux and Demux Mode

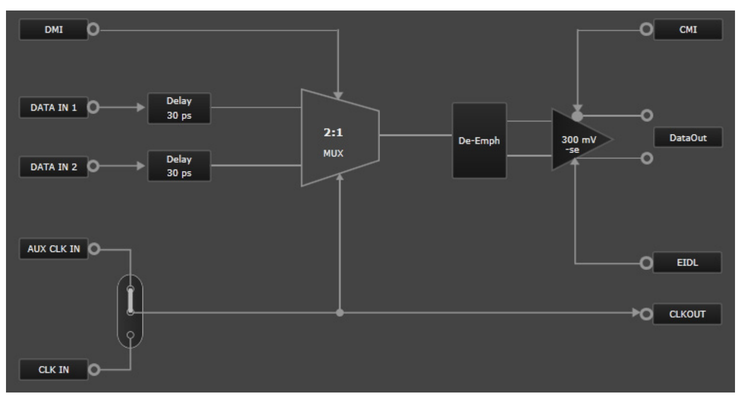
Standalone Mode

The **Standalone** mode is used to set the parameters of the Data Out port of M8041A module and Data In and Clk Gen ports of M8061A module.

The following figure shows the block diagram of M8041A module in **Standalone** mode:



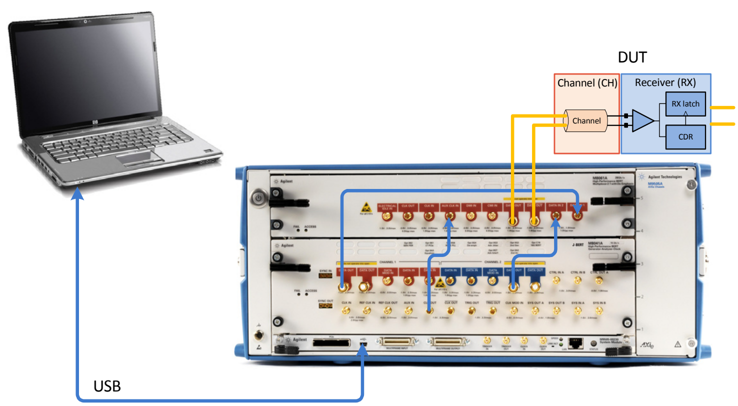
The following figure shows the block diagram M8061A module in **Standalone** mode:



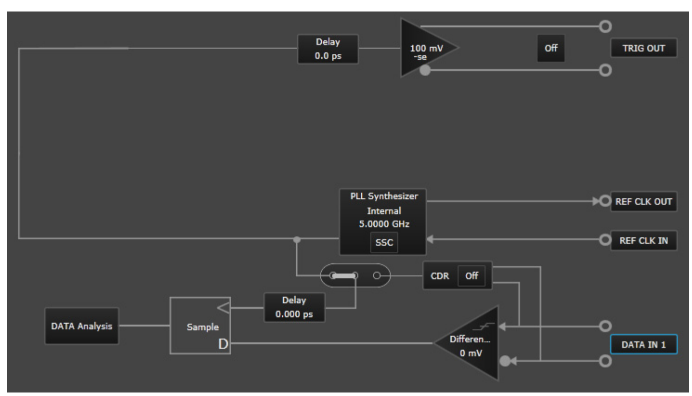
Mux Only Mode

The **Mux Only** mode is used to multiplex the signal in 2:1 ratio. In this mode, M8061A receives the non-inverted signal from the two Data Out ports of M8041A module and multiplex them in one channel in 2:1 ratio. Using this mode, you can set the parameters of Data Out port of M8061A module.

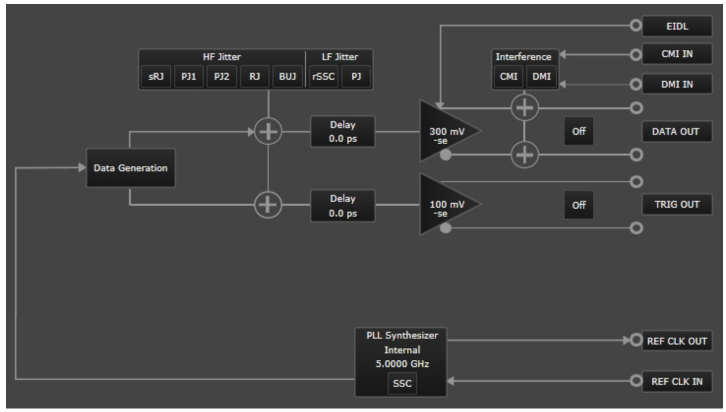
The following figure shows the connection diagram of **Mux Only** mode:



The following figure shows the block diagram M8041A module in **Mux Only** mode:



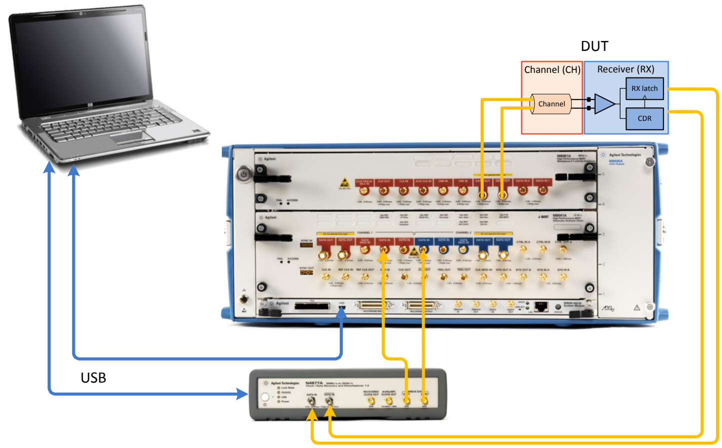
The following figure shows the block diagram M8061A module in **Mux Only** mode:



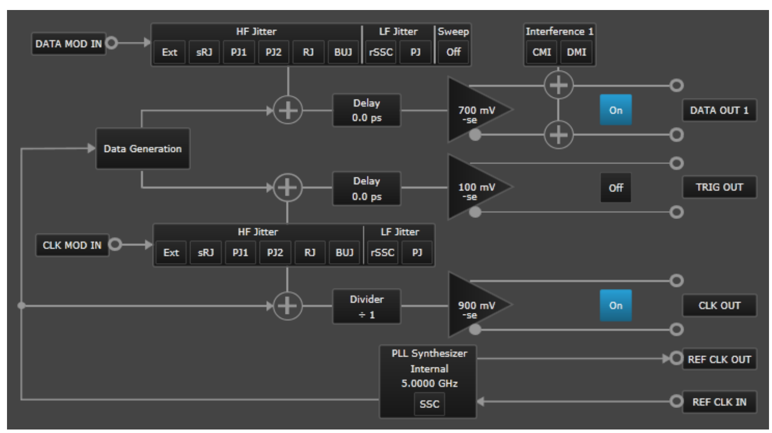
Demux Only Mode

The **Demux Only** mode is used to de-multiplex the signal in 1:2 ratio. In this mode, the M8061A module provides the output to CDR module (N4877A) which de-multiplex the received signal in 1:2 ratio. Using this mode, you can set the parameters of Data Out and Data In ports of M8061A module.

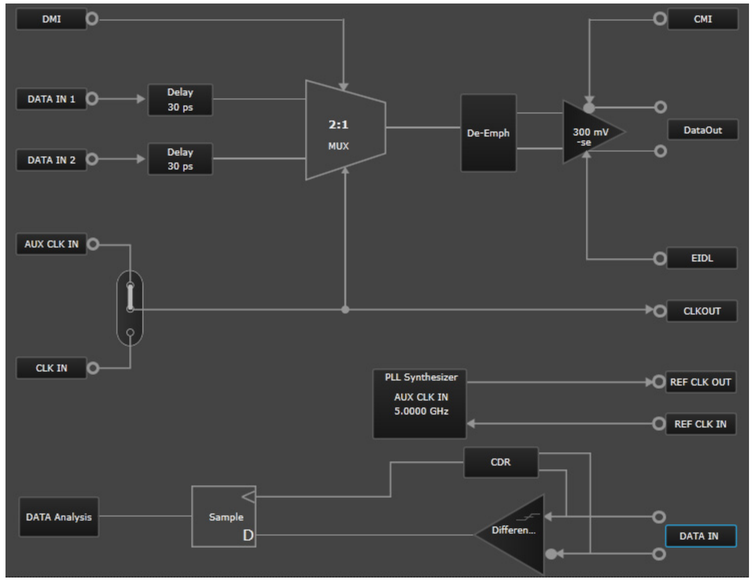
The following figure shows the connection diagram of **Demux Only** mode:



The following figure shows the block diagram M8041A module in **Demux Only** mode:



The following figure shows the block diagram M8061A module in **Demux Only** mode:

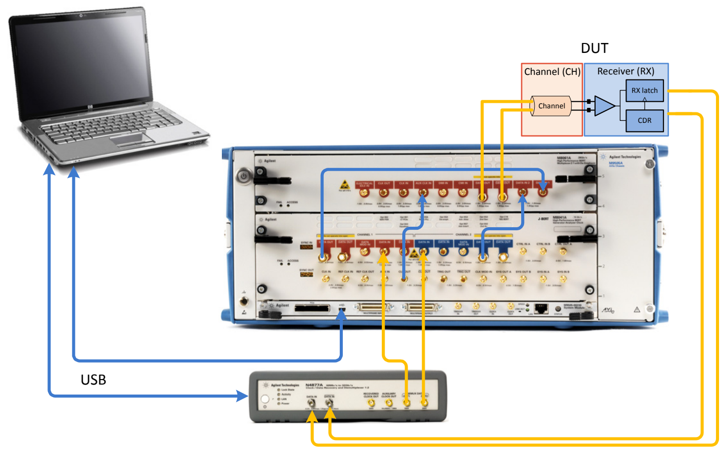


Mux and Demux Mode

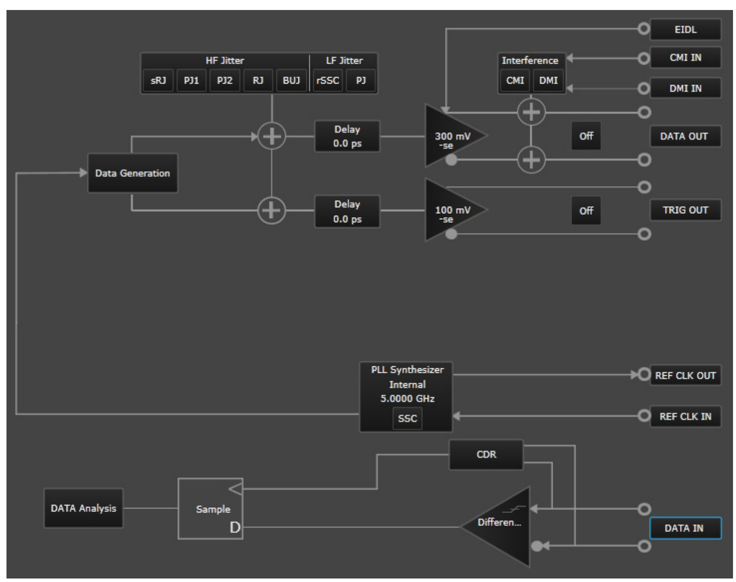
The **Mux and Demux** mode is use to multiplex the signal in 2:1 ratio and again de-multiplex the signal in 1:2 ratio. Using this mode, you can set the parameters of Data Out and Data In ports of M8061A module.

The M8061A expands the data rate up to 32 Gb/s and provides integrated and calibrated 4-tap de-emphasis (expandable to 8 taps).

The following figure shows the connection diagram of **Mux and Demux** mode:



The following figure shows the block diagram in **Mux and Demux** mode:



System View with M8062A Integration

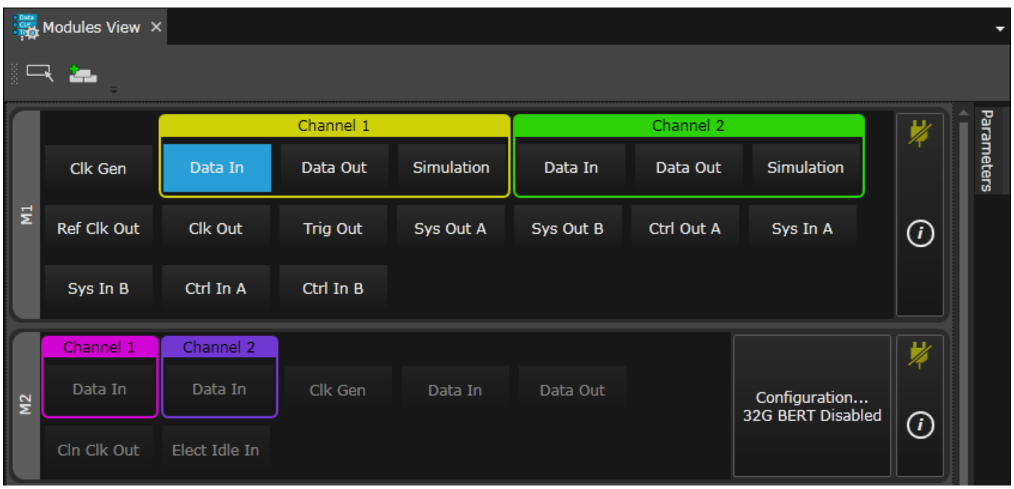
This section describes the block diagrams provided by the **System View** when M8062A is integrated with other modules of M8020A.

M8062A Configuration

The following steps describe the procedure for M8062A configuration:

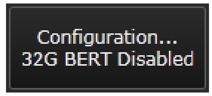
- Launch the M8070A software.
- If the **Modules View** is not already opened, go to **Menu Bar** > **System** and then click **Module View**.
- Locate the **M8062A** module. The following figure shows an example of

Module View when an M8062A (M2) and an M8041A (M1) are installed in the M8020A system:

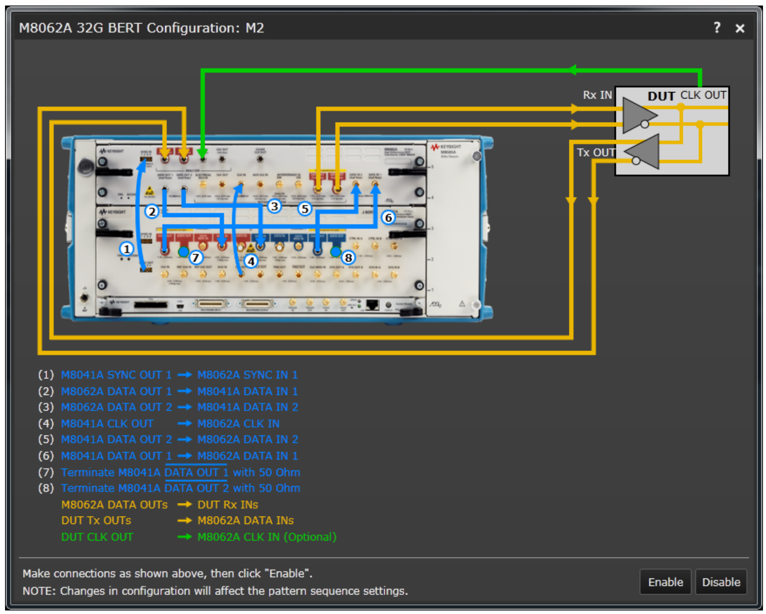


NOTE The M8062A module cannot be used standalone. It must be used in combination with an M8041A module.

- Click the **Configuration...** button present on the M8062A module.



The **M8062A Configuration** dialog will appear as shown in the following figure:

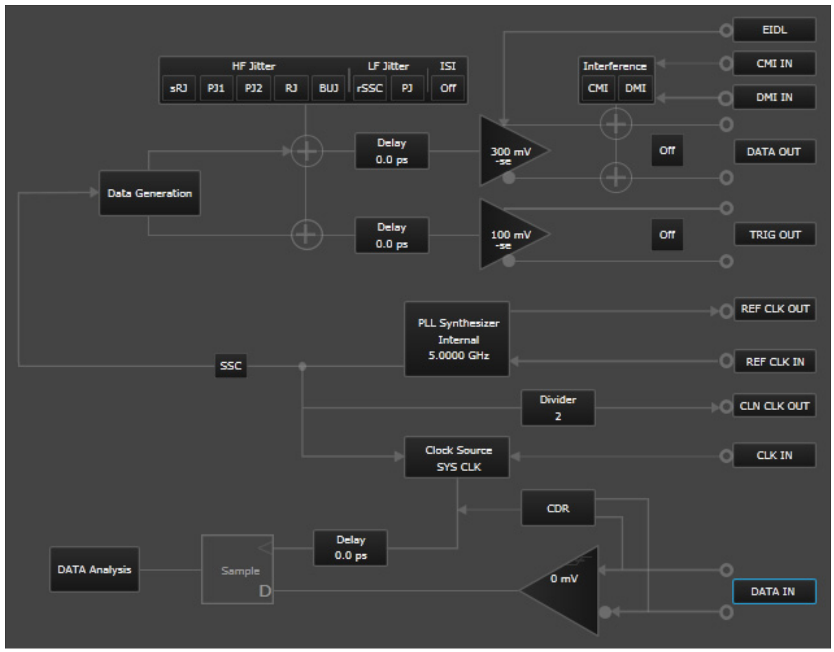


- Make the connections as described in the above figure.
- Click **Enable**.
- For the block diagram representation and interactively modify the settings of the currently mode, switch to **System View**. To do so, go to **Menu Bar** > **System** and then click **System View**.

NOTE System view for M8062A is available only when 32G mode is enabled.

NOTE In 32G mode, access to some M8041A user controls are disabled to facilitate software control of this configuration.

The following figure shows the **System View** of the M8062A module:



In the M8062A **System View**, the principal parts are represented by blocks which are connected by lines/arrows to show the relationship of the blocks. These blocks are highlighted by blue lines around the block on mouse focus. Parameters corresponding to the selected block are displayed in the **Parameters** window, on the right. The block may contain feature elements (for example, LF Jitter contains PJ and rSSC). You can click on the respective button to enable the feature elements. Once enabled, the feature elements are highlighted in blue color. For more details, refer to System View on page 119.

In addition to the blocks that are described in the section System View on page 119, the M8062A **System View** has the following new blocks:

- **Clean Clk Out** - Half-rate, or divided, clock output with no applied jitter.
- **Clk In** - Pattern Generator clock input (half-rate). Connect to clock output of M8041A.
- **Clock Source** - Selects the clock source (CDR, System Clock or Clock In) for the M8062A DataIn port (Analyzer).
- **CDR** - Selecting "CDR" as the M8062A DataIn clock source enables the Analyzer CDR, so that incoming data is sampled using the recovered clock.
- **Auto Re-Lock** - When this feature is enabled (the default setting) the CDR will automatically re-lock when there is a loss of lock. When it is disabled, the CDR will only attempt to re-lock when manually initiated by clicking on the arrow next to the Auto Re-Lock On/Off button. It may be necessary to perform a manual re-lock after a pattern change.
- **High Transition Density** - For data patterns with high transition density, such as 1010 pattern, the CDR may have trouble gaining lock. Enabling the High Transition Density setting reduces the CDR's internal gain to allow it to better lock on patterns with high transition densities. It may be necessary to perform a manual CDR re-lock after a change in the High Transition Density setting.
- **Optimize** - After the CDR has gained lock, certain condition changes, such as increasing applied jitter levels, can cause the analyzer to begin to measure errors. Executing the Optimize function causes the CDR to perform a finer alignment adjustment, which may result in a return to an error-free measurement.

NOTE

When using the Analyzer CDR, it may take a longer time to lock and achieve BER=0 after a frequency change than with the other clock sources.

Group View

The **Group View** allows you to:

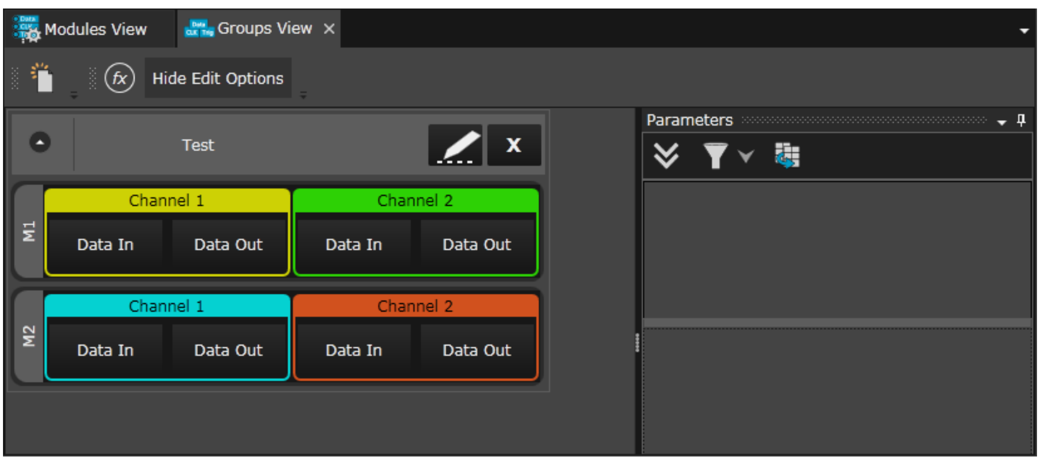
- Add/remove ports of a group.
- Set the properties of functional block of a group.

How to Launch Group View

The **Group View** is automatically launched when you create a group from the **Module View**. For details, refer to Creating Groups in the Module View on page 118.
However, you also launch the **Group View** from the main GUI. To do so:

- Go to the **Menu Bar** > **System** and then select **Group View**.

The **Group View** will appear as shown in the following figure:



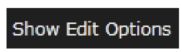
The **Group View** includes the following elements:

- Tool bar
- Group View

Tool bar

The tool bar includes the following elements to perform specific functions:

Table 39

Elements	Name	Description
	Create New Group	Opens the **System Group Editor** which allows you to create a new group. For more details, refer to Create New Groups in Group View on page 138.
	Show Functional Block in Group	Use this toggle button to show functional blocks or locations in a group.
	Show Edit Options	Use this toggle button to show/hide the group edit options.
	Edit Existing Group	Opens the **System Group Editor** which allows you to edit the group. For details, refer to Edit Existing Group on page 140. This icon is only visible when the group is in the edit mode.
	Delete Group	Deletes the group.

Group View

The **Group View** allows you to create, edit and delete a group. In addition, it also allows you to set the properties of functional block of the group. These features are further described in the upcoming sections of this chapter.

Create New Groups in Group View

To create a group in **Group View**:

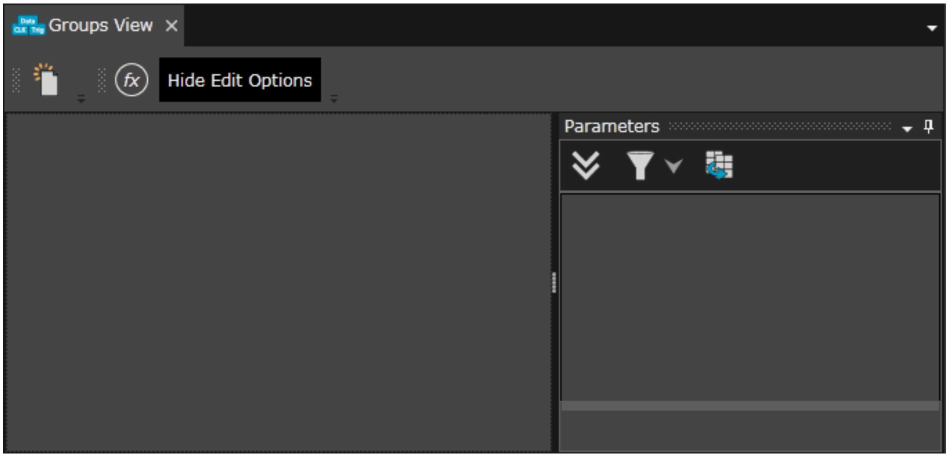
- Click **New Group** icon available on the tool bar. This will open the **System Group Editor**.
- Select the ports or locations from the left side the **System Group Editor** and the preview of selected ports or locations will appear on right side. See the following figure:

- Type a group name in the provided field and click **Create**. A new group will be created under that name.

Show Function Block/Location in Groups

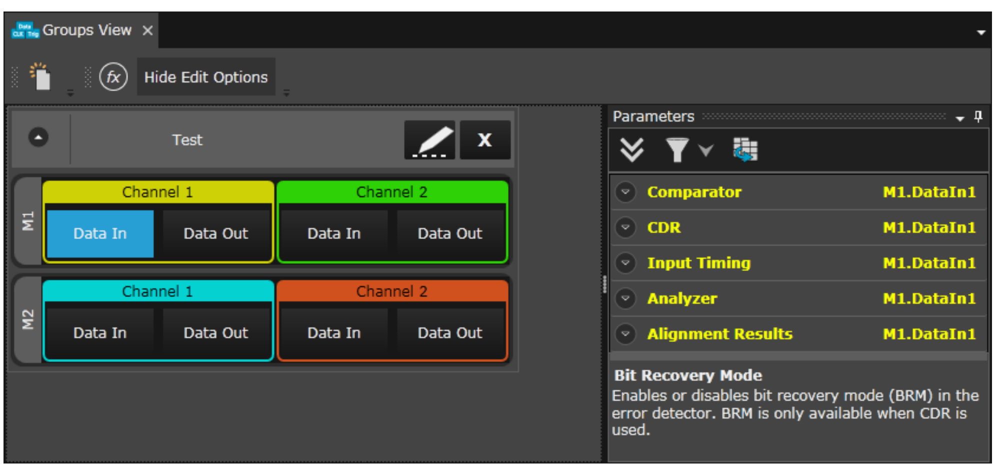
The **Group View** allows you to set the group properties either through functional blocks or through locations in the group.

You can click  icon to switch to show functional blocks in the group. See the following figure:



The group is represented in the form of functional blocks available in the group. Once you click on the function block, the respective properties of the functional block are visible in the **Parameters** window. You can now set the properties of each functional block.

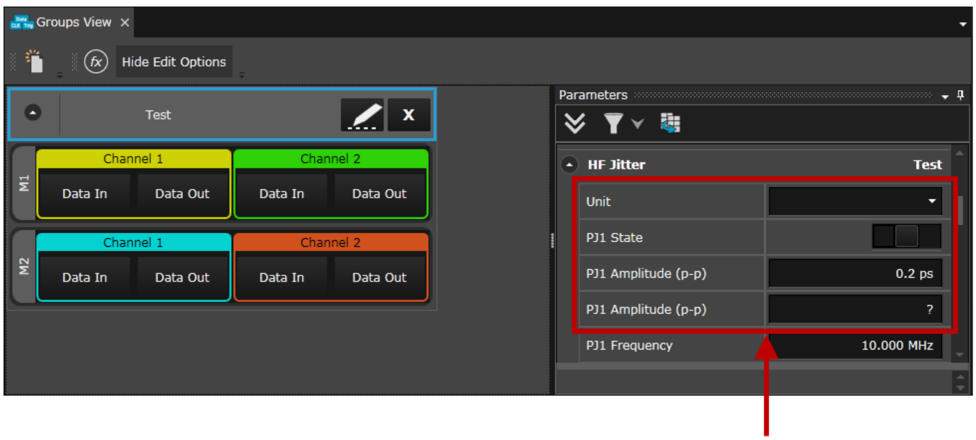
You can click  icon to switch to show locations in the group. See the following figure:



The group is represented in the form of ports or locations available in the group. Once you click on the port or location, the respective properties of the port or location block are visible in the Parameters window.

The main advantage of the **Group View** is that it allows you to set the properties patterns/sequence of similar ports at once.

However, if you set the properties of the similar ports or locations that exists in different channels of a group, the properties of the ports or locations will not be applied. If you try to group the properties by clicking on the header of **Group View**, the corresponding value in the Parameters window will either be blank or show “?”.
See the following figure:



Values not applied in Parameters Window

In this situation, you have to specify the property’s value which will be later applied to both ports or locations.

Edit Existing Group

To edit an existing group;

- Click on the  **Edit Existing Group** icon. This will open the **System Group Editor** window.

- Select the ports or locations from the left side of the **System Group Editor** and the preview of selected ports or locations will appear on right side. See the figure below:

- Click **OK**. The group will be edited. However, if you provide a new group name in the provided field and click **OK**, a new group will be created under that name.

Delete a Group

You can also delete a group. To do so:

- Click on the 

Delete Group icon. It will open a **Delete Group** message box.
- Click **Delete**. The group will be removed from the view.

Adjustable Intersymbol Interference

The M8070A system software provides integrated and adjustable Intersymbol Interference (ISI) capability to test next-generation high-speed digital designs.

The adjustable and programmable ISI function allows emulating channel loss for characterizing and compliance testing of high-speed digital receivers. This function is integrated in each pattern generator channel of the M8020A/M8030A which streamlines test setup by eliminating the need for external cabling and switching of external ISI traces.

As data rates continuously increase, channel loss between transmitter and receiver becomes more important. Channel loss is caused by printed circuit board traces, connectors, and cables in the signal path; resulting in ISI which depends on the channel material and dimensions, the data rate, and bit pattern. All high-speed digital receivers are specified to tolerate a certain amount of total jitter, which typically includes some ISI caused by channel loss. During receiver characterization and compliance test, this loss needs to be emulated.

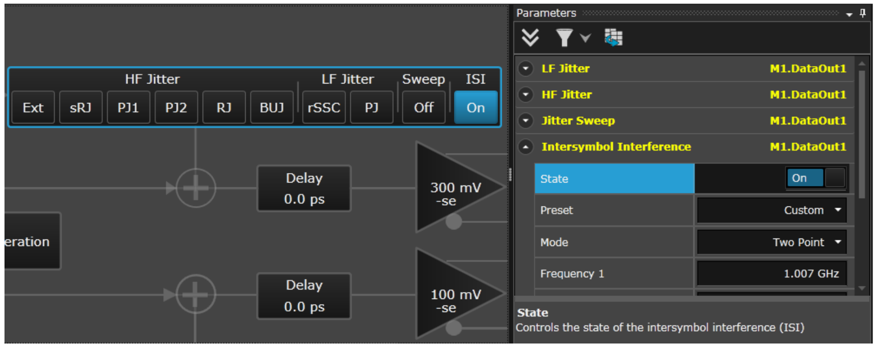
The built-in ISI functionality is programmable for each pattern generator channel. It simplifies receiver test automation for data rates up to 16 Gb/s.

The new adjustable ISI option offers the following advantages:

- Streamlines receiver test setup by providing the highest level of integration.
- Provides accurate and repeatable results for receiver characterization and compliance test by supporting a wide range of loss up to 25 dB and 16 GHz with linear loss curves and fine step resolution.
- Fits into the fully scalable and upgradeable receiver test solution with an upgrade option.

Using System View for ISI Configuration

You can use the **System View** to configure the ISI parameters for a selected '**Data Out**' location. When you select ISI in the **Jitter** block of the **System View**, the corresponding parameters are loaded on the **Parameters** window. You can create your own preset or manipulate frequency and insertion loss for the selected preset.



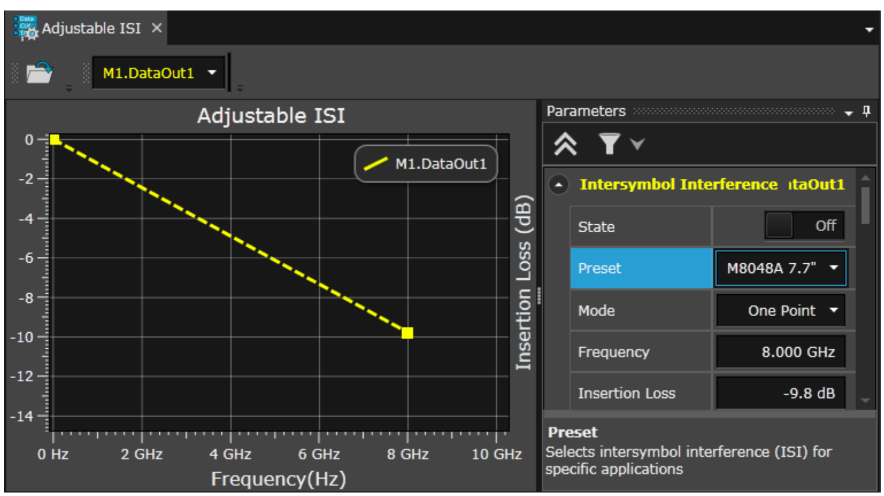
However, the **System View** does not provide flexibility to graphically manipulate the ISI parameters. This can be achieved using the **Adjustable ISI** window.

How to Launch Adjustable ISI Window:

To launch the **Adjustable ISI** window

- Go to the **Menu Bar** > **System** and then select **Adjustable ISI**.

The **Adjustable ISI** window will appear as shown in the following figure:



It includes the following elements:

- Toolbar
- Adjustable ISI Graph
- ISI Parameters

Toolbar

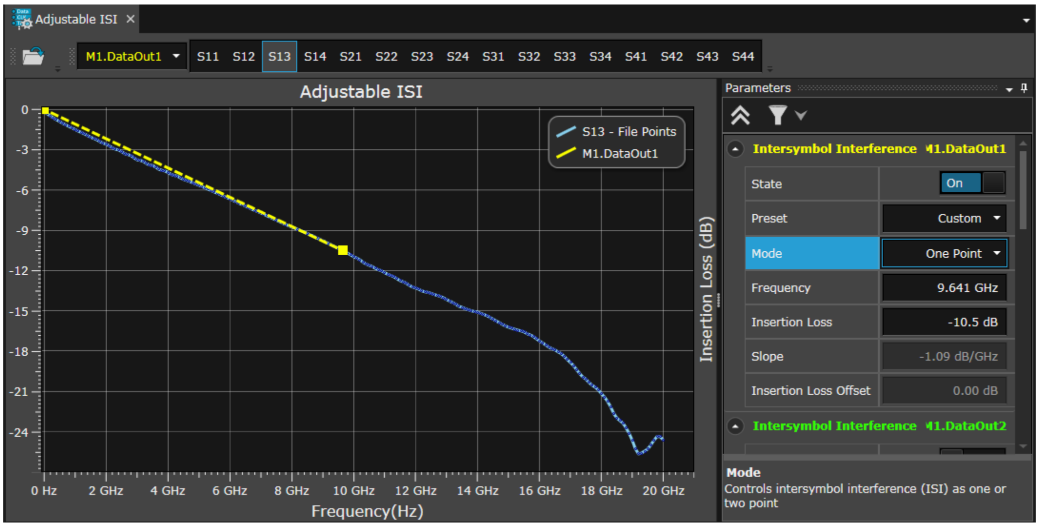
The toolbar provides the following convenient adjustable ISI functions:

Table 40

Elements	Name	Description
	Open SnP File	Click this button to open an s-parameters file (S2P). Currently, the M8070A GUI supports two port (S2P) and four port (S4P) s-parameters files. The s-parameters (also known as scattered parameters) are displayed when the scattered pattern file is loaded. S-parameters are complex matrix that show insertion loss at certain frequency. The two-port device displays four data points while the four-port device displays sixteen data points.
	Channel Selection	Use this drop-down list for channel section for which the graph is plotted for the corresponding values.

Adjustable ISI Graph

The following figure illustrates an ISI graph for 4-port system:



The ISI graph shows the insertion loss for a 2-port or 4-port system. The values on the graph represents insertion loss with respect to frequency. You can select the data points according to which you want to generate the loss and then manipulate points graphically (mouse drag) or by parameter window to draw the data point graph. Please note that if you manipulate the points of the linear graph to the values which are not in the defined limits of Slope and Offset, the pointer will change to icon. In this case, the linear graph will switch to last set values. However, if you manipulate the points of the linear graph to the values which are not in the defined limits of Slope and Offset, the pointer will change to icon. In this case, a message will appear at the bottom of the graph along with the undo option to revert the graph changes.

ISI Parameters

The ISI parameters are summarized in the following list:

- **State** - Enables/disables the ISI state.
- **Preset** - Allows you to select preset for specific applications representing a typical loss characteristic.

- **Mode** - Controls the ISI as one point or two points. For each point you can specify frequency and insertion loss at that frequency. The calculated values are displayed in the **Parameters** window.
  - **One Point**: In this mode, one point of the linear graph is fixed while the another point can be moved graphically to manipulate insertion loss/frequency values.
  - **Two Points**: In this mode, both points of the linear graph can be moved graphically to manipulate insertion loss/frequency values.
- **Slope** - Shows the calculated slope value (Insertion Loss/frequency, dB/ GHz).
- **Insertion Loss Offset** - Shows the Insertion Loss Offset (at frequency= 0 Hz).

Intersymbol Interference (ISI) Setup

The following steps describe the procedure for ISI setup:

1. Connect the DUT in loopback mode. For connection details, refer to *M8020A and M8030A Getting Started Guide*.
2. From the M8070A software, launch **Adjustable ISI** window.
3. Select **M1.DataOut1** channel from drop-down list.
4. From the **Parameters** window, select the **Preset** from the provided list. It will show the linear graph.
5. Select the mode as '**Two Points**'.
6. Manipulate insertion loss/frequency values either graphically or by **Parameters** window.
7. Alternatively, you can also open a S2P or S4P file and graphically manipulate loss and frequency parameters according to data points.
8. Click the toggle button to enable the ISI state.

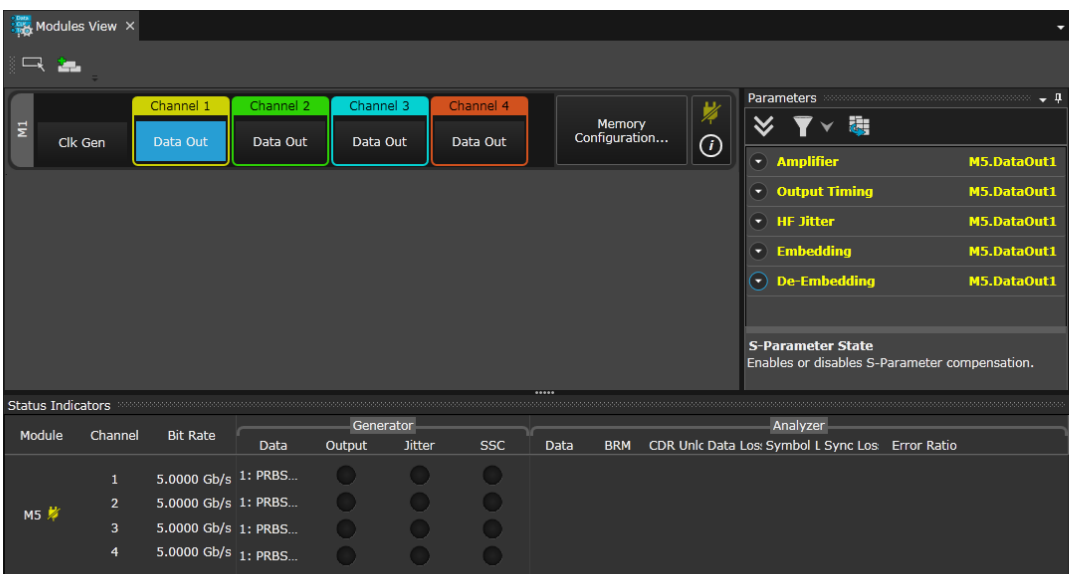
Controlling M8195A AWG from M8070A User Interface for Generating PAM-4 Signals

The M8070A system software (Rev 2.5.0.0 or above) allows to control M8195A along with the M8020A/M8030A modules. This feature is enabled by license M8070A-1TP or M8070A-1NP. Once the M8195A AWG module is installed into an AXIe chassis, it can be accessed via the M8070A software. Ensure that you have latest version of M8070A (Rev 2.5.0.0 or above) and M8195A (Rev 1.3.1.1 or above) software installed in your system.

For complete details on M8195A Arbitrary Waveform Generator, visit www.keysight.com/find/m8195a.

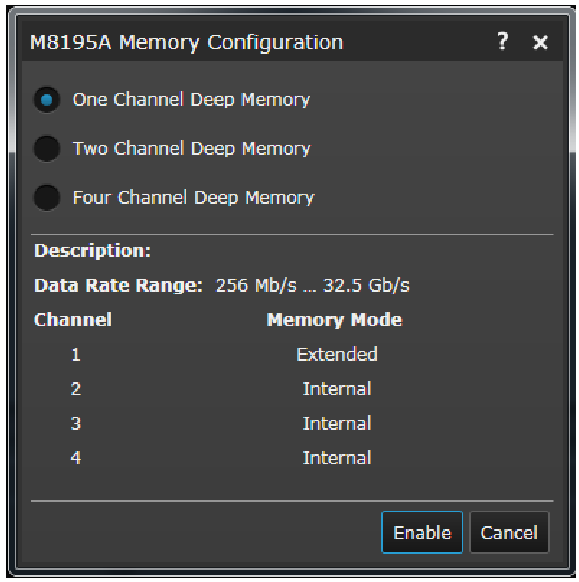
M8195A AWG in Module View

Once the M8195A AWG integration is done, you will see a M8195A module entry in the **Module View** and **Status Indicator** as shown in the following figure:



M8195A Memory Configuration

1. Click the **Memory Configuration** button. Depending upon the available channels in M8195A, the **M8195A Memory Configuration** will appear. The following figure shows the **M8195A Memory Configuration** dialog when four channels in M8195A are available:



2. Click on the radio button to select the memory mode. The following memory modes are available:
   - **One Channel Deep Memory**: The DataOut location of one channel will be sourced from extended memory and the other channels will be sourced from module internal memory. The data range in this mode is 256 Mb/s ... 65 Gb/s.
   - **Two Channels Deep Memory**: The DataOut location of two channels will be sourced from extended memory and the other channels will be sourced from module internal memory. The data range in this mode is 256 Mb/s ... 32.5 Gb/s.
   - **Four Channels Deep Memory**: The DataOut location of all four channels will be sourced from extended memory. The data range in this mode is 256 Mb/s ... 16.250 Gb/s.
3. Click **Enable**.

NOTE

In case the AWG module is configure with 2 or 4 deep memory channels the output signal will contain artefacts if not used with a transition time converter/low pass filter in the range of 16 GHz for 2 deep memory channels or 8 GHz for 4 deep memory channels.

Configuring M8195A Parameters

The M8195A AWG has the **Data Out** and **Clk Gen** ports. You can use the **Parameters** window to configure these ports.

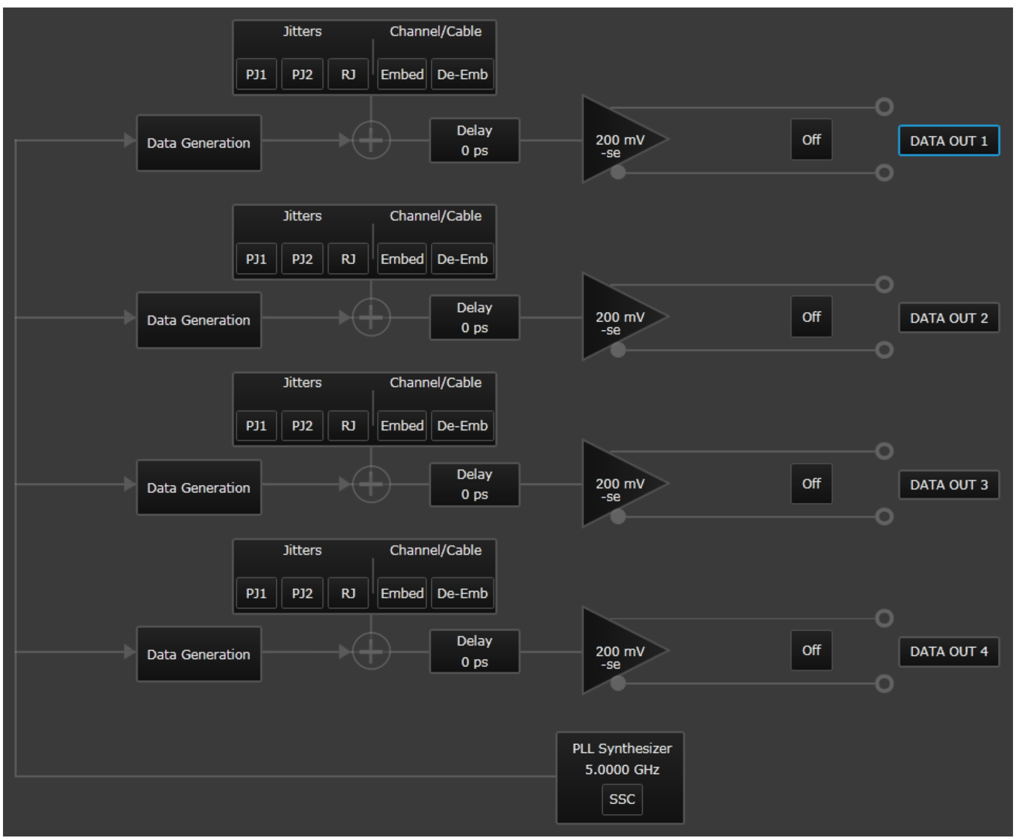
- **Embedding** - Sets the S-Parameter for embedding.
- **Amplifier** - The **Amplifier** function has the following components:
  - Amplitude - Sets the amplitude of the output signal.
  - Offset - Sets the offset voltage of output signal.
  - High - Sets the high voltage of output signal.
  - Low - Sets the low voltage of output signal.
  - Transition Time - Sets the transition time (20%/80%) of data output signal.
- **Output Timing** - Sets the delay of the output data signal.
- **HF Jitter**- Enables and sets the high frequency jitter.
- **De-Embedding** - The **De-Embedding** function has the following components:
  - Standard Cable - Defines whether the signal should be pre-distorted considering the properties of the standard cable set.
  - S-Paramter State - Enables/Disables S-Parameter compensation.
  - S-Paramter Profile - Selects the S-Parameter profile. It also allows you to import and export a S-Parameter profile.
  - S-Paramter Output Port - Selects the output port in S-Parameter profile.
  - S-Prameter Input Port - Selects the input port in S-Parameter profile.
- **Clk Gen -** The **Clk Gen** port has the following components:
  - Synthesizer - Used to set data rate frequency and period. For NRZ line coding frequency may be set up to 32 GHz, and for PAM-4 the value may be set up to 16 GHz. The corresponding output for NRZ line coding will be up to 32 Gb/s, and for PAM-4 it will be up to 32 Gbaud/s.

NOTE The global Jitter button located on the M8070A status bar is presently inactive for M8195A as jitter injection for M8195A is not supported in the M8070A software version 2.5.

System View with M8195A Integration

This section describes the block diagram provided by the System View when M8195A module is integrated in M8070A. The System View for M8195A varies on the number of channels available on M8195A.

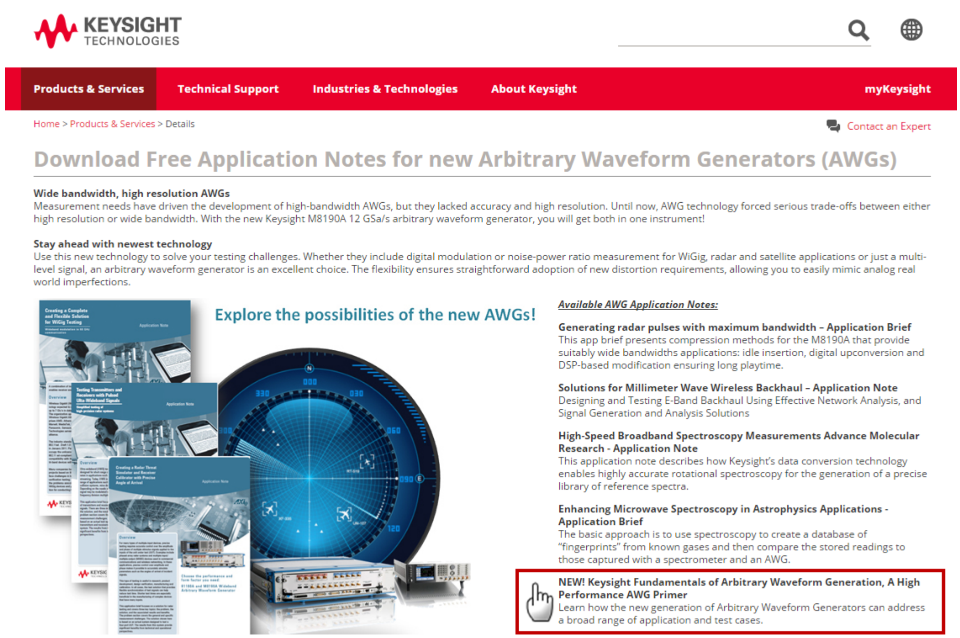
The following figure shows system view of M8195A when all four channels are available.



The M8195A **System View** shows four Data Out channels in the same window. The principal parts of the **System View** are represented by blocks which are connected by lines/arrows to show the relationship of the blocks. These blocks are highlighted by blue lines around the block on mouse focus. On clicking these blocks, the mostly used parameters are available which can be configured using the on-screen numeric keypad. However, parameters corresponding to the selected block are displayed in the **Parameters** window, on the right. The block may contain feature elements (for example, HF Jitter contains PJ1, PJ2 and RJ). You can click on the respective button to enable the feature elements. Once enabled, the feature elements are highlighted in blue color. For more details, refer to System View on page 119.

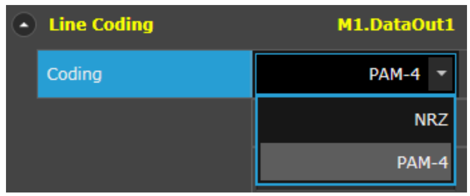
Signal Generation in AWG

For detailed information about signal generation on arbitrary waveform generators of the M81XX class can be found in the document named "Fundamentals of Arbitrary Waveform Generation" which can be downloaded from here: www.keysight.com/find/awg-apps



Selecting Line Coding

You can define the line coding for M8195A **Data Out** port through the **Parameters** window. The M8195A **Data Out** port supports the NRZ and PAM-4 line coding.



NOTE

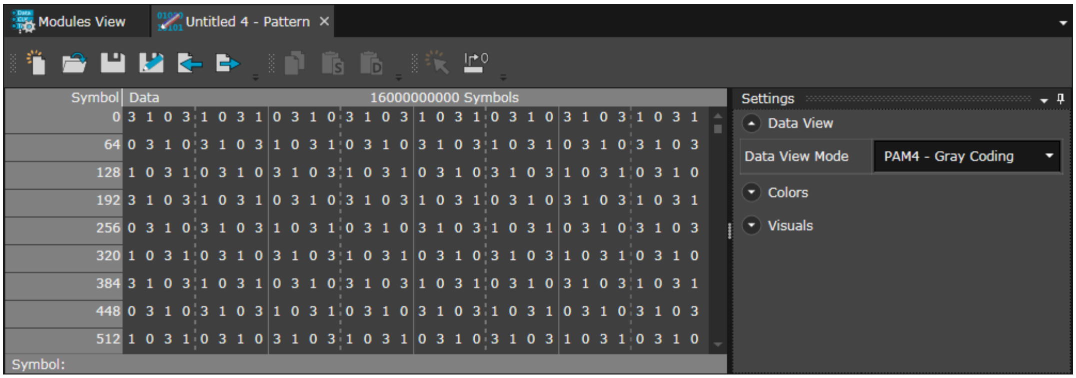
Make sure to use the similar line coding for all Data Out ports of M8195A. An "Auto Correction" window will appear if you select different line coding for Data Out ports. Clicking on the "Allow" button, present on this window applies same line coding on all Data Out ports.

Creating Patterns in M8195A

The M8070A software provides **Pattern Editor** for M8195A module as well. The functionality of the **Pattern Editor** is same as for other modules. You can create, edit, load patterns and perform the pattern settings in the same way as done for other modules. For more details, refer to Pattern Editor on page 250.

However for bit coded patterns, the **Pattern Editor** allows you to view the symbols in the **PAM-4 No Coding** and **PAM-4 Gray Coding**. These options are available in the **Settings** > **Data View Mode**.

The following figure shows an example of bit symbol, viewed in the **PAM-4 Gray Coding.**



In the **PAM-4 No Coding** mode, the bit sequence 00 maps to symbol 0, 10 maps to symbol 2 and 11 maps to symbol 3.

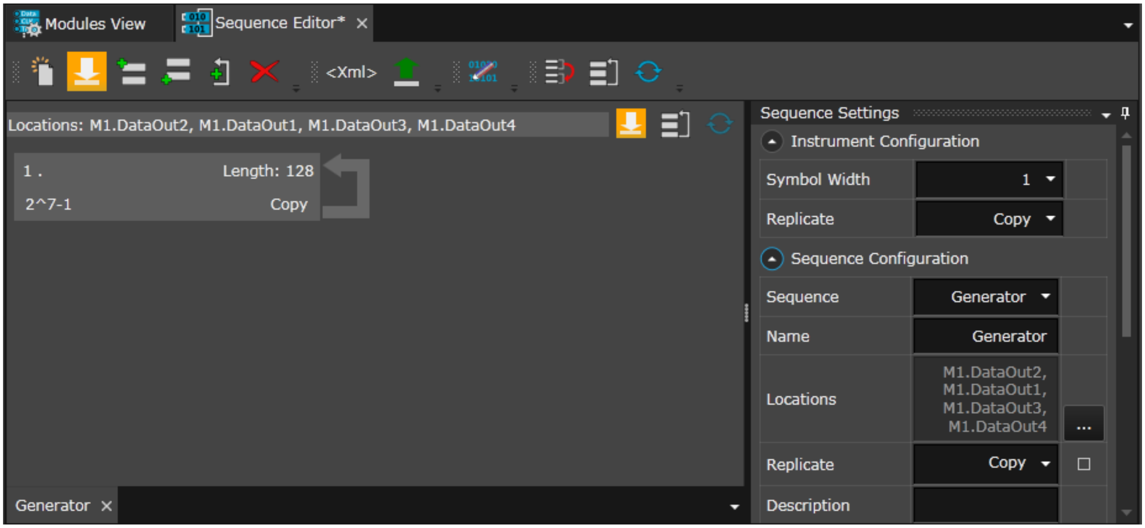
In the **PAM-4 Gray Coding** mode, the bit sequence 00 maps to symbol 0, 00 maps to symbol 0, 01 maps to symbol 1, 11 maps to symbol 2 and 10 maps to symbol 3.

Sequence Settings in M8195A

The M8070A software provides **Sequence Editor** for M8195A module as well. The functionality of the **Sequence Editor** is same as for other modules. You can create, edit, load patterns and perform the sequence settings in the same way as done in M8070A sequence editor. You can change pattern type as static, clock, PRBS, memory pattern, in single block with infinite loop. For more details, refer to Sequence Editor on page 227.

However, in the M8195A **Sequence Editor**, PAM-4 patterns are generated by encoding digital pattern streams (PRBS, Memory Patterns) into PAM-4 Symbol streams.

The following figure shows the M8195A **Sequence Editor**:



You can use the **Sequence Settings** window to reprogram the values of M8195A parameters.

Parameters that are not applicable for M8195A module will be disabled.

The M8195A **Sequence Editor** has the following capabilities:

- A single block can be looped infinitely
- Supports following block types:
  - PRBS (all M8020A/M8030A polynomials that fit into AWG memory)
  - Memory
  - Static 0 or 1
  - Single pulse per block
  - Clock

Jitter Setting in M8195A

The M8195A integration into the M8070A software supports sinusoidal jitter, random jitter and spread spectrum clocking (SSC).

Sinusoidal jitter and random jitter can be turned on and off per channel while SSC can only be activated per module.

Periodic Jitter (Valid for PJ1, PJ2 and SSC)

When defining a sequence referencing a specific pattern it will automatically be generated containing the specified jitter. For example in case the given pattern has a duration of 1 µs for the selected baud rate and the jitter period is 2 µs it would not be possible for this jitter to correctly appear inside the pattern. To handle this a second iteration of the pattern is appended giving the memory segment a total duration of 2 µs. With this it is possible for the resulting waveform to contain the selected jitter.

A side effect of this procedure is that the required sample memory for a selected pattern changes depending on the selected jitter components.

In case the jitter cannot be mapped to the available memory while adhering to the jitter accuracy defined in the data sheet a warning message will be displayed to make the user aware of this shortcoming.

Measuring Periodic Jitter

When measuring periodic jitter on an oscilloscope it is important to note that the jitter period will always be rounded in a way that the effective pattern length (which may contain multiple iterations of the user defined pattern) is a multiple of this jitter period.

Another thing to note is that for short pattern, high jitter frequency and the case where the data rate is an integer multiple of the jitter frequency the effective pattern can become maximally short. This will be observable by a relatively fast start of waveform playback but can have an impact on the measured jitter histogram.

For the above mentioned cases the number of distinct edge deviations from their un-jittered positions can be limited which leads to a histogram containing only a low number of lines.

To tweak the quality of the jitter histogram it is possible to use a larger pattern e.g. by repeating the shorter pattern multiple times in the pattern file or using an odd number for the jitter frequency.

Random Jitter (RJ)

The given pattern will be generated with the selected random jitter parameters. Definable are thereby a low pass and high pass frequency which limits the contained jitter components.

Measuring Periodic Jitter

As the waveform is completely pre-calculated it is not possible to generate truly random jitter with this approach.

Similar to the periodic jitter case it is also true for random jitter that short patterns will not lead to a realistic looking jitter histogram. If a better jitter quality is desired it is therefore necessary to use a longer pattern. In case the actual pattern needs to be short it is possible to have it repeat multiple times in the pattern file and thereby obtain different edge deviations on a single symbol.

5 Setting up Generator

Overview

The M8020A/M8030A generator generates an output signal based on a data pattern. It has the following possibilities for generating an output signal:

- Providing a wide range of clock frequencies

  You can use the generator's internal clock or an external clock for defining the frequency of the outgoing stream.
- Distorting the signal by adding jitter

  You can connect an external delay control device (for example a function generator) to add jitter to the generated signal.
- Adding errors to the output stream

  The M8020A/M8030A can be set up to insert errors into the outgoing stream either internally, according to an external signal, or manually (from the operator).
- Run-time switching between two patterns

  You can set up two patterns and switch between them during runtime either automatically, according to an external signal, or manually.
- Suppressing the output stream

  The output signal can be suppressed according to an external signal.

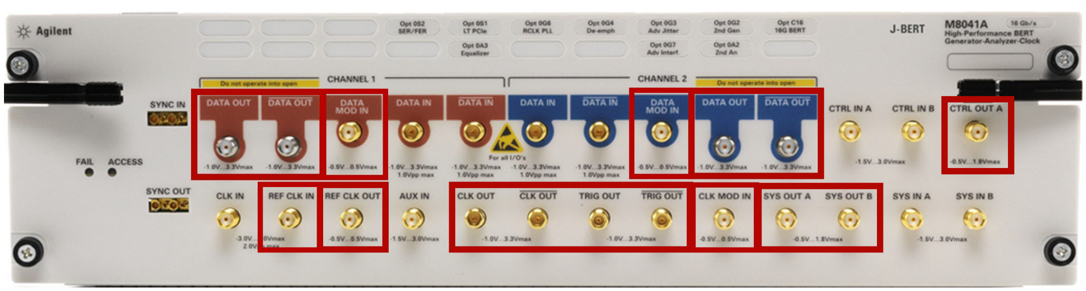
The M8020A/M8030A generator also provides output ports that let you connect an external instrument, such as an oscilloscope.

M8020A/M8030A Generator Ports

The M8020A/M8030A generator ports are used to set the generator's clock frequency and to define the output signal with respect to jitter, error insertion and signal output. In addition, the generator's output ports are also used to supply a clock signal and trigger (for example, analyzer), and an arbitrary data signal for testing your device.

The M8020A/M8030A provides a high level of integration with built-in clock synthesizer, de-emphasis, jitter and level interference. It allows you to set the parameters for the built-in generator parameters such as amplifier, de-emphasis, HF and LF jitter, common mode interference and differential mode interference.

The following figure shows the M8020A/M8030A generator's ports.



The M8020A/M8030A generator's ports include the following:

- Ref Clock In

  The Ref Clk In input can be used as reference frequency or as external system frequency directly for the instrument. An external provided signal can be measured at that input. This input is tightly involved in the system frequency generation.

- Data Out and $\overline{\text{Data Out}}$

  The differential data output serve as device stimuli and can be set up so that they are compatible with a variety of logic families. With respect to Data Out, $\overline{\text{Data Out}}$ has inverted logic.

- Clock Out and $\overline{\text{Clock Out}}$

  The differential clock output serve as frequency (bit rate) reference and can be set up so that they are compatible with a variety of logic families. With respect to Clock Out, $\overline{\text{Clock Out}}$ has inverted logic.

- Trigger Out and $\overline{\text{Trigger Out}}$

  This port allows you to trigger another device (for example, an oscilloscope) and can be set up so that they are compatible with a variety of logic families. Trigger Out has more modes, e.g. sub rate clock to be used as ref clock for a DUT. With respect to Trigger Out, $\overline{\text{Trigger Out}}$ has inverted logic.
- Sys Out A/B

  The system level control outputs used to trigger events to the DUT or external instruments.
- Ctrl Out A

  The control output port provides the Error Output functionality.

The complementary outputs can be used when:

- additional output capability is needed for an instrument such as an oscilloscope or digital communication analyzer.
- your device requires differential inputs.

NOTE **The Generator's Data Out, Clock Out, Trigger Out and ports must be terminated with 50 Ω if they are not connected.**

Understanding the Output Protection Circuit

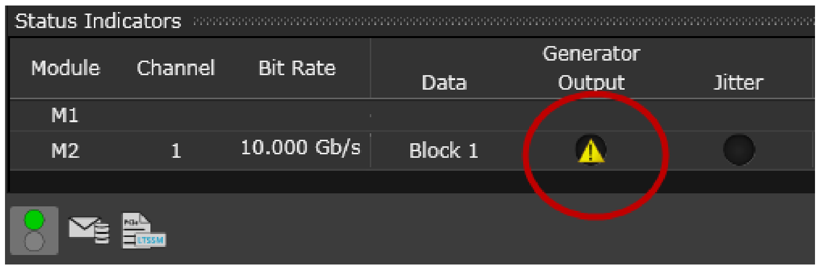
The M8041A and M8051A J-BERT modules offers a huge flexibility for external termination schemes and external termination voltages to address common technologies. For details, please refer to the *M8020A Data Sheet*.

An internal protection circuit continuously monitors the voltages of clock, data and trigger output. It becomes active and turns off the output(s) if the externally applied termination voltage does not match the respective setting (any longer).

NOTE **The M8020A/M8030A output port groups (Data, Clock and Trigger Out) have their own protection circuit. As a consequence, if an output voltage violation occurs at any of the output port(s), then the respective output port group gets disabled or stops working. The output(s) needs to be actively (via GUI or remotely) re-enabled after the fault condition has been removed.**

If an output voltage violation occurs, the output(s) is switched off, which means to a "high impedance" condition. When it gets switched on again (manually by the user) it again follows the standard enabling procedure with termination voltage(s) & impedance checks.

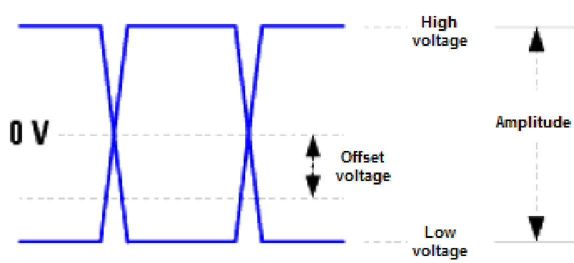
A warning icon appears in the **Status Window** as shown in the following figure:



Understanding the Output Level Parameters

The following figure shows the parameters of a Data, Clock, or Trigger output signal.

As shown in this figure, the signal output levels have the following components:



- High voltage is the upper voltage level of the signal.
- Low voltage is the lower voltage level of the signal.
- Offset voltage is the offset of the average voltage level from 0 V.
- Amplitude of the signal.

When adjusting the output levels, it is important to understand the concept of how the M8020A/M8030A handles voltages.

Changing the Amplitude

Typically, during tests, when you adjust the amplitude, you want to keep the offset constant. This keeps the ideal sampling point within the eye. The M8020A/M8030A handles this by keeping offset voltage constant when amplitude is changed.

Changing the Output Levels

On the other hand, you may want to adjust the output voltage level without changing the amplitude. The M8020A/M8030A handles this by keeping amplitude constant when offset voltage is changed and high/low voltage are accordingly adjusted.

Voltage Level Restrictions

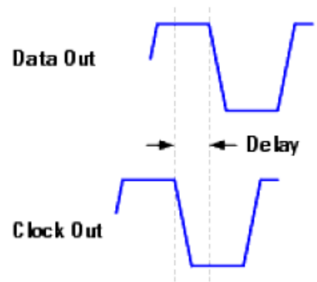
The M8020A/M8030A cannot generate a signal that has voltage levels out of range. If you try to enter a value for one parameter that would put another parameter out of limits, the M8020A/M8030A rejects the change. This could happen, for example, if low voltage is already at the minimum, and you try to lower either of high voltage or low voltage, or increase amplitude.

Understanding Delay and Crossover

The M8020A/M8030A provides the possibility of modifying the output data signal by varying the signal's delay to the clock signal, and the signal's crossover.

Delay

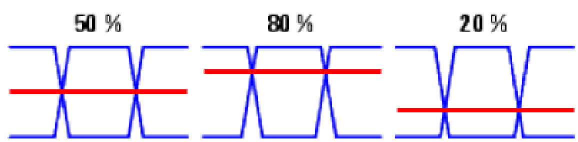
The exact time delay through a test setup can vary. The delay function allows you to compensate for this by adjusting the active edge of the trigger output relative to the clock/data outputs. This varies the phase relationship of the data and clock outputs (causes the data to have a certain time delay after the clock pulse). The higher the delay, the greater the time difference between the clock signal and the data signal. The delay can be adjusted by the generator’s Data Out 1 and Data Out 2 ports.



Crossover

Crossover is the voltage level where the overlapped rising and falling edges of the logic levels intersect. This adjustment varies the widths of the logic highs and lows.

The following figure shows examples of crossover at 50 %, 80 %, and 20 %:



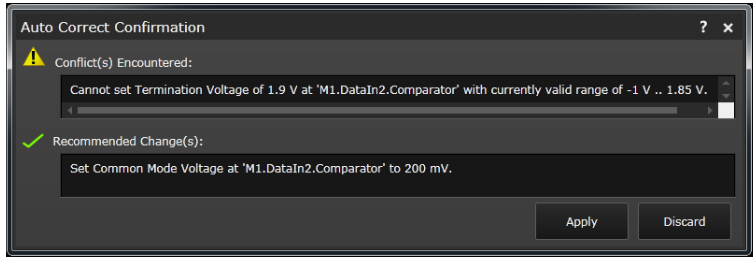
Why Incorrect Terminations Could Damage Your Device

Choosing wrong terminations may cause your device to output voltage levels that are not as expected. It may also cause excessive current or current flow in the wrong direction, which can damage your device.

Note that an internal protection circuit becomes active if the termination voltage is wrongly adjusted. The protection circuit sets the output voltages to safe levels, typically:

Vhi = Vlo = Vterm = externally measured termination voltage.

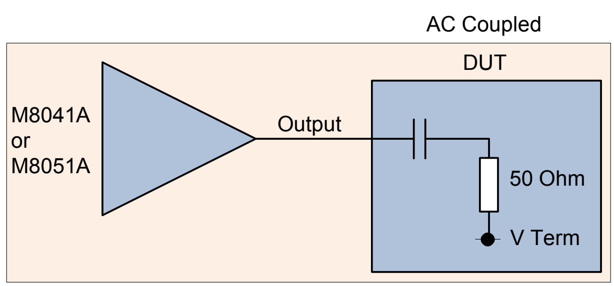
If you adjust the termination voltage, and try to enter value(s) which are outside of the currently allowed window, the **Auto Correction Confirmation** message box will pop up with respective apply and discard options as shown in the following figure:



AC Coupling and Bias Tees

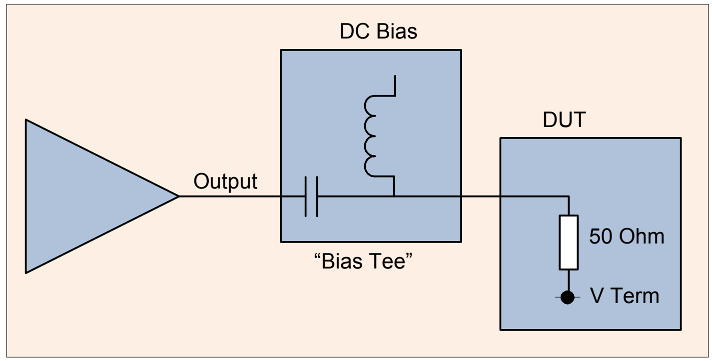
The generator's outputs are always internally DC-coupled; even when AC termination is selected. For this reason, caution must be taken when connecting your instrument to a device or test setup.

The diagram below shows a device that is AC coupled. Note that the capacitor is part of the test setup.

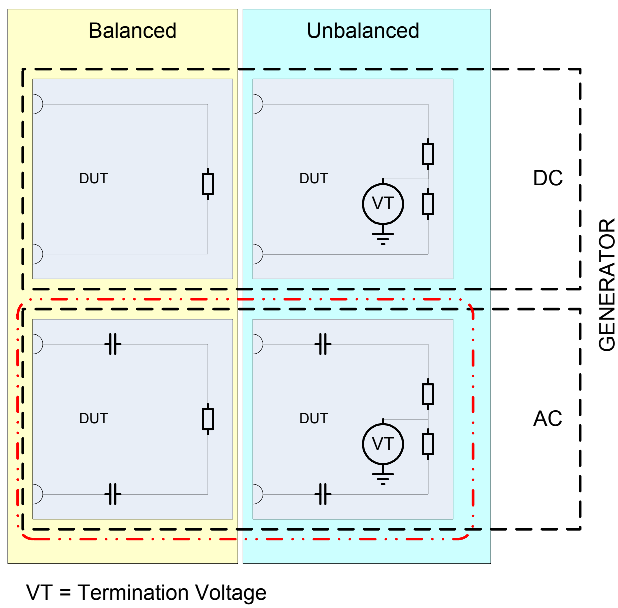


You can use an external bias to power your device. You must ensure, however, that the network is oriented correctly. If it is not, your device or instrument may get damaged.

The following diagram shows a bias tee that is positioned correctly. Notice that the generator's outputs are protected by the blocking capacitor.



The following drawing is an example of the external DUT connected to the generator output for the different possible settings.



For AC-coupled mode (see red box above), the instrument actually cannot and does not need to distinguish between these two termination schemes. Therefore the drop-down box **Termination Model** is not shown and active within the GUI when the AC coupled mode is selected.

There are certain impedance/voltage limits you need to keep in order to be able to turn on (enable) the output(s).

Whenever an output amplifier is turned on, the external DC resistance as well as the external termination voltage is measured and calculated.

These are the impedance/voltage limits for various modes / configurations:

- **DC Coupling Unbalanced:** The measured externally connected resistance should be within the allowed range which is between 40 to 60 Ω here. Also, if an external termination voltage is detected that is not within the allowed ±100 mV tolerance window, the output(s) will not turn on.
- **DC Coupling Balanced:** The measured externally connected resistance should be within the allowed range which is between 80 to 120 Ω here. In this DC Coupling Balanced mode there is an exception implemented, that allows the output(s) to drive high impedance ('into open'). For this, the set maximum amplitude should be below 300 mv, and the offset setting should be between 0-370 mv.
- **AC Coupling:** You can use the AC coupled mode as long as you apply an external DC blocking capacitor. Here the external resistance should be equal or greater than 300 Ω.

Setting up Terminations

Before you can start sending signals to your device, you have to choose the proper termination mode. To do so:

1. Go to the **Menu Bar** > **Generator** and then select **Data Out**.
2. Select **Amplifier** functional block from the **Parameters** window.
3. Provide the DUT's termination settings.
4. Provide termination voltage.

CAUTION Selecting the wrong termination may damage your device.

5. Connect the DUT's input ports to the M8020A/M8030A's output ports.

CAUTION

Do not apply external voltages to the generator outputs.

Output ports of the generator that are not connected to another device must be terminated with 50 Ω to prevent the M8020A/M8030A from damage.

If outputs are disconnected they are usually not turned on because the output enable check detects the misconfiguration. If outputs are disconnected after they have been turned on the protection circuit might get triggered depending on the specific configuration. In this case the output will be switched off. This is an emergency scenario and should never be a normal habit. It can be switched on again (manually by the user) after removing the fault condition, it again follows the standard enabling procedure with termination voltage(s) & impedance checks.

Adjust Output Levels (optional)

Data, Clock and Trigger Out offset and voltage levels can be adjusted. This is typically done when you want to tune your BER measurement or stress the device.

You can adjust the related parameters of the data and clock amplitudes and offsets on the GUI.

To enter specific values for the outputs from the keyboard:

1 Go to the **Menu Bar** > **Generator** and then select **Data Out** and **Trigger Out** ports. Once more, go to the **Menu Bar** > **Generator** and then select **Clock Out** port.
2 Select **Amplifier** functional block from the **Parameters** window.
3 Select coupling as **DC.**
4 Select termination model as **Unbalanced**.
5 Enter the desired termination voltage.

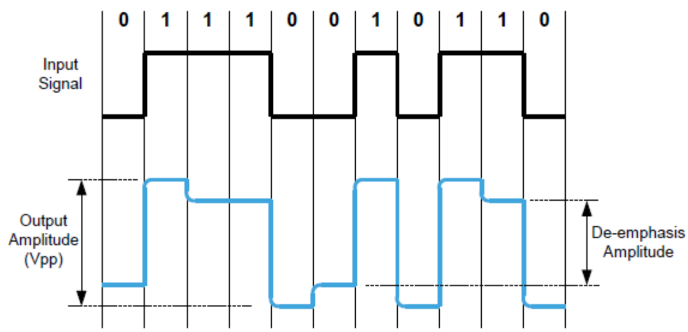
NOTE

If the standard enabling procedure with termination voltage(s) and impedance checks fails the output(s) will stay off. In this state the output resistance is "high impedance". It can be switched on again (manually by the user) after removing the fault condition, it again follows the standard enabling procedure with termination voltage(s) and impedance checks.

De-Emphasis Signal Generator

The M8020A/M8030A has a built-in de-emphasis signal generator that can be connected between the Data Out port of the generator and the DUT.

De-emphasis is a method that reduces the voltage of a digital signal if the generated level is high or low for more than one clock period. The principle is illustrated in the following figure.



The de-emphasis amplitude is specified as a fraction of the output amplitude (in percent or dB).

Post-cursor de-emphasis

The figure above refers to a so-called post-cursor de-emphasis. You may wish to know how that is generated.

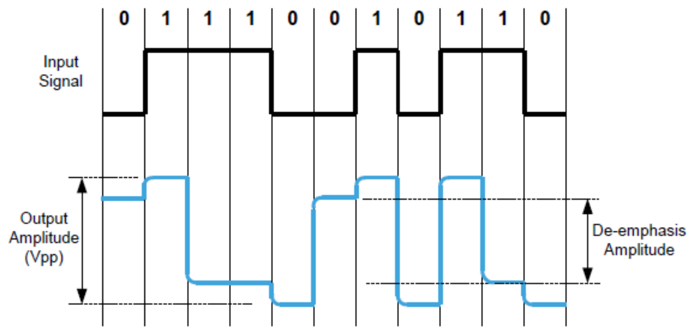
- One branch has a programmable amplifier to produce the desired output voltage (Peak to Peak Voltage).
- The other one has an adjustable delay (automatically set to one signal clock period) and a programmable inverting attenuator/amplifier to produce the delayed signal with a lower voltage swing.

Finally, the signals of both branches are added. This means, the delayed signal voltage is subtracted from the specified peak-to-peak amplitude.

Pre-cursor de-emphasis

It is also possible to convert the input signal to a pre-cursor deemphasized signal.

This can be done by setting the output voltage swing to the desired deemphasis amplitude and specifying a negative amplitude ratio (an amplification). This inverts the roles of the two branches. The delayed signal has now a larger amplitude than the direct signal. A waveform example is illustrated in the following figure.



When pre-cursor de-emphasis is generated this way, the complementary Output of the De-Emphasis Signal Converter becomes the normal output and vice versa.

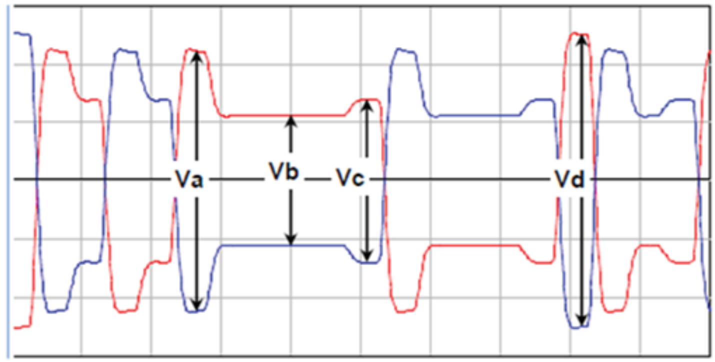
CAUTION

Be very careful if you set the de-emphasis ratio to amplification! In this case, there is no indication of the peak-to-peak voltage applied to the DUT.

You need to calculate or measure the output signal voltage precisely. Otherwise you might damage your device.

De-emphasis on 2 pre-cursor and 5 post-cursors

The following example illustrates the output of the differential signal with variable de-emphasis on 2 pre-cursor and 5 post-cursors generated by M8020A/M8030A De-Emphasis.



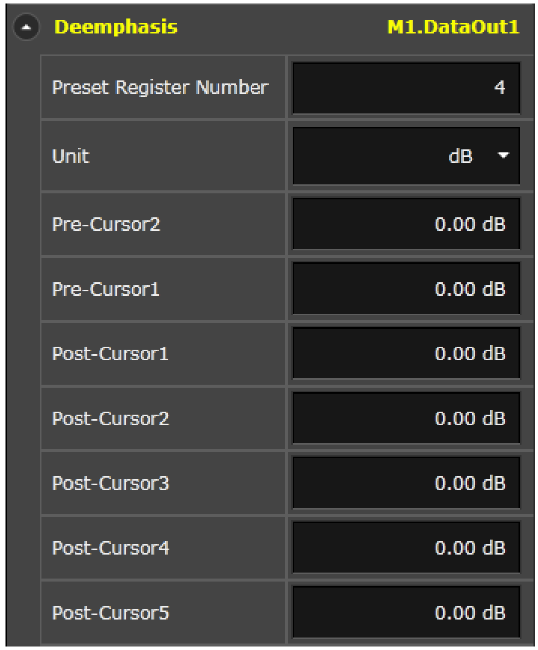
Controlling the De-Emphasis

To control the de-emphasis function:

1 Go to the **Menu Bar** > **Generator** and then select **Data Out**.

2 Select **De-Emphasis** function block from the **Parameters** window.

For M8020A/M8030A, you have two (pre–cursor) and five (post–cursor) and the corresponding unit.



Set the values as following:

- Pre-Cursor2: 0 dB to 6.02 dB
- Pre-Cursor1: 0 dB to 12.04 dB
- Post-Cursor1: 0 dB to 20 dB
- Post-Cursor2: 0 dB to 12.04 dB
- Post-Cursor3: 0 dB to 12.04 dB
- Post-Cursor4: 0 dB to 6.02 dB
- Post-Cursor5: 0 dB to 6.02 dB

You can toggle between dB (decibel) and % (percent). Toggling does not change the value. The **De-Emphasis** function block also allows you to set the “Positive” or “Negative” polarity for de-emphasis.

Setting Up Data Out Port Parameters

The generator produces clock and data outputs that serve as frequency reference and device stimulus for the device under test.

To set the **Data Out** parameters:

1. Go to the **Menu Bar** > **Generator** and then select **Data Out** port.
2. Select **Amplifier** functional block from the **Parameters** window.
3. Set the parameters as described in this section.

Amplitude

This text field allows manual entry of the voltage amplitude and displays the current value.

To modify the value, click inside the text field and enter the desired value.

NOTE

The M8020A/M8030A will not allow you to adjust a voltage beyond its limits. The limit is determined by the M8020A/M8030A's internal hardware. If a limit is encountered, the M8020A/M8030A sends a message to the System Error. For more information about voltage limits, see the M8020A Data Sheet.

NOTE

If you may want to adjust the output voltage level without changing the Amplitude, the J-BERT M8020A handles this by keeping Amplitude constant when Offset voltage is changed and High and Low voltages are accordingly adjusted.

Similarly, if you want to adjust the output voltage level without changing the Offset, the J-BERT M8020A handles this by keeping Offset constant when Amplitude is changed and High and Low voltages are accordingly adjusted.

You can even click **Auto Range** option which will automatically select the amplitude range according to amplitude.

High Voltage

This text field allows manual entry of the logic high voltage level and displays the current value.

To modify the value, click inside the text field and enter the desired value.

Offset Voltage

This text field allows manual entry of the voltage level halfway between logic high and logic low (the offset) and displays the current value.

To modify the value, click inside the text field and enter the desired value.

Low Voltage

This text field allows manual entry of the logic low voltage level and displays the current value.

To modify the value, click inside the text field and enter the desired value.

Delay

This text field allows manual entry of Data and Aux Data output delay and displays the current value in picoseconds.

Crossover

This text field allows manual entry of the data's crossover percentage, and displays the current value.

Clk/2 Jitter

This text field allows manual entry of the Clk/2 Jitter at the Data Out port in units of seconds. It provides half rate clocking; the clock at the clock output runs at half the bit rate.

Data Polarity Inverted

Use this option to invert the logic of the data outputs (Data Out, Trigger Out and Clock Out).

Transition Time

Use this option to control the transition time of the output signal. The choices are smooth, moderate and steep.

Bit Rate

Bit rate is defined as bits per second. The Generator's clock rate sets the bit rate and serves as the frequency reference for the Data, Clock and Trigger outputs, your device, and the Analyzer (if it receives its clock from the Generator). It can be generated internally or supplied from an external source.

When to Use an External Clock Source?

The M8020A/M8030A's internal clock can be used for most testing purposes.

There are some circumstances, however, when an external clock source is required:

- Synchronization with an external clock

  The M8020A/M8030A can be connected to an external clock to allow it to run as part of a larger external system.

- Use of a modulated clock

  A frequency- or delay-modulated clock can be used to provide a small amount of jitter to the clock signal.

- Use of a precision clock

  A precision clock with very low phase noise can be used to enhance the instrument's performance. This is especially interesting for longterm measurements.

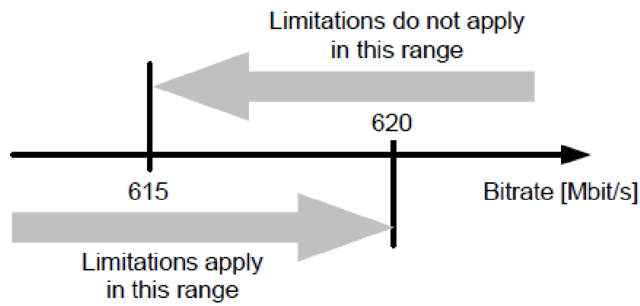
Bit Rate Range

The M8020A/M8030A provides bit rates from 256 Mbit/s up to 16.2 Gbit/s, depending on the instrument's options.

However, several specific properties and limitations need to be taken into account when working at low bit rates. The limitations apply to the instrument according to the following hysteresis curve:

- If the bit rate falls below 615 Mbit/s, the limitations apply.
- If the bit rate exceeds 620 Mbit/s, the limitations no longer apply.

The following figure clarifies the behavior in the range between 615 Mbit/s and 620 Mbit/s:



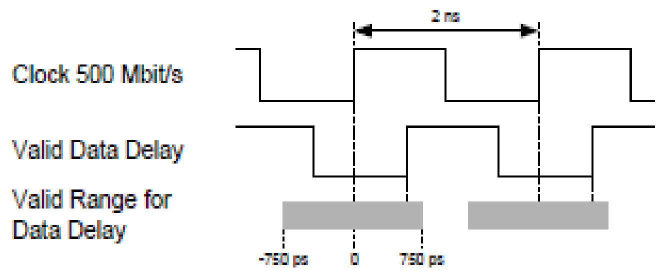
For the generator the following rules apply:

- Below 620 Mbit/s, the Generator can only be operated with an external clock source, because the internal clock source can only produce signals higher than 620 Mbit/s.
- The trigger output cannot be set up to trigger on certain pattern positions or pattern sequences. If this option is enabled (for example in the user interface), the trigger is sent once for each pattern, but the bit position cannot be specified.

  The option to trigger on the divided clock signal is supported as usual.
- There are restrictions to the available clock to data delay values.

  The Generator can vary the clock to data delay only within a range of 10 ns (relative to the clock signal). For frequencies above 666 Mbit/s, this range is sufficient to cover the complete clock cycle (= 1 unit interval).

  For lower frequencies, the valid data delay range is smaller than the clock cycle. The Generator cannot generate signals with a delay outside this range. Therefore, the data delay cannot be set to all values within the clock cycle.

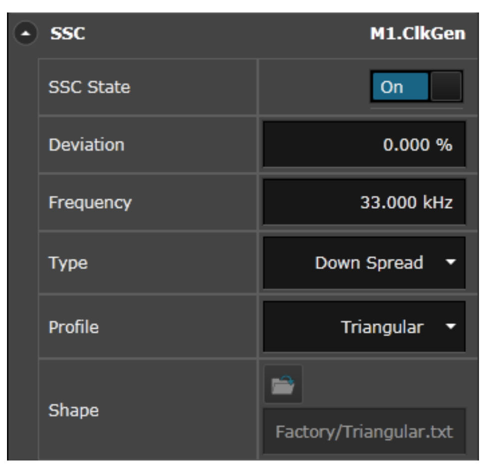


Spread Spectrum Clocking

A **Spread Spectrum Clock** (SSC) is widely used for reducing the peak electromagnetic radiation at the nominal clock frequency. With SSC, the clock pulse is modulated with a relatively low-frequency triangle waveform. This broadens the clock signal spectrum and reduces the peak energy. The M8020A/M8030A allows to emulate such SSC on the data, trigger and clock output signals.

The M8020A/M8030A has a built-in SSC generator.

For the **SSC** settings, go to **Menu Bar** > **Clock** > **Clock Generator** and then select **SSC** function block from the **Parameter** window.



Setting the Bit Rate

You can use an external clock source or the M8020A/M8030A's internal clock to control the bit rate.

To set the bit rate:

1 Go to **Menu Bar** > **Clock** and then **Clock Generator**.

2 Select **Synthesizer** function block from the **Parameter** window.

3 Select the appropriate clock source:

- Internal

  This setting uses the internal clock oscillator.

- Reference

  This locks the clock generator to an external 10/100 MHz reference.

- Direct

  This connects the external clock directly to the clock generator at 8.1 GHz to 16.20 GHz.

- Clock Multiplier

  This enables error counting and error analysis of devices using half-rate clocking.

4 For Direct clock source, click Execute button to measure the incoming clock’s clock rate.

5 Select a clock rate:

- For Internal Clock Source, you can enter a clock rate in the Value and Units field.
- For 10/100 MHz Ref Clock Source, the clock connected to the 10/100 MHz Ref In port must be 10/100 MHz.

The selected clock rate applies to the generator. This is also the clock rate generated at the generator's Clock Out port.

The analyzer internally receives its clock from the generator, it runs at the same clock rate.

External Clock Divider

The internal clock (i.e. the bitrate) is the external clock divided by the value specified in the clock divider field. The external clock divider field is available if you have close an external clock source.

External PLL Clock Divider and Multiplier

The internal clock (i.e. the bitrate) is the external PLL divided by the value specified in the divider field and multiplied with the value specified in the multiplier field. The external PLL clock divider and multiplier field are available if you have chosen a clock source.

Clock Rate Indicators

The Bit Rate indicators are shown in the **Status Indicators** display their current bit rate.

The analyzer bit rate is measured from the incoming clock signal or derived from the data signal.

Trigger Output

The generator's **Trigger Out** port can be used to send a trigger to external devices like an oscilloscope or digital communication analyzer.

In pattern mode, the generator sends a trigger signal that is at least 32 bits long.

In sequence mode, the generator can send a trigger signal whenever a block of the sequence starts or restarts.

Several options are available for the trigger signal. As an example, you can send the trigger as a divided clock signal or as an indicator when the data pattern starts.

Setting Up Trigger Output

To set up the generator's **Trigger Output** port:

1 Connect the external instrument to the **Trigger Out** port.
2 Go to **Menu Bar** > **Generator** > and then select **Trigger Out**.
3 Provide the necessary configuration.

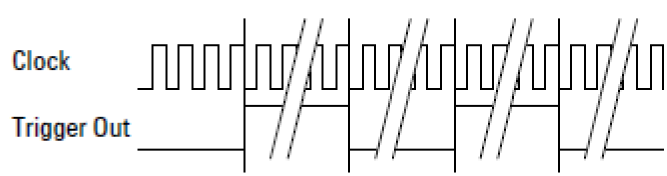
You can always generate a divided clock signal at the **Trigger Out** port.

The alternate trigger signals refer to patterns and are not generated in sequence mode.

To support the generation of a trigger spike at the beginning of a sequence block, the **Trigger Output** can be put into **Sequence** mode.

Clock Divided by n

Select this option to send a trigger signal from the **Trigger/Ref Clock Out** port at every nth clock pulse. Note that the trigger signal itself consists of n/2 bits high followed by n/2 bits low. For example, Clock divided by 8 works as shown below.



NOTE

If the Divider Factor n is uneven (For example 3), the clock's duty cycle will not be 50%, but the signal will stay high for (n+1)/2 and low for (n-1)/2. This results in a Duty Cycle Distortion (DCD) of 0.5 UI.

Sequence Trigger

This function becomes available after a user-defined sequence has been downloaded to the generator.

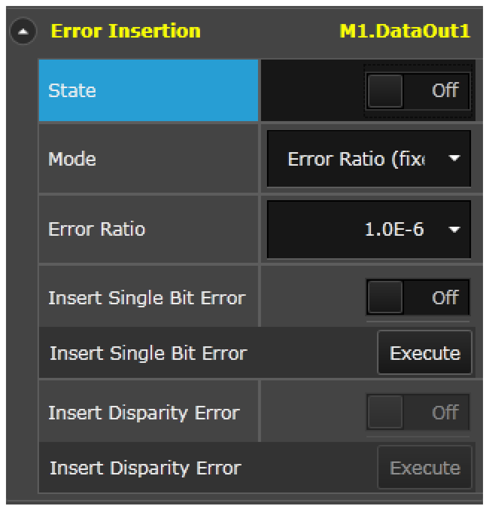
Click this function to switch the **Trigger Out** port to **Sequence** mode.

In Sequence mode, the **Trigger Out** can generate a spike whenever the execution of a block starts or restarts. Whether that happens for a particular block or not is defined for each block individually in the sequence expression.

Error Insertion

To test error correction algorithms, alarms and other functions that are embedded in the data pattern, you can insert logic errors (flipped bits) into the pattern.

For the error insertion, go to **Menu Bar** > **Data Out** and select **Error Insertion** function block from the **Parameters** window.



The instrument provides several options for inserting error bits manually or automatically. You have the following options for inserting errors into the output data stream:

Insert Single Bit Stream

To insert single bit stream:

- Click the slide switch to turn on the **Error Insertion** state.
- Select the supported mode from the drop-down list.
- Select the error ratio from the drop-down list.

Insert Single Bit Error

- To manually insert a single bit error into the output stream:
- Click the **Data Out** port.

- Select **Error Insertion** functional block and enable the **Insert Single Bit Error** state.
- Click **Execute**.

NOTE

TIP - To find out how your DUT reacts on very small bit error rates, set up the pattern generator to enter errors once every 10^{-12} bits and run a longer accumulative test.

You can then find the DUT's true error rate by calculating the difference between the bit error rate set up in the Generator and the accumulated bit error rate found by the Analyzer.

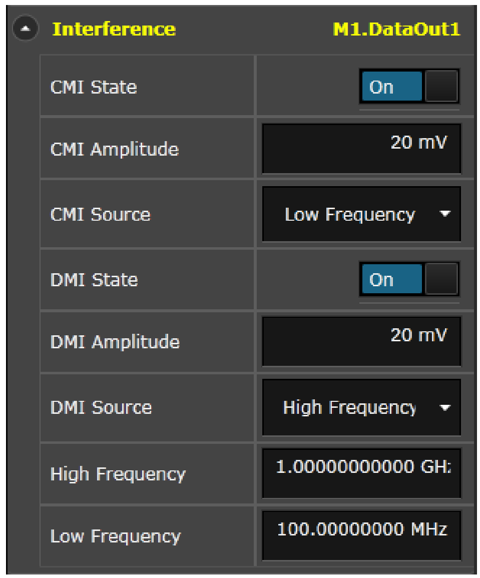
NOTE

If you set up too high an error rate, the Analyzer will not be able to synchronize to the incoming pattern. When setting up an error rate, always make sure that the synchronization threshold is higher than the bit error rate.

Interference

The M8020A/M8030A supports the following two types of interference:

- Common Mode Interference (CMI)
- Differential Mode Interference (DMI)



Common Mode Interference (CMI)

Common Mode Interference (asymmetrical mode) relates to ground (CMI). It has the following characteristics:

- **State**: Enables / disables common mode interference (CMI).
- **Amplitude**: Specifies the amplitude of the common mode interference (CMI).
- **Frequency Source**: Specifies the frequency source for common mode interference (CMI). It can be high frequency source or low frequency source.

Differential Mode Interference (DMI)

Differential Mode Interference (DMI) - Differential Mode Interference (symmetrical mode) is independent of ground (DMI). It has the following characteristics:

- **State**: Enables / disables differential mode interference (DMI).
- **Amplitude**: Specifies the amplitude of the differential mode interference (DMI).
- **Frequency Source**: Specifies the frequency source for differential mode interference (DMI). It can be high frequency source or low frequency source.
- **High Frequency**: Specifies the frequency of the high frequency source.
- **Low Frequency**: Specifies the frequency of the low frequency source.

Jitter Setup

The **Jitter Setup** function is used for composing the total jitter in a defined and calibrated way.

The M8020A/M8030A supports the following types of jitters:

- High Frequency Jitter (HF Jitter)
- Low Frequency Jitter (LF Jitter)

High Frequency Jitter (HF Jitter)

The **High Frequency Jitter** is a composition of the following types of jitters:

- Periodic Jitter 1
- Periodic Jitter 2
- Bounded Uncorrelated Jitter
- Random Jitter
- Spectrally Distributed Random Jitter
- External Jitter Source (Connected to the Delay Ctrl input)

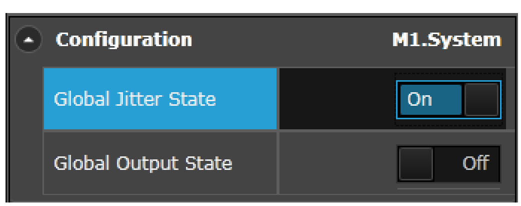
Low Frequency Jitter (LF Jitter)

The **Low Frequency Jitter** is a composition of the following types of jitters:

- Residual Spread Spectrum Clock
- Periodic Jitter

Global Jitter State On/Off Switch

The **Jitter On/Off** switch allows you to globally enable or disable the jitter generation.



How to Enable Global Jitter State

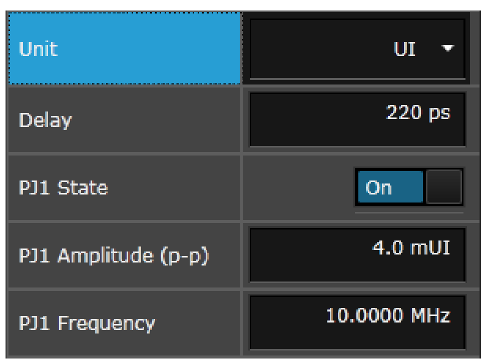
You can enable the global jitter state by pressing the **Jitter** button, present on the status bar of the GUI. When this global jitter is enabled, the button turns green indicating the jitter active state.

Set Jitter Configuration

In this section, an example of **Periodic Jitter 1** is shown to specify the jitter components.

To specify the jitter components:

- **Set Unit**: Defines the unit of jitter amplitude value in either seconds or UI.
- **Set Delay**: Enter a value to define the jitter delay on clock and data.
- **Enable Jitter**: Enable the jitter source (press the corresponding button).
- **Set Parameters**: Set the most commonly used parameter (typically Amplitude and Frequency) directly.

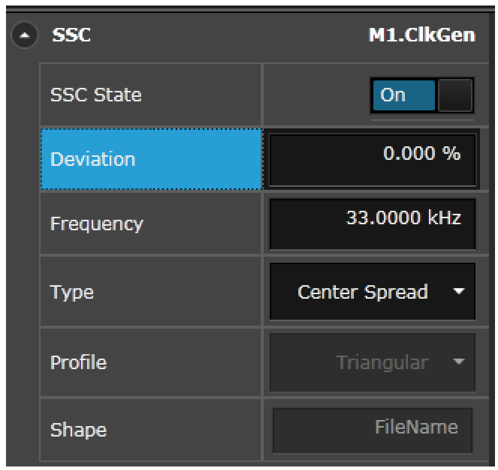


Spread Spectrum Clock

The **Spread Spectrum Clocking** setting controls the generator's spread spectrum (SSC) clocking feature. When the SSC is enabled, it impacts the Data Out, Clock Out, and Trigger Out ports.

The spread spectrum clock is characterized by:

- Deviation
- Frequency
- Type
- Profile
- Shape



- **Deviation**

  If the deviation type **Center Spread** is selected, the deviation can be changed in two different ways:

  - **Change the Center Spread value**: the bit rate remains unchanged, while the upper and lower frequency changes according to the selected deviation. The deviation value specifies ½ p-p value.
  - **Change the Down Spread value**: the upper frequency remains unchanged, while the bit rate is adjusted. The deviation value specifies the p-p value.

  For some setups, the I/Q modulator's range limitations require to use Center Spread instead of Down Spread and adjust the bit rate (and deviation) accordingly.

  If the upper or lower frequency is change, the bit rate will be adjusted according to the selected deviation. The center frequency corresponds to the configured bit rate.

Please note that both the upper and lower frequency and the frequency adjustment of all three frequencies are not reflected in the firmware. The GUI just calculates the resulting bit rate and writes it to the firmware.

The **Deviation** of the clock rate. The ranges of SSC deviation are as follows:

- Down Spread with deviation of 0% ... 1%
- Center Spread with deviation of 0% ... 1%
- Up Spread with deviation of 0% ... 1%

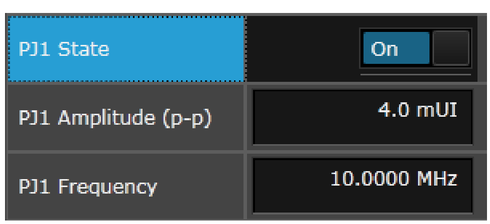
NOTE The deviation in Upspread and Downspread is the p-p value while in Centerspread it is 1/2 p-p.

- **Frequency**: The deviation Frequency. The SCC provides the frequency range of 100 Hz – 200 kHz.
- **Type**: You can click the drop-down list to choose among Down Spread, Center Spread and Up Spread deviation type.
- **Profile**: Controls the profile of the spread spectrum clocking. You can use the drop-down list to choose between the Triangular and Arbitrary profile.
- **Shape**: For Arbitrary profile, you need to specify the Arbitary Waveform file. It is a simple text file that contains the data points which define the arbitrary waveform of the SSC profile.

Periodic Jitter 1

The **Periodic Jitter 1** is characterized by:

- Amplitude
- Frequency

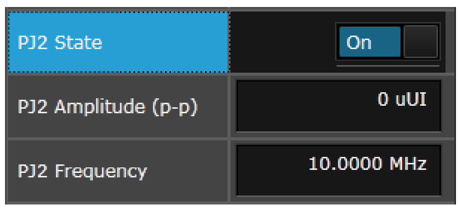


- **Amplitude**: Controls the amplitude of periodic jitter 1.
- **Frequency**: Controls the frequency of the periodic jitter 1.

Periodic Jitter 2

The **Periodic Jitter 2** is characterized by:

- Amplitude
- Frequency



- **Amplitude**: Controls the amplitude of periodic jitter 2.
- **Frequency**: Controls the frequency of the periodic jitter 2.

Specify the Components for Jitter Sweep

The **Periodic Jitter 2** has the following modes:

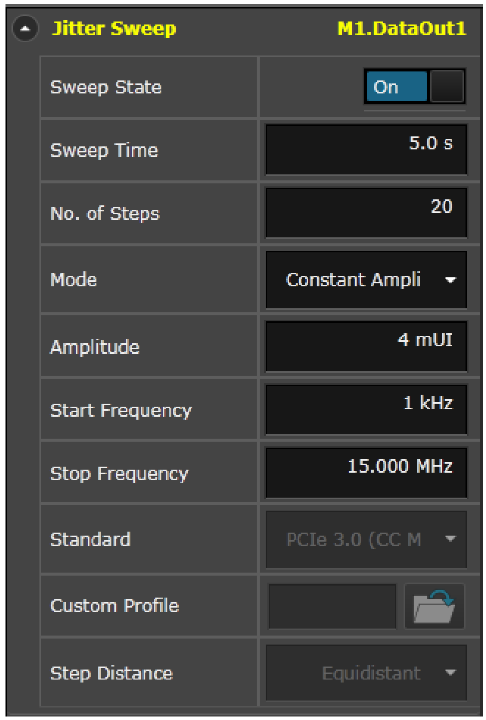
- Constant Amplitude Sweep
- Variable Amplitude Sweep

To specify the components of jitter sweep select **Jitter Sweep** functional block from the **Parameter** window and then select the **Mode** from the drop-down list to specify components the jitter sweep.

Periodic Jitter 2 – Constant Amplitude Sweep Parameters

The **Periodic Jitter 2 – Constant Amplitude Sweep** is characterized by:

- Amplitude
- Frequency Range
- Sweep Time
- No. of Steps



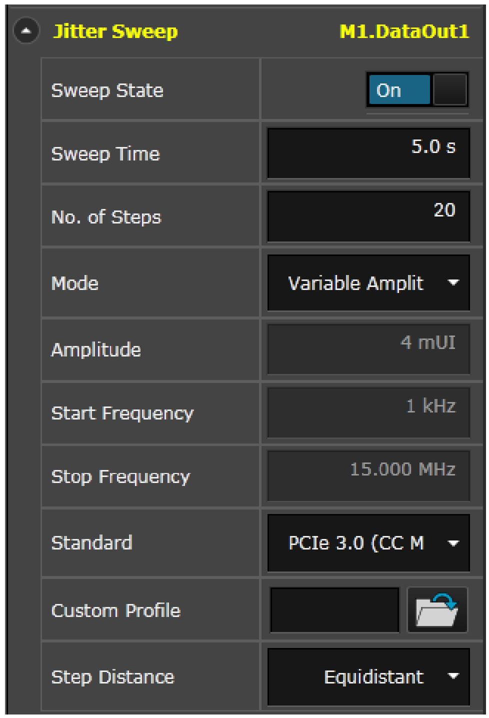
- **Amplitude**: The maximum peak-to-peak Amplitude is limited by the free capacity of the chosen delay line.
- **Frequency Range**: The stop frequency has to be higher than start frequency and the range should be in accordance with the selected waveform.
- **Sweep Time**: You can specify the duration for sweeping the specified frequency range once.
- **No. of Steps**: You can specify the number of steps to fulfill a complete sweep. The start and stop values are included. The valid range is between 2 to 100.

Periodic Jitter 2 – Variable Amplitude Sweep Parameters

The **Periodic Jitter 2 – Variable Amplitude Sweep** is characterized by:

- Standard

- Sweep Time
- No. of Steps
- Step Distance



- **Standard**: You can use this drop-down list to specify whether you want to select a pre-defined standard or a user-defined standard. All the available predefined standards will be shown in this list. However, if you select the user-defined standard, press **Custom Profile** button to locate the **Jitter Tolerance** standard. The user-defined standard uses the same file format like the **Jitter Tolerance Compliance** measurement.

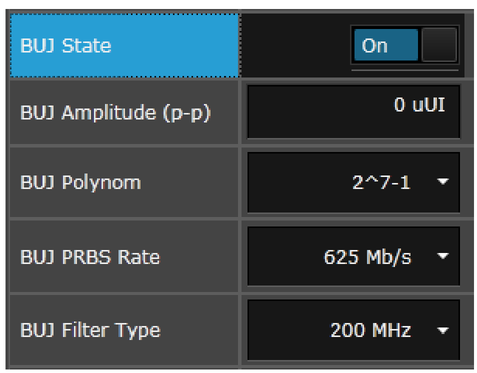
NOTE Selecting the Standard does allocate the required amount of jitter modulation on the delay line being used. To avoid errors when changing the selection, it is recomended to either select the corresponding bit rate first, or enable the PJ2 source after setting the correct bit rate.

- **Sweep Time**: You can specify the duration for sweeping the selected jitter profile/standard once.
- **No. of Steps**: You can specify the number of steps to fulfill a complete sweep. The start and stop values are included. The valid range is between 2 to 100.
- **Step Distance**: You can use this drop-down list to specify whether the frequency steps are log equidistant (EQUidistant) along the periodic jitter curve or a frequency step matches a corner frequency on the periodic jitter curve.

Bounded Uncorrelated Jitter

Bounded Uncorrelated Jitter is characterized by

- Amplitude
- PRBS polynomial
- Data rate
- Filter type

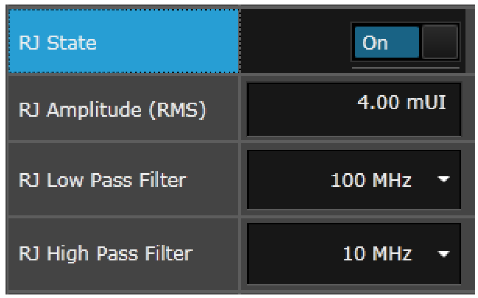


- **Amplitude**: Controls the amplitude of the BUJ jitter source.
- **PRBS polynomial**: The PRBS polynomial can be chosen from a list. Available are eight polynomials, from 2^7-1 up to 2^31-1.
- **Data Rate**: Enter an appropriate **Data Rate**.
- **Filter Type**: The bounded uncorrelated jitter source is equipped with three low-pass filters with cut-off frequencies at 50, 100, and 200 MHz. One of these filters is always active.

Random Jitter

Random Jitter is characterized by:

- Amplitude
- Filter Settings



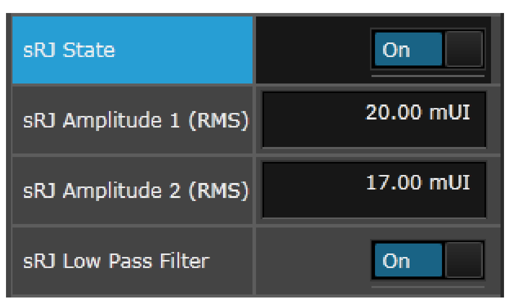
- **Amplitude**: The random jitter Amplitude must be entered as an rms (root mean square) value.
- **Filter**: The random jitter source is equipped with a 10 MHz high pass and a 100 MHz and 500 MHz low pass filter to limit the spectral range.

Spectrally Distributed Random Jitter

Spectrally Distributed Random Jitter is composed of two jitter sources: low frequency jitter and high frequency jitter. It is characterized by the amplitudes of the low and the high frequency jitter.

Spectral Distributed Random Jitter is characterized by

- Amplitude LF
- Amplitude HF



It has the following parameters:

- **sRJ State**: Click this switch to enable the spectrally distributed random jitter.
- **SRJ Amplitude 1 (RMS) LF**: This is the low frequency jitter amplitude as rms value.
- **SRJ Amplitude 2 (RMS) HF**: This is the high frequency jitter amplitude as rms value.
- **sRJ Low Pass Filter**: The spectrally distributed random jitter is equipped with a 100 MHz low pass filter to limit the spectral range, which can be enabled by pressing the corresponding switch.

External Jitter Source

To enable the external jitter source:

- From the **Data Out** port select **HF Jitter** functional block in the **Parameters** window.
- Click the **External** switch to enable external jitter.

NOTE Even if the external jitter source is enabled, you can still add or change internal jitter components.

Keysight M8070A System Software for M8000 Series of BER Test Solutions

User Guide

6 Setting up Analyzer

Overview

The M8020A/M8030A Analyzer examines an incoming bit stream, compares it to the expected pattern, and locates any inconsistencies. The Analyzer requires the following settings to work correctly:

- The expected pattern

  The Analyzer needs to "know" which data to expect so that it can detect bit errors. No expected patterns are required in BRM mode.
- Correct clock frequency

  Required to recognize the bit rate in the data stream.
- Appropriate sampling point

  The sampling point defines where the Analyzer tries to differentiate between 0s and 1s in the data stream. This is necessary so that the Analyzer recognizes the data bits correctly.
- Synchronization to the incoming pattern

  The expected pattern must be synchronized to the incoming pattern so that the Analyzer can find any discrepancies.

The Analyzer provides the following functions to enable you to perform tests:

- Automatic pattern synchronization

  The Analyzer shifts the incoming data stream bitwise to match it to the expected data pattern. A correct BER can only be measured with matching patterns.
- Error accumulation

  You can specify whether a test runs for a specified time or until a specific number of errors has occurred. This lets you carry out longer tests while logging the results to a file.
- BER location mode

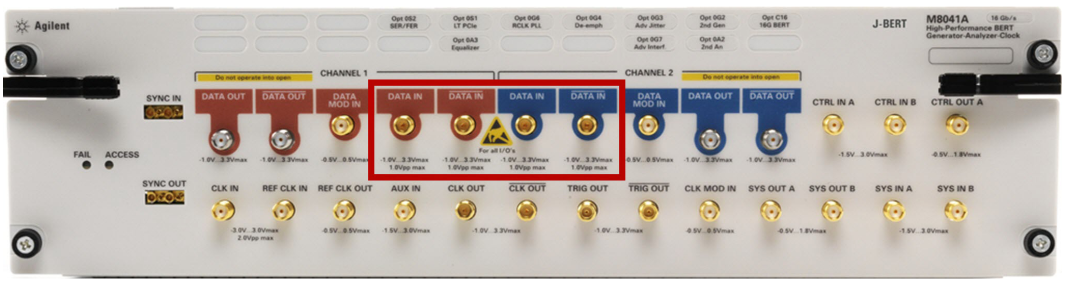
  You can specify whether all errors are counted or only the errors that occurred on a particular bit position or range of positions.
- Trigger output for external measurement instruments

  This allows you to connect other devices for further error analysis.

M8020A/M8030A Analyzer Ports

The M8020A/M8030A analyzer's ports are used for running tests and for connecting external equipment.

The following figure shows the J-BERT M8041A analyzer's ports.



The M8041A analyzer's ports include the following:

- Data In and $\overline{\text{Data In}}$

  This port is connected to the data signal and the inverted data signal.

Data In Port Termination

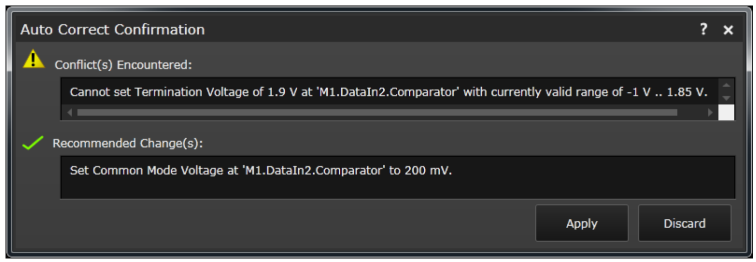
To ensure a valid setup and to protect the devices from damage, proper termination must be specified for both **Data In** connections. You can specify the termination by entering the termination voltage in the respective field.

CAUTION **Selecting the wrong termination may damage your device.**

Why Can Wrong Terminations Damage Your Device?

Choosing wrong terminations may cause your device to output voltage levels that are not as expected. It may also cause excessive current or current flow in the wrong direction, which can damage your device.

If you adjust the termination voltage, and try to enter value(s) which are outside of the currently allowed window, the Auto Correction Confirmation message box will pop up with respective apply and discard options as shown in the following figure:



Setting Up Termination

To select the termination for the Analyzer:

NOTE You must know the termination voltage of the data signal that your DUT sends to the Analyzer.

1 Go to the **Menu Bar** > **Analyzer** and then select **Data In**.
2 Select **Comparator** functional block from the **Parameters** window.
3 In the **Termination Voltage** field, enter the termination voltage that is appropriate for the incoming data signal.

CAUTION Selecting the wrong termination may damage your device.

4 In the **Polarity** field, select **Inverted** if your device inverts data.

You can now physically connect the DUT to the Analyzer.

Compare Mode

The selection in this list defines how the signals arriving at the **Data In** and $\overline{\textbf{Data In}}$ connectors are interpreted. The following options are available:

- Differential

  If differential mode is selected, both input ports need to receive a signal. The actual data signal is measured as the voltage difference between the two incoming signals.

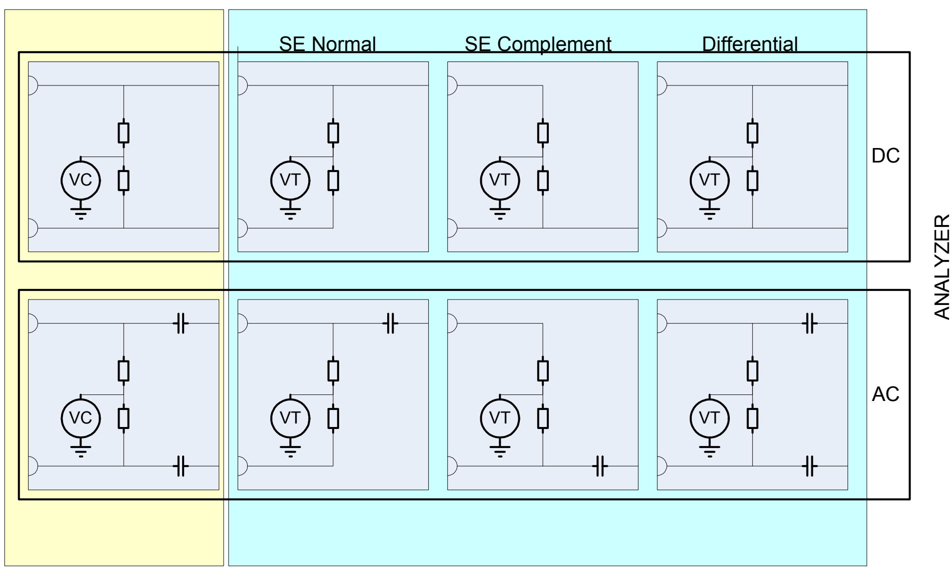
- SE-Normal (Single Ended - Normal)

  In normal mode, only the **Data In** port receives the data signal, the $\overline{\textbf{Data In}}$ port is inactive.

- SE-Complement (Single Ended - Complement)

  In complement mode, only the $\overline{\textbf{Data In}}$ port receives the data signal, the **Data In** port is inactive.

The following drawing is an example how the instrument internal circuitry looks like for different settings.



VT = Termination Voltage

VC = Common Mode Voltage

Data Inverted

To activate data inverted function, select the **Polarity** as **Inverted** in the **Comparator** functional block of the **Analyzer**. This function is required if your device inverts data.

Threshold

Enter a **Threshold** value for applications that do not provide a continuous data stream at the input (for example, any application using bursts), because the averaged threshold voltage will drift from the correct level when there is no input.

Termination Voltage

In this field, enter the termination voltage that is appropriate for the incoming data signal. This selection should be made before the device is connected to the analyzer.

CAUTION Selecting the wrong termination may damage your device.

The **Data In** port is connected to a 50 Ω load impedance (or termination) within the **Analyzer**. Data termination refers to the voltage level at the end of this load. The logic output from a device requires any connected equipment, including the **Analyzer**, to have a specific termination voltage.

Input Range

Before you can synchronize the **Analyzer** to the incoming data stream, you need to define the voltage range within which the eye is located.

Both the high and low level of the data signal must be within this range to find the eye.

NOTE The input voltage range is 2V. When you modify either the high or low voltage, the other voltage is automatically adjusted.

Clock Setup

To measure the **Bit Error Rate** with the **Analyzer**, the bit rate of the data stream must be known. Depending on the options the instrument is delivered with, you could use either an external clock source for the Analyzer (for example, the clock from the generator), or extract the clock signal from the incoming data (CDR mode).

CDR mode does not work for all kinds of data patterns. For example, if the device under test sends only blocks of ones and zeros, there are no transitions in the data stream and the M8020A/M8030A cannot recover the clock.

Also, if you are testing bursts, there are some special considerations for setting up CDR.

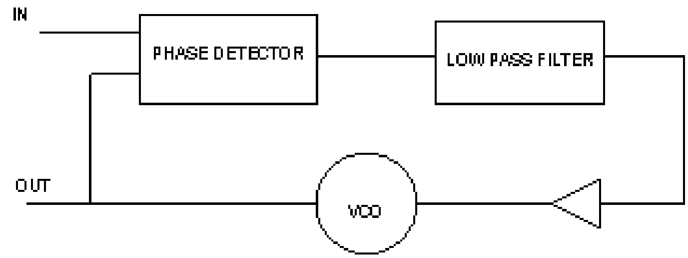
How does Clock Data Recovery Work?

In CDR mode, the CDR has to recover the clock from the incoming data. To do this, the hardware has to decide whether the voltage at the input connector is a logical '1' or '0' and then recover the clock from the detected transitions.

Clock Data Recovery (CDR) is a special kind of Phase Locked Loop (PLL), which recovers clock signal of a data stream. It is a regulatory loop, which synchronizes the local oscillator with an external reference, in this case the incoming data stream.

Phase Locked Loop

A PLL has three parts: a phase detector, a loop filter, and a Voltage Controlled Oscillator (VCO). The phase detector has two inputs, and one output, which is proportional to the phase difference of the inputs. The loop filter is a low pass filter which attenuates the higher frequencies from the output of the phase detector. The VCO is an adjustable oscillator which changes the output frequency depending on its input voltage. The diagram below shows a simple PLL.



One of the most important characteristics of a PLL is its loop transfer function.The loop bandwidth is defined as the integrated magnitude of the PLL's frequency transfer function over the entire frequency spectrum. The loop bandwidth describes how the regulatory loop tracks the VCO to a sine wave FM modulated input signal. Above the bandwidth the loop cannot track such a modulation completely, and thus, the response to the modulation is attenuated.

The other loop parameter is peaking. This describes how much a modulation is exaggerated (mostly close to the loop bandwidth).

Data Rate and Loop Parameters

The data stream contains multiple frequencies, and the CDR needs to know the expected data rate. You can choose between 2 types of CDR; 1st order and 2nd order.

The M8020A/M8030A has an internal CDR which has the following additional input parameters:

- **Loop Bandwidth**: It is the input parameter to set the characteristics of the loop.
- Additionally, you can set the value for peaking in the provided field.
- **Transition Density**: It affects the loop parameters, and it must be either entered, or measured. Some standards define a loop bandwidth for a specific transition density.
- **Peaking**: It defines the second order CDR characteristics by defining a peaking value which is valid for jitter transfer function. The peaking parameter is only available for the second Loop Order.

Threshold

The regular threshold voltage is not only used to determine the optimum sampling for the data, but also to perform measurements.It is not possible to use it for the clock recovery.

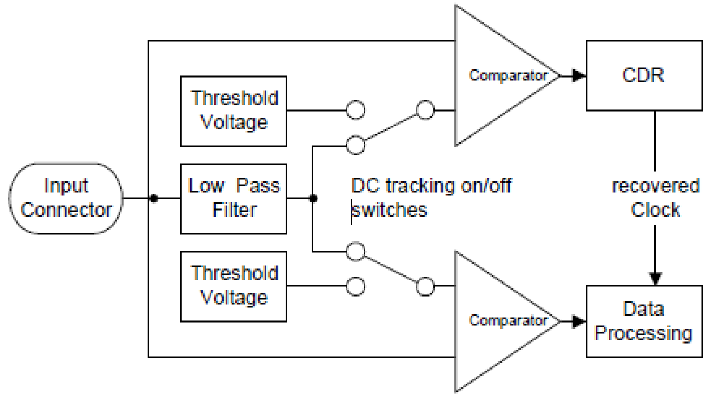
For this reason, the clock recovery circuitry has its own comparator for the incoming data. This comparator also needs to know the threshold voltage.

The threshold voltage can be derived from the input signal via a lowpass filter. This will work fine for most applications. But applications that do not provide a continuous data stream at the input (for example, any application using bursts) cannot use this low-pass filter, because the threshold voltage will drift from the correct level when there is no input. In such cases, the threshold can be specified manually. It is then no longer derived from the input signal. The manually set threshold voltage must of course be within the input range.

The difference between the data path and the CDR path is that the comparator of the CDR is always single-ended. Thus, this comparator always needs a threshold voltage that lies between the high and low levels of the incoming signal.

The differential threshold of the data path comparator has no relation to the single-ended threshold of the CDR path comparator. This means that in differential mode, the two thresholds will be different and in single-ended mode (either normal and complement) they will/can be equal (except during measurements).

The following figure shows a simplified block diagram. It does not reflect the different input modes (especially the differential case), but it matches both single-ended cases.



Bit Rate Range

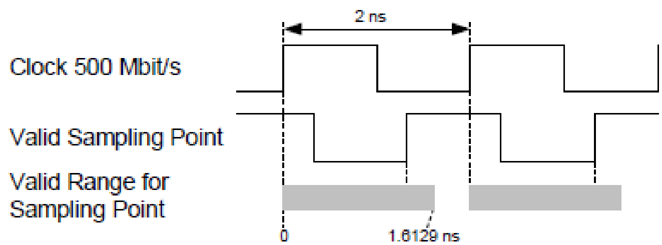
The M8020A/M8030A provides bit rates from 256 Mbit/s up to 16.207 Gbit/s, depending on the instrument's options.

For the Analyzer the following rules apply:

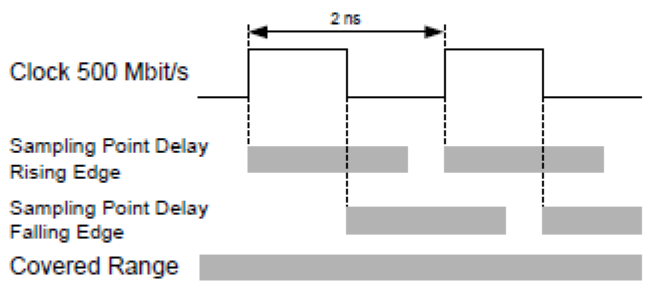
- For low frequencies, you cannot use the automatic data alignment functions (**Auto Align** and **Data Center**). Instead you need to align the Analyzer manually. You can also go to the **Menu Bar** > **Analyzer** or **Module View**, and then select **Data In** port. From the **Properties** window, select **Analyzer** functional block and then click **Execute** button for the **Auto Alignment**
- There are few restrictions to the available sampling point delay values.

  The **Analyzer** can vary the sampling point delay only within a range of 0.0 ps to 1.6129 ns (relative to the clock signal). For frequencies above 620 Mbit/s, this range is sufficient to cover the complete clock cycle (= 1 unit interval).

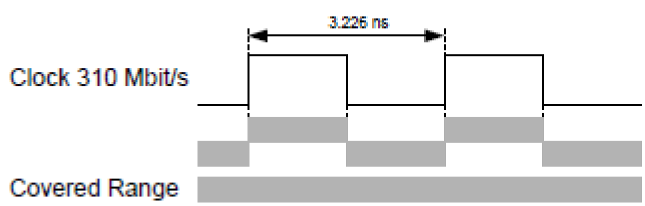
  For lower frequencies, the maximum sampling point delay is smaller than the clock cycle. Therefore, the sampling point cannot be set everywhere within the clock cycle.



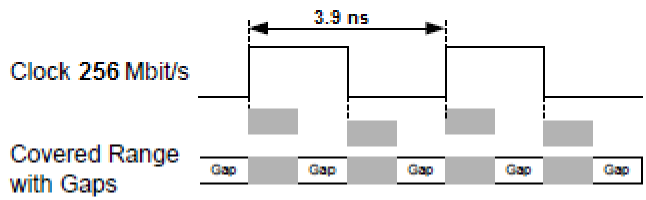
If you cannot find the optimum sampling point in the valid range of sampling points, you can switch from the rising to the falling clock edge.



With this method you can still place the sampling point anywhere in the clock cycle to find the optimum sampling point, even at low frequencies.



However, if the bit rate falls below approximately 310 Mbit/s, even with this method gaps occur in the range of possible sampling points.



As a result, you might not be able to find the optimum sampling point, if it falls in one of the gaps.

CDR Setup

To perform the **CDR Setup**:

1. Go to the **Menu Bar** > **Analyzer** and then select **Data In**.
2. Select **CDR** functional block from the **Parameters** window.
3. Click **CDR State** switch to turn on the CDR state.
4. Specify the **Loop Bandwidth** within the range of 500 kHz to 20 MHz.

Transition Density

The transition density is defined by the number of transitions in the incoming data divided by the total number of bits transmitted. In this field enter the transition density in (%).

This parameter affects the loop bandwidth, and thus must be entered correctly. Some standards specify the loop bandwidth for a given transition density. In such a case enter the value given in the specification, so that the CDR behaves according to the standard. If a standard from the preset list is selected, this field is also preset.

Loop Bandwidth

This is the range of the CDR loop bandwidth. In this field the user should enter the loop bandwidth value; the range is within 313 kHz to 20 MHz.

When the M8020A/M8030A Analyzer is used to characterize a data stream instead of a receiver, the loop parameters should be set according to the used standard.

If the CDR is used to recover the bit stream from a receiver to be characterized due to a lack of a clock output, choose the loop bandwidth significantly higher than the receiver's bandwidth. To characterize a DUT's CDR it is the best practice to use its recovered clock output instead of the J-BERT's built in CDR. Choose a low loop bandwidth to measure the jitter on an incoming data stream as the CDR will track the incoming jitter up to the loop bandwidth and thus make it invisible to the Analyzer.

CDR Spread Spectrum Clocking

This control is used to adapt the CDR to an input bit stream with SSC.

Enabling SSC state widens the loss of lock detection window, and sets the peaking to optimum SSC performance. Enter the **Expected Deviation** and the type of deviation (**Up-Spread**, **Down-Spread** or **Center-Spread**) to set the locking window to an optimum.

SSC is mostly used **Down-Spread** which means, the clock signal is modulated to a lower frequency and back. Thus the average frequency is lowered by half of the maximum deviation. The CDR is adapted to that value. Enter the maximum deviation as specified in the standard.

Sampling Point Setup

This section provides basic information on the sampling point setup.

How Does the Sampling Point Setup Work?

The sampling point of a data signal is defined by two values: a point in time and a voltage level. Each bit of the data signal is sampled at this point in time and in reference to this voltage level. The point in time (in reference to the clock signal) is referred to as the data input delay, and the voltage level is referred to as the threshold.

The location of the sampling point is the decision factor as to whether the incoming bits are identified as logic 0's or 1's. To measure the accurate bit error ratio at the input port, false readings of logic 0's or 1's must be avoided. Therefore, the sampling point must be set to the optimum location within the data eye.

The functions within the **Sampling Point Setup** window allow you to:

- Prepare the Analyzer for the incoming data signal regarding the connector termination.
- Graphically display the eye diagram in terms of voltage, input delay and a BER threshold.
- Adjust the location of the sampling point.

Auto Alignment

Use this option to automatically set the optimum sampling point.

To perform the **Auto Alignment**:

1. Go to the **Menu Bar** > **Analyzer** and then select **Data In**.
2. Select **Analyzer** function block from the **Parameters** window.
3. Click **Execute** button for the auto alignment.

This routine will not stop if the optimum sampling point cannot be found. If no optimum sampling point is found after a reasonable time, you can click **Cancel**.

The following settings may affect the result of the auto align function:

- Polarity (Data Inverted)
- Threshold (BER Threshold)

Threshold Center Alignment

To perform the **Threshold Center Alignment**:

1. Go to the **Menu Bar** > **Analyzer** and then select **Data In**.
2. Select **Comparator** function block from the **Parameters** window.
3. Click **Execute** button for the threshold center alignment.

The threshold center alignment starts an auto-search function that sets the threshold to the optimum point of the incoming data eye on the vertical voltage axis without changing the data input delay. This function can be used for determining the optimum threshold for asymmetric data eyes, or for patterns with an unequal mark density.

NOTE This function uses the alignment BER Threshold to determine the top and bottom eye edges.

Data Center Alignment

To perform the **Data Center Alignment**:

1. Go to the **Menu Bar** > **Analyzer** and then select **Data In**.
2. Select **Input Timings** function block from the **Parameters** window.
3. Click **Execute** button for the data center alignment.

This button starts an auto-search function that aligns the data signal with the clock signal so that the **Analyzer** samples at the optimum point of the data eye in the time axis. This automatically compensates for delays in the clock/data paths, preventing unnecessary errors. The decision threshold is not changed.

NOTE Ensure that the received clock frequency is stable before using Data Center.

NOTE

The clock/data alignment process time is pattern dependent, and with some large user patterns the alignment can take several minutes. If you encounter such a long time with a user pattern, it may be possible to first perform clock/data alignment on a pure PRBS pattern. This generally does not affect alignment accuracy, and can minimize measurement time.

This tip does not apply in cases of severe pattern dependent jitter or with devices that do not work with PRBS patterns.

NOTE

This function uses the alignment BER threshold to determine the left and right eye edges.

Canceling Auto Align

Click this button to cancel the **Auto Align**, **Threshold Center**, or **Data Center** functions while they are in progress. The following parameters will be returned to their previous value or status:

- Auto Align Canceled
  - Data Delay and Threshold values returned to previous.
- Threshold Center Canceled
  - Threshold value returned to previous.
- Data Center Canceled
  - Data Delay value returned to previous.

Alignment BER Threshold

In this list, select an alignment BER threshold that is appropriate for your application.

The alignment BER threshold is the pre-defined threshold used by the **Auto Align**, **Threshold Center**, and **Data Center** functions to define the edges of the data input eye in the time and voltage axes. You may wish to change the threshold for the following reasons:

- Choosing smaller alignment BER thresholds will cause the auto-search functions to set more accurate sampling points. However, if the BER threshold is set lower than the residual BER of the measurement, the auto-search functions will fail. 1E-9 is the smallest BER threshold available.

Pattern Synchronization

The M8020A/M8030A calculates bit error rates by comparing the received data with the expected data patterns. To do this, it needs to know where the start of the pattern is located in the data stream.

Introduction to Pattern Synchronization

Pattern synchronization (sync) refers to aligning the incoming data pattern with the internal reference pattern.

Hardware-Generated Patterns

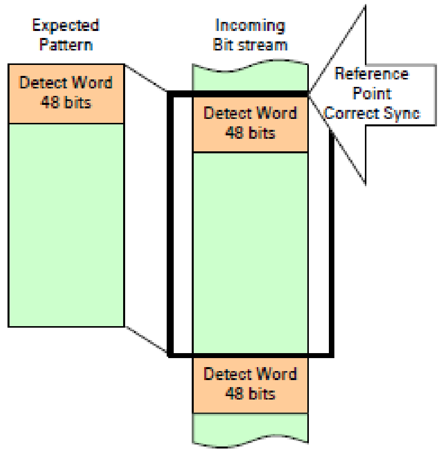
For 2^n-1 PRBS patterns, bits from the incoming data pattern "seed" the Analyzer's Generator, causing it to generate a precisely aligned internal reference pattern.

Memory-Based Patterns

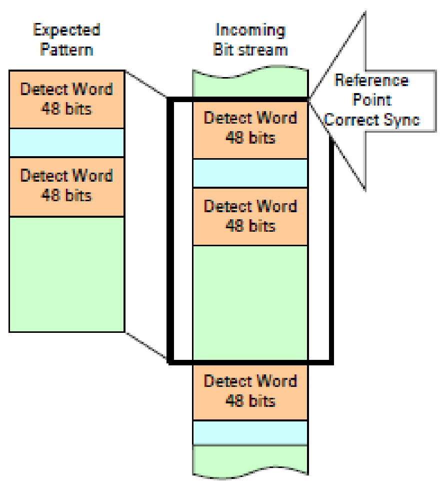
For software-generated and user patterns, a 48-bit pattern from the pattern is used as a detect word. Optimally, this detect word should be unique within the entire pattern. The Analyzer searches for this detect word within the incoming data stream, and uses the point in the data stream as a reference, and compares all following bits with the pattern. If the measured BER is better than the synchronization BER, the Analyzer will be synchronized.

There are thus three possible outcomes for a synchronization:

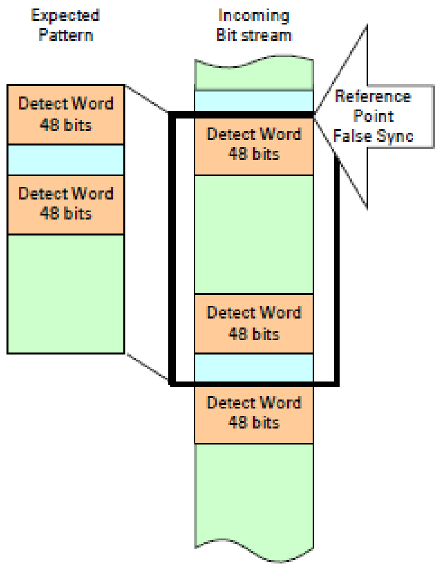
- Single instance of the detect word in the data stream



- Multiple instances of the detect word with correct synchronization.



- Multiple instances of the detect word with false synchronization.



If the **Analyzer** attempts to synchronize on the incorrect detect word, the BER will be unacceptably high, and, if automatic synchronization is selected, the **Analyzer** attempts another re-sync.

The detect word on which the **Analyzer** attempts to re-sync is chosen strictly by chance. So if there are two instances of the detect word in the pattern, the Analyzer has a 50% chance of selecting the correct one.

The more instances of the detect word exist in the pattern, the higher are the chances for incorrect synchronization. The software attempts in any case to identify a 48-bit pattern that occurs as seldom as possible in the pattern. For very large patterns, this can unfortunately take a very long time, and the software ends the search if it expects that it would take longer to find an adequate detect word than it would to attempt to synchronize. If the search for a detect word is ended, the most unique detect word identified is used.

Patterns must always be synchronized in order to do accurate BER testing. If patterns are out of alignment by just one bit, errors can be as high as 50% (5E-1) for PRBS patterns, and 100% (1E+0) for custom patterns.

By default, the **Analyzer** is in automatic sync mode with a sync threshold of 1E-3. This setting is recommended for most applications, and usually allows the synchronization function to be "transparent", requiring no attention. However, for special applications, changes can be made to the sync mode and sync threshold.

What Type of Synchronization Should You Use?

The type of synchronization you use affects how errors are measured and displayed. A **Sync Loss** is recognized when the BER is greater than the sync threshold. This can be caused by a high error rate, pattern misalignment, or clock loss. Choose the sync mode setting that is appropriate for the type of errors you anticipate.

- Automatic Sync with a sync threshold BER of 1E-3 is recommended for most applications.

  With this mode selected, the synchronization algorithm starts whenever the BER exceeds the threshold. However, it is not possible to make accurate BER measurements higher than the sync threshold.

- Manual sync can be used for synchronizing once, confirming proper pattern alignment, and then measuring BERs higher than the sync threshold. This is useful for the following applications:
  - To monitor the integrity of clock signals. You may wish to measure BERs that exceed the sync threshold to confirm clock slip.
  - To collect data for constructing eye contour information. You may wish to move the sampling point to locations in the data eye that have BERs exceeding the sync threshold.

  This mode doesn't allow the analyzer to automatically synchronize if the BER becomes greater than the sync threshold. For example, the analyzer will not re-synchronize after momentary clock loss.

NOTE Adjusting the data input delay may cause momentary clock loss. If you select Manual Sync mode, this may also result in sync loss.

- Burst sync mode is a special operating mode for measuring data in bursts of bits, rather than one continuous stream of bits.

NOTE

If the Analyzer is in Manual Sync mode, it is recommended that you keep an eye on the **Sync Loss** indicator, present on the **Status Indicator**. There are various actions that can lead to loss of synchronization. Check the sync loss every time you make changes to the instrument.

What is False Synchronization?

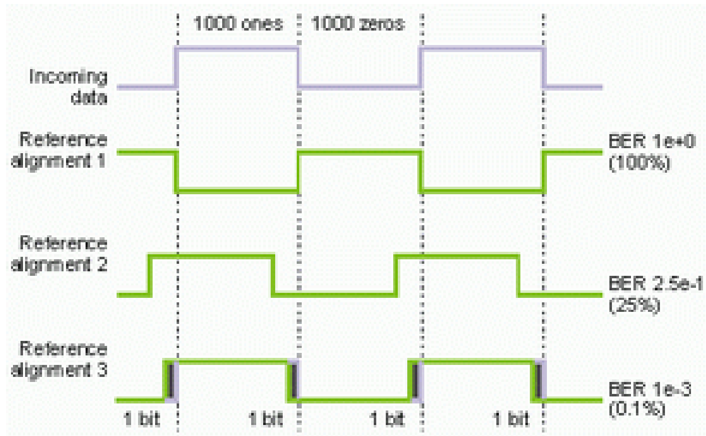
For patterns other than PRBS, the **Analyzer** may gain sync at a point in the pattern that meets the sync threshold, but is not the correct point where the internal reference pattern and the received data pattern match. This is called false synchronization.

NOTE

False synchronization cannot occur with PRBS patterns because a 1 bit misalignment would cause a measurement of 50% or more errors. Thus, the BER during a misalignment would always be greater than the sync threshold BER.

For example, consider a pattern of 1000 ones and 1000 zeros as shown in the following figure. With reference alignment 1 the patterns are totally out of phase and the Analyzer is measuring 100% errors.

But as the reference moves closer to optimum alignment, the percentage of errors gradually approaches zero (reference alignment 2 and 3). For exact alignment, the sync threshold must be set lower than the BER caused by a 1 bit misalignment, in this case 1E-3.

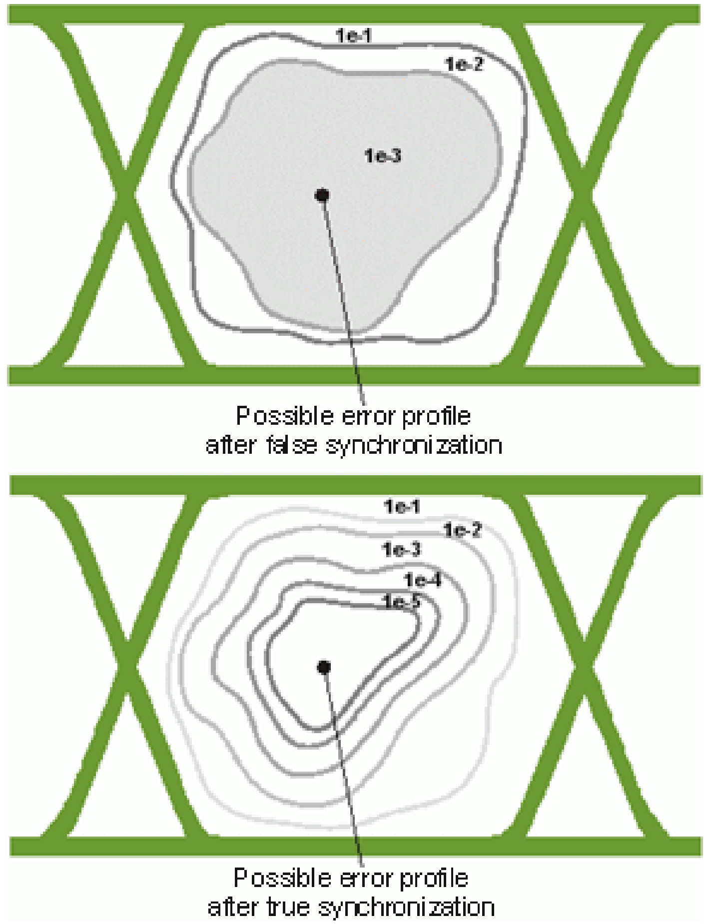


How Can You Tell if Your Synchronization is False?

You may suspect false synchronization under the following conditions:

- You are using a pattern other than PRBS and the Analyzer gains sync, but it measures a constant, fixed error ratio.
- You are using a pattern other than PRBS and the Analyzer gains sync, but auto-search functions (Auto Align, Clock/Data Center, Threshold Center) repeatedly fail.

  In a false sync, the sync threshold BER of 1E-3 may be met, but eye edges at BER 1E-3 (required by an auto-search function) may not be found. This is because BERs less than 1E-3 do not exist within the data eye.



If you suspect a false sync, try re-synchronizing at a sync threshold BER lower than the fixed error ratio. If sync is acquired without the problems listed above, then your previous sync was false. Your current sync should be on an exact pattern alignment.

NOTE While auto-search functions are in progress, the sync threshold BER is changed to the same value as the alignment BER threshold. If you are using these functions and want to consistently re-synchronize at a lower sync threshold, you must set the alignment BER threshold to the same value as the sync threshold BER.

How to Synchronize an Incoming Pattern

To synchronize the incoming pattern to the expected pattern:

1 Go to the **Menu Bar** > **Analyzer** and then select **Data In**.

2 Select the **Analyzer** function block from the **Parameters** window.

3 Select **Re-Sync**. The choices are **Automatic** or **Manual**.

- Automatic Sync

  When this option is selected, the **Analyzer** constantly tries to synchronize the patterns when the BER threshold is exceeded.

- Manual

  Manual synchronization can be selected, for example, if the signal delay is very unstable, and you want to avoid that, the resynchronization process affects the measurement results.

Bit Recovery Mode

Understanding Bit Recovery Mode (BRM)

Traditionally, bit error rate testing compares the bits from a Device Under Test (DUT) against a reference data set, called the expected data. The user of Bit Error Ratio Tester (BERT) has to provide this expected data and load it into the M8020A/M8030A.

The M8020A/M8030A system then samples the incoming data signal with a sampling point that can be varied over time and voltage, to measure the BER. The M8070A system software is capable of creating graphics, such as the eye opening, from the information gathered during sampling. A compare circuit counts the differences between the bits of the incoming data stream and the expected data.

Now there is a mode that removes the need for the user to provide expected data, while still allowing one and two-dimensional measurements, such as the eye opening.

This is the Bit Recovery Mode (BRM).

This has two benefits:

- the user does not need to worry about the expected bits, which makes a setup easier and faster.
- the M8020A/M8030A system can measure non-deterministic data streams. This means it is no longer necessary to force a device into a specific test mode.

Bit recovery mode uses a second sampling circuit in the M8020A/M8030A analyzer to always sample at the sweet spot of the eye (typically at 50% of eye opening in time and voltage). The sampled data from this second sampling circuit acts as a reference and is passed to the compare circuitry, instead of the expected data.

This means the BER is now a relative figure. Taking the bit from the sweet spot of the eye cannot verify if this bit is correct in itself, it can just use it as a reference for any bit sampled in the border area of the eye. Bit recovery mode makes one and two-dimensional sweeps possible to sample in the border area of the eye and find out how the BER value increases. From this we can derive the random and deterministic (Rj and Dj) characteristics.

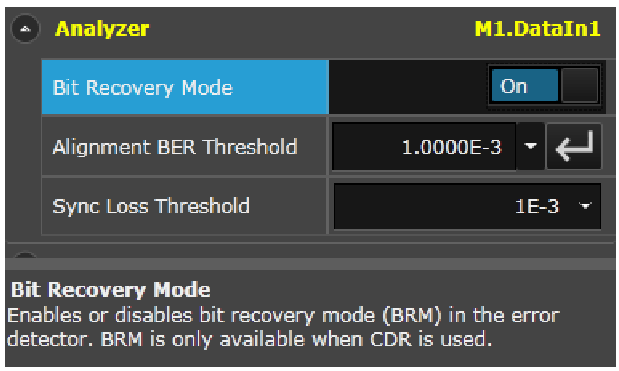
We can also take the eye opening with a known, deterministic test pattern and compare it with operational data, where a device idles or provides scrambled data, or there are asynchronous events like hand-shake signals.

The Bit Recovery mode also has a limitation that it requires a minimum eye opening: an eye that is too narrow cannot be processed. Care also needs to be taken if there is a good eye opening but there is a finite BER inside the eye.

Setting up Bit Recovery Mode

To set up the **Bit Recovery Mode** mode:

1. Go to the **Menu Bar** > **Analyzer** and then select **Data Input** or if you are in **Module View**, then click on **Data In** location.
2. Select **Analyzer** functional block from the **Parameters** window.
3. Click **Bit Recovery Mode** switch to turn on the **Bit Recovery Mode**.



NOTE The bit recovery mode can be enabled only when CDR state is on.

Once the **Bit Recovery Mode** is enabled, the BRM indicator on the **Status Indicator** turns green. See Status Indicators Window on page 92.

BRM can be used with all measurements supported by M8070A system software.

7 Setting up Patterns

KEYSIGHT TECHNOLOGIES

Pattern Overview

The purpose of data patterns is to simulate the type of data that your device might receive in the real world. Different patterns present different data loads to your instrument, which can cause variations in the bit error ratio. A bit pattern is sent from the generator to your device. At the same time, the expected output pattern of your device is internally generated in the analyzer (to provide a reference).

Selecting a pattern is the first step in setting up a BER measurement. The M8020A/M8030A provides various patterns to fulfill most standard testing needs. It supports the following types of patterns:

- PRBS
- Pulse
- Clock
- Static
- Memory Patterns

Patterns consist of a sequence of symbols. A symbol can have the following type of coding:

- Binary (Bit)
- 8B/10B
- 128B/130B
- 128B/132B

Sequence Editor

Overview

The sequence editor allows you to create and maintain sequences. In addition to this, it also allows you to edit the memory patterns.

A sequence consists of up to 500 blocks that can be looped. Single or multiple blocks can be looped. The sum of the blocks and the counted loops must not exceed 500. An overall loop restarts the sequence after it has come to its end.

You can also upload predefined sequences for PCIe, USB and SATA using the **Recall Setting** dialog. For details, refer to Recall/Save Instrument State on page 103.

When to Use a Sequence

You may wish to test a device that uses a certain protocol for processing data.

For example, the device might expect synchronization data, a preamble, payload data, and a suffix.

All this can be provided by a user-defined sequence.

How a Sequence is Defined

The sequence is defined by a SequenceExpression which is formulated in its own language, checked by the Sequence Editor.

The SequenceExpression specifies:

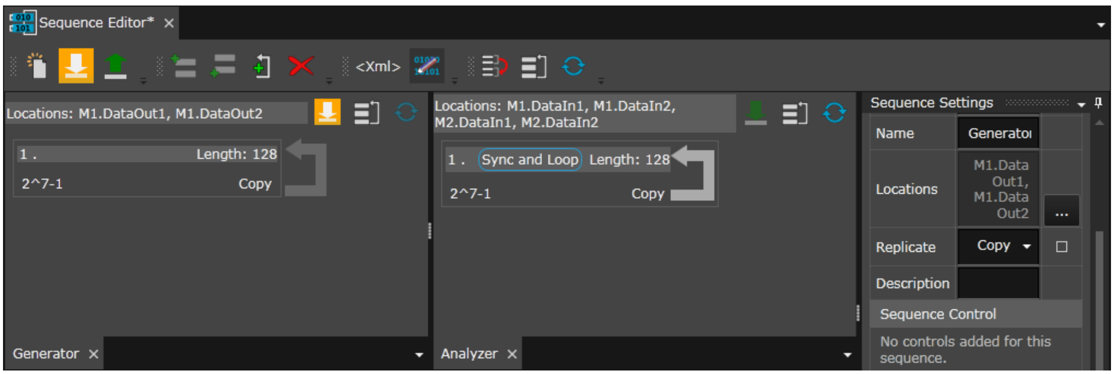
- the sequence start (and break) conditions
- the blocks, their contents, and trigger output
- the loops

How to Launch a Sequence Editor

To launch the **Sequence Editor**:

- Go to the **Menu Bar** > **Patterns** and then select **Sequence Editor**.

  The **Sequence Editor** will appear as shown in the following figure:



The **Sequence Editor** user interface includes the following elements:

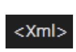
- Toolbar
- Sequence Control Pane
- Sequence Settings Window
- Pattern Edit Pane

These GUI elements are described in the section that follows.

Toolbar

The toolbar provides the following convenient sequence editing functions:

Table 41

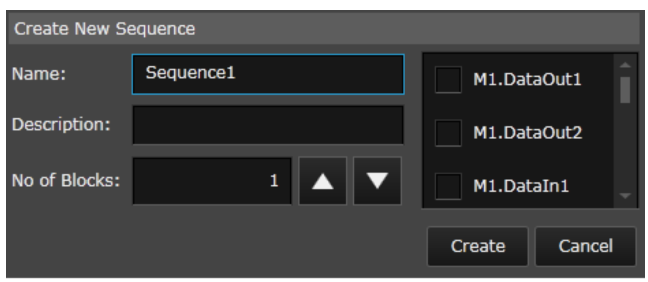
Elements	Name	Description
	New	Click this icon to create a new sequence. Refer to Creating New Sequence on page 230.
	Download	Click this icon to save the modifications that are done using the **Sequence Settings** window on the module. An  orange icon indicates that modifications are not yet applied on the module. You can also apply the changes on either generator, or analyzer or both.
	Reload Running Sequence	Click this icon to reload the running sequence settings. Reloading a sequence will discard the changes made in the sequence editor and will reload the current sequence with its factory settings.
	Add Block Before	Click this icon to add a block before selected sequence block.
	Add Block After	Click this icon to add a block after selected sequence block.
	Loop	Click this icon to create a loop in a sequence block. Refer to Creating a Loop on page 231.
	Delete	Click this icon to delete the selected block from the sequence. Deleting a block will also remove all loops that are associated with this block.
	<xml>	Click this icon to toggle between the user interface and xml code. The changes made in user interface are also reflected in xml code and vice versa.
	Show Patterns	Shows the patterns in the **Pattern Edit Pane** if the selected block has memory patterns.

Elements	Name	Description
	Break	Click this icon to terminate an infinite loop that is set to "manual" break condition. Sequence execution continues with the next block.
	Reset	Click this icon to interrupt and re-initialize a running sequence.
	Sync	Click this icon to sync the sequences globally.

Creating New Sequence

To create a new sequence:

1 Click on the  **Create New Sequence** icon present on the toolbar. A sliding window will appear which allows you to create a new sequence. See the figure below:



2 Perform the following settings:

- **Name** - Provide a sequence name.
- **Description** - Provide a description of the sequence.
- **No. of Blocks** - Enter the no. of blocks to be added. You can add upto 500 blocks in a sequence.
- **Sequence Locations** - Use the check-box to select the sequence location. You can either select a single sequence location or multiple locations.

3 Click **Create** to create a sequence.

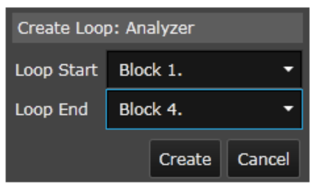
Creating a Loop

To create a loop (if desired):

- Click the **Add Block Before** icon to add the blocks before the selected sequence blocks.

- Click the **Add Block After** icon to add the blocks after the selected sequence blocks.

- Click the **Create Loop** icon. A **Create Loop** dialog will appear as shown in the following figure:



- Specify the start and end block of the loop for the specified sequence (for e.g. generator, analyzer or user-defined sequence).
- Click **Create** to create a loop in the specified blocks.
- Click on the loop indicator. You will see a **Loop Setting** functional block in the **Sequence Setting** window where you can specify the loop count and enable the looping option.

Loop Within Sequences

A loop defines the transition from the end of a block to the beginning of the same or a previous block. It is not possible to jump into an existing loop. It is also not possible to specify loops within loops (except the default overall loop).

For information on how to create a loop, refer to Creating a Loop on page 231.

Deleting a loop - It is possible to delete a loop. To delete a loop, select the loop indicator and click **Delete** icon.

Modifying the Existing Sequences

By default, the **Generator** and **Analyzer** sequences already exist whenever you launch a **Sequence Editor**. You can however modify these sequences as per the requirements. Using these options, you can:

- Add and delete the blocks
- Add loops in the blocks
- Specify the settings for each block using the **Sequence Settings** window. For more details, refer to Sequence Setting Window on page 232.

NOTE

Please note that there should be at least one block in the sequence.

Remember that you are not allowed to perform delete operation in the block if there is only one block in sequence.

Sequence Setting Window

The **Sequence Setting** window allows you to set the properties for the selected block and sequence. Using this window, you can specify the following settings:

- **Instrument Configuration**. For details, refer to Instrument Configuration on page 233.
- **Sequence Configuration**. For details, refer to Sequence Configuration on page 238.
- **Block Data**. For details, refer to Block Data on page 239.
- **Block Settings**. For details, refer to Block Settings on page 242.
- **Block Branches**. For details, refer to Block Branches on page 243.
- **Block Controls**. For details, refer to Block Controls on page 244.

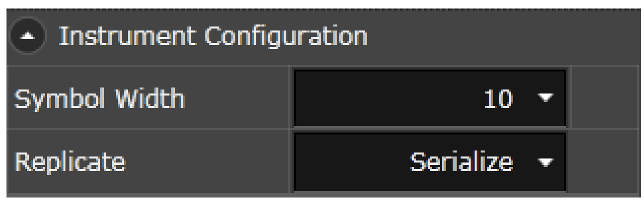
Set to Default Check-Box

Most of the parameters in the **Sequence Setting** window contains a **Set to Default** check-box which gets highlighted when some modifications are done. You can click on this check-box to change the settings to their default values.

The following figure shows how the check-box gets highlighted when you modify the symbol width to 10.

However, when you click on the **Set to Default** check-box, the value changes to 1 (default value). See the following figure.

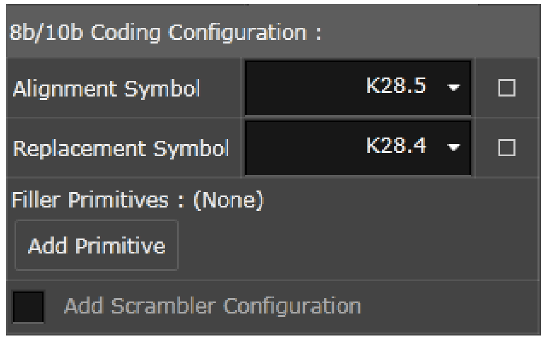
Instrument Configuration



The instrument configuration section provides the following options:

- **Symbol Width**: Use this option to select the symbol width e.g, 1, 10, 130 or 132 bit. Depending upon the symbol width you have selected, you can specify the coding configuration for that particular symbol width.
- **Replicate**: Select the replicate option (Serialized, Copy or Copy plus Phase Adjust).

If the symbol width is 10, you can specify the 8B/10B coding configuration. Using this, you can define alignment symbol, replacement symbol and filler primitives.

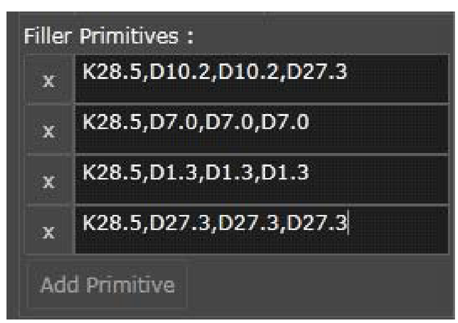


The **Alignment Symbol** contains the K28.1, K28.5 and K28.7 symbols.

The **Replacement Symbol** contains the K28.0, K28.4, K28.7, K23.7, K27.7, K29.7 and K30.7 symbols.

Filler Primitives are inserted or deleted for clock tolerance compensation. These are not compared and therefore cannot be counted as errors. **Filler Primitives** contain symbols. A maximum of four alternative filler primitives can be used. Each filler primitive can consist of up to 4 filler symbols. Filler symbols are separated by comma (,). To add filler primitive;

- Click the **Add Primitive** button.
- Enter the **Symbol**. The filler primitive can consist of up to 4 filler symbols.



Wild Cards for Filler Primitives

The wild cards allow you to set one or more out of the maximum of four symbols of a filler primitive as don't care. In this case, all allowed D and K symbols will match and are removed from incoming DUT data.

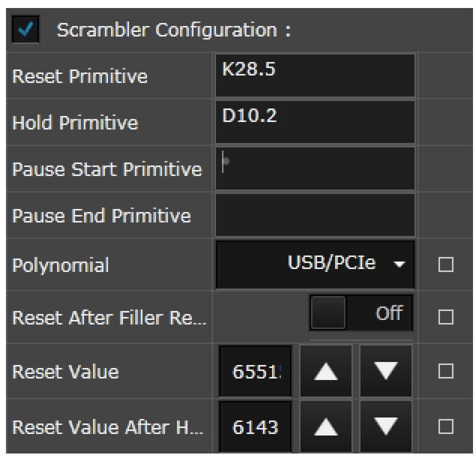
For example:

K28.5, ●

Here the dot (●) symbol will be treated as don't care.

Add Scrambler Configuration

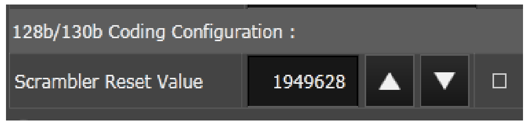
Select the **Scrambler Configuration** check-box to add scrambler configuration to the sequence.



It allows you to:

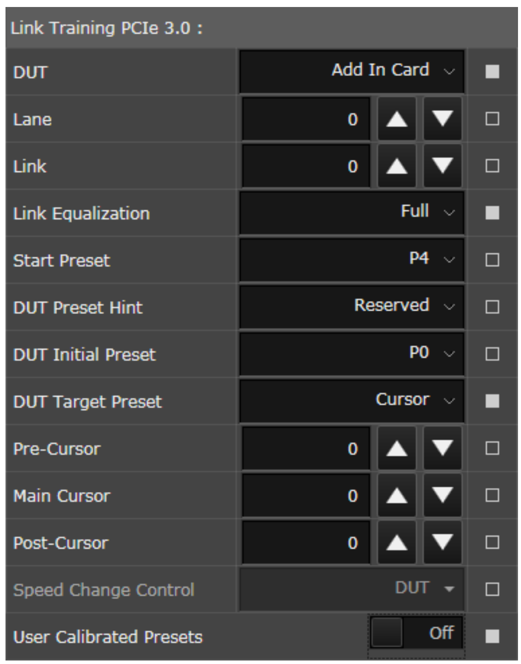
- Reset Primitive
- Hold Primitive
- Pause Start Primitive
- Pause End Primitive
- Polynomial
- Reset After Filler Remove
- Reset Value
- Reset Value After Hold

If the symbol width is 130, you can specify the 128B/130B coding configuration. Using this, you can define scrambler reset value.



Link Training Configuration

If the symbol width is 130, you can configure the link training parameters to control LTSSM ((Link Training and Status State Machine).



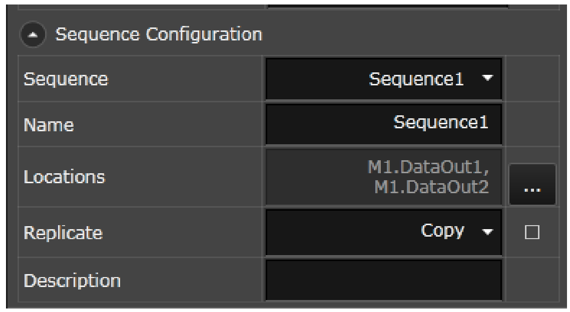
The following parameters can be configured as an attributes.

- **DUT Type** - Specifies which role the DUT should play during link training. It can either be an upstream device for testing a downstream port or vice versa.

- **Lane** - Specifies the lane number being used.
- **Link** - Specifies the link number being used.
- **Link Equalization** - Determines whether link equalization should be performed. It can either be aborted after phase 1 (Bypass) or fully executed. In the second case it can be determined whether only preset or all (i.e. individual cursor) requests should be accepted.
- **Start Preset** - Specifies the preset used by the J-BERT's TX port after switching to Gen 3 operation and when operating as an upstream device.
- **DUT Preset Hint** - Specifies the preset hint being sent by the J-BERT to the DUT during phase 0 of the link equalization procedure. It is only used when the BERT operates as upstream device.
- **DUT Initial Preset** - Specifies the preset the J-BERT transfers to the DUT in phase 0 of the link equalization procedure. It is only used when the BERT operates as an upstream device.
- **DUT Target Preset** - Specifies the preset the J-BERT requests the DUT to switch to during link equalization. Depending on the role the J-BERT is playing, this is done in either phase 2 or 3 of the link equalization training.
- **Speed Change Control** - Specifies whether the speed change to Gen3 (i.e. 8 Gbps) speed will be initiated by DUT or BERT during link training. It is only used when the BERT operates as a downstream device.
  If not specified DUT will initiate the speed change and will also request BERT for the same. It is only used for the DUT type as System Board.
- **User Calibrated Presets** - Specifies whether BERT's Data Out should use user-calibrated presets or standard presets during link training. Enabling it means that BERT's Data Out will use de-emphasis/pre-shoot values that had been previously calibrated, otherwise it will use standard de-emphasis/pre-shoot values defined by the PCIe3 specification. By default this option is turned off which means that standard presets will be used.

For further details on link training for PCIe 3.0, refer to Interactive Link Training on page 285.

Sequence Configuration



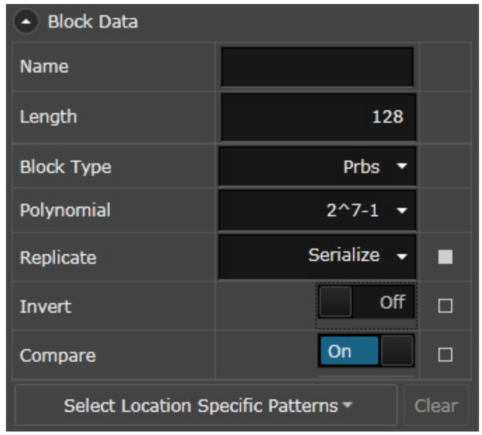
The sequence configuration section provides the following options:

- **Sequence**: Allows you to apply a sequence configuration on either Generator, Analyzer or user-created sequences.
- **Name**: Provide the sequence name.
- **Locations**: Specify the locations for the sequence. Use the button available to select the locations.

NOTE Make sure not to use the unused location which are not being used in the test setup. However, if it is used, ensure that the 'Re-Sync' parameter of these locations is set to manual to reduce overall system load caused by unnecessary pattern re-sync attempts.

- **Replicate**: Select the replicate option (Serialized, Copy or Copy plus Phase Adjust).
- **Description**: Add a description to the sequence.

Block Data



The **Block Data** section allows you to:

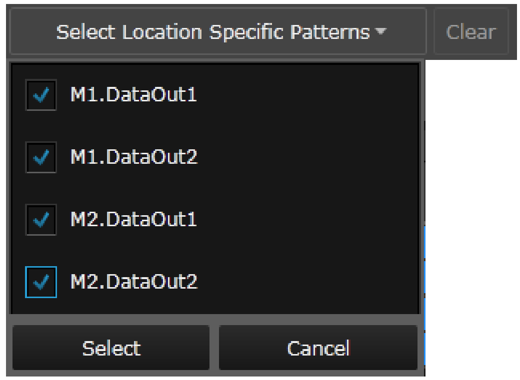
- Provide a block name.
- Provide a block length.
- Select **Block Type.** The available options are **Clock**, **Pulse**, **PRBS**, **Static**, **Memory Pattern** and **Link Training**.
    - For block type as **Clock**, you need to specify the **Divider**, **Replicate** and the **Compare** feature (refer Compare Feature on page 241**)**.
    - The **Replicate** feature shows how the serial patterns are split to multiple locations. It has the option options:
      Serialize: In this a pattern is split and distributed to the locations. This is for the parallel side of a serial bus.
      Copy: In this each location gets a copy of the pattern.
      Copy Plus Phase Adjust: In this each location gets a copy of the pattern. Scrambler phases of the different locations are set differently. This is for the parallel side of a serial bus.
    - For block type as **Pulse**, you need to specify the **Width**, **Compare** and **Offset** feature.
    - For block type as **PRBS**, you need to specify the **Polynomial**, **Replicate**, **Invert** and **Seed (Hex)** feature.

      The following table shows the PRBS polynomial that is used to generate the selected PRBS 2^n-1.

Table 42

PRBS	Shift Register Length	Polynomial	PRBS Length
2^7-1	7	x^7+x^6+1	127
2^9-1	9	x^9+x^5+1	511
2^11-1	11	$x^{11}+x^9+1$	2047
2^13-1	13	$X^{13} + X^{12} + X^2 + X^1 + 1$	8191
2^15-1	15	$x^{15}+x^{14}+1$	32767
2^23-1	23	$x^{23}+x^{18}+1$	8388607
2^23p-1	23	$x^{23}+x^{21}+x^{18}+x^{15}+x^7+x^2+1$	8388607
2^31-1	31	$x^{31}+x^{28}+1$	2147483647
2^33-1	33	$X^{33} + X^{20} + 1$	8589934591
2^35-1	35	$X^{35} + X^{33} + 1$	34359738367
2^39-1	39	$X^{39} + X^{35} + 1$	549755813887
2^41-1	41	$X^{41} + X^{38} + 1$	2199023255551
2^45-1	45	$X^{45} + X^{44} + X^{42} + X^{41} + 1$	35184372088831
2^47-1	47	$X^{47} + X^{42} + 1$	140737488355327
2^49-1	49	$X^{49} + X^{40} + 1$	562949953421311
2^51-1	51	$X^{51} + X^{50} + X^{48} + X^{45} + 1$	2251799813685247

  - For block type as **Static**, you need to specify the **Single Value** and the **Compare** feature.
  - If you select **Block Type** as **Memory Pattern**, a **Select Pattern** window will open which allows you to load the memory patterns.
  - For block type as **Link Training**, you need to specify the **Direction.**
- Click **Select Location Specific Patterns**. This opens with a drop-down selection where you can specify the ports or locations of specific patterns.



- Select the ports or locations and then click **Select**. The settings option for each port or location will be added to the **Block Data** functional block.
- Click **Clear** if you want to remove port or location specific patterns and use single pattern for the block.

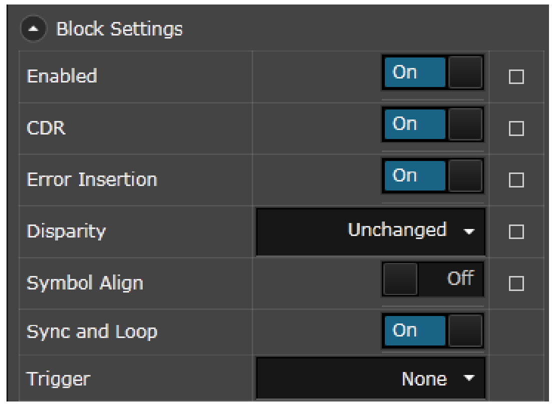
Compare Feature

The **Compare** feature allows you to compare the block data for a particular sequence. It provides you the freedom to modify the sequence without deleting the blocks.

When to use/not use this feature

Suppose you created a sequence with multiple blocks. Now, if you want to exclude particular block(s) from that sequence while comparing, instead of deleting the whole sequence, just set the compare functionality “OFF” which in-turn will disable that particular block(s) from the sequence for comparison.

Block Settings



The **Block Settings** section allows you to:

- Click on the corresponding ON/OFF toggle button to turn on the following features:
  - CDR
  - Error Insertion
- Click on the ON/OFF toggle button to enable the block settings on the sequence.
- Click on the ON/OFF toggle button to turn on the **Symbol Align** feature.
- Select disparity whether positive, negative or unchanged.
- Click on the ON/OFF toggle button to turn on the **Sync and Loop** feature.

NOTE

The Sync feature will be automatically turned on once you turn on the Sync and Loop feature.

- Use the drop down option to specify whether you want to apply trigger on either **Pulse** or **Pulse or PRBS Match**.

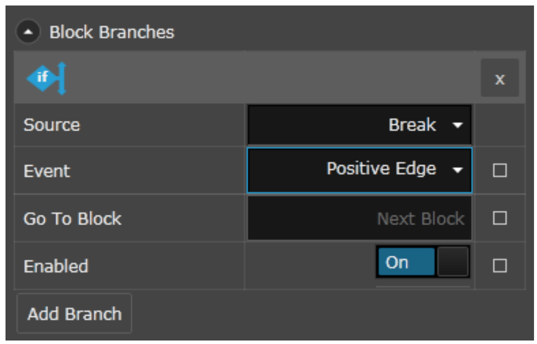
NOTE

The changes made in the **Block Settings** section are reflected on the selected block.

Block Branches

The **Block Branches** are used to add the branches within the sequence. You can add up to two branches in a sequence.

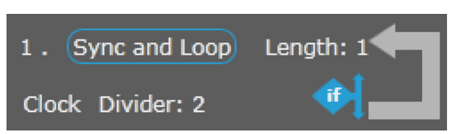
To add a **Block Branches**, click on the **Add Branch** button. The **Block Branches** section will appear as shown in the following figure:



The **Block Branches** section provides the following settings:

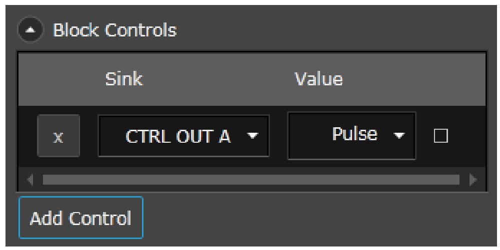
- **Source**: Use the drop-down list to specify the source for the branch.
- **Event**: Specify the event for the branch.
- **Go to Block**: Specify the block name to jump.
- **Enabled**: Use the ON/OFF toggle button to enable the branching option.
- Click **Add Branch** if you want to add another branch. Up to two branches can be added within the same block.
- Click **Delete** icon to delete the branch.

Once the branching is enabled in a block, the **Block Branches** icon will appear on the sequence block as shown in the following figure:



Block Controls

The **Block Controls** section allows you to provide sink value at Ctrl Out A and Sys Out A/B. These values help you to trigger the given sink at different values that are defined i.e. Low, High or Pulse. Press **Add Control** to add more block controls. You can add up to four block controls.



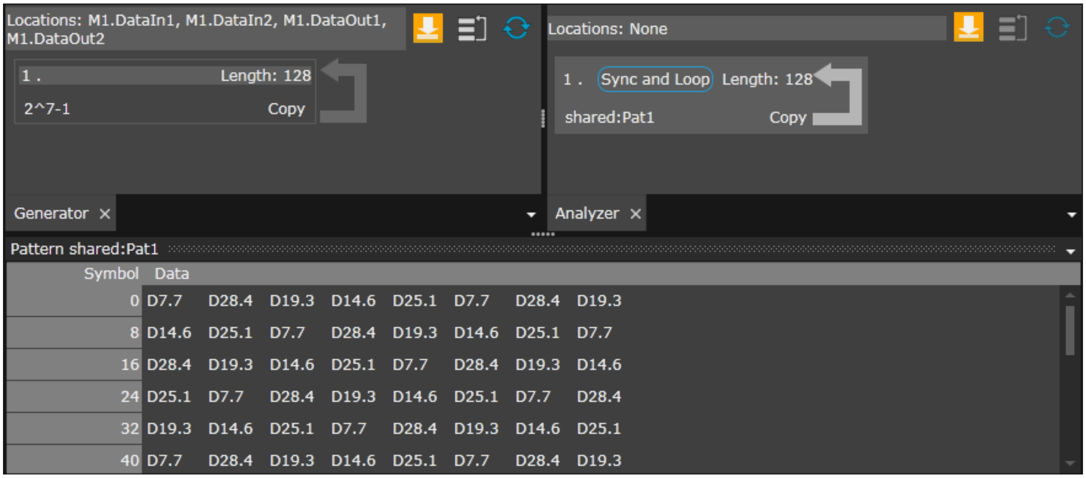
Editing a Pattern in a Sequence Editor

The **Sequence Editor** user interface contains **Pattern Edit Pane** that allows you to edit the memory patterns.

Make sure to enable the **Show Patterns** option by clicking on the  **Show Pattern** icon, present on the toolbar. This will display the **Pattern Edit Pane**, in case it is not visible in the **Sequence Editor**.

To edit a pattern, you have to first load it in the sequence block. To do so:

1 Select the block on which the patterns are loaded.
2 Go to **Sequence Setting** window and then select **Block Data** functional block.
3 Select the **Block Type** as **Memory Patterns**. A **Select Pattern** window will open which allows you to load the memory patterns.
4 Select the desired patten and click **Select**.
5 The pattern will be loaded into the selected block as well as on the **Pattern Edit Pane**. See the following figure:



6 Edit the patterns as explained in the section Pattern Edit Pane on page 266.

Sharing Sequences

The M8070A software currently does not allow the functionality to share the sequences. However, there is a workaround to share sequences.

Follow the given steps:

1 Go to **File** > and click **Save Instrument State..**. Once you save the instrument state, the sequence settings are also saved in the **Settings** folder which is created at the following path:

"Documents\Agilent\M8070A\Workspaces\Default\User\Settings"

2 Share **Settings** folder.

3 Copy the **Settings** folder at the same path on another system.

4 Go to **File** > and click **Recall Instrument State...**. The sequences will now appear in the **Sequence Editor**.

User-Defined Sequences

This section describes the basics of user-defined sequences.

A sequence is created and maintained by means of the Sequence Editor. A sequence consists of up to 500 blocks that can be looped. Each block can generate a pause signal (constant 0 or 1), a divided clock signal, a 2^n -1 PRBS, or a user pattern.

Single or multiple blocks can be looped. The sum of the blocks and the counted loops must not exceed 500. An overall loop restarts the sequence after it has come to its end.

Sequence Block Display

A new sequence consists of one block that is infinitely repeated (looped). By default, this block has a length of 512 bits and generates a Pause 0 signal (a continuous stream of zeros). All this is shown on the display.

Sequence Block Parameters

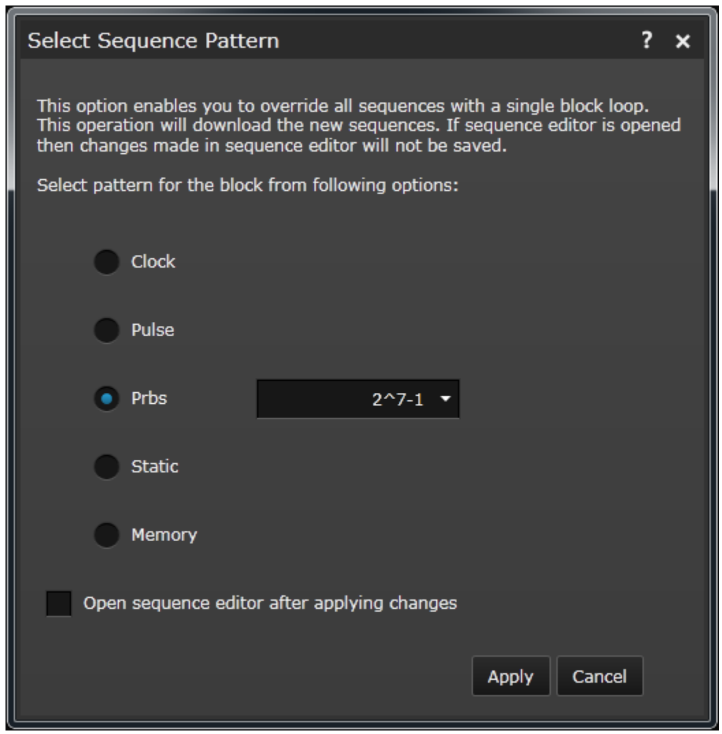
The **Sequence Setting** window allows you to change the contents and the trigger generation of a sequence block.

Choices are:

- **Block No.** - Can be up to 500 blocks.
- **Length** - This is the length of the pattern in bits for a standard pattern of 128 bits.
- **Block Type** - Clock, PRBS, Pulse, Static or Memory Patterns.
- **PRBS** - You can choose a PRBS of polynomial 2^n -1. The range of "n" is 7, 10, 11, 15, 23, or 31. You can change the block length, if desired.
- **Divided Clock**: You can use this option to generate the signal at every nth clock pulse.
- **Memory Pattern**: When you select pattern type as memory pattern, a **Select Pattern** dialog will appear. You have to locate the pattern file and then click **OK**. The patterns will be loaded in the sequence.

Select Sequence Pattern Dialog

The **Select Sequence Pattern** dialog allows you to override all sequence with a single block loop. This operation will download the new sequences. If the **Sequence Editor** is already opened, then changes made in the **Sequence Editor** will not be saved.



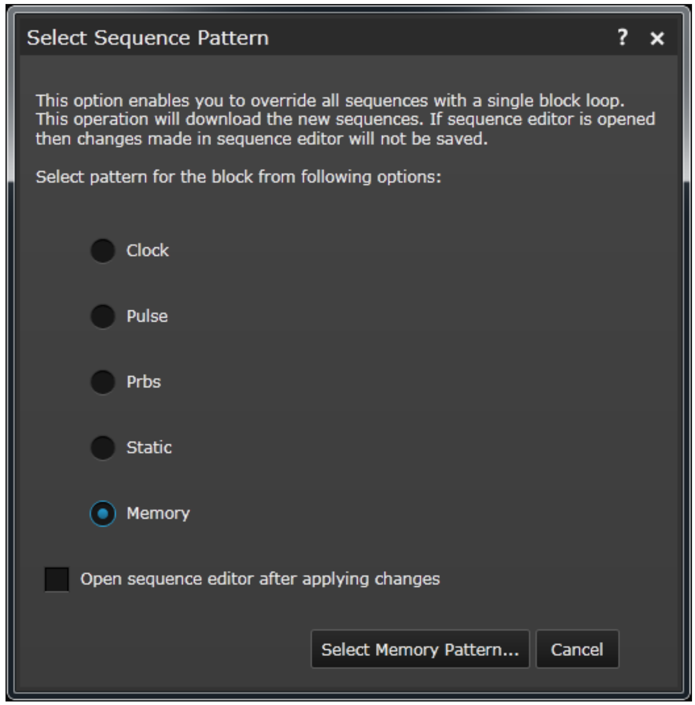
This dialog allows you to select pattern for the block from the following options:

- Clock
- Pulse
- PRBS
- Static
- Memory

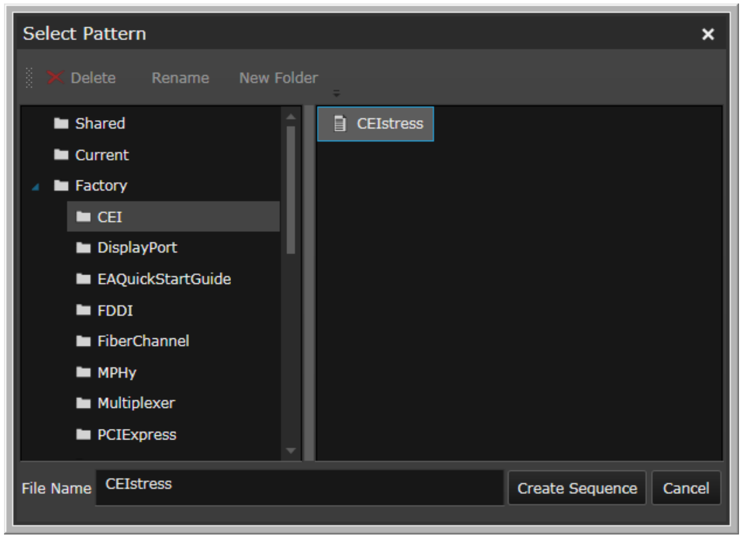
Select the check box if you wish to open the **Sequence Editor** with the selected patterns.

Click **Apply** to apply the changes.

However, when the **Memory** option is selected the **Apply** button changes to **Select Memory Pattern...** button.



When you click this button, the **Select Pattern** dialog will appear.



Once you select the pattern in **Select Pattern** dialog, click **Create Sequence** button to create the new sequences.

Pattern Editor

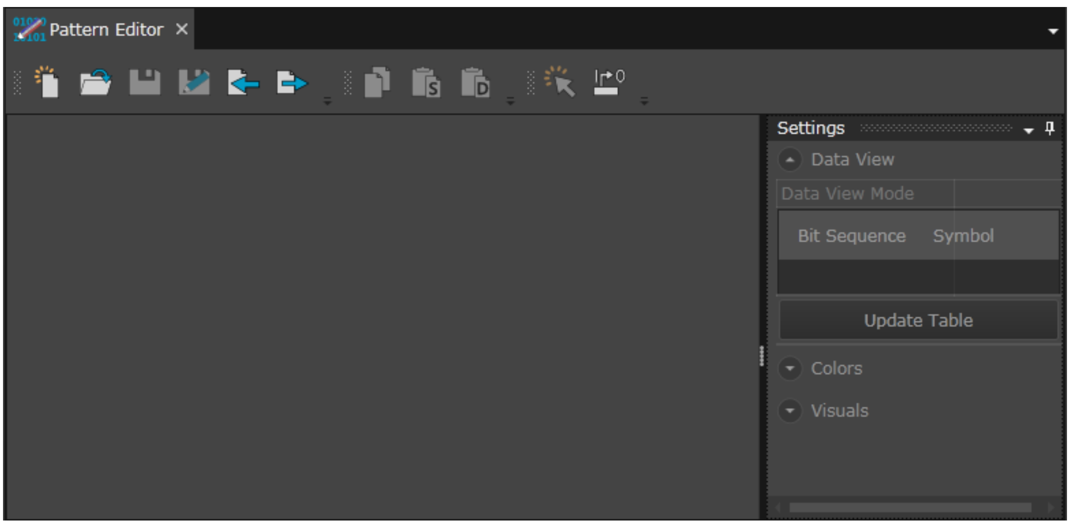
The pattern editor provides an interactive user-interface for creating, editing and importing the patterns.

How to Launch Pattern Editor

To launch the **Pattern Editor**:

- Go to the **Menu Bar** > **Patterns** and then select **Pattern Editor**.

The **Pattern Editor** will appear as shown in the following figure:



The **Pattern Editor** user interface includes the following elements:

- Toolbar
- Settings Window
- Pattern Edit Pane

Toolbar

The toolbar provides the following convenient pattern editing functions:

Table 43

Elements	Name	Description
	New	Click this icon to create a new pattern. For details, refer to Creating New Patterns on page 253
	Open	Click this icon to open a pattern from a file. For details, refer to Opening Existing Patterns on page 254
	Save	Click this icon to save the current pattern. For details, refer to Saving Patterns on page 256.
	Save As	Click this icon to change the properties of current pattern and then save it under different name. For details, refer to Saving Patterns on page 256.
	Import	Click this icon to open **J-BERT Pattern** dialog. You can import the patterns from J-BERT. For details, refer to Importing Patterns on page 259.
	Export	Click this icon to export patterns in the desired location. These patterns can be used by other instruments for it for testing and analysis. For details, refer to Exporting Patterns on page 260.

Elements	Name	Description
	Copy	These functions follow Microsoft Windows copy/paste functionality. You can perform the copy/paste operations in the following ways: ▪ Click on Copy/Paste icons ▪ Use keyboard shortcuts (Ctrl+C, Ctrl+V) ▪ Right click and use the context menu options The Copy/Paste function allows you to: ▪ Select either the partial data or the complete symbols in a pattern and copy/paste it anywhere in the pattern. ▪ Use the copy/paste operations for the partial data across multiple instances of Pattern Editor, irrespective of the pattern coding. ▪ Use the paste operation to and from clipboard to Pattern Editor. Please note that the paste operations for the complete pattern symbol is only allowed in the same pattern symbol coding. This icon is only available when the partial data is copied from the pattern. Whenever a paste operation is done using this function, the underline data will be pasted.
	Paste as String	
	Paste as Data	
	Select All	Click this icon to select all the symbols in the pattern.
	Block Edit	Click this icon to perform block edit operations on currently selected pattern. For details, refer to Block Edit Operations on page 261.

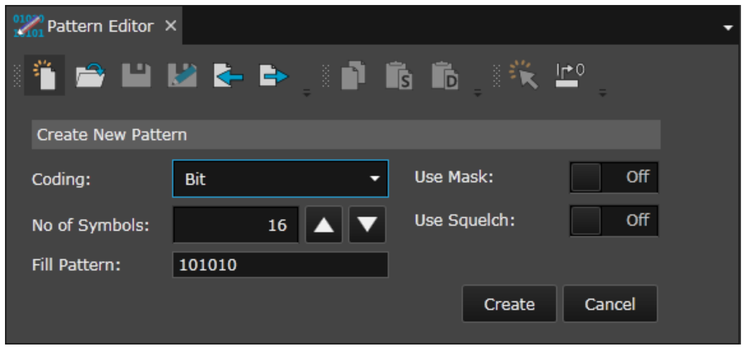
The **Pattern Editor** allows you to perform following tasks on patterns:

- Create a new patterns. For details, refer to Creating New Patterns on page 253.
- Open and already existing patterns. For details, refer to Opening Existing Patterns on page 254.
- Save a patterns. For details, refer to Saving Patterns on page 256.
- Import supported patterns. For details, refer to Importing Patterns on page 259.
- Perform block edit operations. For details, refer to Interactive Link Training on page 285.
- Edit a patterns on **Pattern Edit Pane**. For details, refer to Pattern Edit Pane on page 266.

Creating New Patterns

To create a new pattern:

1 Click the **New** icon present on the tool bar. This opens with the **Create New Pattern** sliding window as shown in the following figure:



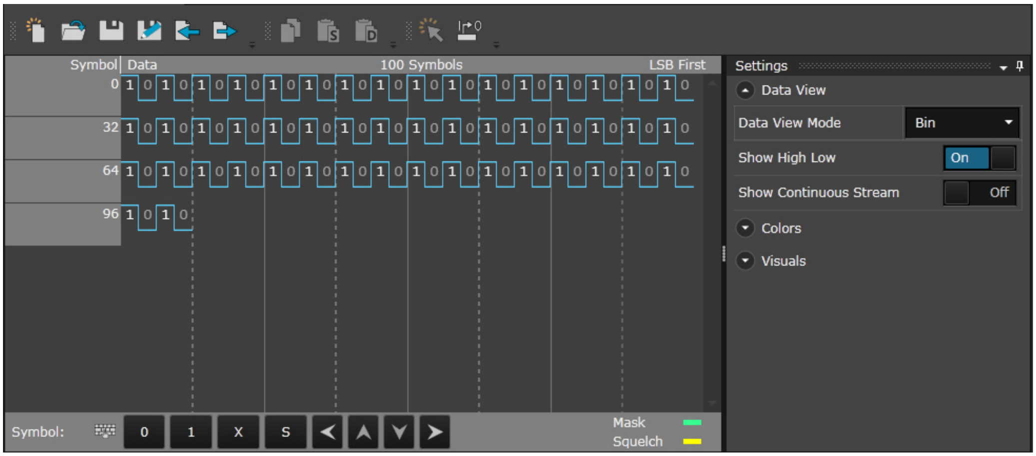
2 Select the coding type. The **Pattern Editor** currently supports the following types of coding:
- Bit (Binary)
- 8B/10B
- 128B/130B
- 128B/132B

3 Specify the number of symbols. Use the **UP** and **DOWN** button to increase or decrease the number of symbols.

4 Specify the fill patterns.

5 Specify by using the ON/OFF toggle switch whether you want to use **Mask** and **Squelch** (Electric Idle). In **Squelch**, the amplifier output is zero.

6 After you have entered these parameters, click **Create**. The newly created pattern will appear in the **Pattern Editor** pane.



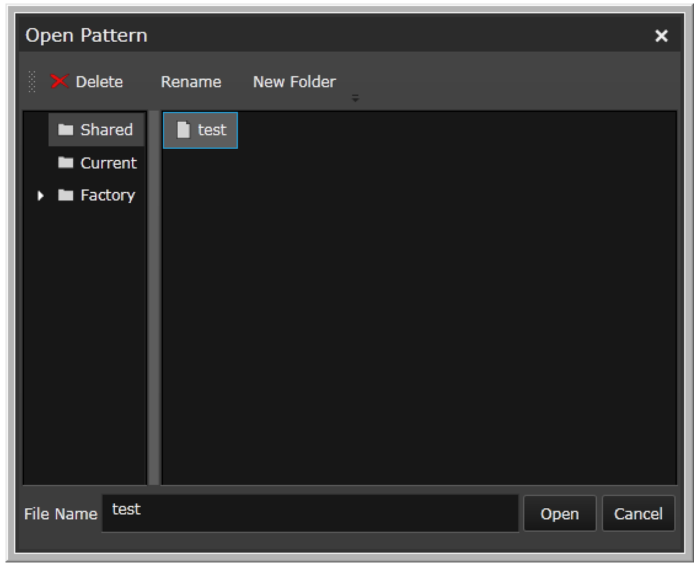
7 Click the **Save** icon to save the pattern. For more details, refer to Saving Patterns on page 256.

Opening Existing Patterns

To open an existing user pattern:

1 Click the **Open** icon present on the tool bar.

This opens the **Open Pattern** dialog, where you can locate and open the desired pattern. You can even perform the operations such as renaming and deleting the pattern file and creating new folders.



You can select the pattern from the following default folders:

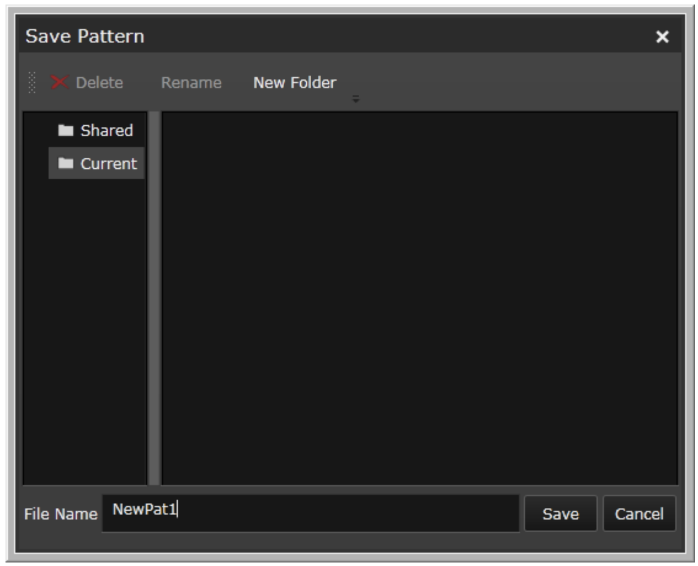
1 **Shared**: Patterns that are shared between settings.
2 **Current**: Patterns that are local to current setting.
3 **Factory**: Factory supplied standard patterns. These patterns are read only and cannot be modified.

2 Click **Open** to load the patterns.
3 To rename a file, select the file and click **Rename**. The filename will become editable.
4 To delete a file, select the file and click **Delete**.
5 To add new folder, select location you want to create your folder and then click **New Folder**.
6 To rename a folder, select the folder and click **Rename**. The folder name will become editable.
7 To delete a folder, select the folder and click **Delete**.

Saving Patterns

To save the current user pattern:

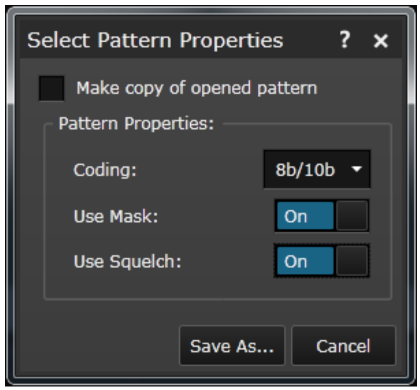
1 Click the  **Save** icon present on the tool bar. A **Save Pattern** dialog will appear which allows you to save the patterns under the defined file name and location.



You can use the **Save Pattern** dialog to perform the operations such as renaming and deleting the patterns. You can even create a new folder or save the patterns in the **Current** or **Shared** folders. The current folder is for current users while the shared folder is accessible by all users. If the pattern has already been saved earlier, the saved file will be updated.

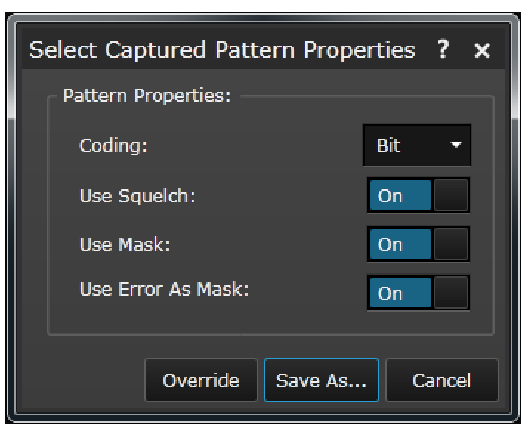
2 Click on the  **Save As** icon if you wish to change the properties of current pattern and then save it under different name. The current pattern can be either a non-captured pattern or a captured pattern.

a For saving a non-captured pattern, a **Select Pattern Properties** dialog will appear. This dialog provides the following options:



- Make a copy of current pattern with no change in pattern properties.
- Change the properties of current pattern. Using this option, you can change the pattern coding and enable/disable Mask and Squelch.

b For saving a captured pattern, a **Select Captured Pattern Properties** dialog will appear. This dialog provides the following options:

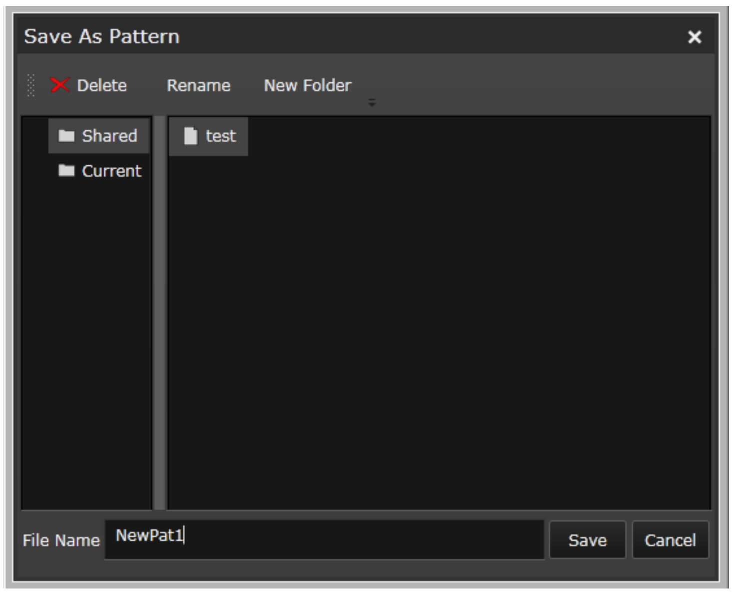


- Change the properties of current pattern. Using this option, you can change the pattern coding and enable/disable **Mask**, **Squelch** and **Error as Mask**.

The **Use Error As Mask** option is only visible when the **Use Mask** option is enabled and coding is same as captured pattern.

- Change the properties of current captured pattern. Using this option, you can change the pattern coding and enable/disable Mask, Squelch and Error as Mask.
- Click **Override** to remove all errors from the pattern.

3 Press **Save As...** button. A standard **Save As Pattern** dialog box will appear.



4 Specify the name and location and click **Save**.

Importing Patterns

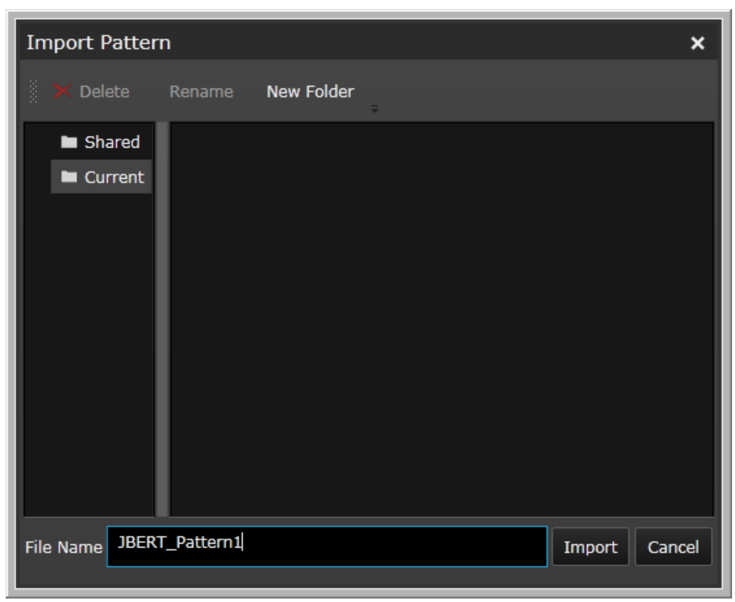
The M8070A **Pattern Editor** allows you to import the J-BERT N4903B patterns, edit them and use it for testing and analysis.

NOTE The length of J-BERT N4903B pattern format (Version=EPA 2.0) is limited to 4194304 bytes (33554432 bits). However, M8070A pattern format (Version=M8000 1.0.0) can be much longer and has a different header.

To import a pattern:

1 Click the **Import** icon. This opens the **Select J-BERT Pattern** dialog, where you can locate the J-BERT N4903B pattern. You are allowed to import patterns with any file extension.

2 Select the JBERT pattern and click **Open** in the **Select J-BERT Pattern** dialog. An **Import Pattern** dialog will appear as shown in the following figure:



3 Choose a location (shared or current) to save the pattern. You can also create a new folder.

4 Provide a file name and click **Import**. The imported patterns will appear in the **Pattern Edit Pane**.

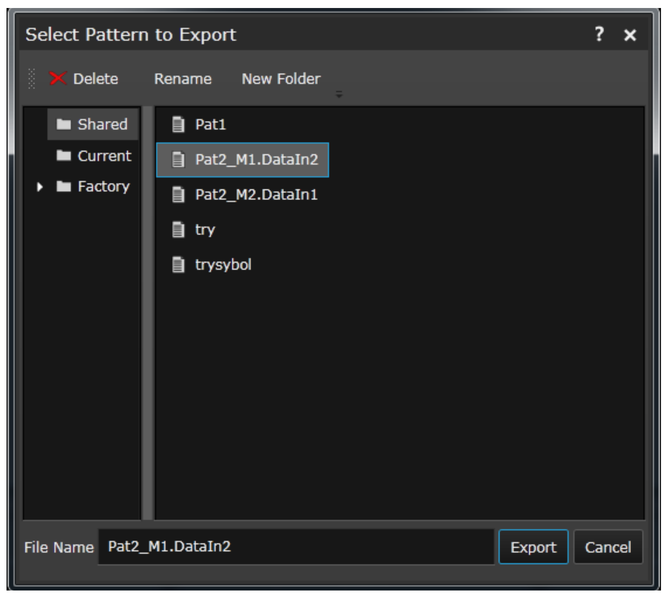
5 You can use the **Import Pattern** dialog to perform the operations such as renaming and deleting the patterns.

Exporting Patterns

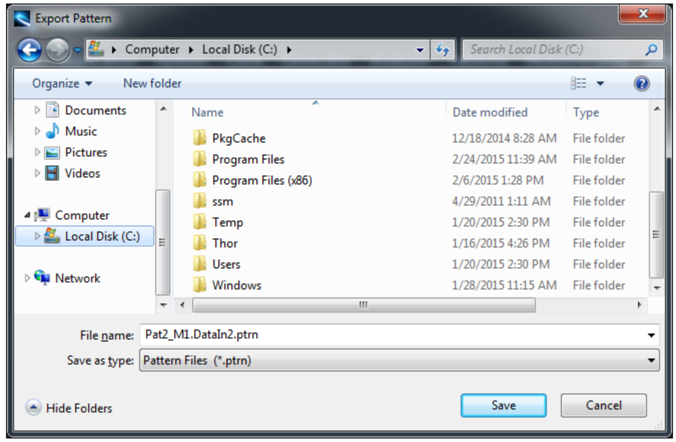
The M8070A **Pattern Editor** allows you to export patterns and use it for testing and analysis.

To export a pattern:

1 Click the  **Export** icon. This opens the **Select Pattern to Export** dialog.



2 Select the pattern and click **Export**. This opens the **Export Pattern** dialog.



3 Locate the path and file name for the *.prtn* file.

4 Click **Save** to export the data to the specified destination.

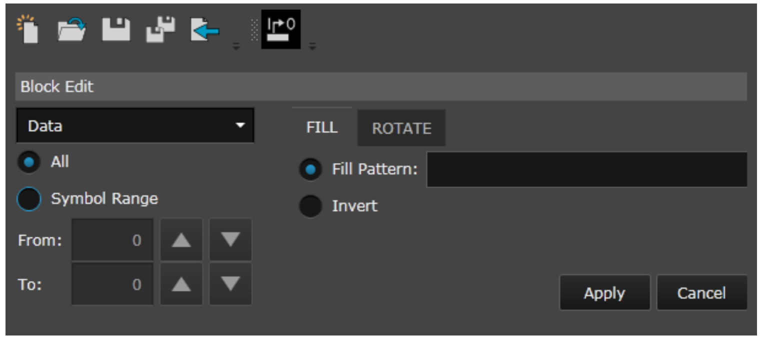
Sharing Patterns

You can share patterns to other users by exporting the pattern to a shared location and later importing that pattern to Pattern Editor. For details on how to export and import patters, refer to Exporting Patterns on page 260 and Importing Patterns on page 259.

Block Edit Operations

The **Block Edit** window provides an easy way to modify parts of the pattern or the entire pattern at once. This can be used when setting up a new pattern. It can also be used as an optional technique for editing existing patterns.

Click on the  **Block Edit** icon present on the toolbar. The **Block Edit** sliding window will appear as shown in the following figure:



With the **Block Edit** window, you can use the drop-down list to specify whether the block edit operation has to be performed on data or listed attributes. You can also define the range that is to be modified. The available options for the range are:

- **All**

  Choose this option to edit the entire pattern.
- **All**

  Choose this option to edit the entire pattern.
- **Range**

  Choose this option to select the range of symbols entered in the **From** and **To** fields.

The **Block Edit** window contains the following tabs:

- **Fill**

  This tab allows filling the given range in the pattern with the specified value. The available options for the **Fill** tab are:
  - **Fill Pattern**

  Fills the specified value to the given range in the pattern.
  - **Invert**

  Invert the bits. 0 becomes 1 and 1 becomes 0.
- **Rotate**

  This tab provides the following options:

- **Rotate Symbol Left by**:

  Treats the pattern data in the specified range as a circular buffer and rotates the bits to the left by the specified amount. No data will be lost and what is at the start of the buffer will be at the end of the buffer after the rotation.

- **Rotate Symbol Right by**:

  Treats the pattern data in the specified range as a circular buffer and rotates the bits to the right by the specified amount. No data will be lost and what is at the end of the buffer will be at the start of the buffer after the rotation.

- **Align to Sequence**:

  Aligns the pattern data in the specified range to a specified pattern sequence.

Settings Window

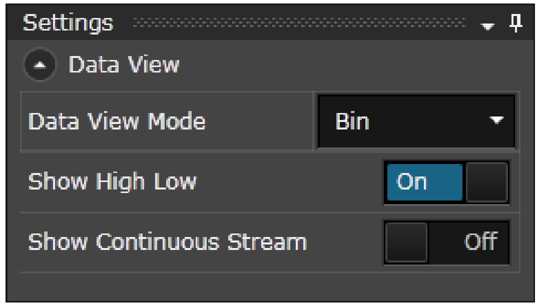
The **Settings Window** allows you to set the visualizations of the currently selected patterns. This helps you to easily identify the data and attributes in the pattern.

The **Settings Window** provides the following options:

Data View

The **Data View** option allows you to set the following:

- **Data View Mode** - Select the data view mode as Bin, Hex or PAM4.
- **Show High Low** - Toggle the ON/OFF switch to display either high or low transition in the patterns. This option is only available in Bit coding.
- **Show Continuous Stream** - Toggle the ON/OFF toggle switch to show continuous stream in the patterns. This option is only available in Bit coding.

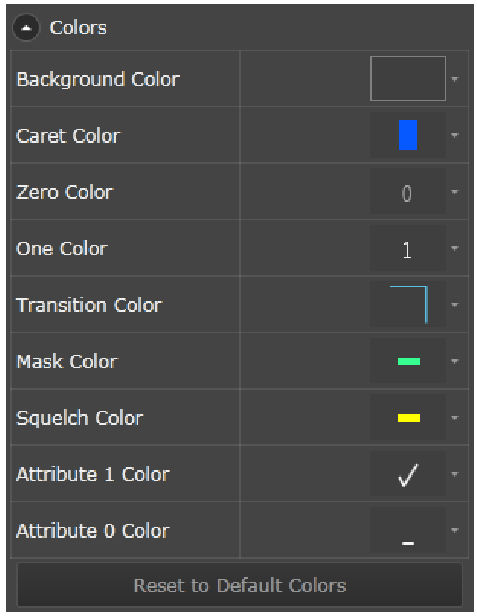


Colors

The **Colors** option allows you to set the following:

- **Background Color** - Changes the background color of the pattern editor pane.
- **Caret Color** - Changes the color of the carat in the pattern.
- **Zero Color** - Changes the color of all occurrences of zeros (0) in the pattern.
- **One Color** - Changes the color of all occurrences of ones (1) in the pattern.
- **Transition Color** - Changes the transition color in the pattern.
- **Mask Color** - Changes the color of the bits that are masked.
- **Squelch Color** - Changes the color of the bits that are squelch.
- **Attribute 1 Color** - Changes the color of attribute 1.
- **Attribute 0 Color** - Changes the color of attribute 0.

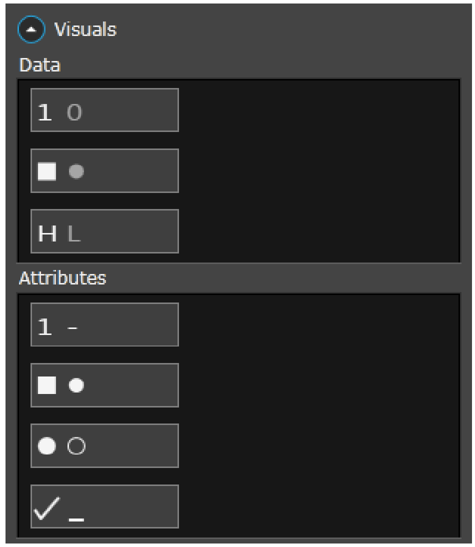
You can at anytime reset the color settings to default colors by pressing the **Reset to Default Colors** button.



Visuals

The **Visual** option allows you to set the following:

- **Data** - Changes the data appearance as selected from the data visual options.
- **Attributes** - Changes the attribute appearance as selected from the attributes visual options.



Pattern Edit Pane

The pattern edit pane displays the pattern and also allows you to edit it.

The pattern edit pane shows the symbol and data. You can edit selected data using the keyboard keys. Remember, the pattern edit pane does not allow you to enter any wrong data.

The pattern edit pane also allows the copy/paste functionality. Once you select the symbols in a pattern, you can copy them and paste it anywhere in the pattern. You can perform the copy/paste operations in the following ways:

- Click on Copy/Paste icons
- Use keyboard shortcuts (Ctrl+C, Ctrl+V)
- Right click and use the context menu options

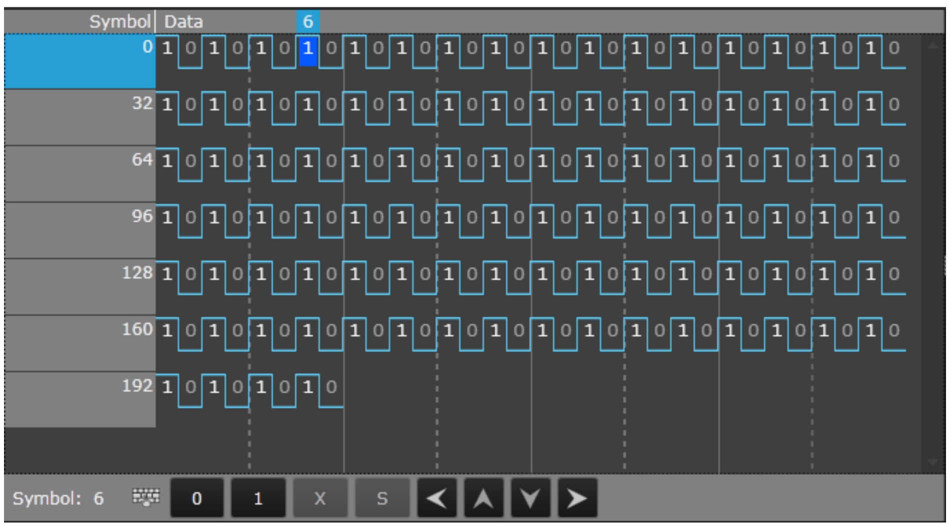
Please note that the copy operation only allows a complete symbol to be copied.

It is possible to use the copy/paste operations across multiple instances of Pattern Editor window with same pattern coding.

Patterns consist of a sequence of symbols. A symbol can have the following type of coding:

1 Bit (Binary)
2 8B/10B
3 128B/130B
4 128B/132B

The following figure shows the **Pattern Edit Pane** with a bit coded pattern (Binary) pattern:



The bottom of the **Pattern Editor Pane** indicates the position of the selected symbols and buttons to edit them. These buttons are only available when you create a pattern in bit coding and the data view mode is binary (BIN). You can use these buttons to:

0 - Set data 0

1 - Set data 1.

X - Click this toggle button to add/remove mask on a data.

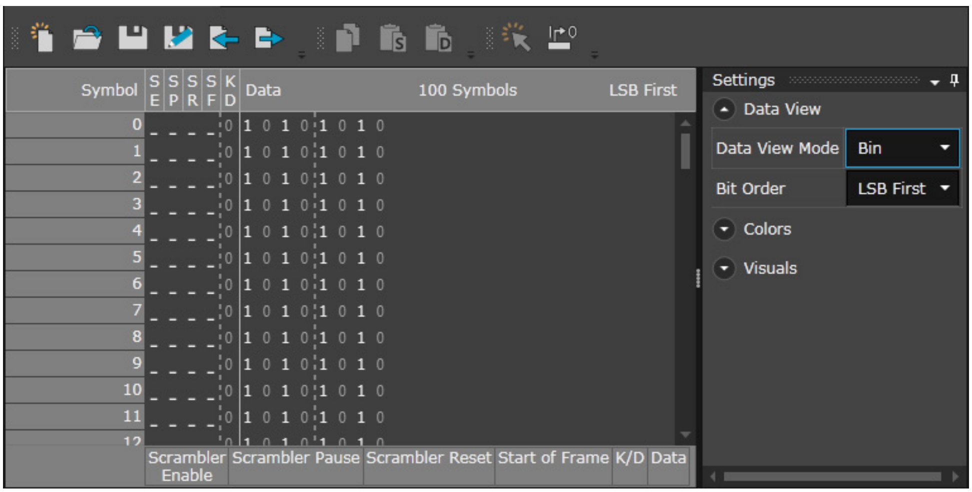
S - Click this toggle button to add/remove squelch on a data.

You can use the  buttons as well as the keyboard keys to move the pointer position to the left, right, up and down in the pattern.

In binary mode, when you click on any symbol, its position is displayed as following:

- The top of the **Pattern Editor Pane** displays the index of cursor (symbol) in current row.
- The bottom of the **Pattern Editor Pane** displays the symbol number.

The following figure shows the **Pattern Editor Pane** with a symbol coded (8B/10B) coded pattern:



You can use the **Settings Window** to enhance the visualizations of the current pattern. For details, refer to "Settings Window" on page -263.

Symbol Attributes

A symbol can have additional attributes to modify behavior. They are supported by all symbols. These are:

1. **Mask**: This attribute only affects the **Data In** ports. It specifies if the symbol is actually compared or masked (excluded from compare).
2. **Squelch**: This attribute only affect the **Data Out** ports. If this attribute is 1, a squelch (out of band) level is used.

In addition there are coding specific attributes, to control aspects of the coding like bypassing or using a scrambler.

Editing a Pattern

To edit a pattern:

1 Open an already existing pattern. Refer to Opening Existing Patterns on page 254.

2 If no pattern is available, a new pattern can be created. Refer to Creating New Patterns on page 253.

3 Select the bits to be edited in the patterns

4 Use the keyboard to enter the required bits. It can be in form of 0 or 1.

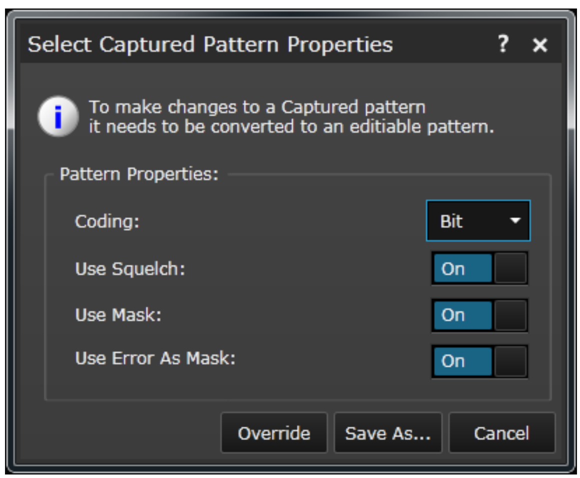
Editing a Captured Pattern

To edit a captured pattern, it needs to be converted to an editable pattern. Follow the given steps make the captured pattern editable:

1 Open an already saved captured pattern. Refer to Opening Existing Patterns on page 254.

2 Select the bits to be edited in the patterns

3 Use the keyboard to enter the required bits. It can be in form of 0 or 1. A **Select Captured Properties** dialog will appear:



4 Change the properties of current pattern. Using this option, you can change the pattern coding and enable/disable **Mask**, **Squelch** and **Error as Mask**.

The **Use Error As Mask** option is only visible when the **Use Mask** option is enabled and coding is same as captured pattern.

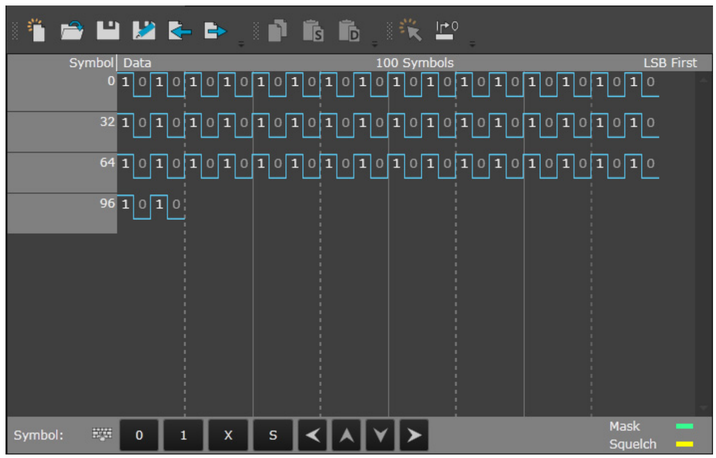
5 Click **Override** to remove all errors from the pattern.

6 Click **Save As**. A Save Pattern As dialog will appear.

7 Specify the name and location and click **Save**.

Bit Coding

The following figure shows the pattern edit pane with a bit coded pattern in binary (BIN) mode:



The bottom of the pattern edit pane indicates the position of the selected symbols and buttons to edit them. These buttons are only available when you create a pattern in bit coding and the data view mode is binary (BIN). You can use these buttons to:

0 - Set data 0

1 - Set data 1.

X - Click this toggle button to add/remove mask on a data.

S - Click this toggle button to add/remove squelch on a data.

You can use the  buttons as well as the keyboard keys to move the pointer position to the left, right, up and down in the pattern.

The bottom of the pattern edit pane also displays the symbol number which indicates the number.

You can use the **Settings Window** to enhance the visualizations of the current pattern. For details, refer to "Settings Window" on page -263.

To encode a bit coded symbol 1, 2 or 3 bits are needed. This depends on the use of mask and squelch:

Table 44 Plain bit coding without using mask or squelch

Bit offset range	Bit (range) name	Description
0	Data	Data bit

Table 45 Mask is used

Bit offset range	Bit (range) name	Description
0	Data	Data bit
1	Mask	Mask (ignored on DataOut)

Table 46 Squelch is used

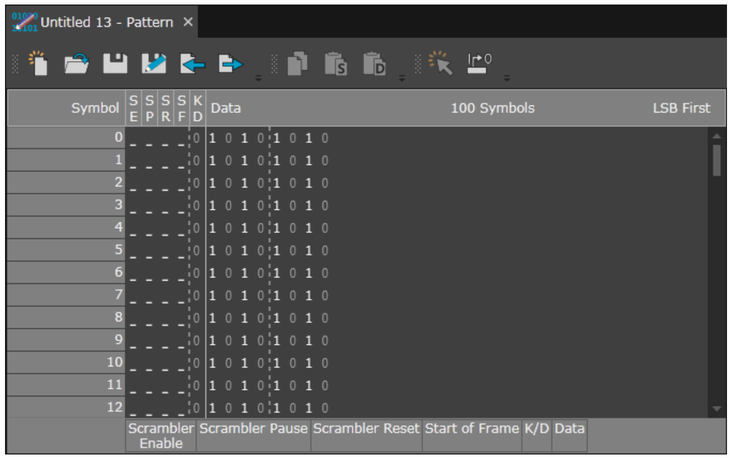
Bit offset range	Bit (range) name	Description
0	Data	Data bit
1	Squelch	Squelch (ignored on DataIn)

Table 47 Mask and Squelch are used

Bit offset range	Bit (range) name	Description
0	Data	Data bit
1	Mask	Mask (ignored on DataOut)
2	Squelch	Squelch (ignored on DataIn)

8B/10B Coding

The following figure shows the pattern edit pane with a 8B/10B coded pattern:



You can use the **Settings Window** to enhance the visualizations of the current pattern. For details, refer to Settings Window on page 263.

This type of pattern contain:

- **Symbol**: 8B/10B coding have 8 bits in one symbol.
- **Attributes**: It has the following attributes:
  - **Mask** - This attribute only affects the **Data In** ports.
  - **Squelch** - This attribute only affects the **Data Out** ports.
  - **Scrambler Enable** - This attribute enables/disables the scrambler.
  - **Scrambler Pause** - This attribute is used to pause scrambler for some special symbol.
  - **Scrambler Reset** - This attribute is used to reset scrambler LFSR (Linear feedback shift register).
  - **Start of frame** - This attribute enables/disables the start of frame marker.
  - **K/D** - This attribute specifies whether the symbol bits are control characters or data.
- **Data** - Specifies data bits.

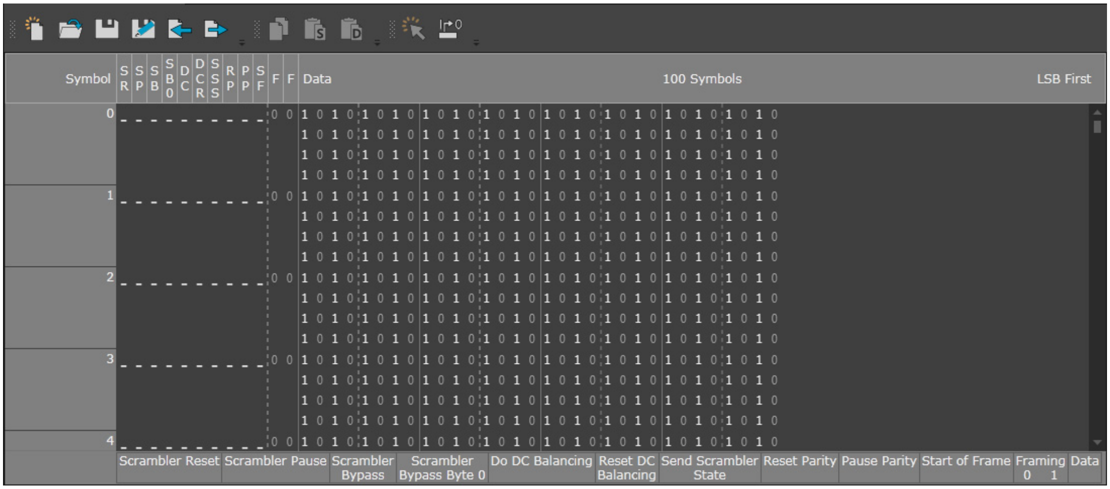
To encode an 8B/10B symbol, 16 bits (2 bytes) are used:

Table 48

Bit offset range	Bit (range) name	Description
7:0	Data	Symbol data
8	K/D	0 = D-character, 1 = K-character
9		Reserved for future use. Must be set to 0
10	Mask	Mask (if present, ignored on DataOut)
11	Squelch	Squelch (if present, ignored on DataIn)
12	Enable Scrambler	Enable Scrambler (ignored on DataIn)
13	Pause Scrambler	Pause Scrambler (ignored on DataIn)
14	Reset Scrambler	Reset Scrambler (ignored on DataIn)
15	Start of Frame	Start of Frame (ignored on DataOut)

128B/130B Coding

The following figure shows the pattern edit pane with a 128B/130B coded pattern:



You can use the **Settings Window** to enhance the visualizations of the current pattern. For details, refer to Settings Window on page 263.

This type of pattern contain:

- **Symbol**: 128B/130B coding having 128 bit in one symbol.
- **Attributes**: It has the following attributes:
- **Mask** - This attribute only affects the **Data In** ports.
- **Squelch** - This attribute only affects the **Data Out** ports.
- **Scrambler Enable** - This attribute enables/disables scrambler.
- **Scrambler Pause** - This attribute is used to pause the scrambler for some special symbol.
- **Scrambler Reset** - This attribute is used to reset the scrambler LFSR (Linear feedback shift register).
- **Scrambler Bypass** - This attribute is used to bypass the scrambling over the symbol.
- **Scrambler Bypass Byte 0** - If this bit is enabled, it does not allow scrambling over the symbol.
- **Do DC Balancing** - This attribute is used to set DC balancing over symbol.

    - **Reset DC Balancing** - This attribute resets the DC balancing state.
    - **Send Scrambled State** - This attribute specifies whether to send the scrambled state.
    - **Reset Parity** - This attribute is used to reset the parity bit.
    - **Pause Parity** - This attribute is used to pause the parity bit.
    - **Start of frame** - This attribute enables/disables the start of frame marker.
    - **Framing** - This attribute enables/disables the start of frame marker.
- **Data** - Specifies the data bits.

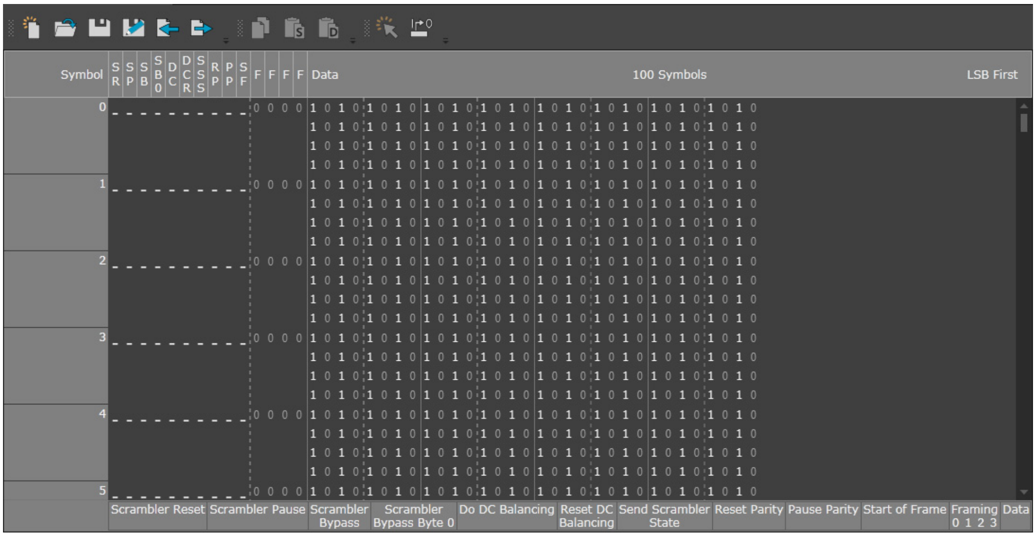
To encode a 128B/130B symbol, 144 bits (18 bytes) are used:

Table 49

Bit offset range	Bit (range) name	Description
1:0	Framing	Framing bits
129:2	Data	Data
130	Mask	Mask (if present, ignored on DataOut)
131	Squelch	Squelch (if present, ignored on DataIn)
132	Reset Scrambler	Scrambler reset (ignored on DataIn)
133	Pause Scrambler	Scrambler pause (ignored on DataIn)
134	Bypass Scrambler	Scrambler bypass
135	Bypass Byte 0 Scrambler	Scrambler bypass byte 0
136	Do DC Balancing	Do DC balancing (ignored on DataIn)
137	Reset DC Balancing	Reset DC balancing (ignored on DataIn)
138	Send Scrambler State	Send scrambler state (ignored on DataIn)
139	Reset Parity	Reset Parity (ignored on DataIn)
140	Pause Parity	Pause Parity (ignored on DataIn)
141	Start of Frame	Start of Frame (ignored on DataOut)
143:142		Reserved for future use. Must be set to 0

128B/132B Coding

The following figure shows the pattern editor pane with a 128B/132B coded pattern:



You can use the **Settings Window** to enhance the visualizations of the current pattern. For details, refer to Settings Window on page 263.

This type of pattern contain:

- **Symbol**: 128B/132B coding having 128 bit in one symbol.
- **Attributes**: It has the following attributes:
  - **Mask** - This attribute only affects the **Data In** ports.
  - **Squelch** - This attribute only affects the **Data Out** ports.
  - **Scrambler Reset** - This attribute is used to reset the scrambler LFSR (Linear feedback shift register).
  - **Scrambler Pause** - This attribute is used to pause the scrambler for some special symbol.
  - **Scrambler Bypass** - This attribute is used to bypass the scrambling over the symbol. If this bit is enabled, it will not allow scrambling over the symbol.
  - **Scrambler Bypass** Byte 0 - If this bit is enabled, it does not allow scrambling over the symbol.

  - **Do DC Balancing** - This attribute is used to set DC balancing over the symbol.
  - **Reset DC Balancing** - This attribute resets the DC balancing state.
  - **Send Scrambled State** - This attribute specifies whether to send the scrambled state.
  - **Reset Parity** - This attribute is used to reset the parity bit.
  - **Pause Parity** - This attribute is used to pause the parity bit.
  - **Start of frame** - This attribute enables/disables the start of frame marker.
  - **Framing 0 1 2 3** - This attribute enables/disables the start of frame marker.
- **Data** - Specifies the data bit.

You can edit selected data using the keyboard.

To encode a 128B/132B symbol, 144 bits (18 bytes) are used:

Table 50

Bit offset range	Bit (range) name	Description
3:0	Framing	Framing bits
131:4	Data	Data
132	Mask	Mask (if present, ignored on DataOut)
133	Squelch	Squelch (if present, ignored on DataIn)
134	Reset Scrambler	Scrambler reset (ignored on DataIn)
135	Pause Scrambler	Scrambler pause (ignored on DataIn)
136	Bypass Scrambler	Scrambler bypass
137	Bypass Byte 0 Scrambler	Scrambler bypass byte 0
138	Do DC Balancing	Do DC balancing (ignored on DataIn)
139	Reset DC Balancing	Reset DC balancing (ignored on DataIn)
140	Send Scrambler State	Send scrambler state (ignored on DataIn)

Bit offset range	Bit (range) name	Description
141	Reset Parity	Reset Parity (ignored on DataIn)
142	Pause Parity	Pause Parity (ignored on DataIn)
143	Start of Frame	Start of Frame (ignored on DataOut)

Pattern Capture

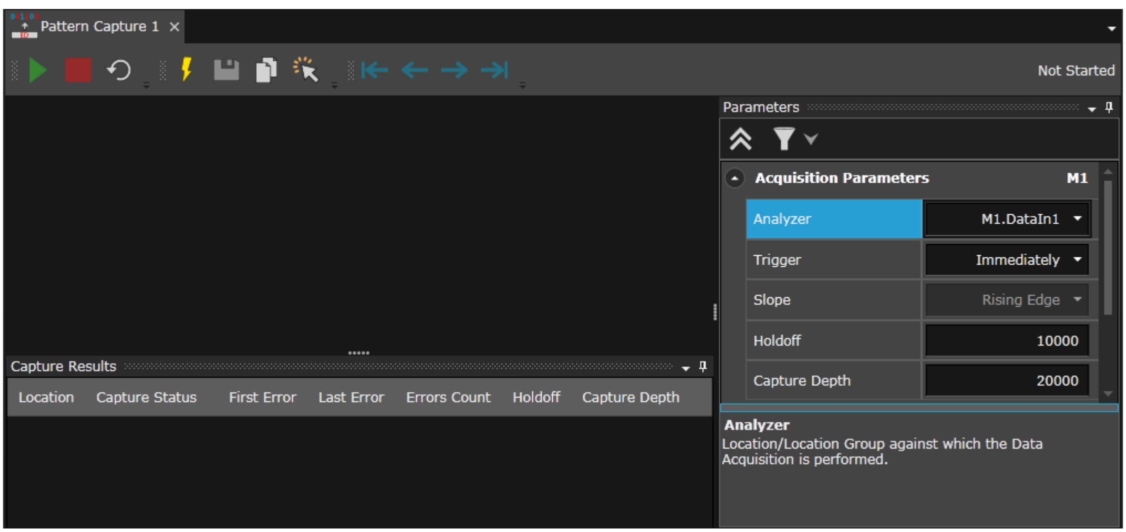
The M8020A/M8030A Analyzer captures the data received from the device under test. The captured data bits are displayed in the pattern capture pane in binary or 8b/10b symbol coding. The received data is compared with the expected data and the errored bits/symbols are highlighted. The captured data can be saved for post processing. The maximum capture memory is
2 Gb. However, it also depends on the holdoff length which represents the amount of symbols in which the trigger events will be ignored.

How to Launch Pattern Capture Window

To launch the **Pattern Capture** window:

- Go to the **Menu Bar** > **Patterns** and then select **Pattern Capture**.

The **Pattern Capture** window will appear as shown in the following figure:



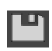
The **Pattern Capture** window includes the following elements:

- Toolbar
- Pattern Captured Pane
- Captured Results
- Parameters Window

Toolbar

The toolbar provides the following convenient pattern capture functions:

Table 51

Elements	Name	Description
	Start	Starts capturing the current pattern
	Stop	Stops capturing the current pattern
	Reset	Resets the values of acquisition parameters
	Manual Trigger	Manually starts the event that is triggering the capture logic.
	Save Captured Data	Opens the “Save As” dialog which allows to save the captured data patterns as a separate pattern in the workspace (shared/current).
	Copy	Copies the selected data.
	Save all Analyzers Captured Data	Opens the “Save As” dialog which allows to save all analyzers captured data patterns as a separate pattern in the workspace. This button will only be available when data acquisition is perform using the location group.
	Error Navigation Buttons	▪ First Error - Takes to the first errored bit in the captured pattern. ▪ Previous Error - Takes to the previous errored bit before the current one in the captured pattern. ▪ Next Error - Takes to the next errored bit after the current one in the captured pattern. ▪ Last Error - Takes to the last errored bit in the captured pattern.

Parameters Window

The **Parameters** window have the following acquisition and show parameters for pattern capture.

Acquisition Parameters

- **Analyzer** - Use this drop-down menu to select the channel against which the data capture has to be performed.
- **Trigger** -Selects the event that is triggering the captured logic. It can be captured on the following four stop events:
  - **Immediately** – Starts capturing the data immediately and displays the captured data.
  - **Error** – Starts capturing the data when it receives an errored bit and displays the captured data.
  - **CTRL IN A** – Waits for a trigger signal from CTRL IN A port and displays the captured data.
  - **CTRL IN B** – Waits for a trigger signal from CTRL IN B port and displays the captured data.
- **Slope** - Selects the edge (rising edge or falling edge) of CTRL IN A or CRL IN B that is triggering the captured logic.
- **Holdoff** - Defines the minimum of data bits to capture before the trigger event occurs. This value is adjusted to a multiple of the current symbol granularity.
- **Capture Depth** - Defines the minimum of data bits to capture including holdoff. This value is adjusted to a multiple of the current symbol granularity.

Show Parameters

- **View Bit Pattern As** - Displays the captured bit coed pattern in Binary, Hex or Symbol view.
- **View Coded Pattern As** - Display the coded pattern (8B/10B) in Binary, Hex or Symbol view.

How to Capture a Pattern

The following example explains the steps to capture a pattern to display errored bits.

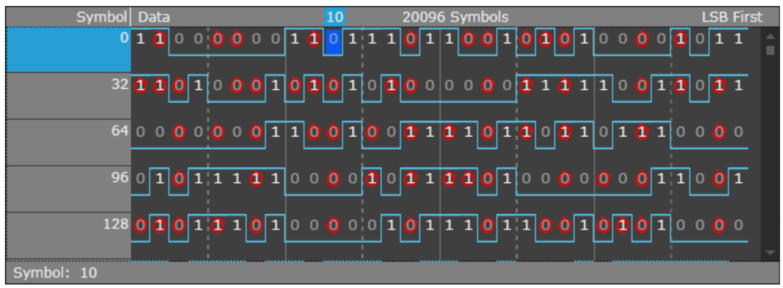
1 Switch to **Parameters** window.

2 Set the acquasation parameters are shown below:

- Location: *M1.DataIn1*
- Trigger: *Error*
- Holdoff: *500*
- Capture Depth: *10000*

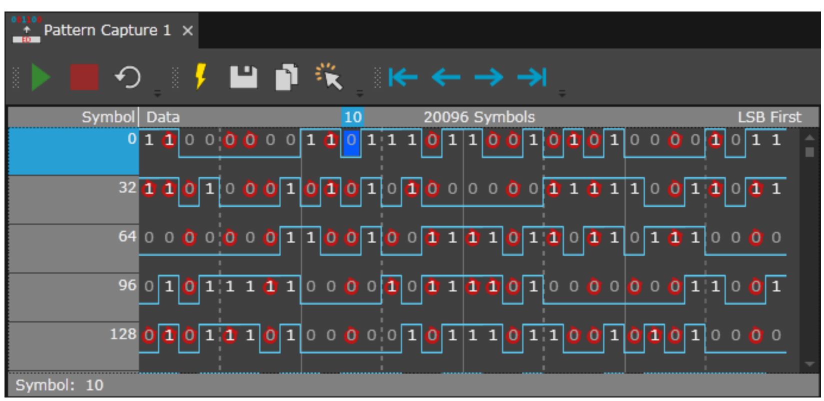
3 Click **Start** button. A message “Waiting for Trigger” will appear. You can click **Manual Trigger** to manually trigger the captured logic.

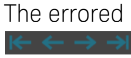
4 The captured data will be displayed in the **Pattern Captured** pane. The errored bits are marked red as shown in the following figure:



Pattern Capture Pane

The **Pattern Capture Pane** displays the captured pattern in the binary or 8b/10b symbol coding.



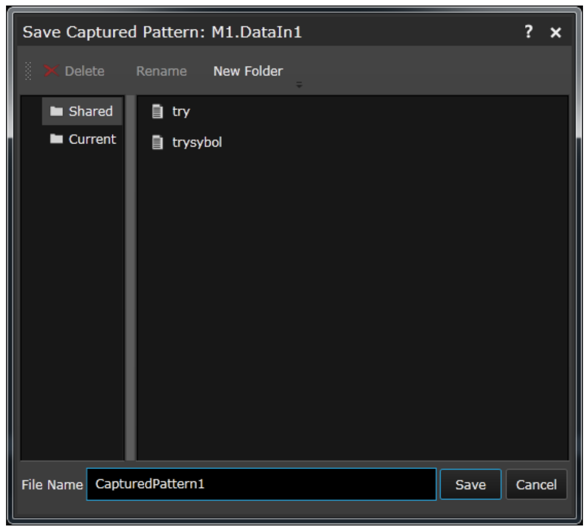
The errored bits in the captured patterns are marked red. You can use the  **Error Navigation** buttons to jump to next, previous, first or last error.

Please note that the **Pattern Capture** pane does now allow to editing feature. This means you cannot edit the captured pattern. However, if you want to edit a captured pattern, you have to first save it and then open in the **Pattern Editor**. For details on how to edit pattern, refer to Editing a Pattern on page 269. Using a **Pattern Editor**, you can also save the results in different encoding schemes.

Saving a Captured Pattern

You can save the captured pattern for post processing. To do so;

- Click **Save** icon. This will open a **Save Captured Pattern** dialog.

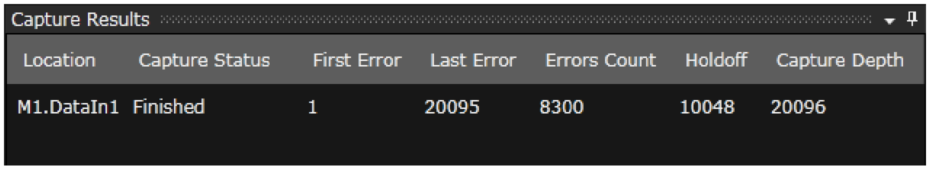


- Specify the folder where it will be saved.
- Enter a file name and click **Save**. The current pattern will be saved under the filename.

You can open the saved file it in the **Sequence Editor** and then download it to module to create a sequences. For details, refer to Sequence Editor on page 227.

Capture Results Pane

The **Capture Results** pane displays the results of the patterns captured.



The results are summarized as following:

- **Location** - Displays the location on which pattern capture is performed.
- **Capture Status** - Displays the current status of the capturing event as 'Finished', 'Stopped' or 'Waiting for Trigger'.
- **First Error** - Displays the position of first error bit.
- **Last Error** - Displays the position of last error bit.
- **Errors Count** - Displays the total number of errored bits.
- **Holdoff** - Displays the minimum number of data bits to capture before the trigger event occurs.
- **Capture Depth** - Displays the minimum number of data bits to capture including holdoff.

Interactive Link Training

PCI Express 3.0 Testing

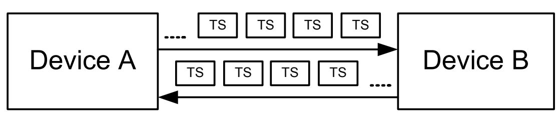
It provides the following types of testing:

- Transmitter Testing for PCIe 3.0
- Receiver Jitter Tolerance Testing for PCIe 3.0
- Tx/Rx Link Equalization Testing for PCIe 3.0
- PLL Loop Bandwidth Testing for PCIe
- Differential Impedance Testing

LTSSM (Link Training and Status State Machine)

Link Training and Status State Machine (LTSSM) is the sub-block that drives and controls the link initialization, and training process for a PCIe device to enable the normal data exchange between the two devices over the link. LTSSM operates at the physical layer level and exchanges physical layer packets (Ordered sets such as TS1 and TS2) to initialize, train, and manage the link.

Two PCIe 3.0 instruments exchange Training Sequences as shown in the following diagram:



Training Sequences are also used to switch the link to low power states.

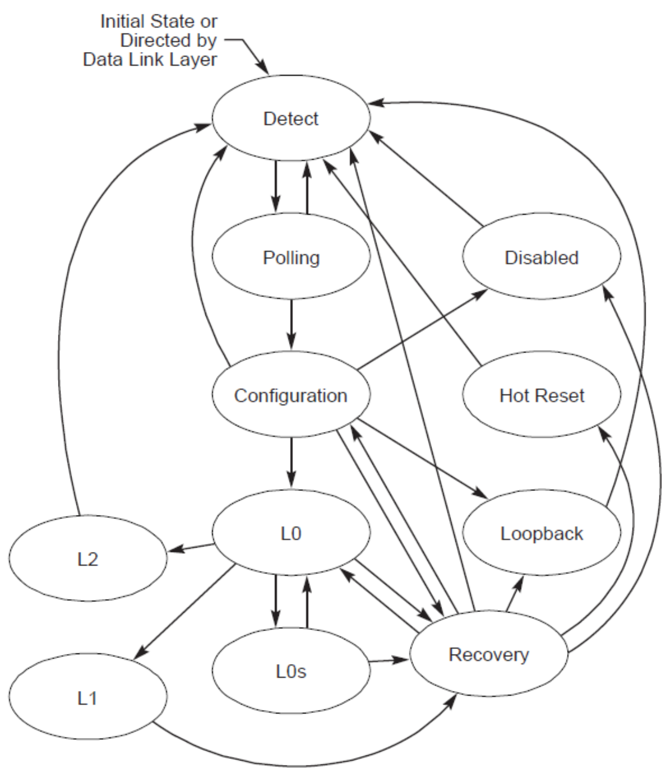
LTSSM States

LTSSM transits through various states and substates during link initialization, training, and management. The entry and exit of each of these states and the exchange of packets (Ordered sets) between the two devices during each state is as per the PCIe specifications.

As per the PCIe specifications, LTSSM has 11 states and further sub-states. These states are referred to frequently in this chapter to describe LTSSM testing using the Keysight LTSSM Tester tool. Following is a list of these states followed by a diagram to illustrate the sequence of these states as per the PCIe specifications.

- Detect
- Quiet
- Polling
- Configuration
- L0
- Recovery
- L0s
- L1
- L2
- Hot Reset
- Disabled
- Loopback

The following figure displays the top-level states of LTSSM.



The Link Training starts in state “Detect”. The purpose of this state is to detect when a far end termination is present. In the “Polling” state, bit lock and Symbol lock are established and Lane polarity is configured. In “Configuration”, both the Transmitter and Receiver are sending and receiving data at the negotiated data rate. In “Recovery”, both the Transmitter and Receiver are sending and receiving data using the configured Link and Lane number as well as the previously supported data rate(s). An active link that can transport transaction layer packets is in state “L0”. All power management states are entered from this state.

The intent of the “Disabled” state is to allow a configured Link to be disabled until directed or Electrical Idle is exited (i.e., due to a hot removal and insertion) after entering “Disabled”. The “Loopback” is intended for test and fault isolation use.

LTSSM Tests for PCIe 3.0

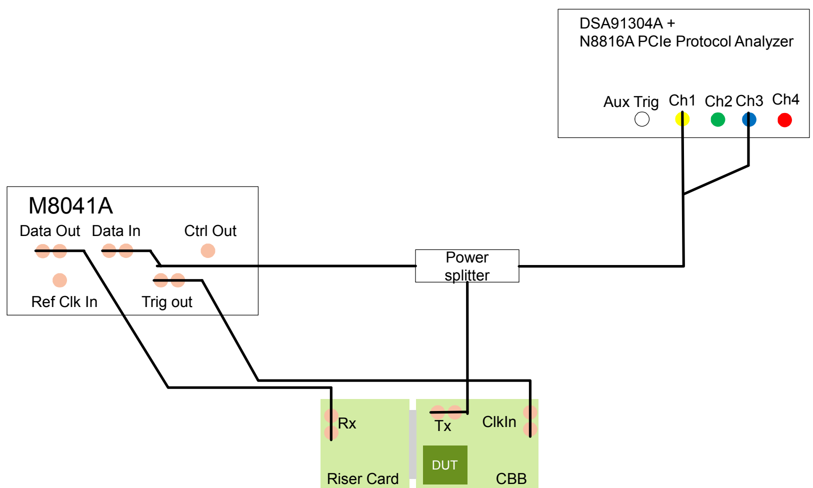
You can execute the following types of LTSSM tests for PCIe 3.0:

Table 52

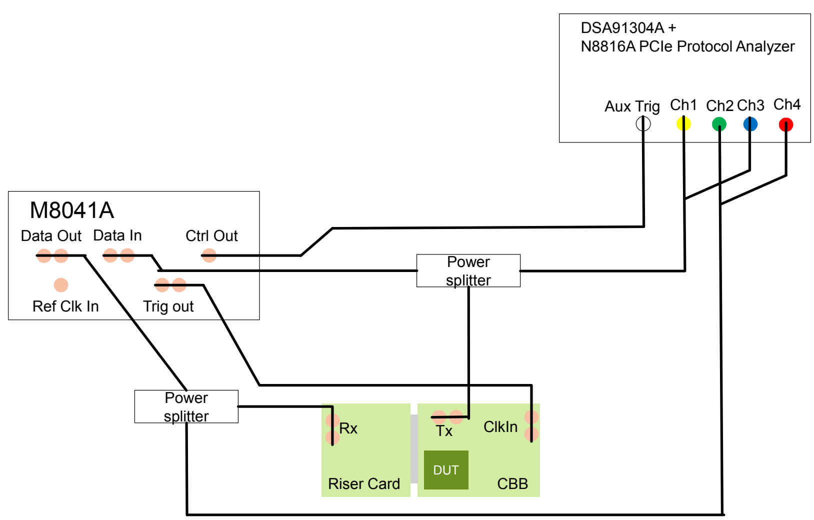
Test Number	Test Name
2.3	Add-in Card Transmitter Initial Tx EQ Test for 8.0GT/s
2.4	Add-in Card Transmitter Link Equalization Response Test for 8GT/s
2.7	System Board Transmitter Link Equalization Response Test for 8GT/s
2.10	Add-in Card Receiver Link Equalization Test for 8GT/s
2.11	System Board Receiver Link Equalization Test for 8GT/s

Test Setup

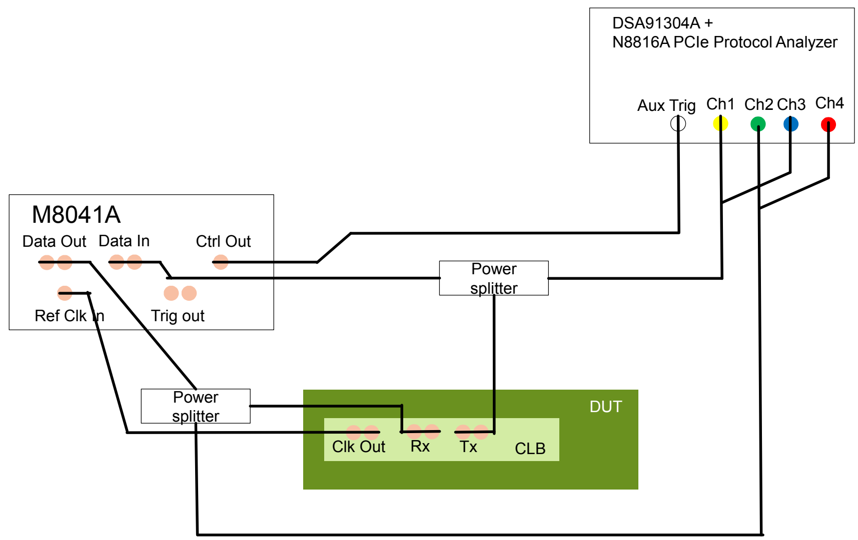
The following block diagram shows the setup for test 2.3:



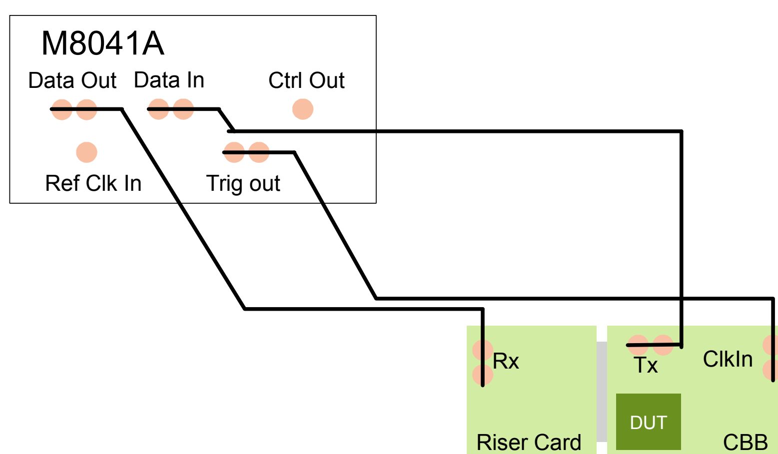
The following block diagram shows the setup for test 2.4:



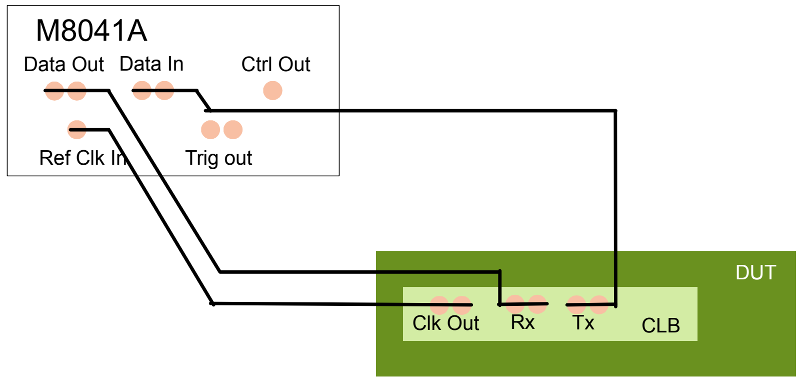
The following block diagram shows the setup for test 2.7:



The following block diagram shows the setup for test 2.10:



The following block diagram shows the setup for test 2.11:



Link Training Configuration

You can configure the link training parameters to control LTSSM using the **Sequence Settings** window. For details on link training parameters, refer to Link Training Configuration on page 236.

Test Procedure

Follow the given steps to perform link training for test 2.3:

1. In the M8070A software load the instrument state **PCIe_PHY2.3_Test**.
2. Power on the DUT.
3. Enable the **Outputs** of the M8041A.
4. Reset the DUT by pressing the reset button on the CBB.
5. With this setup everything is prepared for capturing the data for the P0 preset. Hit the **Break** button in the **Sequence Editor** to start the link training. If link training was successful the **Status Indicator** should show that the sequencer of the generator is executing block 4 or 5 and that there are no bit errors.
6. Set up the oscilloscope.
7. For shutting down the PCIe link hit the **Break** button again. The sequencer of the generator should return to block 2.
8. Before taking the measurement for P1 the test pattern has to be changed. For this select block 4 of the generator sequence. Under **Sequence Settings** > **Block Data** change the memory pattern to compliancePatternHeaderP1Lane0. This pattern can be found in the

Current directory. Not changing the pattern will cause the SIG test software to show the wrong preset number, but won't affect the test results otherwise.

9 Do the same for block 4 of the error detector sequence.

10 Under **Sequence Settings** > **Instrument Configuration** > **Link Training PCIe 3.0** change the **DUT Initial Preset** to P1.

11 Download the changed sequence.

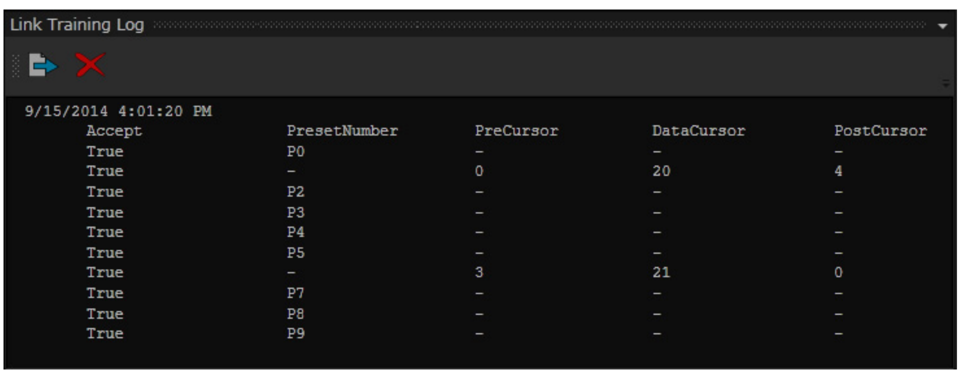
12 Repeat the test as for P0 and then repeat the test procedure for P2 through P9.

13 Start the SIG software and load all captured files into the **Preset Test**.

Link Training Log

The **Show Link Training Log** window displays the log generated while initiating the link training test. All information regarding the executed tests and their status are displayed with the date and time stamp. This helps to identify the root cause of a problem. For instance, the **Show Link Training Log** shows test failure due to an unexpected LTSSM state change.

The following figure shows the log generated by the **Show Link Training Log** window.



Accept	PresetNumber	PreCursor	DataCursor	PostCursor
True	P0	-	-	-
True	-	0	20	4
True	P2	-	-	-
True	P3	-	-	-
True	P4	-	-	-
True	P5	-	-	-
True	-	3	21	0
True	P7	-	-	-
True	P8	-	-	-
True	P9	-	-	-

You can open/close the **Show Link Training Log** window by clicking on the **Show Link Training Log** icon present in the status bar. The **Show Link Training Log** icon with orange background indicates an update or new entry in the link training log.

User Calibrated Presets

It specifies whether BERT's Data Out should use user-calibrated presets or standard presets during link training. Turning it on means that BERT's Data Out will use de-emphasis/pre-shoot values that had been previously calibrated, otherwise it will use standard de-emphasis/pre-shoot values defined by the PCIe3 specification (defined below). By default this option is turned off which means that standard presets will be used.

The following table shows standard preset from PCIe3 specification:

Preset Number	De-emphasis (dB)	Preshoot (dB)
P0	-6.0	0.0
P1	-3.5	0.0
P2	-4.5	0.0
P3	-2.5	0.0
P4	0.0	0.0
P5	0.0	2.0
P6	0.0	2.5
P7	-6.0	3.5
P8	-3.5	3.5
P9	0.0	3.5

Keysight M8070A System Software for M8000 Series of BER Test Solutions

User Guide

8 Working with Measurements

KEYSIGHT TECHNOLOGIES

Overview

The M8070A software provides the following measurements:

- Error Ratio
- Output Timing
- Output Level
- Jitter Tolerance
- Eye Diagram

Exploring Measurement User Interface

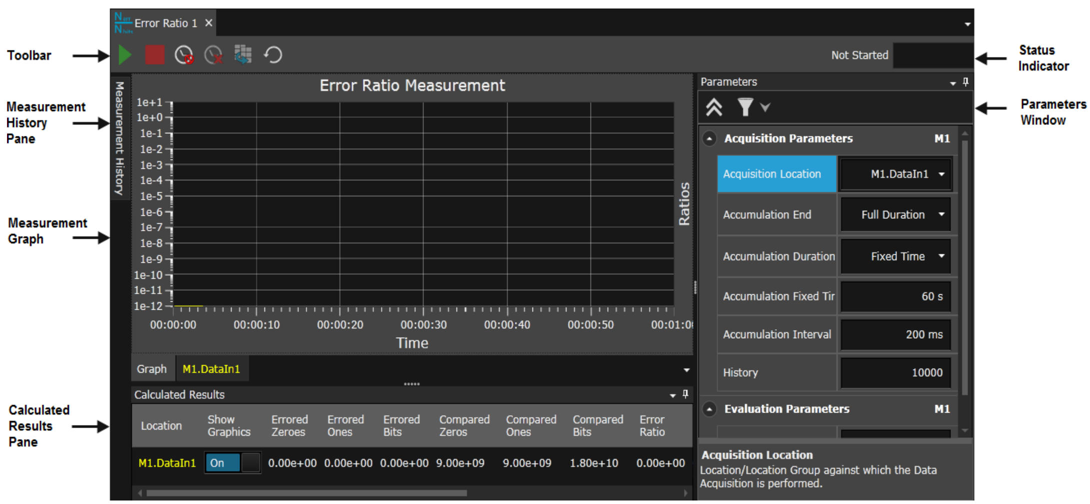
This section describes the functionality provided by the measurement user interface.

Launching the Measurement User Interface

To launch the measurement user interface:

- Go to the **Menu Bar** > **Measurements** and then select the respective measurements (**Error Ratio**, **Output Timing, Output Level** or **Jitter Tolerance**) to launch the measurement user interface.

The following figure shows the measurement user interface:



The measurement user interface has the following GUI elements which are common to all measurements:

- Toolbar
- Status Indicator
- Measurement History Pane
- Measurement Graph
- Parameter Window
- Calculated Results Pane

Lets discuss these GUI elements in the following sections.

Toolbar

The toolbar contains the following icons:

Table 53

Elements	Name	Description
	Start /Continue Measurement	Starts a measurement.
	Break Measurement	Halts the measurement at that point. Once paused, you can continue the measurement again by pressing **Continue Measurement** icon. Note: This option is not available in **Error Ratio Measurement**.
	Stop Measurement	Stops the measurement.
	Step Into Measurement	Steps further into the measurement. Note: This option is not available in **Error Ratio Measurement**.
	Enable/Disable Measurement Run History	Enables or disables the measurement run history. For details, refer to Measurement History Window on page 297.
	Clear Measurement History	Clears the measurement run history.
	Copy Measurement History Properties	Copies the measurement history properties to the currently running measurement.
	Reset Measurement	Resets the measurement to its default values.

Status Indicator

The status indicator shows the current state of a measurement. There can be various states of a measurement, depending on the type of measurement. These may be as follows:

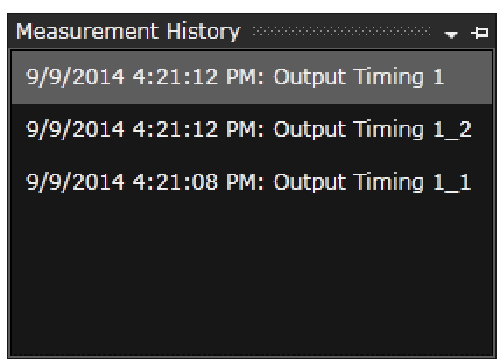
- **Not Started**: Indicates that the measurement has not yet started.
- **Running**: Indicates that the measurement is currently running.
- **Stop**: Indicates that the measurement is stopped.
- **Error**: Indicates an error while executing the measurement which is caused due to invalid parameter settings.
- **Suspended**: Indicates that the measurement is suspended.
- **Finished**: Indicates that the measurement is completed.

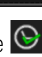
The following figure shows the status indicator while the measurement is running:

Measurement History Window

The **Measurement History** window maintains the history of executed measurement along with their time stamp. This allows you to refer to the previously run measurements and compare their results.

The **Measurement History** window is shown in the figure below:



Click the  icon to toggle between the enable/disable measurement run history in the Measurement History window.

Copy Measurement History Properties

This feature allows you to copy the properties of run measurement to currently running measurement.

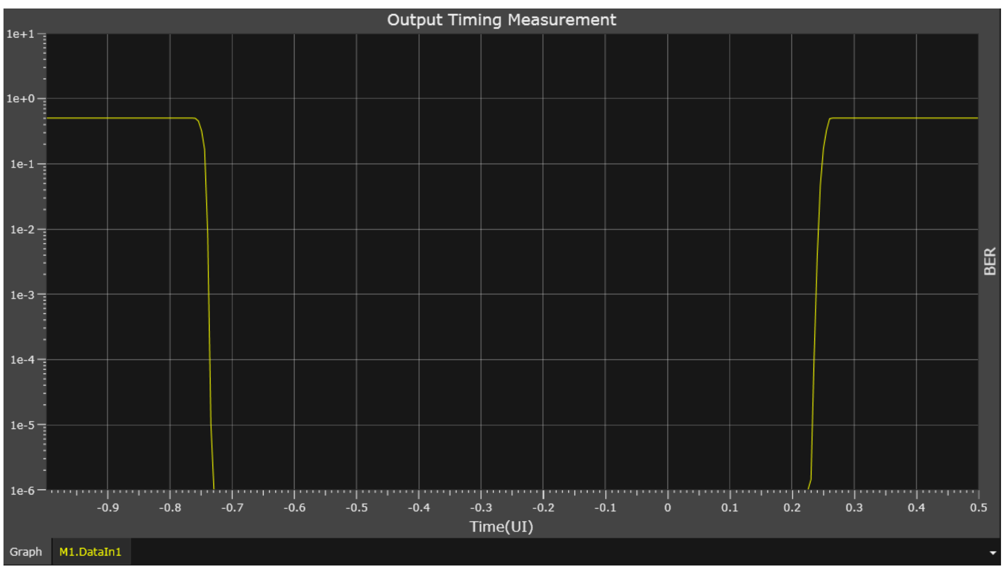
To do so,

- Select a measurement history from the list shown in the **Measurement History** window.
- Click the  **Copy Measurement History Properties** icon. The properties of the selected measurement will be copied to the current measurement.

Measurement Graph

The **Measurement Graph** displays the calculated graph and results.

The following figure displays the **Measurement Graph** of the **Output Timing Measurement**.



The **Measurement Graph** contains the following tabs:

- **Graph**: Displays the graphical representation of the measurement. The graph varies from measurement to measurement. The details of each measurement graph are further described in their respective sections.
- **Location**: Displays the raw measurement data for that location. However, if you are running measurement for a group, multiple tabs will appear that display the raw measurement for each location.

When you right-click on the **Measurement Graph**, a context menu appears which provides the following options:

- **Turn ON/OFF Fit to view (Ctrl+Home)**- Turns ON/OFF **Fit to view option**.
- **Fit to view (Home)** - Makes the visible area fit to display entire contents.
- **Copy screenshot (F11)** - Copies the screenshot of charts to clipboard.
- **Save screenshot (Ctrl+S)** - Saves the screenshot as an image (PNG) under a name.
- **Quick Help (Alt+F1)** - Opens a window that provides brief information about the dynamic display.

Parameters Window

The **Parameter** window allows you to set the parameters for a location or a group. For each measurement, it contains two types of parameters:

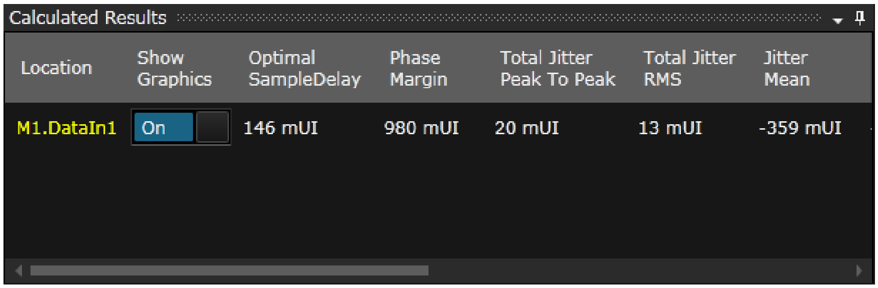
- **Acquisition Parameters** - Pre-Parameters influence how the data for a measurement is collected; changes require a re-run in order to be effective. It also allows you to select a location or location group against which the data acquisition is performed.
- **Evaluation Parameters** - Post-Parameters influence how the collected measurement data is evaluated. Changes do not require a re-run in order to be effective.

The acquisition and evaluation parameters differ from measurement to measurement. The detailed description of these parameters are explained in the sections that follow.

Calculated Results Pane

The **Calculated Results** pane displays the calculated results in the form of measurement parameters for each location. The calculated measurement parameters varies from measurement to measurement.

The **Calculated Results** pane is shown in the following figure:



For each location, you can click on the 

slide button to show/hide the measurement graph.

Error Ratio Measurement

Overview

The **Error Ratio Measurement** allows you to collect measurement data over a specific period. This can be used to create test scenarios that are reproducible and comparable. Also, you can let tests run over long times and then evaluate the results afterwards.

NOTE While the error ratio measurement is running, you should not modify the measurement setup, as the measured bit errors do not represent the performance of your DUT under real circumstances.

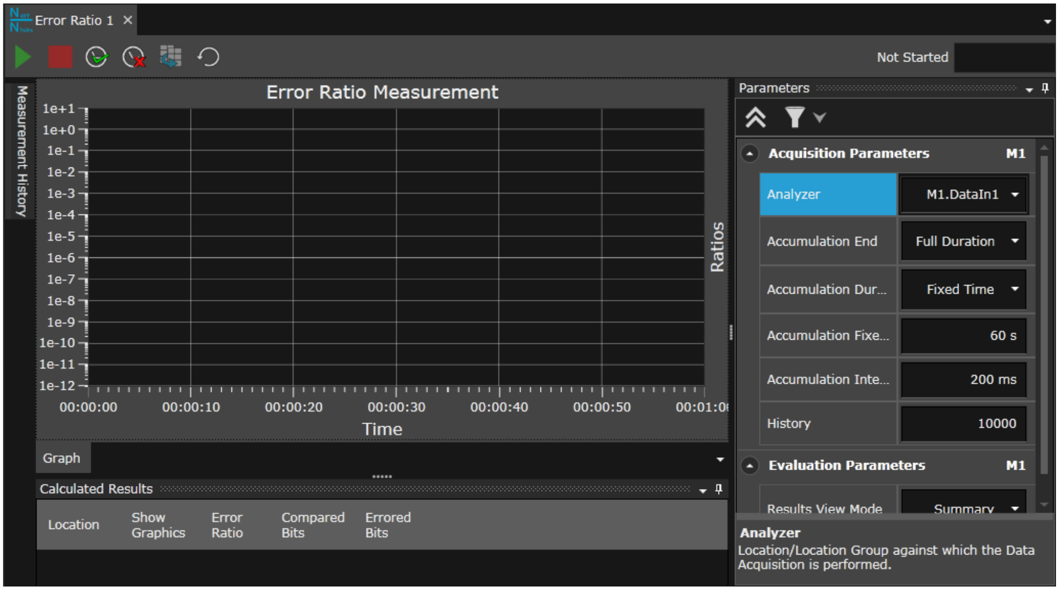
The period of time can be set through the **Parameters** window, an absolute time setting, or the time it takes to measure a specified number of bits or bit errors. The accumulation period should be long enough to make a statistically valid BER measurement.

Launching Error Ratio Measurement

To launch the **Error Ratio Measurement**:

- Go to the **Menu Bar** > **Measurements** and then select **Error Ratio**.

The user interface of **Error Ratio Measurement** will appear as shown in the following figure:



The **Error Ratio** user interface includes the following elements:

- **Toolbar**: For details, refer to Toolbar on page 296.
- **History Pane**: For details, refer to Measurement History Window on page 297.
- **Measurement Graph**: For details, refer to Measurement Graph on page 305.
- **Parameter Pane**: For details, refer to Acquisition and Evaluation Parameters for Error Ratio on page 303.
- **Calculated Results**: For details, refer to Calculated Results on page 306.

Acquisition and Evaluation Parameters for Error Ratio

The **Parameters** window allows you to set the acquisition and evaluation parameters for **Error Ratio** measurement:

Table 54

Parameters	Description	Values
Acquisition Parameters		
Analyzer Location	Location or location group against which the data acquisition is performed.	Use the drop-down list to specify the location or location group.
Accumulation End	Specify the criteria to end accumulation.	Pass/Fail | Full Duration | Number of Bits
Accumulation Duration	Specify the acquisition duration.	Fixed Time | Indefinitely
Accumulation Fix Time	Specifies the duration of the accumulation. This is a conditional parameter and appear when the **Accumulation Duration** as selected as **Fixed Time**.	Min - 1 s Max - 31.5 Ms
Accumulation Interval	Specify the accumulation interval on which the error ratio sample is taken.	Min - 100 ms Max - 2 Ms
Number of Compared Bits	Specify the number of compared bits for which the error ratio sample is taken. This is a conditional parameter and appear when the **Accumulation End** as selected as **Number of Bits**.	Min - 1E+6 Max - 1E+18
Target Error Ratio	Specifies the target error ratio of the accumulation results. This is a conditional parameter and appear when the **Accumulation End** as selected as **Pass/Fail**.	Min - 1E -18 Max -1E-3
Target Confidence Level	Specifies the target confidence level of the accumulation results. This is a conditional parameter and appear when the **Accumulation End** as selected as **Pass/Fail**.	Min - 0.1 % Max - 99.9 %
History	Defines the number of accumulation values that are kept in memory.	Min - 1 Max - 100000

Parameters	Description	Values
Evaluation Parameters		
Results View Mode	Allows to view the calculated results in either detailed or summarized mode.	Detailed | Summary
Display Error Ratio	Displays the error ratio of the accumulated results. This is a conditional parameter and appear when the **Results View Mode** is selected as "Detailed".	Erroneous Zero Ratio | Erroneous One Ratio | Error Ratio

How to Run a Measurement

The **Error Ratio** measurement immediately starts calculating the error ratio as soon as it receives a valid signal and the respective error ratio settings are done.

To run an **Error Ratio** measurement, perform the following steps:

- Use the **Parameters** window to select the location or location group for which the data acquisition has to be performed.
- Set the acquisition parameters in the **Parameters** window. For details, refer to Acquisition and Evaluation Parameters for Error Ratio on page 303.
- Click the  **Start Measurement** icon to run the measurement. The measurement status indicator will indicate **Running**.

NOTE Please note that once you run the measurement you cannot modify the acquisition parameters. However, if you try to modify acquisition parameters by stopping the measurement and then run the measurement, a new instance of measurement will be executed.

How to Stop a Measurement

To stop an **Error Ratio** measurement:

- Click the  **Stop Measurement** icon to stop the measurement.

Measurement Graph

Once you run an **Error Ratio** measurement for a specified duration, the calculated graph and the raw data is shown on the measurement graph as follows:

This graph displays the delta errored 1's ratio, delta errored 0's ratio, and total delta error ratio at data points over the entire accumulation period. The error ratios on the y-axis are set to a range of 1E+0 (100% errors) to 1E-12. The accumulation period is on the x-axis.

Display Change

During accumulation, data will appear to move from left to right on the ratios graph. When the graph is completely filling the display, the x-axis time scale will double. The data graph is then occupying only half of the display and will continue to move to the right again. This will repeat until the accumulation period has ended.

Measurement Data

The following figure shows the raw measurement data for the selected location.

TimeStamp	Error Ratio	Compared Bits	Errored Bits	Error Zero Ratio	Compared Zeros	Errored Zeroes	Error One Ratio
00:00:00:211	1.33e-08	1.06e+09	1.40e+01	7.64e-09	5.24e+08	4.00e+00	1.88e-08
00:00:00:415	5.89e-09	1.02e+09	6.00e+00	9.89e-09	5.06e+08	5.00e+00	1.95e-09
00:00:00:637	9.01e-09	1.11e+09	1.00e+01	1.27e-08	5.51e+08	7.00e+00	5.36e-09
00:00:00:845	9.61e-09	1.04e+09	1.00e+01	7.75e-09	5.16e+08	4.00e+00	1.14e-08
00:00:01:075	1.31e-08	1.15e+09	1.50e+01	8.78e-09	5.70e+08	5.00e+00	1.73e-08
00:00:01:241	1.56e-08	8.33e+08	1.30e+01	1.94e-08	4.13e+08	8.00e+00	1.19e-08
00:00:01:424	1.09e-08	9.16e+08	1.00e+01	1.32e-08	4.54e+08	6.00e+00	8.67e-09

Graph M1.DataIn1

Test Times and Confidence Levels

A true **Error Ratio** measurement must be statistically valid. Because it is not possible to predict with certainty when errors will occur, your device must be tested long enough to have confidence in its **Error Ratio** performance.

Calculated Results

The following results are displayed when the bit coded patterns are loaded in **Sequence Editor**:

Calculated Results

Location	Show Graphics	Error Ratio	Compared Bits	Errored Bits	Error Zero Ratio	Compared Zeros	Errored Zeroes	Error One Ratio	Compared Ones	Errored Ones
M1.DataIn1	On	5.76e-12	1.74e+11	1.00e+00	0.00e+00	8.61e+10	0.00e+00	1.14e-11	8.75e+10	1.00e+00

- **Errored Zeros**: Displays the number of errored zeros during the accumulation period.
- **Errored Ones**: Displays the number of errored ones during the accumulation period.
- **Error Bits**: Displays the number of errored bits during the accumulation period.

- **Compared Zeros**: Displays the number of compared zeros during the accumulation period.
- **Compared Ones**: Displays the number of compared ones during the accumulation period.
- **Compared Bits**: Displays the number of compared bits during the accumulation period.
- **Error Ratio**: Displays the ratio of the number of errors to the number of bits.
- **Error Zero Ratio**: Displays the ratio of the number of error zero to the number of bits.
- **Error One Ratio**: Displays the ratio of the number of errors one to the number of bits.
- **Total Words**: Displays the total number of received sequencer words.
- **Frames**: Displays the number of frames received in a time interval.
- **Errored Frames**: Displays the number of errored frames received in a time interval.
- **Frame Error Ratio**: Displays the ratio of the number of frame errors to the number of frames received in the current (or last completed) accumulation period.
- **ConfidenceLevel@TargetErrorRatio**: Displays the percentage of confidence level achieved at specified target error ratio at certain point of time.
- **ErrorRatio@TargetConfidenceLevel**: Displays the number of error ratio achieved at specified confidence level at certain point of time.
- **Results**: Display the measurement result either **Pass** or **Fail**. It is only available when the “Accumulation End” is selected as “Pass/Fail”.

However, when the symbol coded patterns are loaded in **Sequence Editor**, the following results are displayed:

Calculated Results

Location	Show Graphics	Symbol Error Ratio	Received Symbols	Compared Symbols	Errored Symbols	Illegal Symbol Ratio	Illegal Symbols	Filler Symbol Ratio	Filler Symbols	Disparity Error Ratio	Wrong Disparity
M1.DataIn1	On	9.15e-11	2.19e+10	2.19e+10	2.00e+00	0.00e+00	0.00e+00	0.00e+00	0.00e+00	4.58e-11	1.00e+00

- **Symbol Error Ratio**: Displays the symbol ratio of the number of errored symbols to the total number of symbols received.

- **Received Symbols**: Displays the total number of Received Symbol Count (all incoming symbols excluding incoming filler symbols) received in a time interval.
- **Compared Symbols**: Displays the number of compared symbols considered for the accumulation period.
- **Errored Symbols**: Displays the number of errored symbols measured during the accumulation period.
- **Illegal Symbol Ratio**: Displays the ratio of the number of Illegal Symbol count to the number of symbols received in the accumulation period.
- **Illegal Symbols**: Displays the total number of Illegal Symbols received in the accumulation period.
- **Filler Symbols Ratio**: Displays the ratio of total no of filler symbols to the number of symbols received during the accumulation period.
- **Filler Symbols**: Displays the total no of filler symbols received during the accumulation period.
- **Disparity Error Ratio**: Displays the ratio of the number of illegal disparity change count to the number of symbols received in the accumulation period.
- **Wrong Disparity**: Displays the total number of wrong disparity received in the accumulation period.
- **Error Ratio**: Displays the ratio of the number of errors to the number of bits.
- **Compared Bits**: Displays the number of compared bits during the accumulation period.
- **Errored Bits**: Displays the number of errored bits during the accumulation period.
- **ConfidenceLevel@TargetErrorRatio**: Displays the percentage of confidence level achieved at specified target error ratio at certain point of time.
- **Results**: Display the measurement result either **Pass** or **Fail**. It is only available when the “Accumulation End” is selected as “Pass/Fail”.
- **Total Words**: Displays the total number of received sequencer words.
- **Frame Error Ratio**: Displays the ratio of the number of frame errors to the number of frames received in the accumulation period.
- **Frames**: Displays the number of frames received in a time interval.
- **Errored** Frames: Displays the number of errored frames received in a time interval.
- **ErrorRatio@TargetConfidenceLevel**: Displays the number of error ratio achieved at specified confidence level at certain point of time.

Output Timing Measurement

Overview

The **Output Timing** measurement is used to measure the timing and jitter behavior for a device under test (DUT). It uses a bit error rate (BER) measurement to evaluate the shape of the eye for the output signal of the DUT. It also analyzes the jitter, separates the random jitter and deterministic jitter components, and estimates the total jitter.

A direct result is the determination of the optimum sampling point delay for receiving data from the DUT with maximum confidence.

DUT **Output Timing/Jitter** includes the **Fast Total Jitter** measurement that can be used to measure the total jitter for devices which generate a very low error density in a reasonable time span.

Output Timing Characteristics

The sampling point is swept automatically within a 1.5 clock period to generate a "bathtub" curve. The resulting graph is centered around the optimum sampling point of the port.

In addition, the results are available in a tabular view. If a clock signal is defined, the software measures the data to clock alignment and displays the absolute delay.

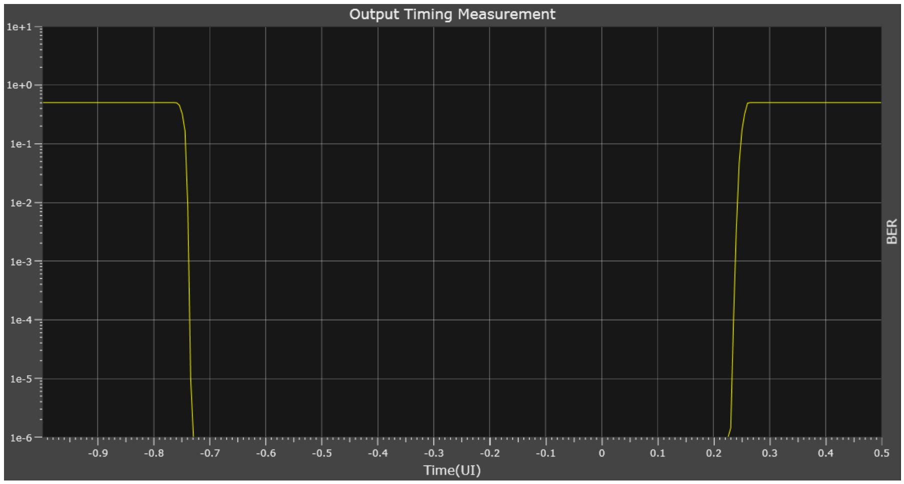
Jitter Characteristics

The DUT Output Timing/Jitter measurement calculates the different components making up the jitter:

- Random Jitter (RJ)
- Deterministic Jitter (DJ)

Example Results

The following illustration shows the resulting graph of a typical DUT **Output Timing** measurement:



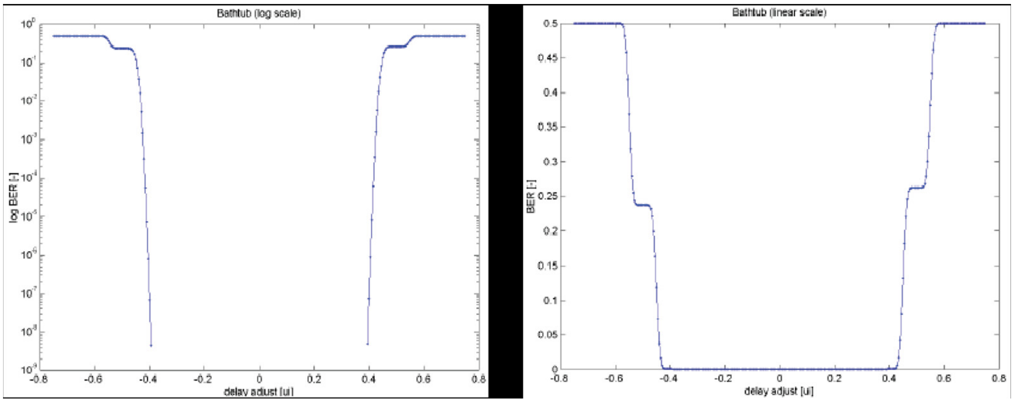
Understanding the Jitter Calculation

After the output timing behavior of the DUT is measured, the M8020A/M8030A calculates the different jitter components:

Random and Deterministic Jitter

To understand the RJ and DJ results, it is helpful to first understand how the software generates the results:

1 The bathtub curve is measured.



2 All measurement points that have BER between the BER Threshold and Minimum BER for RJ/DJ Separation are transformed into Q-space.

The Q-factor describes the signal-to-noise ratio at the decision circuit.

3 Linear regression is performed for both the left and right edges.

4 The mean and sigma are calculated for both lines:

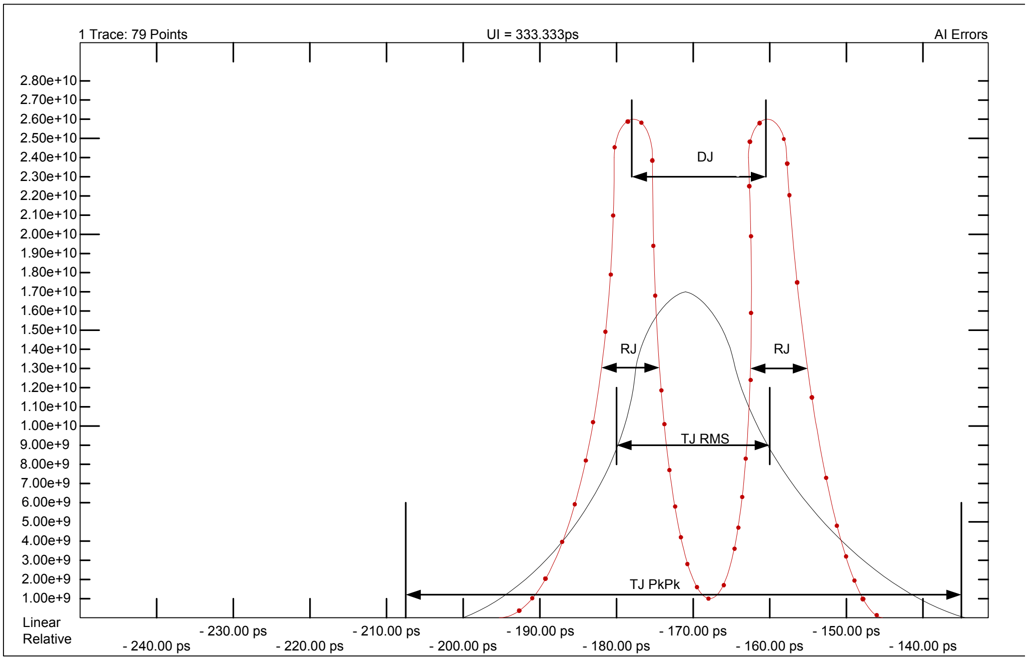
- RJ is calculated as the mean of the two sigmas.
- DJ is calculated as the period minus the difference of the two means.

5 The estimated TJ is calculated:

- Linear regression is used to extrapolate the bathtub curve to lower BER values.
- The intersections of the resulting lines with the Residual BER for RJ/DJ Separation are located.
- The eye opening is calculated.

The estimated TJ is the period minus the width of the eye opening.

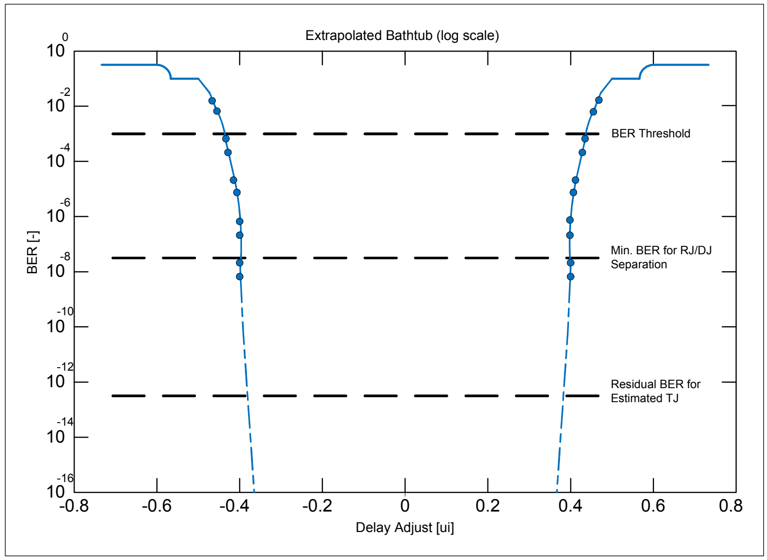
The following illustration shows a jitter curve where both RJ and DJ are present. It also shows how the TJ peak-to-peak and RMS are calculated.



Estimated Total Jitter

The **Estimated Total Jitter** (TJ) allows you to predict the jitter expected for very low bit error rates that would take a long time to measure. It is obtained by extrapolating the measured BER curves.

The TJ is estimated by extending the BER curves (based on the points detected between the BER Threshold and the Minimum BER for RJ/DJ Separation) to the Residual BER for RJ/DJ Separation level. The estimated TJ is the period minus the width of the measured eye.



Explanation of the Fast Total Jitter Measurement

The **Fast Total Jitter** measurement is an optimized method to determine the total jitter for devices that generate a very low error density (BER well below 10^{-10}).

To measure (not estimate) the total jitter for a device with a BER of 10^{-12} with conventional methods, one usually needs to compare more than 10^{12} bits for each sample point. To measure a full eye opening this way with appropriate timely resolution takes time (maybe days or weeks, depending on the data rate), and the probability of seeing one or no error in 10^{12} bits is not higher than 37%.

If one would compare 10^{13} bits for the same device, the probability of seeing ten errors is even lower (12%), but the probability of observing no error is almost zero.

The **Fast Total Jitter** measurement implements a method that reduces the measurement time considerably and provides a higher accuracy. It is based on statistical and probability calculations.

Measurement Duration

The duration of a **Fast Total Jitter** measurement depends on:

- the BER threshold
- the bit rate
- the sample delay step size
- the contribution of random jitter
- the contribution of deterministic jitter

Compared to a conservative bathtub measurement, the **Fast Total Jitter** measurement can reduce the measurement time by more than a factor of 40.

Typical test times are approximately 20 minutes at 10 Gbit/s and slightly more than one hour at 2.5 Gbit/s for a measurement at the 10^{-12} BER threshold with an overall confidence level better than 90%.

Acquisition Parameters

You can use the **Properties** window to specify the following acquisition parameters:

Set the criteria for moving to the next sample point:

- Number of Compared Bits

  After this number of compared bits, the measurement stops for the current sample point and moves to the next one.
- Number of Errors

  After this number of errors, the measurement stops for the current sample point and moves to the next one. This allows you to speed up the measurement. You can switch off this option if only the number of compared bits is important.

Set the criteria for the sample delay:

- Sample Delay Resolution

  Specifies the time distance between sampling points. A smaller value yields more sampling points in a unit interval. You can enter the resolution in UI.

  If the resolution used for the measurement is not high enough, the bathtub curve does not clearly show the edges. For example, you may wish to change the Resolution from 0.01 to 0.005 and run the measurement again.
- Sample Delay Optimization

  You can choose between Fast Total Jitter at BER or None.

- Fast Total Jitter at BER

  This enables the **Fast Total Jitter** measurement. Before enabling this measurement, you need to know the BER floor of the device and to specify a BER threshold that is above that floor.

Evaluation Parameters

You can use the **Properties** window to specify the following evaluation parameters:

- Analyze Errors

  You can analyze for:

  - All Errors

  To calculate the BER values from all bits/errors.

  - Errored Zeros

  To calculate the BER values if "0" is expected, but "1" received.

  - Errored Ones

  To calculate the BER values if "1" is expected, but "0" received.

- BER Threshold

  To calculate the parameters for the given BER threshold. This is the BER level for which output timing numerical values (phase margin, skew, etc.) are calculated. It is also the upper limit of the BER range for RJ/DJ separation.

  The BER threshold influences some of the parameters of the DUT Output Timing measurement. You can also drag and drop the horizontal BER threshold in the graphical display to change this value.

- Min BER for RJ/DJ Separation

  Lower limit of the BER range for RJ/DJ separation.

- Residual BER for Estimated Total Jitter

  BER level for which the estimated total jitter is calculated.

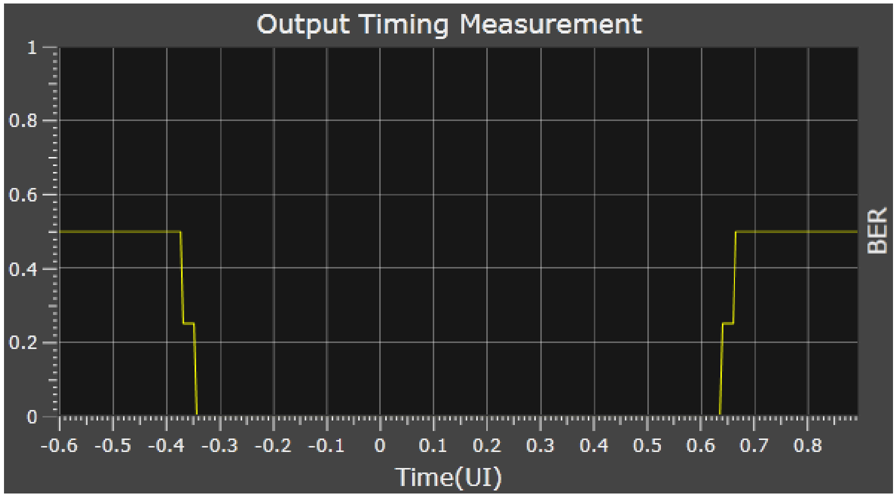
- Display Unit

  Choose between Unit Interval and Seconds to select the timebase for the display's x-axis.

- Vertical Axis Scale

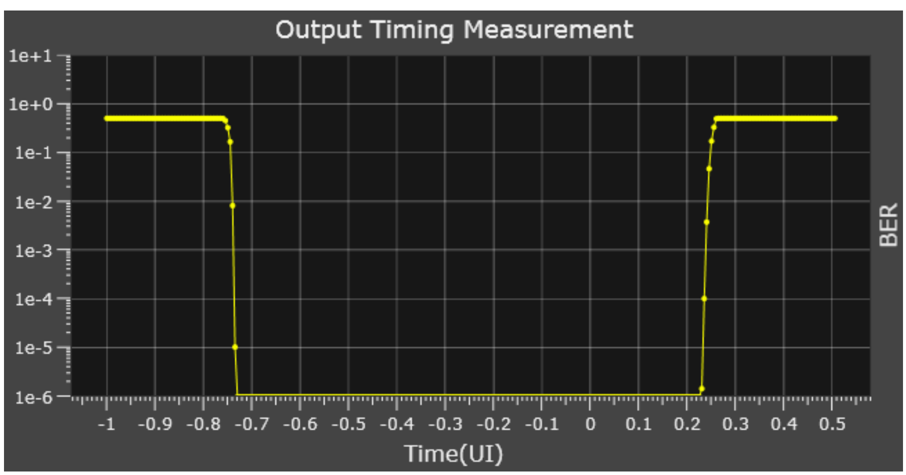
  Choose between Logarithmic and Linear to select the scale for the display’s y-axis.

  For example, a DUT Output Timing measurement displayed on a linear scale may look like this:



- Show Measured Points

  If you want to see the points that have actually been measured, click the **Show Measured Points** slide button.

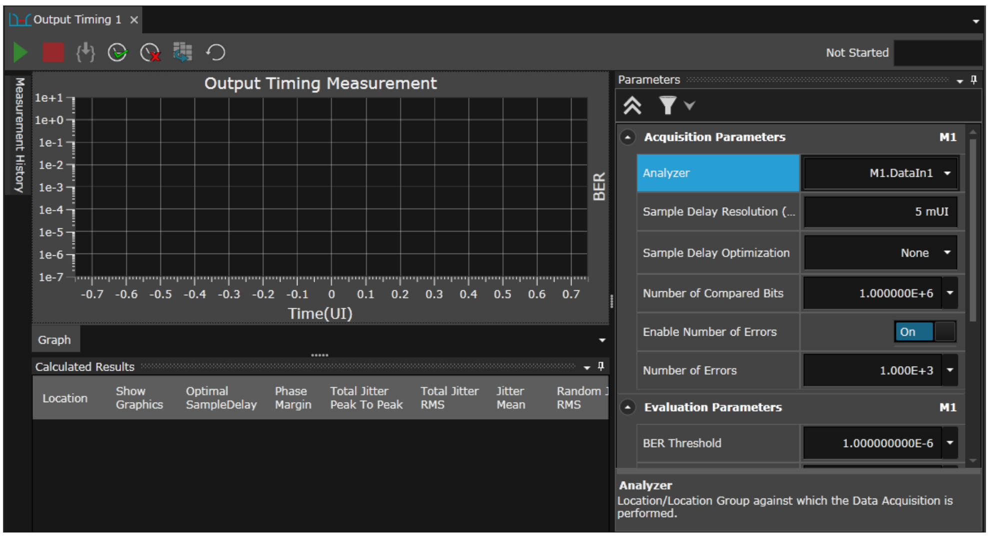


Launching the Output Timing Measurement

To launch the **Output Timing** user interface:

- Go the **Menu Bar** > **Measurements** and then select **Output Timing**.

The **Output Timing** user interface will appear as shown in the following figure:



The **Output Timing** user interface includes the following elements:

- **Toolbar**: For details, refer to Toolbar on page 296
- **History Pane**: For details, refer to Measurement History Window on page 297.
- **Measurement Graph**: For details, refer to Measurement Graph on page 320
- **Parameters Window**: For details, refer to Parameters Pane on page 318.
- **Calculated Results**: For details, refer to Calculated Results on page 321.

Parameters Pane

The **Parameters** window shows the acquisition parameters and evaluation parameters for output timing measurement.

Table 55

Parameters	Description	Values
Acquisition Parameters		
Acquisition Location	Location or Location Group against which the data acquisition is performed.	
Sample Delay	Specify the sample delay resolution.	Min - 5 mUI Max - 200 mUI
Sample Delay Optimization	Specify the sample delay optimization.	None | Fast Total Jitter
Fast Total Jitter at BER	Specifies the Fast Total Jitter BER. This is a conditional parameter and appear when **Sample Delay Optimization** is selected as “Fast Total Jitter”	Min - 1E-15 Max - 1E-9
No. of Compared Bits	Specify the criteria for moving to the next measurement and is only enabled when the Sample Delay Optimization is “None”.	Min - 1 E+0 Max - 1E+18
Enable Number of Errors	Use the slide switch to enable this property. Once this property is enabled, you can specify the additional number of errored bits to consider while moving to the next measurement	ENABLE | DISABLE
Number of Errors	Specify the criteria for moving to the next measurement and is only enabled when the Sample Delay Optimization is “None”. This parameter gets enabled when you enable the “Enable Number of Errors”.	Min - 1 E+0 Max - 1E+18
Evaluation Parameters		
BER Threshold	Specify the BER threshold	Min - 1E-15 Max - 1E-1
Min BER for RJ/DJ Separation	Specifies the minimum BER for random and deterministic jitter components separation.	Min - 1E-18 Max - 1E+0
Residual BER for Estimated Total Jitter	Specify the residual BER for estimated total jitter.	Min - 1 E-12 Max - 1E-6

Parameters	Description	Values
Analyze Errors	Specify options to analyze errors	All Errors | Errored Zeros | Errored Ones
Vertical Axis Scale	Specify the Y axis scale.	Linear | Logarithmic
Display Unit	Specify the unit to be displayed.	Seconds | UI
Show Measured Points	Show/hides the measured points on the graph.	Show | Hide

How to Run Output Timing Measurement

To run an **Output Timing** measurement, perform the following steps:

- Use the **Parameters** window to select the location or location group for which the data acquisition has to be performed.
- Set the acquisition and evaluation parameters for **Output Timing** in the **Parameters** Window. For details, refer to Acquisition Parameters on page 314 and Evaluation Parameters on page 315.
- Click the **Start Measurement** icon to run the measurement. The measurement status indicator will indicate **Running**.

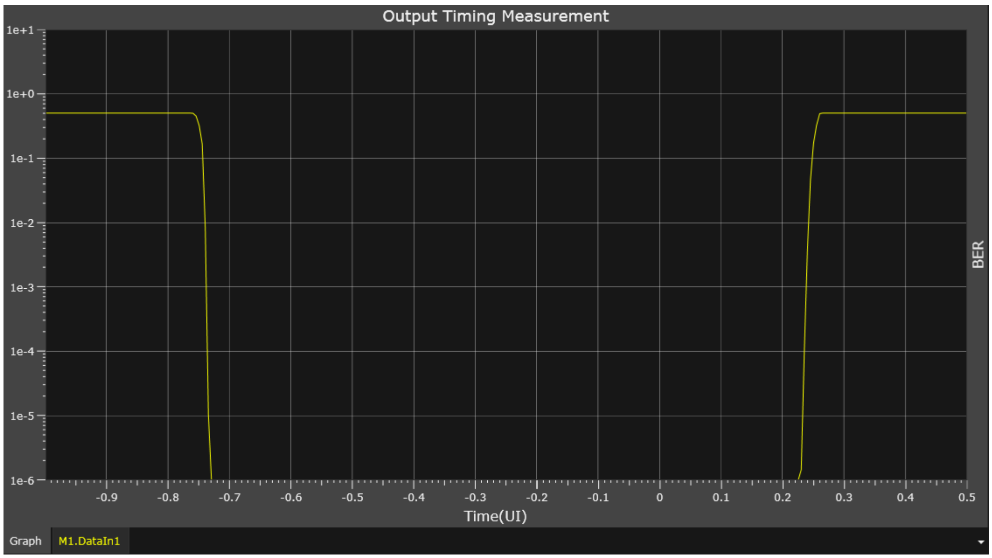
How to Stop Output Timing Measurement

To stop a measurement:

- Click the **Stop Measurement** icon to stop the measurement.

Measurement Graph

Once you run an **Output Timing** measurement for a specified duration, the following graph is shown on the measurement graph:



The bathtub curve in the graph shows the overall jitter distribution over a unit interval and serves as the basis for bit error rate estimation.

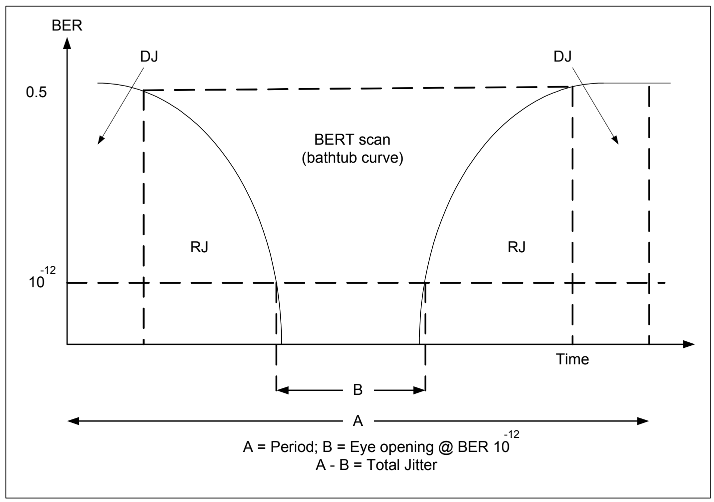
How an Output Timing Measurement Works

The sampling point is swept automatically within a 1.5 clock period to generate a “bathtub” curve. The resulting graph is centered around the optimum sampling point.

The BER Bathtub graph plots TJ (Total Jitter) data and BER values across an entire bit period, which is labeled as the unit interval (UI) on the horizontal axis. BER values are calculated using DJ and RJ and are used to extrapolate to low BER levels. In general, the BER values will differ from the measured TJ values at high probabilities, but the two should converge at low probabilities. Jitter can be measured by moving the sampling point across the eye pattern into the crossing region step by step.

The sampling point- the amplitude decision threshold and the time at which the bits are measured – is scanned across the eye pattern over the time period while monitoring the BER. The bathtub curve can also be used to separate random from the deterministic jitter.

The slope of the bathtub curve is the measure of the random jitter, whereas the slope offset positions on the time axis are set by deterministic jitter. Total jitter is quantified by noting points where BER reduces to at both the eye edges and subtracting this interval from the bit period.



Calculated Results

The calculated results for the **Output Timing** measurement are displayed in the tabular format. It includes the following elements:

- **Location**: Location or Location/Group against which the data acquisition is performed.
- **Show Graphics**: Displays the measurement graph in the measurement output window.
- **Optimal Sample Delay**: Sample delay coordinate of the center of a bounding box around the BER threshold contour line.
- **Phase Margin**: Period of time where the bit error rate is lower than the BER threshold.
- **Total Jitter Peak-to-Peak**: Peak-to-peak value for total jitter. Calculated as the pulse period (unit interval) minus the Phase Margin.

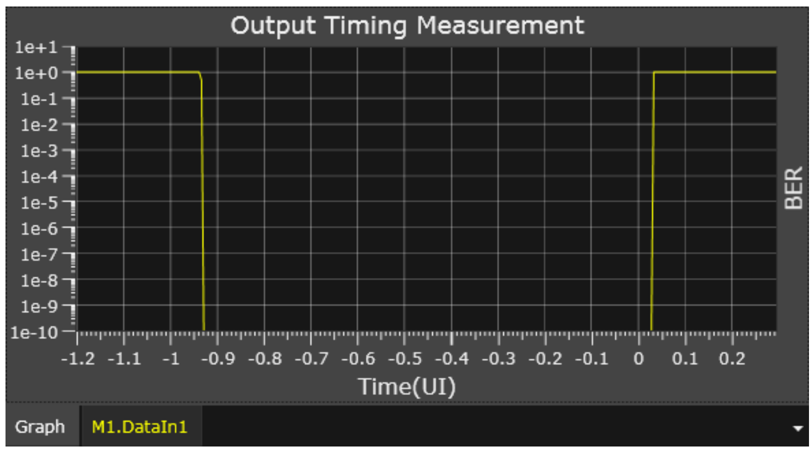
- **Total Jitter RMS**: The average of the left and right jitter histogram root mean squared values.
- **Jitter Mean**: Mean value for total jitter. Calculated as the weighted average of the left edge jitter histogram.
- **Random Jitter RMS**: The total jitter component with Gaussian distribution. After transforming a contiguous range of measured points into Q space and performing a linear regression, it is calculated as the mean of the sigmas of the two straight lines. The contiguous range is limited by the BER Threshold and the Min. BER for RJ/DJ Separation threshold.
- **Deterministic Jitter**: The total jitter component with non-Gaussian distribution. After transforming a contiguous range of measured points into Q space and performing a linear regression, it is calculated as the period minus the difference between the means of the two straight lines.
- **Estimated Total Jitter**: A forecast of the expected jitter for very low bit error rates. After extrapolating the measured BER curves, it is calculated as the period minus the expected width of the eye opening.
- **Left Edge No. Points**: The the number of points that has been measured between the BER Threshold and the Min. BER for RJ/DJ Separation threshold. It is displayed for both slopes. This number has to be greater than 2 for the RJ, DJ, and estimated TJ values to be applicable.
- **Left Edge R^2**: The R^2 values are calculated for both slopes of the bathtub curve. They are a measure of how well the transformed points between BER Threshold and Min. BER for RJ/DJ Separation fit to the linear regression. They have to be greater than 0.75 for the RJ, DJ, and estimated TJ values to be applicable.
- **Right Edge No. Points**: The number of points that has been measured between the BER Threshold and the Min. BER for RJ/DJ Separation threshold. It is displayed for both slopes. This number has to be greater than 2 for the RJ, DJ, and estimated TJ values to be applicable.
- **Right Edge R^2**: The R^2 values are calculated for both slopes of the bathtub curve. They are a measure of how well the transformed points between BER Threshold and Min. BER for RJ/DJ Separation fit to the linear regression. They have to be greater than 0.75 for the RJ, DJ, and estimated TJ values to be applicable.

Fast Total Jitter Measurement Results

The **Fast Total Jitter** measurement provides both graphical and numerical results:

The example below shows a copied result, and the display of measured points was enabled.

The results of the recent measurement are disabled, because they refer to a standard "bathtub" measurement that provides many more results. By actuating the Show indicator, you can inspect both alternatively.



The resulting graph shows you the points in time that have been investigated and whether the actual BER at these points was higher or lower than the BER threshold specified for the measurement.

NOTE For coded patterns, the Fast Total Jitter measurement will only work when the bit recovery mode is enabled.

Calculated Results for Fast Total Measurement

The calculated results for the **Fast Total Jitter** measurement are displayed in the **Calculated Results** pane. It includes the following elements:

- **Location**: Location or Location/Group against which the data acquisition is performed.

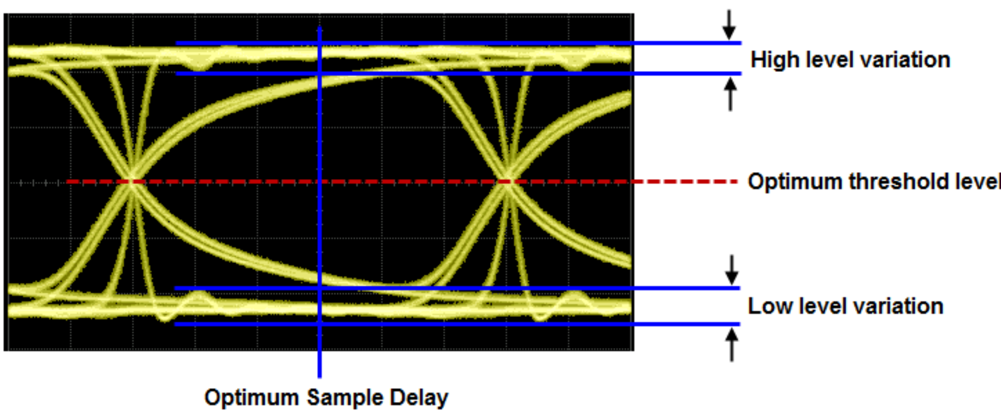
- **Show Graphics**: Displays the measurement graph in the measurement output window.
- **Optimal Sample Delay**: The mean value of the left and right bathtub/Total Jitter BER Threshold intersections.
- **Phase Margin**: The period of time where the bit error rate is lower than the Total Jitter BER Threshold.
- **Total Jitter Peak-Peak**: Peak-to-peak value of the total jitter. Calculated as the pulse period (unit interval) minus the Phase Margin at the Total Jitter BER Threshold.
- **Total Jitter Uncertainty**: Displays the total jitter uncertainty.

Output Level Measurement

Overview

The **Output Levels** measurement allows you to characterize the behavior of the output levels of a device under test (DUT). The sampling delay is fixed. The analyzer's decision threshold is automatically swept within a user-defined range.

A direct result is the determination of the optimum decision threshold level for receiving data from the DUT with maximum confidence.

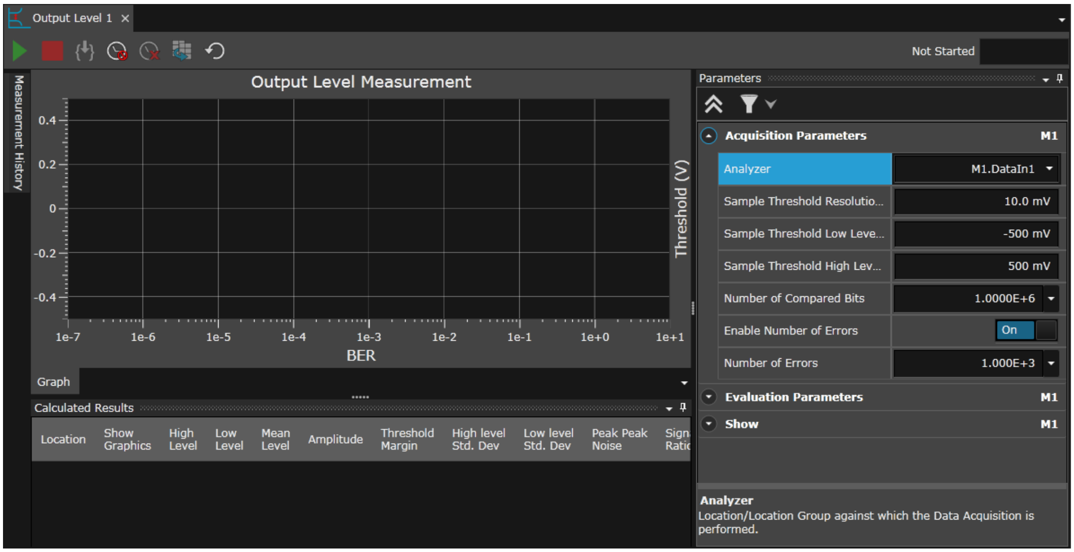


Launching the Output Level Measurement

To launch the **Output Level** user interface:

- Go the **Menu Bar** > **Measurements** and then select **Output Level**.

The **Output Level** user interface will appear as shown in the following figure:



The **Output Level** user interface includes the following elements:

- **Toolbar**: For details, refer to Toolbar on page 296
- **History Pane**: For details, refer to Measurement History Window on page 297.
- **Measurement Graph**: For details, refer to Measurement Graph on page 320
- **Parameters Window**: For details, refer to Parameters Window on page 326.
- **Calculated Results**: For details, refer to Calculated Results on page 333.

Parameters Window

The **Parameters** window have the following acquisition, evaluation and show parameters for output level measurement.

Acquisition Parameters

- **Analyzer**: Specifies location/location group against which data acquisition is performed.

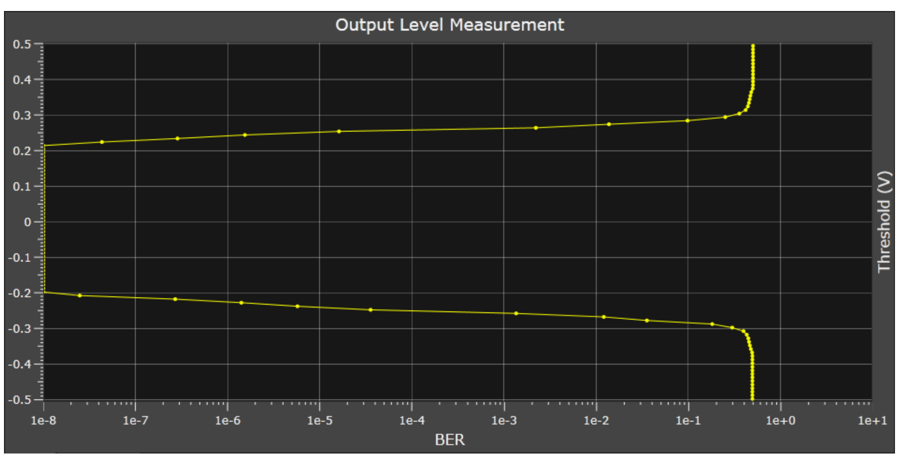
- **Resolution**: Specifies the distance between the sampling points. The lower this value is, the more sampling points you have in the selected voltage range. The minimum step width is hardware-dependent. At the time being, the minimum is 1 mV.
- **Low Level**: This is the lower end of the measured voltage range.
- **High Level**: This is the upper end of the measured voltage range.
- **Number of Compared Bits**: After this number of compared bits, the measurement stops for the current sample point and moves to the next one.
- **Number of Errors**: After this number of errors, the measurement stops for the current sample point and moves to the next one. This allows you to speed up the measurement. You can switch off this option if only the number of compared bits is important.

Evaluation Parameters

- **BER Threshold**: This is the bit error rate threshold at which the Threshold Margin is determined. It is also the upper threshold for the Q-factor calculations. The BER Threshold is displayed in the BER vs. Threshold graph.
- **Min BER for Q**: This is the lower threshold for the Q-factor calculations.

Show Graphics View

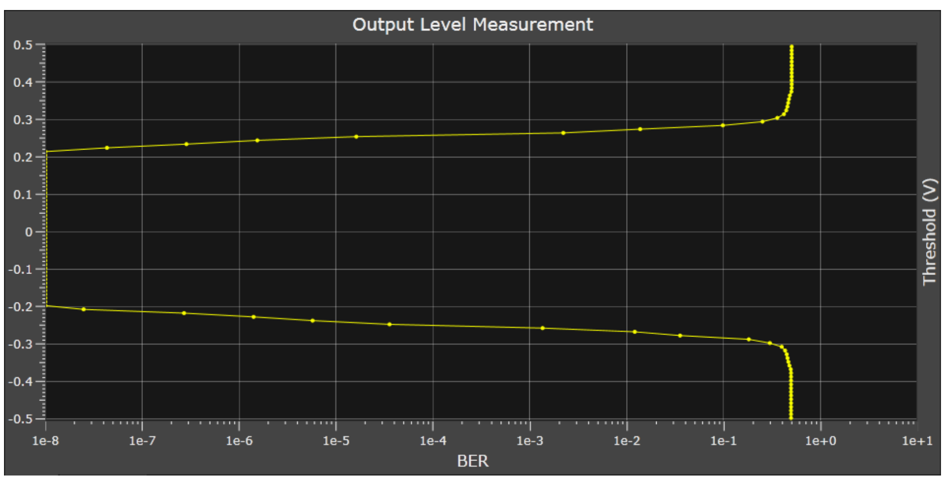
BER vs. Threshold Graph: This graph shows the relationship between the analyzer decision threshold and the measured BER.



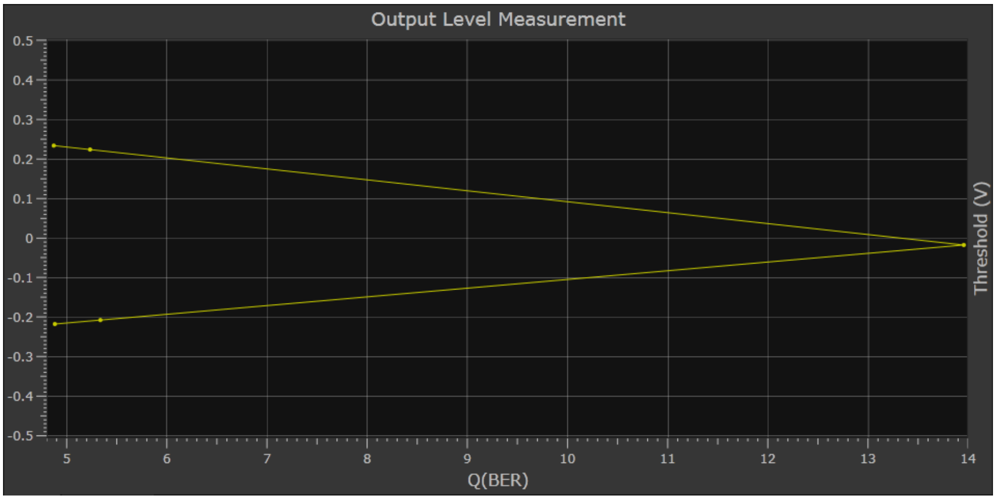
The BER considers all errors. It is calculated as:

$$BER_{AllErrors} = \frac{(\sum Error1s + \sum Error0s)}{(total\ \#\ of\ Bits)}$$

- **QBER vs. Threshold Graph**: This graph shows the extrapolation of the optimum Q-factor and the optimum threshold level from a limited number of measured points.
  The measured data points to be used for the calculation have to be within a contiguous BER range. This range is defined by specifying the Min BER for Q (lower threshold) and the BER Threshold (upper threshold). Both thresholds can be set in the lower section of the View tab.
  The Q-factor can only be calculated if for both high and low level rails two or more points fall within the defined BER range. For reliable results use at least five measured points.
  To ensure proper settings, enable the **Show Measured Points** function and switch to the **BER vs. Threshold** graph. Move the upper BER threshold marker (vertical line) so that a sufficient number of measured points is included for the calculation:



This graph illustrates the calculation of the Q-factor as the best fit line through the calculated points.



Changing the Measurement Setting

- If you change the acquisition settings through **Parameter** window after the measurement has been run, please note that the changes on the parameters tab will take only effect if you run the measurement again.
- However, if you change the parameters that change the display of the measured data (graphics, show measurement points and axis scale), there is no need to repeat the measurement.

Available Views

The **Output Levels** measurement provides three different graphical views to visualize the calculated results:

- **BER versus Threshold**:
  This graph shows the relationship between the analyzer decision threshold and the resulting BER. It presents the raw data.
- **Q from BER versus Threshold**:
  This graph shows the extrapolation of the Q-factor and the optimum threshold level from a limited number of measured points.

How to Execute Output Level Measurement

To run an **Output Level** measurement, perform the following steps:

- Use the **Parameters** window to select the location or location group for which the data acquisition has to be performed.
- Set the acquisition and evaluation parameters for **Output Level** in the **Parameters** Window. For details, refer to Parameters Window on page 326.
- Click **Start Measurement** icon to execute the measurement. The measurement status indicator will indicate **Running**.

How to Stop Output Level Measurement

To stop a measurement:

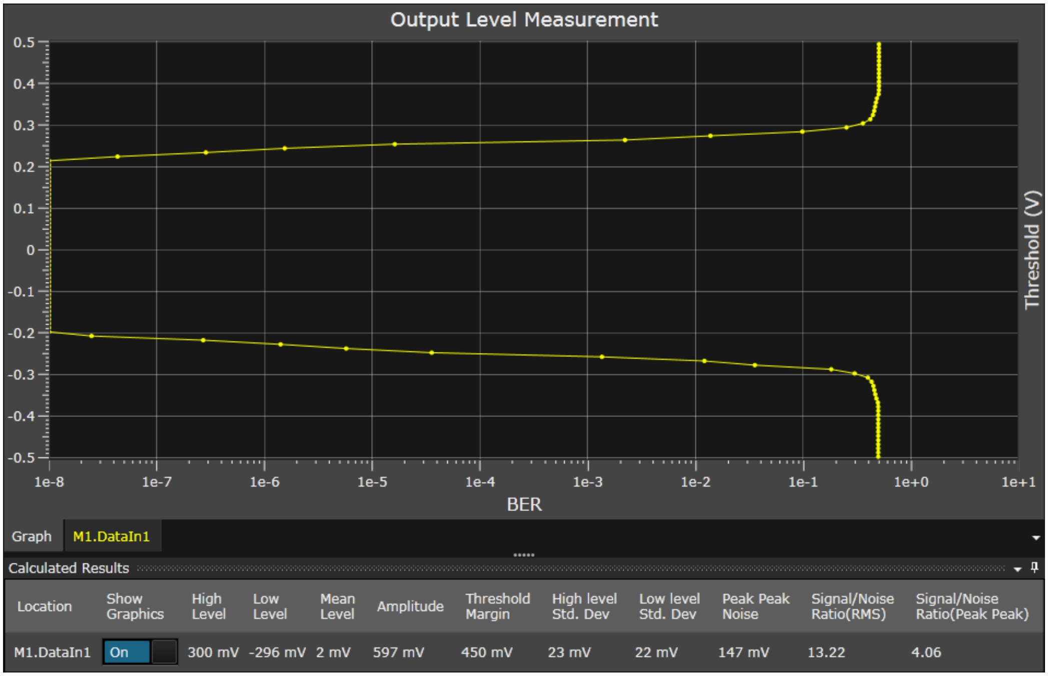
- Click **Stop Measurement** icon to stop the measurement.

Measurement Graph

The **Measurement Graph** displays the absolute values of the derivative of the bit error rates over the thresholds (dBER/dTh). It visualizes the data that forms the basis for the calculations of the level and noise values.

The **Output Level** measurement provides Q from BER versus Threshold graph. This graph refers to the Q-factor calculations.

The following illustration shows the **BER versus Threshold** graph of a simple **Output Level** measurement:



The **Measurement Graph** contains the following tabs:

- **Graph**: Displays the graphical representation of the measurement. The graph varies from measurement to measurement. The details of each measurement graph are further described in their respective sections.
- **Location**: Displays the raw measurement data for that location. However, if you are running measurement for a group, multiple tabs will appear that display the raw measurement for each location.

When you right-click on the **Measurement Graph**, a context menu appears which provides the following options:

- **Turn ON/OFF Fit to view (Ctrl+Home)** - Turns ON/OFF Fit to view option.
- **Fit to view (Home)** - Makes the visible area fit to display entire contents.
- **Copy screenshot (F11)** - Copies the screenshot of charts to clipboard.
- **Save screenshot (Ctrl+S)** - Saves the screenshot as an image (PNG) under a name.

- **Quick Help (Alt+F1)** - Opens a window that provides brief information about the dynamic display.

How to Improve the Output Levels Display

You can consider the following points to change the display of an existing measurement:

- **Measurement Points**: If you wish to see more details to investigate the graph, select **Show Measured Points** on the **Graph** option of the **Properties** window. With this option selected, all measured points are indicated in the graph with small squares.
- **Zoom**: Several zoom factors are available. When you show the zoom graph, you can also allow the zoom graph to track the mouse
- **Scale**: Choose between **Logarithmic** and **Linear** to select the scale for the displays x-axis.
  The scale of the **QBER vs. Threshold** graph is always linear.
- **Graph Context Menu**: When you right-click on the **Measurement Graph**, a context menu appears which provides the options such as fit to view, copy and save screen shot and quick help of dynamic display.

How to Change the Output Levels Properties

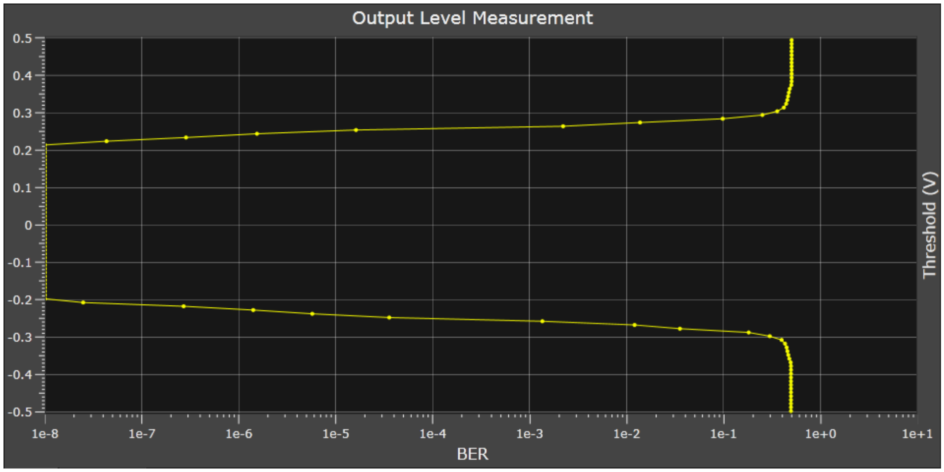
In the example measurement, we have set the focus on speed: 100 threshold levels and 1,000,000 compared bits per measuring point. You may wish to obtain more precise results.

1. Switch to the **Parameters** window.
2. Increase the **Number of Compared Bits** to 100,000,000.

   Remember: One failure per 1 million bits yields a BER resolution of 10-6. One failure per 100 million bits yields a BER resolution of 10-8.
3. Decrease the **Sample Threshold Resolution** to 2mV.

   This gives us 500 steps per Volt.
4. Press the **Start Measurement** icon to repeat the measurement with the new parameters.

The measurement now takes more time than the previous, but it is also much more precise.



Calculated Results

The following figure illustrates the calculated results:

Calculated Results

Location	Show Graphics	High Level	Low Level	Mean Level	Amplitude	Threshold Margin	High level Std. Dev	Low level Std. Dev	Peak Peak Noise	Signal/Noise Ratio(RMS)	Signal/Noise Ratio(Peak Peak)
M1.DataIn1	On	300 mV	-296 mV	2 mV	597 mV	450 mV	23 mV	22 mV	147 mV	13.22	4.06

The calculated results are divided into three groups:

- Level Results
- Noise Results
- Q-factor Results

Level Results

The level results are defined as follows:

- **High Level**: The High Level is the mean of the upper dBER/dTh distribution. It is calculated as:

$$Mean = \frac{\sum (dBer \cdot threshold)}{\sum dBer}$$

- **Low Level**: The Low Level is the mean of the lower dBER/dTh distribution. It is calculated as:

$$Mean = \frac{\sum (dBer \cdot threshold)}{\sum dBer}$$

- **Mean Level**: The Mean Level is the middle between the High and Low Levels, calculated as:

$$Mean\ Level = \frac{LowLevel + HighLevel}{2}$$

- **Amplitude**: The Amplitude is the difference between its High and Low Levels.
- **Threshold Margin**: The Threshold Margin is the distance between the upper and the lower BER curves at the position given by the BER Threshold setting.

Noise Results

The noise parameters are defined as follows:

- **High Level Std. Dev.**: The standard deviations are derived from the dBER/dTh histogram.
  The High Level Standard Deviation is calculated as:

$$StdDev = \sqrt{\frac{\sum \left((threshold - Mean)^2 \cdot dBer\right)}{\sum dBer}}$$

  where Mean is the High Level of the terminal.
- **Low Level Std. Dev.**: The Low Level Standard Deviation is calculated as:

$$StdDev = \sqrt{\frac{\sum \left((threshold - Mean)^2 \cdot dBer\right)}{\sum dBer}}$$

  where Mean is the Low Level of the terminal.

- **Peak Peak Noise**: The peak-to-peak Noise is calculated as:

$$Peak\ Peak\ Noise = \mathrm{High}Level - \mathrm{Low}Level - Threshold\ Margin$$

Note that the **Threshold Margin** depends on the position of the **BER Threshold**.

- **Signal/Noise Ratio (RMS)**: The RMS Signal-to-Noise Ratio is calculated as:

$$SNR\ (RMS) = \frac{HighLevel - LowLevel}{StdDev\ 1 + StdDev\ 0}$$

- **Signal/Noise Ratio (Peak Peak)**: The peak-to-peak Signal-to-Noise Ratio is calculated as

$$\mathrm{SNR\ (PeakPeak)} = \frac{HighLevel\ \text{-}\ LowLevel}{PeakPeakNoise}$$

Note that the **Peak-to-Peak Noise** depends on the position of the **BER Threshold**.

Q Factor Results

The numerical Q-factor parameters are defined as follows:

- **Q Factor**: The Q-factor is calculated as:

$$Q = \frac{\mu_1 - \mu_0}{\sigma_1 + \sigma_0}$$

where µ1,0 is the mean level of the 1 and 0 rails, respectively, and σ1,0 is the standard deviation of the noise distribution on the 1 and 0 rails.

- **Q Optimum Threshold**: The Q Optimum Decision Threshold is calculated as:

$$\text{Optimum Decision Threshold} = \frac{\sigma_0\mu_1 + \sigma_1\mu_0}{\sigma_1 + \sigma_0}$$

- **Q Residual BER**: The Q Residual BER is the expected BER at the Optimum Decision Threshold. It is calculated as:

$$BER = \frac{e^{-(Q^2/2)}}{Q_{opt}\sqrt{2\pi}}$$

Numbers below 1e-255 are expressed as zero.

- **Q High Level**: The Q High Level is the mean, calculated from the linear regression curve for the high level data:

$$\mu = \frac{-A}{B}$$

- **Q High Level Std.Dev**: The Q High Level Standard Deviation is the σ (Sigma), calculated from the linear regression curve for the high level data:

$$\sigma = \left| -\frac{1}{B} \right|$$

- **Q High Level Nr. Points**: This is the number of data points used for the calculation of the Q High Level value. It depends on the setting of the BER Threshold and also on the setting of the Min BER for Q parameter. The minimum for calculating Q-factor values is two points. It is recommended to use more than 5 points.
- **Q High Level R^2**: The R^2 parameter is an indicator that shows how well the converted data points fit to the straight line. It is calculated as:

$$R^2 = \frac{\left(\Sigma XY - \frac{(\Sigma X)(\Sigma Y)}{n}\right)^2}{\left(\Sigma X^2 - \frac{(\Sigma X)^2}{n}\right)\left(\Sigma Y^2 - \frac{(\Sigma Y)^2}{n}\right)}$$

The R^2 parameter should be examined before trusting the Q-values. Its maximum value is 1.0. It must be seen in conjunction with the number of data points.

For example: Two data points always fit perfectly well, but the resulting Q-factor calculations are not reliable. On the other hand, 50 data points may reveal a poor R^2 value. This tells you that the linearization is prone to errors.

If the R^2 value falls below 0.75, the Q-factor calculations are not applicable.

- **Q Low Level**: The Q Low Level is the mean, calculated from the linear regression curve for the low level data:

$$\mu = \frac{-A}{B}$$

- **Q Low Level Std.Dev**: The Q Low Level Standard Deviation is the σ (Sigma), calculated from the linear regression curve for the low level data:

$$\sigma = \left| -\frac{1}{B} \right|$$

- **Q Low Level Nr. Points**: This is the number of data points used for the calculation of the Q Low Level value. It depends on the setting of the BER Threshold and also on the setting of the Min BER for Q parameter.

  The minimum for calculating Q-factor values is two points. It is recommended to include more than 5 points.

- **Q Low Level R^2**: The R^2 value can also be seen as an indicator of how well the noise distribution fits to Gaussian shape. It will not fit, for example, if the received signal is dominated by cross-talk or modal noise.

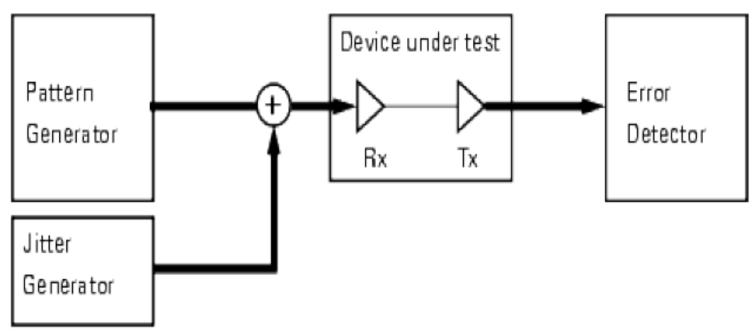
Jitter Tolerance Measurement

Overview

A **Jitter Tolerance** measurement is used to determine the ability of a device or system to maintain communication quality in the presence of jitter. It comes in two varieties:

- **Jitter Tolerance Characterization** determines the jitter levels where the device under test can no longer maintain a desired bit error ratio (BER).
- **Jitter Tolerance Compliance** verifies that the device under test is able to maintain a BER level at pre-defined jitter levels and jitter frequencies, as defined by a standard.

The basic setup of a jitter tolerance test is illustrated below:



Understanding Jitter Tolerance

The behavior and the passed/failed classification of data receiving devices or circuits are determined by their ability to withstand:

- Jitter
- Intersymbol interference
- Level noise

This refers particularly to the performance of phase-locked loops (PLLs) or clock data recovery circuits (CDR).

Jitter tolerance can be measured by applying a distorted data signal to the DUT and measuring the resulting bit error ratio. To make jitter tolerance tests reproducible, the signal distortion must also be reproducible. This requires some definitions.

Jitter

Receivers react differently on

- different types of jitter
- the jitter composition
- the jitter frequency spectrum
- the combination of jitter frequencies and data rate

Various test standards specify the jitter composition to be used for jitter tolerance tests.

The M8020A/M8030A provides the following means for generating artificial (reproducible) jitter:

- Two types of voltage-controlled signal delay lines
- A phase shifter for modulating the generated clock

Intersymbol Interference

Conductors on PC boards have a limited bandwidth. This causes intersymbol interference which changes the shape of the received eye opening. The eye becomes asymmetrical.

Level Noise

Level noise affects the voltage amplitude of the eye opening.

PLL Performance Test Testing a PLL (or CDR) is not complete until the entire frequency range of the PLL has been checked under worst-case conditions.

For this purpose, the M8020A/M8030A provides the Jitter Tolerance Characterization measurement.

PLL Standards and Compliance Tests

The Optical Internetworking Forum (OIF) as well as other institutions have proposed standards for testing the performance of data receivers and receiver circuits in the presence of jitter.

For this purpose, the M8020A/M8030A provides the Jitter Tolerance Compliance measurement.

Types of Jitter

The standards for jitter tolerance tests prescribe combinations of certain jitter types. This section gives an overview of these types.

Random Jitter

Random jitter (RJ) is always present. Resistors, transistors, and other components generate noise that impacts transmitters and receivers.

This kind of jitter has a Gaussian distribution over time.

Bounded Uncorrelated Jitter

Similar to random jitter, bounded uncorrelated jitter (BUJ) has also a Gaussian distribution, but this distribution is cut (bounded) at both sides.

This kind of jitter can be caused, for example, by crosstalk on a parallel bus or by intersymbol interference of random or long pseudo random binary bit sequences.

To simulate bounded uncorrelated jitter for jitter tolerance tests, it can be generated from a filtered (frequency-limited) PRBS.

Periodic Jitter

Periodic jitter (PJ) is generally picked up from other periodic sources.

- **Rectangular Jitter**: Rectangular jitter may be caused, for example, by a switching power supply. The jitter amplitude is usually specified in UI. One UI (Unit Interval) is always the reciprocal of the present data rate.
- **Sinusoidal Jitter**: This kind of jitter can be picked up from any adjacent signal or clock. The jitter histogram has a U-shape.
- **Triangular Jitter**: Triangular jitter is always generated by a spread spectrum clock but can also be caused by other periodic sources. Triangular jitter has a uniform jitter distribution.

Intersymbol Interference

Intersymbol interference (ISI) is caused by bandwidth limitations of cables and backplanes. It cuts off higher frequencies and hence changes the shape of the eye opening.

Intersymbol interference can be simulated by inserting a defined transmission path between the Generator and the DUT.

Crosstalk

Crosstalk between adjacent signal paths modulates the vertical eye amplitude. This narrows the voltage range of the receiver needed for capturing data correctly.

For jitter tolerance tests, amplitude modulation is most often simulated by modulating the data output with a sinewave signal. This is called Sinusoidal Interference (SI).

Total Jitter

Total jitter is the sum of all kinds of jitter.

BER Setup

Table 56 BER setup description

Name	Description
Data Rate	Displays the Data Rate for the measurement.
Target BER	Sets the required target BER. The default is 1.0E-09.
Confidence Level	Confidence Level = 1 – exp(-data rate * gate time * measurement depth) The range is 0 % to 99.9 %. The default is 95 %.
Frequency Relax Time	Sets the amount of time to pause the measurement after a change in jitter modulation frequency.
Amplitude Relax Time	Sets the amount of time to pause the measurement after a change in jitter modulation amplitude.

Graph Setup

Table 57 Graph setup description

Name	Description
Template Limits	Show/hides the maximum and minimum jitter amplitude limits in the graph.
Template Points	Show/hides the template points in the graph.
Compliance Limits	Show/hides the compliance curve in the graph.

Instrument Setup

Table 58 Instrument setup description

Name	Description
Generator	The Generator drop down list box shows all available Data Out locations that are used as a data source. A generator must be specified to perform the measurement.
Analyzer	The Analyzer drop down list box shows all available Data In locations that are used for data acquisition. A detector must be specified to perform the measurement.

Measurement Setup

Table 59 Measurement setup description

Name	Description
Measurement Template	Click the  **Open** icon to open the **Jitter Tolerance Measurement** template.
Start Frequency	Sets the start frequency of the **Jitter Tolerance Measurement**.
Stop Frequency	Sets the stop frequency of the **Jitter Tolerance Measurement**.
Num Points	Sets the number of measurement points.
Mode	Sets the Jitter Tolerance Measurement mode. The options are **Compliance** or **Characterization**. In **Characterization** mode you have to set the search algorithms. For details, refer to Characterization on page 343. To set Compliance margins, see Compliance Margin on page 348.
Step Size	Sets the step size for characterization algorithms. The option are Binary Step Size, Linear Step Size and Log Step Size. For details, see Step Size on page 347.
CDR LBW Auto	Automatically sets the external CDR Loop Bandwidth.

Characterization

Table 60 Characterization

Name	Description
Algorithm	Search algorithms apply to characterization measurements only.

Name	Description
Binary	The Binary algorithm uses a variable step size to find the highest passing SJ amplitude at each SJ frequency in the template. Starting at the maximum SJ amplitude determined by the upper limit in the template file or the instrument limit, whichever is lower, followed by the minimum SJ amplitude determined by the lower limit in the template file, the Binary algorithm then sets subsequent SJ amplitudes at the logarithmic midpoint between the highest passing and lowest failing SJ amplitudes. A test point is determined to be a passing point if the BER measured is below the BER threshold set by the user. The equation for the midpoint (c) between points a and b on a log scale is: c = 10^((log(a) + log(b)) / 2). The algorithm exits if the channel is tolerant of the maximum jitter amplitude or is intolerant of the minimum jitter amplitude. Otherwise, the algorithm continues until the last step size is less than the minimum step size set by the user for Binary algorithms. The jitter tolerance result at each jitter modulation frequency is the highest jitter amplitude at which the measurement was made below the threshold BER. However, if the device is intolerant of even the minimum jitter value, the result is not valid and is not plotted. In many cases, the Binary algorithm is the quickest. However, devices with hysteresis may not have consistent results when using the Binary algorithm since the approach direction (up vs. down) will cause the result to vary. This occurs because the PLL in the clock data recovery unit will typically hold lock longer as the jitter is increased, then it would establish lock as the jitter is reduced. For example, a receiver may have a higher passing SJ amplitude when the previous step was a lower (passing) SJ amplitude, compared to a lower passing SJ amplitude when the previous step was a higher (failing) SJ amplitude. If this is a problem, see the Binary + Down Linear and Binary + Up Linear algorithms.

Name	Description
	The following is an example of the Binary algorithm. At a single SJ frequency, if the true pass/fail SJ amplitude is 0.7 UI, the minimum step size is 0.1 UI, the template minimum is 0.1 UI, and the template maximum is 1.0 UI, then the binary algorithm would follow these test points: **1** Template maximum 1.0 UI - FAIL **2** Template minimum 0.1 UI - PASS **3** Log midpoint between 1.0 UI (FAIL) and 0.1 UI (PASS) = 0.32 UI - PASS **4** Log midpoint between 1.0 UI (FAIL) and 0.32 UI (PASS) = 0.57 UI - PASS **5** Log midpoint between 1.0 UI (FAIL) and 0.57 UI (PASS) = 0.75 UI - FAIL **6** Log midpoint between 0.75 UI (FAIL) and 0.57 UI (PASS) = 0.65 UI - PASS **7** Log midpoint between 0.75 UI (FAIL) and 0.65 UI (PASS) 0.70 UI - PASS **8** Last step size < 0.1 UI, highest passing SJ amplitude is 0.70 UI
Binary + down linear	The Binary + Down Linear algorithm is the same as the Binary algorithm followed by the Down Linear algorithm. Refer to the descriptions for Binary and Down Linear in this table.
Binary + up linear	The Binary + Up Linear algorithm is the same as the Binary algorithm followed by the Up Linear algorithm. Refer to the descriptions for Binary and Up Linear in this table.
Down linear	The Down Linear search algorithm starts at the maximum jitter value determined by the template. If a BER measurement has errors above the measurement threshold, the jitter amplitude is adjusted lower by the linear step size. This algorithm exits when the BER is measured below the measurement threshold BER, or if the last amplitude measurement point was at the minimum jitter value in the template. The jitter tolerance result at each jitter modulation frequency is the highest jitter amplitude at which the measurement was made below the threshold BER. However, if the device is intolerant of even the minimum jitter value, the result is not valid and is not plotted.

Name	Description
Down logarithmic	The Down Logarithmic search algorithm starts at the maximum jitter value determined by the template. If a BER measurement has errors above the measurement threshold, the jitter amplitude is adjusted lower by the coefficient calculated from the logarithmic step size. For example, if the coefficient is 10%, then the next amplitude equals the previous amplitude minus 10% of the previous amplitude. This algorithm exits when the BER is measured below the measurement threshold BER, or if the last amplitude measurement point was at the minimum jitter value in the template. The jitter tolerance result at each jitter modulation frequency is the highest jitter amplitude at which the measurement was made below the threshold BER. However, if the device is intolerant of even the minimum jitter value, the result is not valid and is not plotted.

Name	Description
Up linear	The Up Linear search algorithm starts at the minimum jitter value determined by the template. If a BER measurement has errors below the measurement threshold, the jitter amplitude is adjusted higher by the linear step size. This algorithm exits when BER is measured above the measurement threshold BER, or if the last amplitude measurement point was at the max jitter value in the template. The jitter tolerance result at each jitter modulation frequency is the highest jitter amplitude at which the measurement was made below the threshold BER. However, if the device is intolerant of even the minimum jitter value, the result is not valid and is not plotted.
Up logarithmic	The Up Logarithmic search algorithm starts at the minimum jitter value determined by the template. If a BER measurement has errors below the measurement threshold, the jitter amplitude is adjusted higher by the coefficient calculated from the logarithmic step size. For example, if the coefficient is 10%, then the next amplitude equals the previous amplitude plus 10% of the previous amplitude. This algorithm exits when the BER is measured above the measurement threshold BER, or if the last amplitude measurement point was at the maximum jitter value determined by the template. The jitter tolerance result at each jitter modulation frequency is the highest jitter amplitude at which the measurement was made below the threshold BER. However, if the device is intolerant of even the minimum jitter value, the result is not valid and is not plotted.
Up Log + Linear	The Up Log + Linear search algorithm performs the same algorithm as Up Logarithmic, but in addition, it returns to the last passing amplitude and steps linearly up until it reaches a fail point. The algorithm continues to increase the SJ amplitude until failure, even if it surpassed the original failed point.

Step Size

Table 61 Step size descriptions for characterization algorithms

Name	Description
Binary Step Size	The Step Size for the Binary algorithm defines the exit criteria for the algorithm. The Binary search algorithm stops once its step size falls below this user defined Step Size. The Binary Step Size applies to the Binary algorithm and the binary portions of the Binary + Down Linear and Binary + Up Linear algorithms..
Linear Step Size	This defines the step size for each step of the Up Linear and Down Linear algorithms. When Down Linear is enabled as the search algorithm, the measurement starts from the maximum (which depends on the jitter frequency). A step size of 100 mUI, for example, may result in a sequence of 1000 UI, 999.9 UI, 999.8 UI, and so on. The test for one frequency stops when the BER limit is met or minimum specified amplitude is reached. When Up Linear is enabled as the search algorithm, the measurement starts from the minimum. A step size of 100 mUI, for example, will result in a sequence of 0.1 UI, 0.2 UI, 0.3 UI, and so on. The test for one frequency stops when the BER limit is exceeded or the maximum amplitude is reached.
Log Step Size	This defines the step size for each step of the Up Logarithmic and Down Logarithmic algorithms. When Down Logarithmic is enabled as the search algorithm, the measurement starts from the maximum (which depends on the jitter frequency). A percentage of 50%, for example, may result in a sequence of 1000 UI, 500 UI, 250 UI, and so on. The test for one frequency stops when the BER limit is met or the specified minimum amplitude is reached. When Up Logarithmic is enabled as the search algorithm, the measurement starts from the specified minimum amplitude. A percentage of 50%, for example, may result in a sequence of 0.1 UI, 0.15 UI, 0.23 UI, and so on. The test for one frequency stops when the BER limit is crossed or the maximum amplitude (which depends on the jitter frequency) is reached.

Compliance Margin

Table 62 Compliance Margin

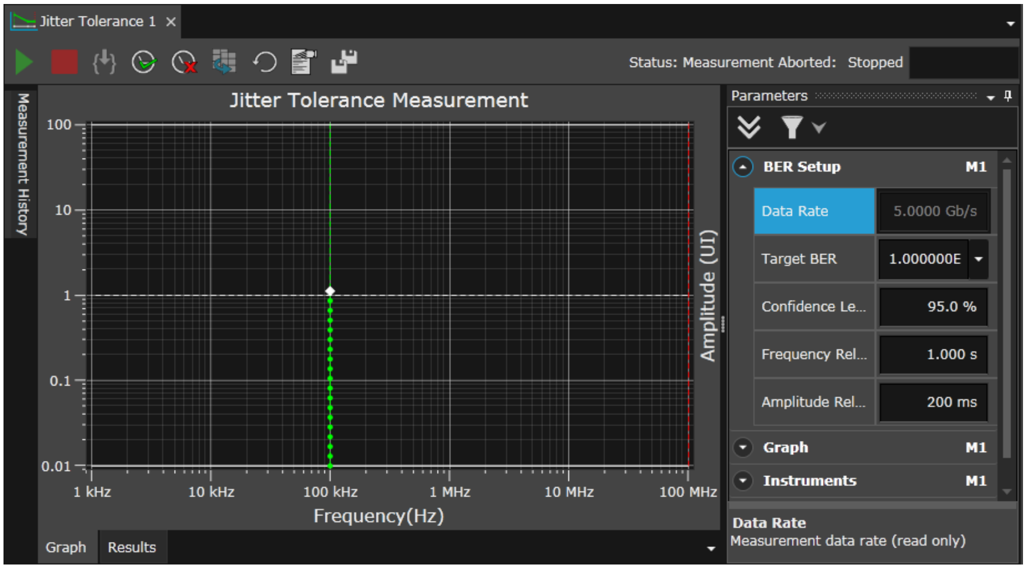
Name	Description
Compliance Margin	In Compliance mode, the test starts at points defined on the jitter tolerance curve and is increased or decreased by the specified margin percent. The compliance template is adjusted according to the equation SJ Amplitude (with margin) = original SJ Amplitude * (1.0 + margin(%)/100). This can be used to test if a receiver can withstand a certain percentage more jitter than dictated by the compliance template. Conversely, if a receiver is failing, the margin can be lowered to test if a receiver can pass a compliance test with a less stringent template.

Launching Jitter Tolerance Measurement

To launch the **Jitter Tolerance Measurement**:

- Go to **Menu Bar** > **Measurements** and then select **Jitter Tolerance**.

The **Jitter Tolerance** user interface will appear as shown in the following figure:



The **Jitter Tolerance** user interface includes the following GUI elements:

- **Toolbar**: For details, refer to Toolbar on page 296.
- **Status Indicator**: For details, refer to Status Indicator on page 297.
- **Measurement History Window**: For details, refer to Measurement History Window on page 297.
- **Measurement Graph**: For details, refer to Measurement Graph on page 350
- **Results Pane**: For details, refer to Viewing the Jitter Tolerance Results on page 351.

How to Run Jitter Tolerance Measurement

Once you have specified the Frequency, BER, and Search criteria, press the **Start Measurement** icon to run the measurement. The measurement status indicator will show the progress.

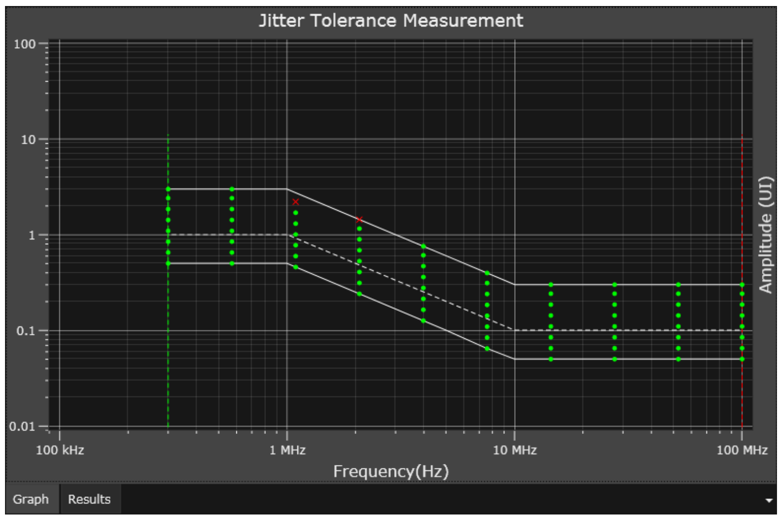
How to Stop/Abort Jitter Tolerance Measurement

To stop a measurement, click the **Stop Measurement** icon.

Measurement Graph

After a jitter tolerance measurement has been run, results can be viewed by clicking on the **Graph** tab. The graph displays the sinusoidal jitter frequency on the x-axis, and the sinusoidal jitter amplitude on the y-axis.

The following figure shows the measurement results in the graph view.



Results Points

The graph displays the BER measurement results of each tested sinusoidal jitter point. Passing results are displayed as a green dot. In the graph, the green "+" means that you've reached the instruments max jitter capability without a fail. The red cross "x" means that you have failed to reached the instruments max jitter capability.

Maximum and Minimum Template Limit Lines

The solid white lines on the graph show the search range during characterization. This feature reduces test time by eliminating points where the DUT is likely to pass when its performance range is known.

Template Points

The green dots on the graph are the measurement points defined in the Template File.

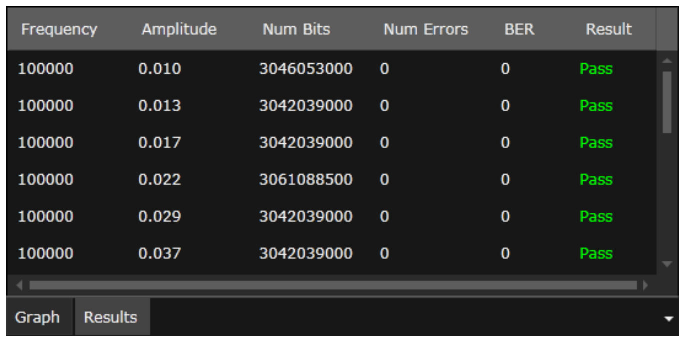
Compliance Limit Line

The dashed white line on the graph shows the compliance test defined in the Template File.

Viewing the Jitter Tolerance Results

Measurement results can also be viewed in tabular form by clicking on the **Results** tab.

The following figure shows the results at each measurement point in tabular form.

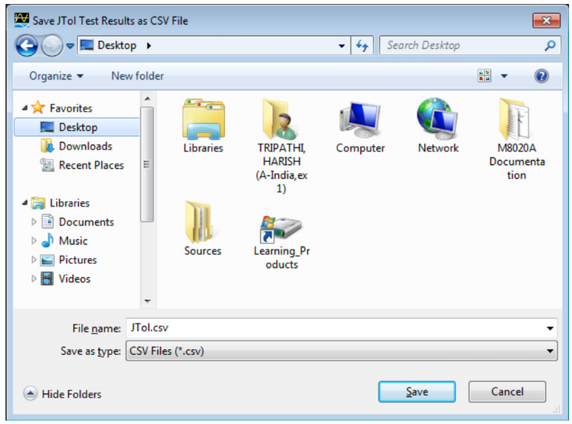


Frequency	Amplitude	Num Bits	Num Errors	BER	Result
100000	0.010	3046053000	0	0	Pass
100000	0.013	3042039000	0	0	Pass
100000	0.017	3042039000	0	0	Pass
100000	0.022	3061088500	0	0	Pass
100000	0.029	3042039000	0	0	Pass
100000	0.037	3042039000	0	0	Pass

Saving Jitter Tolerance Measurement Results

You can also save the **Jitter Tolerance Measurement** results in to a CSV file. To do so:

- Click on the Save icon. A standard **Save As** dialog will appear.

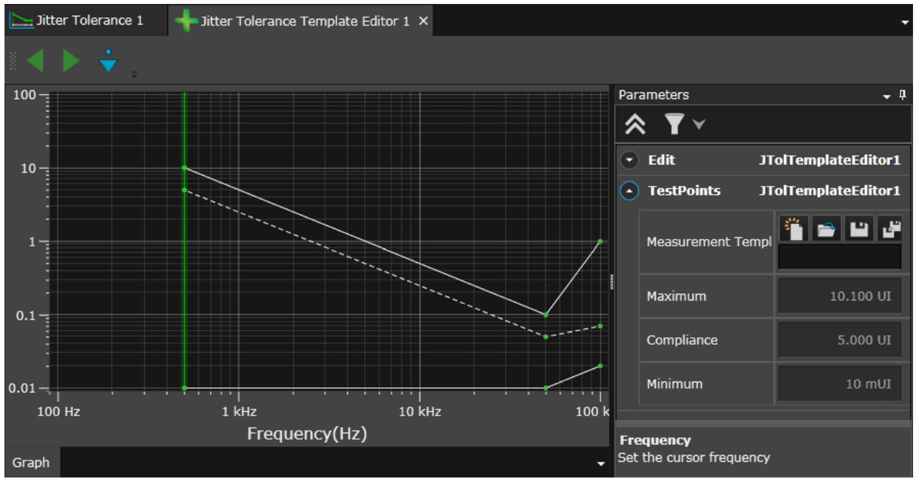


- Provide a filename and location.
- Click **Save**. The **Jitter Tolerance Measurement** results will be saved in *CSV* format.

Using the Jitter Tolerance Template Editor

The **Jitter Tolerance Template Editor** is used to modify the currently loaded template file. It can be accessed by clicking on the **Open Jitter Tolerance Template Editor** icon, present on the toolbar.

The following is an example of jitter tolerance template editor showing the measurement point values (frequency and amplitude) for the currently loaded template file.



Template Editor Toolbar

Table 63

Elements	Name	Description
	Previous /Next Measurement Point	The left and right arrow buttons cause the vertical marker to jump to the next/previous measurement point.
	Auto Scale Graph	Scales the graph to the optimum setting.

Template Editor Functions

Table 64

Function	Description
Edit	
Frequency	The frequency at the current vertical marker position is displayed in this window. Changing this frequency using the keypad, allows new measurement points to be defined.
Add Test Point/Remove Test Point	Use the drop-down menu to add or remove test points and then click on the **Execute** button.
Test Points	
Measurement Template	Selects the **Jitter Tolerance Measurement** template file. It provides the following functions:  **New** - Opens a new template file.  **Open...** - Opens an existing template file which includes limits, compliance, and jitter tolerance measurement state.  **Save** - Saves the currently open template file to the current filename.  **Save As...** - Saves the currently open template file to a different filename and/or location.
Maximum Changes the maximum amplitude of the measurement point at the current vertical marker position. These are the points along the solid white line on the graph.	
Compliance Changes the compliance amplitude of the measurement point at the current vertical marker position. These are the points along the upper solid white line on the graph.	
Minimum Changes the minimum amplitude of the measurement point at the current vertical marker position. These are the points along the lower solid white line on the graph.	

Eye Diagram Measurement

What is an Eye Diagram?

The M8070A System Software provides quick design analysis with the Eye Diagram capability.

The Eye Diagram allows a quick check for the DUT's signal output, and determines the signal quality. The eye contour lines display the measured eye at a deeper BER level, for accurate results.

The Eye Diagram feature provides the following measurement capabilities:

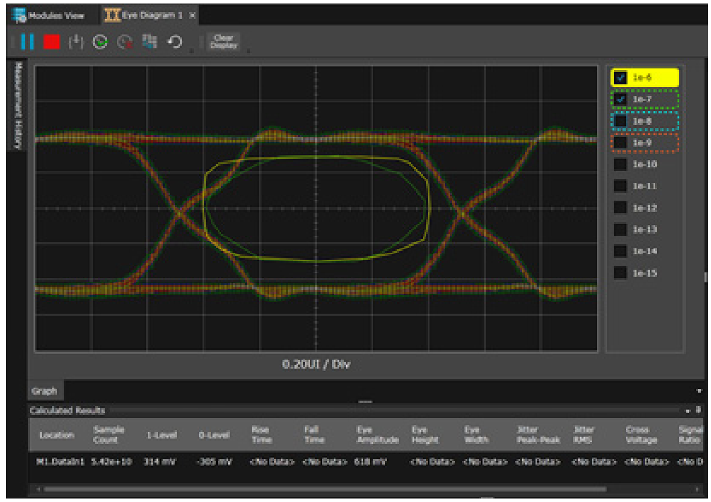
- Eye Diagram Waveform
- BER Contour Lines
- Automatic Measurement for the most relevant eye parameters:-
  - Eye Height
  - Eye Width
  - Jitter P-P
  - Jitter RMS
  - Cross Voltage

The Eye Diagram generates a three dimensional graph of the bit error rate (BER).

This measurement helps in determining and analyzing the quality of the DUT's signal output. The Eye Diagram results comprise of voltage(y), time(x), and BER(z).

The Eye Diagram, and the table of Calculated Eye Measurement Results form simultaneously. They become more precise with the increasing number of measured bits.

The following graph shows the Eye Diagram integrated in the M8070A system software:



Methods of Representation of Eye Diagram

There are two methods of representing the eye diagram:

- Waveform
- Contour

Waveform

Waveform is the shape, and form of a signal. The waveform graph shows the periodical variation of voltage against time.

The waveform in the M8020A/M8030A is similar to the one in the oscilloscope. In this case, the waveform initially gives a coarse, but quick picture of the signal quality; while the 'smooth waveform' quickly generates a high resolution graph. The waveform is displayed in an incremental way, showing the coarse picture quickly, and then refining it further.

The BER Threshold is configurable. The BER Threshold is the level up to which the signal is represented as waveform, and BER values below this threshold are represented as contours.

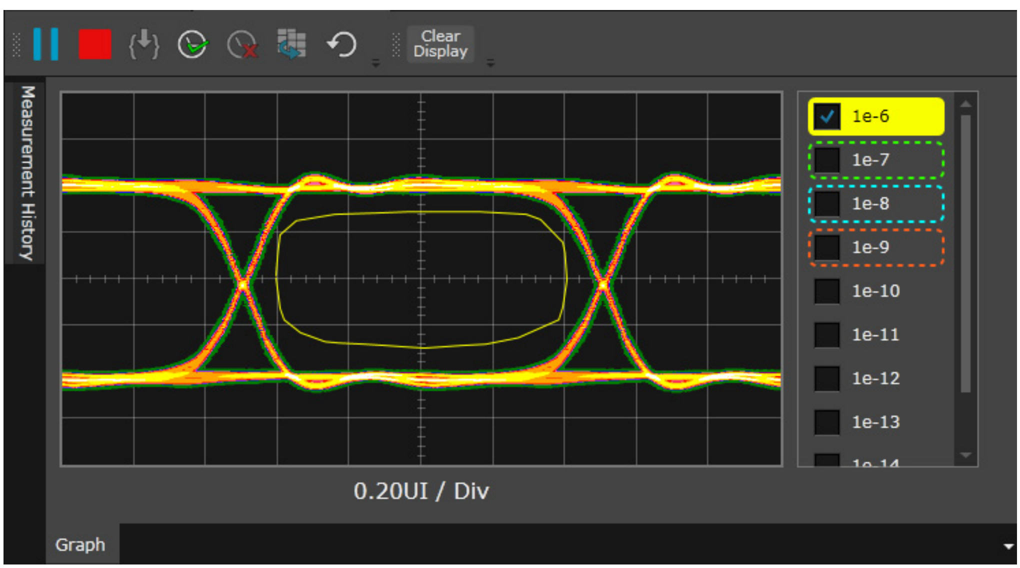
BER Contour

Contour is a curve connecting points where the BER has a same particular value.

The contour graph is plotted within the Eye Diagram, and it helps to determine the Eye Opening at deep bit error rates, such as 1e-10, 1e-12, and so forth. Depending upon the user's requirement the deep BERs can be calculated. The extrapolated eye contour lines display the eye opening for lower BER levels, such as 1e-15. The BER is displayed as a function of sampling delay, and sampling threshold.

The eye opening is one of the main characteristics of a high speed device.

The legend on the right side of the screen represents the contour BER values in different colors. When you select a BER value from the legend, it displays the contour for the selected BER in the same color. Initially all the BER values on the legend are struck across. When the contour is measured, the BER value changes into normal font; and when the contour is extrapolated the BER value changes into italics. A BER value for which a Contour does not exist is struck across. You can select multiple BER values. The screen shot below displays the contour and the legend.



The outer contour represents the measured BER contour, while the inner lines are extrapolated.

The Minimum Number of Errors indicates the minimum number of hits required to plot the contour.

Eye Diagram Measurement Parameters

Acquisition Parameters

Name	Description
Analyzer	Location or location group against which the data acquisition is performed.
Persistence	Set the criteria for Persistence.

Name	Description
Infinite	With 'Infinite', the eye appears with the Automated Eye Parameter Measurement, and the results keep on improving in accuracy, as more number of points get measured and represented on the diagram along with the deeper BER values. The BER threshold in the Graph Tab decides how long the measurements will be done at the boundaries of the eye. Once the measured points exceed the BER Threshold defined by the user, the measurement is done deep inside the eye. This is a default setup for persistence.
Fixed Time (Secs)	With 'Secs', the measurement runs for the specified time, and the measurement automatically restarts showing only the refined eye diagram. To stop the measurement, click the abort button.
Number of eyes	Set the criteria for the number of eyes. 1.5ui This option displays '1.5' eyes on the graph. The transitions of the complete eye are placed such, that, they show '0.25' eyes on both sides. So, the total number of displayed eyes is 1.5. 2ui This option displays '2.0' eyes on the graph. The transitions of the complete eye are placed such, that, they show '0.5' eyes on both sides. So, the total number of displayed eyes is 2.0.

Evaluation Parameters

Name	Description
Calculate Results For	Set the criteria for calculating the eye width, eye height, JPP, JRMS, cross voltage according to the BER threshold selected.
0 Errors	To see '0 errors' select the option. This gives the results according to the last measured contour for the current measurement. This is more significant in the case of deep BERs.
BER Threshold	Enter the BER threshold at which the five results, eye width, eye height, JPP, JRMS, and cross voltage, will be calculated. The show checkbox displays the contour at which the results are calculated.
Transition Time	Set criteria for transition time 10/90 Measures horizontal scan from 10% to 90% of the signal amplitude. 20/80 Measures horizontal scan from 20% to 80% of the signal amplitude.

Graph Settings

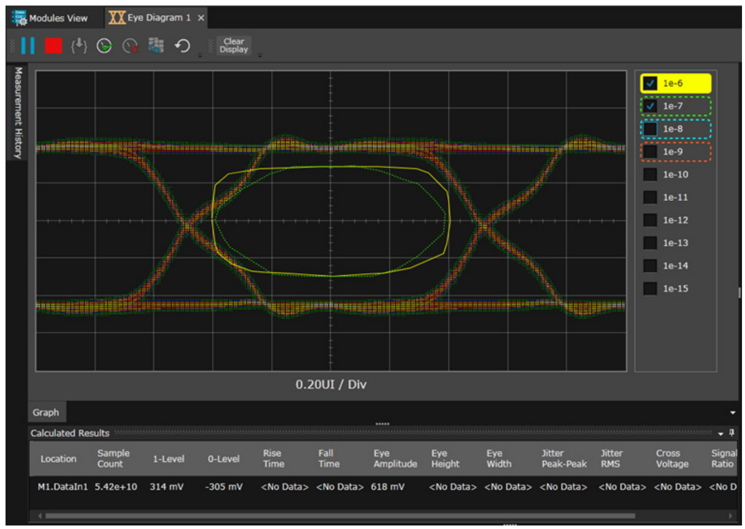
Name	Description
Display Unit	Set the criteria for the Timing Units: Unit Interval Unit used to measure delay relative to the eye width. Seconds Unit used to measure eye width in absolute terms of time.
Show Waveform	This option shows the waveform graphic.
Smooth Waveform Graphics	The smooth waveform gives a finer waveform output. It logarithmically interpolates between the measured points to give a finer waveform.
Waveform BER Threshold	The BER Threshold value allows the user to measure around the eye boundaries first, and then, deeper inside the Eye for BERs beyond the specified threshold.
Contours	This option plots the contour.
Contour Legend	This shows the list of BER values.

Lunching the Eye Diagram Measurement

To launch the **Eye Diagram Measurement**:

- Go to **Menu Bar** > **Measurements** and then select **Eye Diagram**.

The **Eye Diagram** user interface will appear as shown in the following figure:



The **Eye Diagram** user interface includes the following GUI elements:

- **Toolbar**: For details, refer to Toolbar on page 296.
- **Status Indicator**: For details, refer to Status Indicator on page 297.
- **Measurement History Window**: For details, refer to Measurement History Window on page 297.
- **Results Pane**: For details, refer to Calculated Eye Diagram Results on page 362.

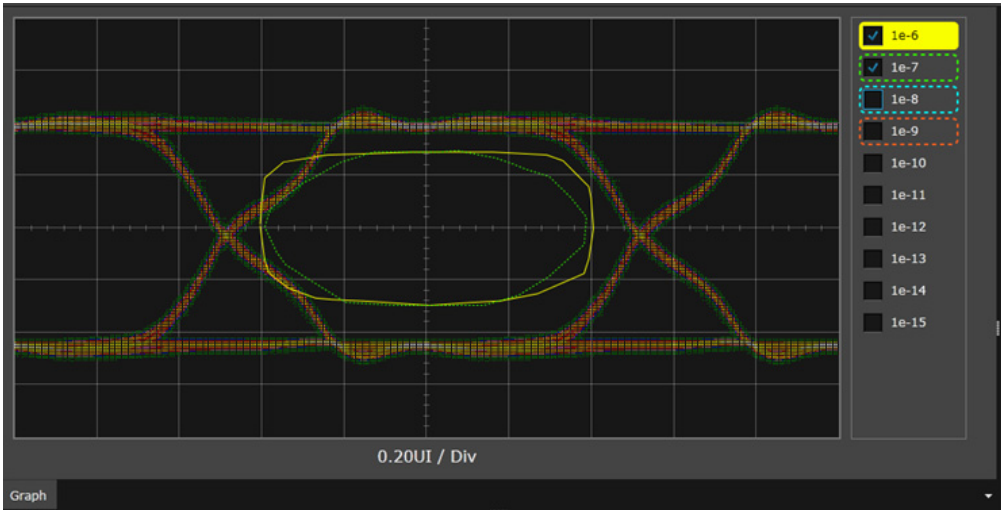
How to Start/Abort an Eye Diagram Measurement

To initiate the eye diagram measurement:

1. Click the Eye Diagram menu item from the Analysis submenu.
2. Click the Start button to execute the measurement.

The measurement runs, and the eye diagram along with the Calculated Eye Diagram Results get updated constantly.

The following graph shows the eye diagram.



3 To stop the measurement click on Abort.

How to Change the Default Settings

To achieve desired results, you can change the measurement parameters through the Parameters window.:

1 Click the Properties button to open the Properties dialog box.

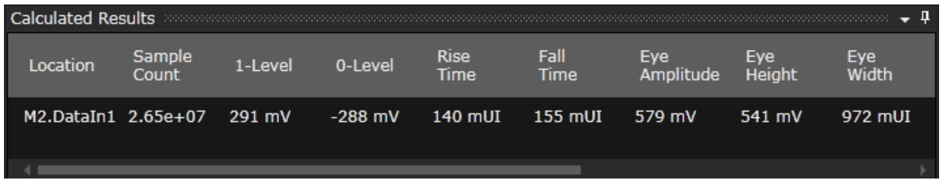
2 Use the different tabs in this dialog box to make the required settings:

For details, see Acquisition, Evaluation and Graph Settings.

Please note that you can only change the Evaluation and Graph Settings while the measurement is running.

Calculated Eye Diagram Results

The following results are displayed when an eye diagram measurement is completed:



Location	Sample Count	1-Level	0-Level	Rise Time	Fall Time	Eye Amplitude	Eye Height	Eye Width
M2.DataIn1	2.65e+07	291 mV	-288 mV	140 mUI	155 mUI	579 mV	541 mV	972 mUI

- **Location**: Location or Location/Group against which the data acquisition is performed.
- **Sample Count**: The number of points measured for the eye diagram. If persistence, ‘infinite’, is enabled, the value will gradually increase.
- **Error Bits**: Displays the number of errored bits during the accumulation period.
- **1-Level:** One Level is a measure of the mean value of the logical 1 of an eye diagram.
- **0-Level:** Zero level is the measurement of the mean value of the logical 0 of an eye diagram.
- **Rise time:** Measures the mean transition-time taken by the data on the rising edge of the eye diagram. The data crosses three thresholds: lower, crossover point, upper and the eye transition.

  Note: Rise time is dependent on the Transition Time of the View Tab. The two options are 10/90 and 20/80.
- **Fall time:** Measures the mean transition-time taken by the data on the falling edge of the eye diagram. The data crosses three thresholds: lower, crossover point, upper and the eye transition.

  Note: Fall time is dependent on the Transition Time of the View Tab. The two options are 10/90 and 20/80.
- **Eye Amplitude:** The difference between the logic 1 level and the logic 0 level.
- **Eye Height:** Measures the vertical opening of an eye diagram with respect to BER threshold. This determines “eye closure” due to noise.

  Note: The Eye Height is calculated according to the BER Threshold set in the View Tab. Eye Height is calculated with 'Height at' in the View tab.

- **Eye Width:** The horizontal measurement of the eye opening at a specified BER Threshold.

  Note: The Eye Width is calculated according to the BER Threshold set in the View Tab. Eye Width is calculated as per the selection of either Width at Crossing Point or Custom defined Width.

- **Jitter Peak-Peak:** Full width of the histogram at the eye crossing point with respect to BER threshold.

  Note: The Jitter P-P is calculated according to the BER Threshold set in the Parameter window.

- **Jitter RMS:** A standard deviation of the crossing point histogram with respect to BER threshold.

  Note: The Jitter RMS is calculated according to the BER Threshold set in the Parameter window.

- **Cross Voltage:** Crossing percentage is a measure of the amplitude of the crossing points relative to the 1 level and 0 level.

  Note: The Cross Voltage is calculated according to the BER Threshold set in the Parameter window.

- **Signal to Noise Ratio:** In signal-to-noise, the "signal" is the information power of the signal indicated by the difference between the ‘1’, and ‘0’ level. The "noise" is the combined standard deviations of the ‘1’ level spread, and the ‘0’ level spread.
- **Duty Cycle Distortion:** This value is the difference between the period of a 1 bit and a 0 bit.

9 Utilities

Overview

The M8070A software provides the following utilities:

- Script Editor
- DUT Control Interface
- SCPI Editor
- Self Test Utility
- Licenses Window
- Settings Window
- Logger Window
- SCPI Server Information
- Plug-in Manager Window

Script Editor

The **Script Editor** provides flexibility to the programmers to automate their plug-ins thus allowing them to do everything they want in the measurements.

The **Script Editor** uses the IronPython programming language which is a flexible and powerful language suitable to a range of tasks. The IronPython is an open-source implementation of the Python programming language which is tightly integrated with the .NET Framework. IronPython can use the .NET Framework and Python libraries, and other .NET languages can use Python code just as easily. Whether you're interested in writing full applications, tackling scripting tasks, or embedding a scripting language into another application, IronPython has something to offer you.

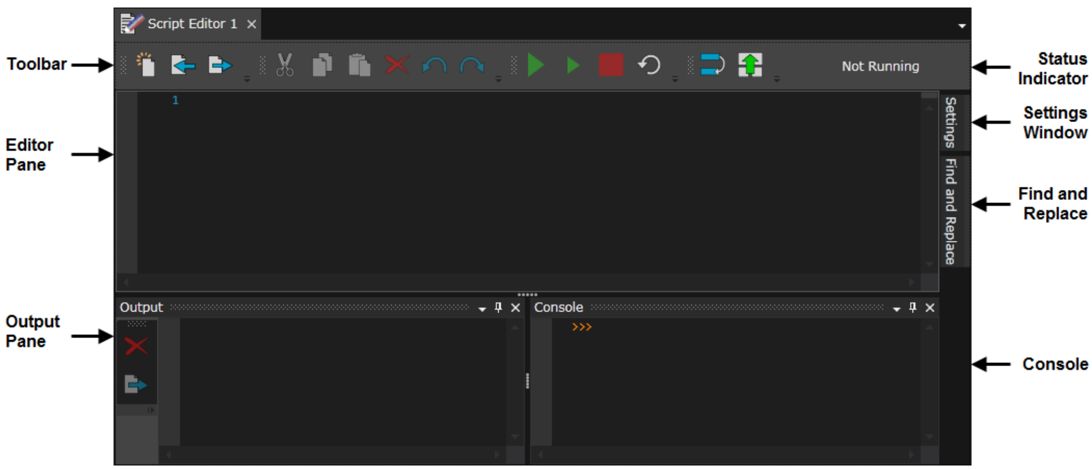
For more details on the IronPython, please refer to the following web-page: http://ironpython.net

Launching the Script Editor

To launch the **Script Editor**:

- Go to the **Menu Bar** > **Utilities** and then select **Script Editor**.

The **Script Editor** interface will appear as shown in the following figure:



The **Script Editor** interface consists of the following GUI elements:

- Toolbar
- Editor Pane
- Output Pane
- Console
- Settings Window
- Find and Replace Dialog

The listed GUI elements are described in the section that follows.

Toolbar

The **Script Editor** toolbar provides the following convenient script editing functions:

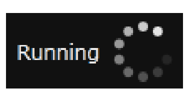
Table 65

Icon	Name	Keyboard Shortcut	Description
	New	CTRL + N	Click this icon to create a new project/module. For details, refer to Creating a Script on page 376
	Save	CTRL + S	Click this icon to save the current project/module. For details, refer to Saving a Script on page 376.
	Import	CTRL + O	Click this icon to import external project/module (.PY) in to the current project/module. For details, refer to Importing a Script on page 376.
	Export	CTRL + E	Click this icon to export the project/module (.PY) to the local drive. For details, refer to Exporting a Script on page 377.
	Cut	CTRL + X	
	Copy	CTRL + C	These functions follow Microsoft Windows conventions.
	Paste	CTRL + V	
	Delete	DEL	
	Undo	CTRL + Z	Click this icon to undo your last action.
	Redo	CTRL + SHIFT + Z	Click this icon to redo your last action.

Icon	Name	Keyboard Shortcut	Description
	Start Script	F5	Click this icon to execute the script. For details, refer to Running a Script on page 377.
	Start Selected	F8	Click this icon to execute the select portion of the script.
	Stop Script	F6	Click this icon to stop the currently running script.
	Reset local variables and loops		Click this icon to reset the local variable and loops in the script.
	Duplicate Row/Sele ction	CTRL + D	Click this icon to duplicate the code of the current cursor position or selected code to a new row. For details, refer to Adding Duplicate Row/Selection in the Code on page 378.
	Restore default window layout		Click this icon to restore the default window layout.

Status Indicator

The **Status Indicator** indicates the current state of the script. It is located on the right side of the toolbar.



It shows the following status:

- **Running** - When the script is being executed.
- **Not Running** - When the script is either not executed or the execution is completed.
- **Failed** - When the script execution fails.

Editor Pane

The **Editor** pane allows you to:

- Write, edit, debug and run the scripts
- Open, save, import and export the scripts
- Debug the script code
- Run the entire script or selected lines from the script
- Apply settings to the editor using the **Settings** window

The **Editor** pane is an area where you enter code and execute it immediately. It also allows the execution of a single line of script code, with the resulting messages printed to the **Output** pane. For details, refer to Output Pane on page 372. In addition, the **Editor** pane has many features to help you enter, edit, check syntax, and debug script code. Beside this, it also allows you to use the Microsoft Window's standard functionally such as cut, copy, paste, undo and redo operations in the script code.

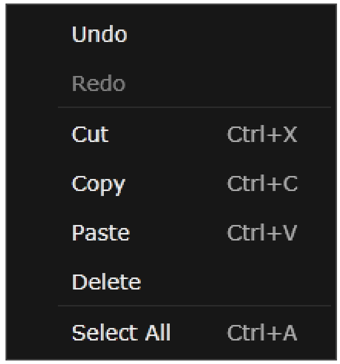
The following figure shows a simple program written in the **Editor** pane.

```
prices = {'apple': 0.40, 'banana': 0.50}
my_purchase = {
    'apple': 1,
    'banana': 6}
grocery_bill = sum(prices[fruit] * my_purchase[fruit]
                   for fruit in my_purchase)
print 'I owe the grocer $%.2f' % grocery_bill
```

Context Menu

The **Context** menu appears when you right-click on the editor pane. It provides the Window's standard functionally such as undo, redo, cut, copy, paste, delete and select all operations in the script code.

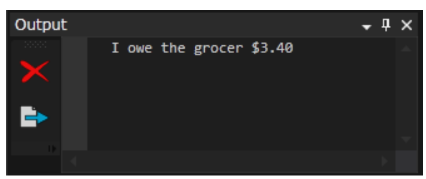
The following figure shows the options provided by the **Context** menu:



Output Pane

The **Output** pane displays the result of the executed script code. However, if the interpretor finds any error in the code, it is displayed in this pane.

The following figure shows an output of a program shown in an **Output** pane:



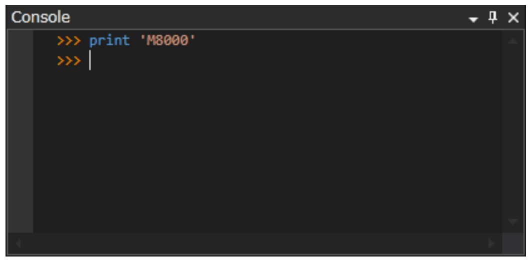
It provides the following options:

Clear Window - Click the  **Clear Window** icon to clear the output pane.

Export - Click the  **Export** icon to open a Window's standard **Save As** dialog to save the content of the **Output** pane in a log file.

Console

The **Console** allows you to enter, interact with and visualize data, inside a command interpreter. All the commands entered in the **Console** are executed in a separate process, thus allowing you to interrupt any process at any time.



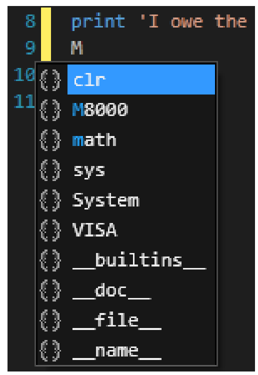
Settings Window

The **Settings** window provides settings for **Editor** pane, **Output** pane and **Console**.

The following table lists the settings for **Editor** pane:

Table 66

Settings	Description
Show Whitespace	Click this switch to show whitespace in the code.
Convert Tabs to Spaces	Click this switch to convert tabs to spaces in the code.
Tab Size	Use this setting to increase/decrease the tab size in the code.

Settings	Description
Word Wrapping	Click this switch to enable word wrapping in the code.
Zoom	Use this setting to zoom in or zoom out the code.
Code Completion	Click this switch to enable code completion feature. It saves the key strokes for caret movement when coding which can greatly reduce typing required for coding. The following figure shows how the code completion feature can reduce the typing while coding: 

The following table lists the settings for **Output** pane:

Table 67

Settings	Description
Word Wrapping	Click this switch to enable word wrapping.
Zoom	Use this setting to zoom in or zoom out the output of the program executed.
Autoscroll Output	Click this switch to enable autoscroll feature.

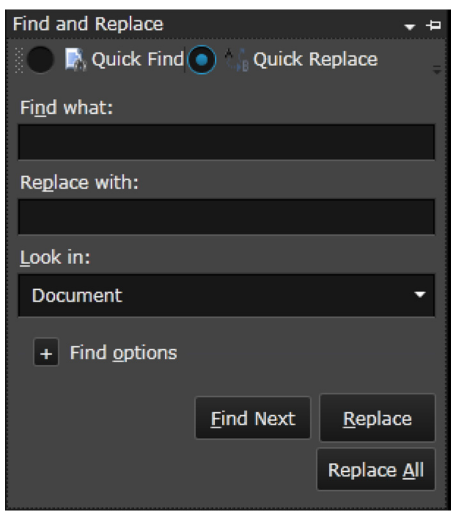
The following table lists the settings for **Console**:

Table 68

Settings	Description
Zoom	Use this setting to zoom in or zoom out.
Code Completion	Click this switch to enable code completion feature. It saves the key strokes for caret movement when coding which can greatly reduce typing required for coding.

Find and Replace

The **Find and Replace** dialog provides the search and replace capability in the script. The **Quick Find** option allows you to locate a specified text string while the **Quick Replace** option allows you to locate and replace a specified text string.



The **Find and Replace** dialog provides the following options:

- **Find What** - Displays the search string you want to locate.
- **Replace With** - Displays the string that will replace the string you searched for.

- **Look In** - Provides the search operation either in the entire document (script) or in the selection (selected part of the script).
- **Find Options** - Provides several other options (Match case, Match whole word, Search up and Search type) in the find dialog that you can use to customize your search.
- **Find Next** - Locates the next instance of the search string.
- **Replace** - Displays the string that will replace the string you searched for.
- **Replace All** - Replaces all instances where the search string is found.

Creating a Script

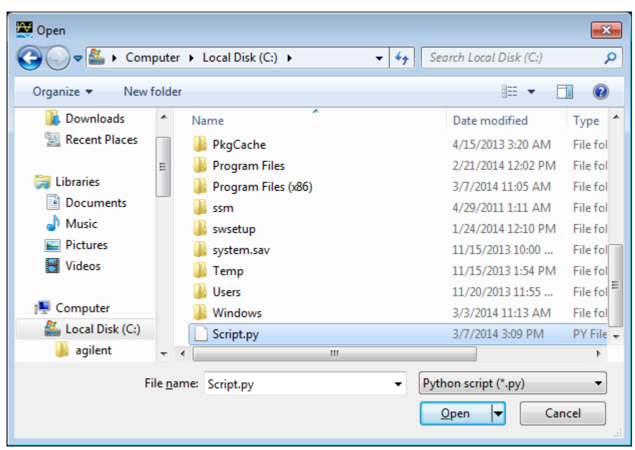
To create a script, click the **New** icon present on the toolbar. You can use **Editor** pane to write a script code.

Saving a Script

To save a current script, click the **Save** icon present on the toolbar.

Importing a Script

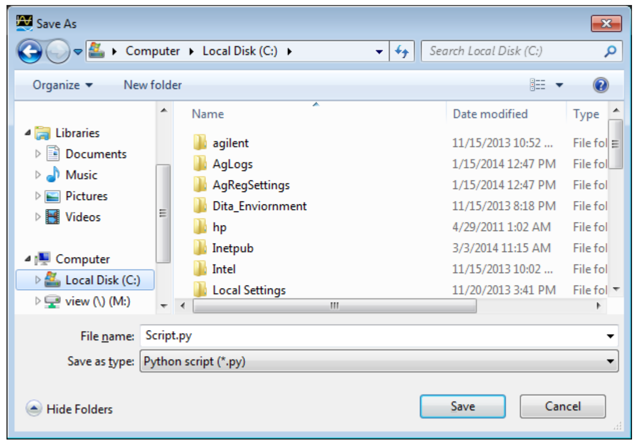
To import a script, click the **New** icon present on the toolbar. A Window's standard **Open** dialog will appear.



- Locate the python script file (.PY) you want to import.
- Click **OK**.

Exporting a Script

To export a script, click the **New** icon present on the toolbar. The Window's standard **Save As** dialog will appear.



- Select the location where you want to export the script.
- Enter a file name in the **File Name** text field.
- Click **Save**.

Running a Script

You can run either the complete script code or the selected part. To do so;

- Click the **Start Script** icon present on the toolbar to execute the entire script.
-
- Click the **Start Selected** icon present on the toolbar to execute the selected part of the script.

Verify the status of the executed scripts though **Status Indicator**. Refer to Status Indicator on page 370.

The output of the scripts will be displayed in the **Output** pane. However, if the scripts fails, the error will also be shown in the **Output** pane.

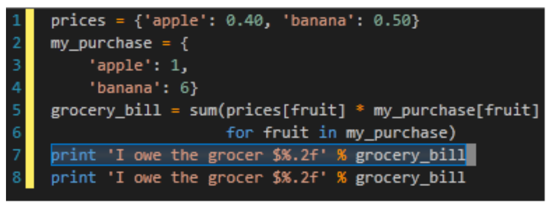
Stopping a Script

To stop the currently running script, click the **Stop** icon present on the toolbar.

Adding Duplicate Row/Selection in the Code

To create a copy of the current row of the code or the selected part, click the **Duplicate Row/Selection** icon.

The following figure shows the row being duplicated using the **Duplicate Row/Selection** option.



DUT Control Interface

The **DUT Control Interface** feature allows you to control a device under test at well defined positions within the measurements of the M8070A software. It allows you to:

- Read out error count information from devices with built-in error counters.
- Read out and write devices status registers
- Import and edit of Python scripts to access DUT from M8070A system software
- Integrate devices built-in error count information in error monitoring and jitter tolerance test routines of M8070A.

The **DUT Control Interface** editor provides inserting code snippets of the supported DUT. This allows you to tailor the supported measurements in a way that the DUT is controlled at critical points during the measurement and also provide error information for the measurement. Currently, the **DUT Control Interface** supports using a DUT's built-in error counters in the following measurements:

- Error Ratio Measurement
- Jitter Tolerance Measurement

All other measurements provide callback functions as well. These are useful to tailor the measurement to the DUT (e.g. settling times in dependency of measurement parameters), or to control the device under test during the measurement.

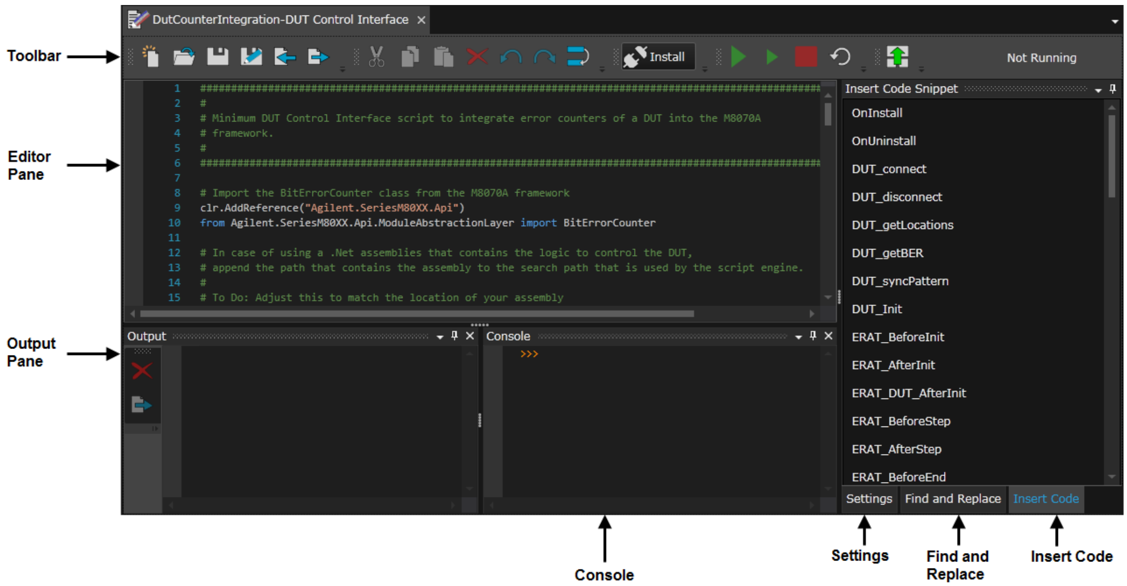
The **DUT Control Interface** feature is enabled by the license M8070A-1TP or M8070A-1NP.

Launching the DUT Control Interfacer

To launch the **DUT Control Interface**:

- Go to the **Menu Bar** > **Utilities** and then select **DUT Control Interface**.

The **DUT Control Interface** interface will appear as shown in the following figure:



The **DUT Control Interface** interface consists of the following GUI elements:

- Toolbar
- Editor Pane
- Output Pane
- Console
- Insert Code
- Settings Window
- Find and Replace Dialog

The listed GUI elements are described in the section that follows.

Toolbar

The toolbar provides the following convenient script editing functions:

Table 69

Icon	Name	Description
	New	Click this icon to open a new DUT script. For details, refer to Creating a Script on page 376
	Open	Click this icon to open an existing DUT script.
	Save	Click this icon to save the current DUT script. For details, refer to Saving a Script on page 376.
	Save As	Click this icon to save the current DUT script under different file name at either shared or current locations. For details, refer to Saving a Script on page 376.
	Import	Click this icon to import external DUT script (.PY) in to the editor. For details, refer to Importing a Script on page 376.
	Export	Click this icon to export the DUT script (.PY) to the local drive. For details, refer to Exporting a Script on page 377.
	Cut	These functions follow Microsoft Windows conventions.
	Copy	
	Paste	
	Delete	
	Undo	Click this icon to undo your last action.

Icon	Name	Description
	Redo	Click this icon to redo your last action.
	Duplicate Row/Selection	Click this icon to duplicate the code of the current cursor position or selected code to a new row. For details, refer to Adding Duplicate Row/Selection in the Code on page 378.
	Install	Click this button to install or uninstall the DUT control interface. Installing the DUT Control Interface script integrates the implemented functions into the M8070A software and makes the built-in error counters available for measurements and the remote interface. Editing, or interactive execution of the script is no longer possible when the script is installed.
	Start Script	Click this icon to execute the script. For details, refer to Running a Script on page 377. Running the script in the editor is useful to debug the script functionality. The individual functions can be executed from the Console after the script has been executed.
	Start Selected	Click this icon to execute the select portion of the script. This is useful to update the functionality available on the console after making changes to parts of the script, without executing all of the script. This avoids other side effects like resetting global variables that can be unwanted during testing the script.
	Stop Script	Click this icon to stop the currently running script.
	Reset local variables and loops	Click this icon to reset the local variable and loops in the script.
	Restore default window layout	Click this icon to restore the default window layout.

Status Indicator

The **Status Indicator** indicates the current state of the script. It is located on the right side of the toolbar.

It shows the following status:

- **Running** - When the script is being executed.
- **Not Running** - When the script is either not executed or the execution is completed.
- **Failed** - When the script execution fails.

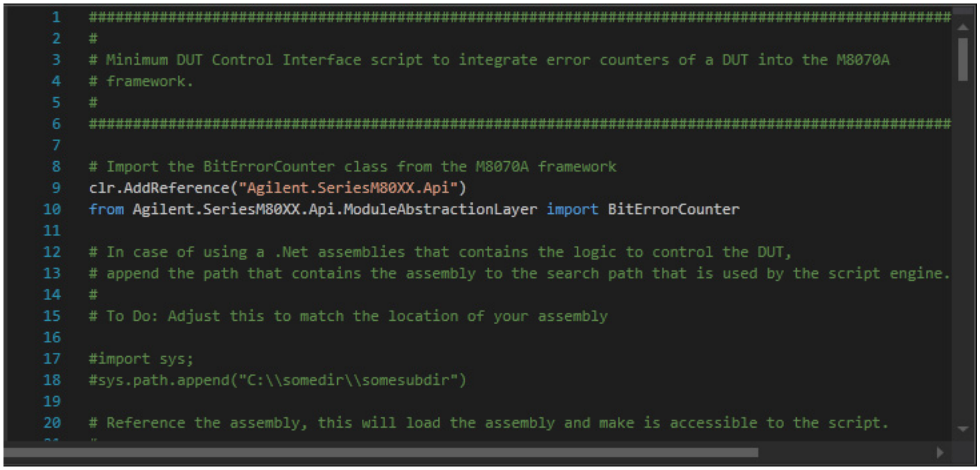
Editor Pane

The **Editor** pane allows you to:

- Write, edit and run the scripts
- Open, save, import and export the scripts
- Run the entire script or selected lines from the script
- Apply settings to the editor using the **Settings** window

The **Editor** pane is an area where you enter code and execute it immediately. It also allows the execution of a single line of script code, with the resulting messages printed to the **Output** pane. For details, refer to Output Pane on page 372. In addition, the **Editor** pane has many features to help you enter, edit and check syntax of the code. Beside this, it also allows you to use the Microsoft Window's standard functionally such as cut, copy, paste, undo and redo operations in the script code.

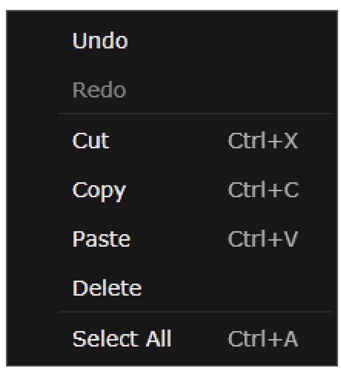
The following figure shows a simple program written in the **Editor** pane.



Context Menu

The **Context** menu appears when you right-click on the editor pane. It provides the Window's standard functionally such as undo, redo, cut, copy, paste, delete and select all operations in the script code.

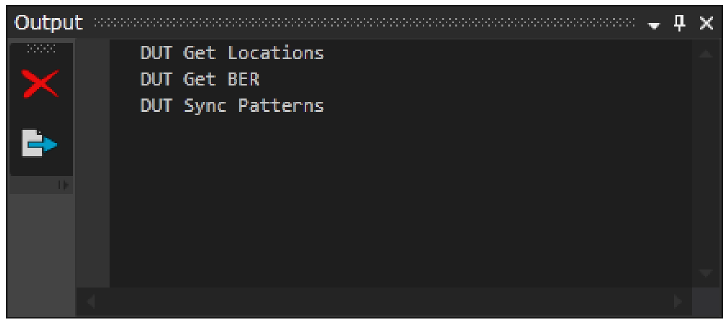
The following figure shows the options provided by the **Context** menu:



Output Pane

The **Output** pane displays the result of the executed script code. However, if the interpretor finds any error in the code, it is displayed in this pane.

The following figure shows an output of a program shown in an **Output** pane:



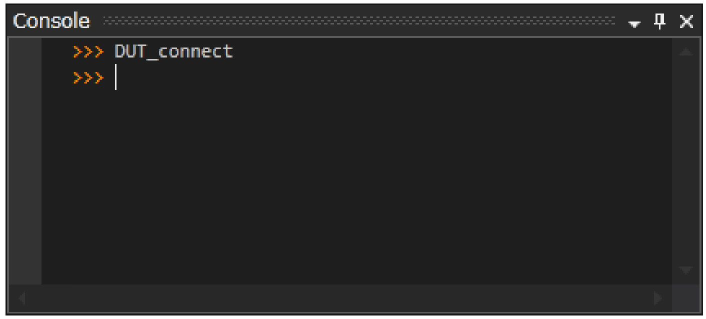
It provides the following options:

Clear Window - Click the  **Clear Window** icon to clear the output pane.

Export - Click the  **Export** icon to open a Window's standard **Save As** dialog to save the content of the **Output** pane in a log file.

Console

The **Console** allows you to enter, interact with and visualize data, inside a command interpreter. All the commands entered in the **Console** are executed in a separate process, thus allowing you to interrupt any process at any time.



Insert Code Window

The **Insert Code** window lists code snippets that are supported by the DUT Control Interface. You can double-click on the code snippet to insert them in the **Editor** pane. The code snippets ensure correct signature of the supported functions and contain documentation of the expected functionality and the function's arguments.

Please note that you are not allowed to modify the script when the DUT control script is installed.

Also, the **Output** pane and **Console** will not be available when DUT control script is installed.

Settings Window

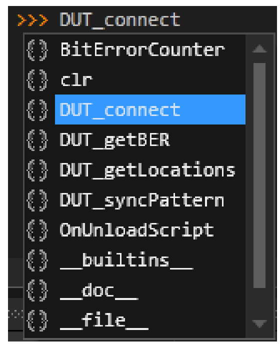
The **Settings** window provides settings for the **Editor** pane, **Output** pane and **Console**.

The following table lists the settings for the **Editor** pane:

Table 70

Settings	Description
Show Whitespace	Click this switch to show whitespace in the code.
Convert Tabs to Spaces	Click this switch to convert tabs to spaces in the code.
Tab Size	Use this setting to increase/decrease the tab size in the code.

Settings	Description
Word Wrapping	Click this switch to enable word wrapping in the code.
Zoom	Use this setting to zoom in or zoom out the code.
Code Completion	Click this switch to enable code completion feature. It saves the key strokes for caret movement when coding which can greatly reduce typing required for coding. The following figure shows how the code completion feature can reduce the typing while coding:



The following table lists the settings for the **Output** pane:

Table 71

Settings	Description
Word Wrapping	Click this switch to enable word wrapping.
Zoom	Use this setting to zoom in or zoom out the output of the program executed.
Autoscroll Output	Click this switch to enable autoscroll feature.

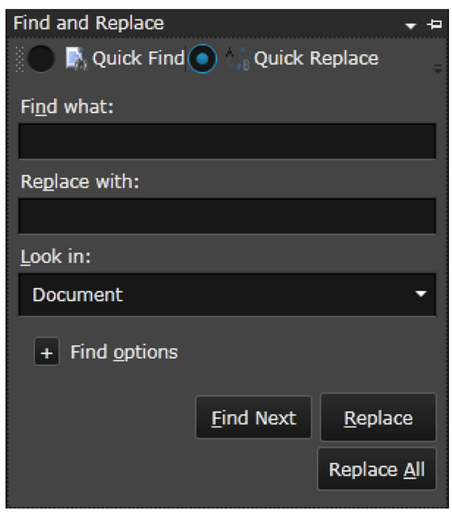
The following table lists the settings for **Console**:

Table 72

Settings	Description
Zoom	Use this setting to zoom in or zoom out.
Code Completion	Click this switch to enable code completion feature. It saves the key strokes for the caret movement when coding which can greatly reduce typing required for coding.

Find and Replace

The **Find and Replace** dialog provides the search and replace capability in the script. The **Quick Find** option allows you to locate a specified text string while the **Quick Replace** option allows you to locate and replace a specified text string.



The **Find and Replace** dialog provides the following options:

- **Find What** - Displays the search string you want to locate.
- **Replace With** - Displays the string that will replace the string you searched for.

- **Look In** - Provides the search operation either in the entire document (script) or in the selection (selected part of the script).
- **Find Options** - Provides several other options (Match case, Match whole word, Search up and Search type) in the find dialog that you can use to customize your search.
- **Find Next** - Locates the next instance of the search string.
- **Replace** - Displays the string that will replace the string you searched for.
- **Replace All** - Replaces all instances where the search string is found.

Creating a New Script

To create a new script, click the **New** icon present on the toolbar. A default script will be loaded in the **Editor** pane which you can edit to create a new script.

Open a Script

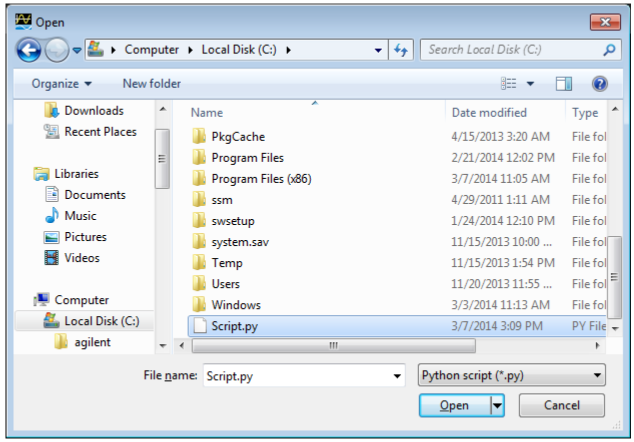
To open a saved script, click the **Open** icon present on the toolbar. A **Open Hooks Script** dialog will appear which allows you to open the DUT scripts.

Saving a Script

To save the current script, click the **Save** icon present on the toolbar. A **Save Hooks Script** dialog will appear which allows you to save the scripts on either shared or current folder.

Importing a Script

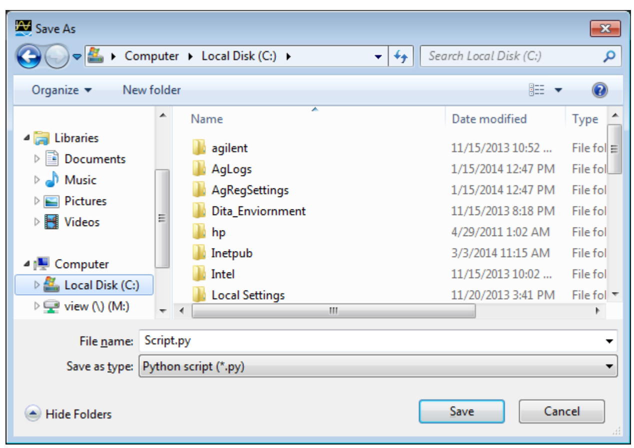
To import a script, click the **Import** icon present on the toolbar. A Window’s standard **Open** dialog will appear.



- Locate the Python script file (.PY) you want to import.
- Click **OK**.

Exporting a Script

To export a script, click the  **Export** icon present on the toolbar. The Window's standard **Save As** dialog will appear.



- Select the location where you want to export the script.
- Enter a file name in the **File Name** text field.
- Click **Save**.

Running a Script

You can run either the complete script code or the selected part. To do so;

- Click the **Start Script** icon present on the toolbar to execute the entire script.
- Click the **Start Selected** icon present on the toolbar to execute the selected part of the script.

Verify the status of the executed scripts though **Status Indicator**. Refer to Status Indicator on page 370.

The output of the scripts will be displayed in the **Output** pane. However, if the scripts fails, the error will also be shown in the **Output** pane.

Stopping a Script

To stop the currently running script, click the **Stop** icon present on the toolbar.

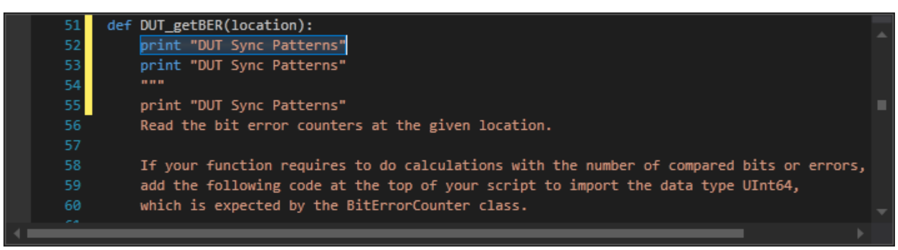
Stopping the script does not always work. If the script is using non-Python code (e.g. calling functions in .Net libraries), then the execution of these functions cannot be stopped in all cases.

Adding Duplicate Row/Selection in the Code

To create a copy of the current row of the code or the

selected part, click the **Duplicate Row/Selection** icon.

The following figure shows the row being duplicated using the **Duplicate Row/Selection** option.

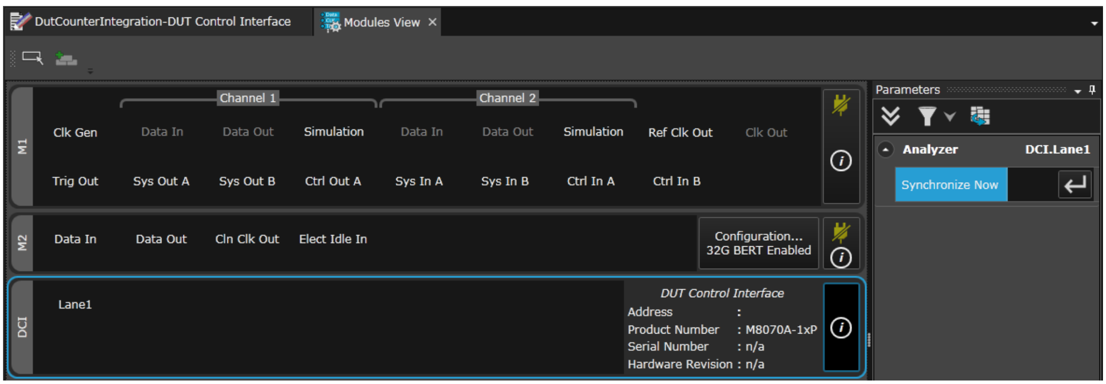


Effects on M8070A GUI on Installing a DUT Control Script

On successful installation of DUT control scripts, the following changes are reflected in M8070A GUI.

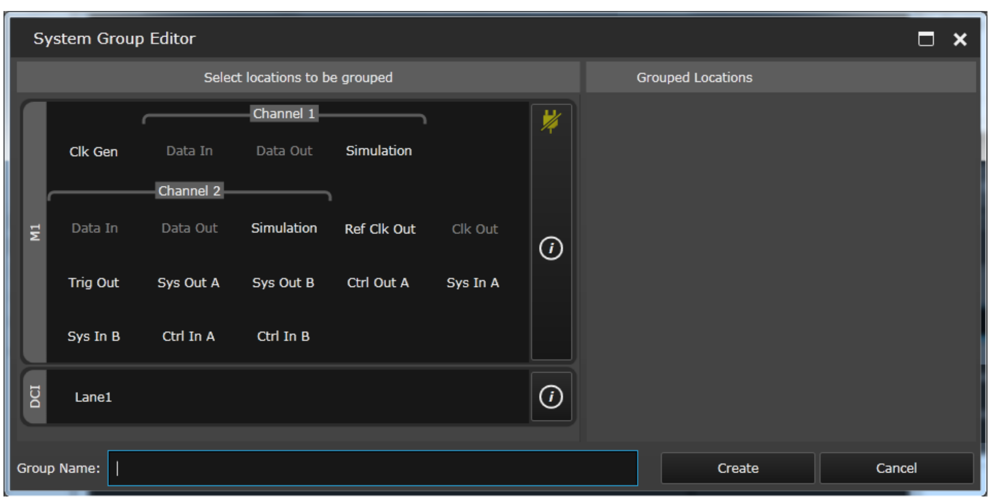
Module View

A new DUT control along with the supported parameters is now added in the **Module View**:



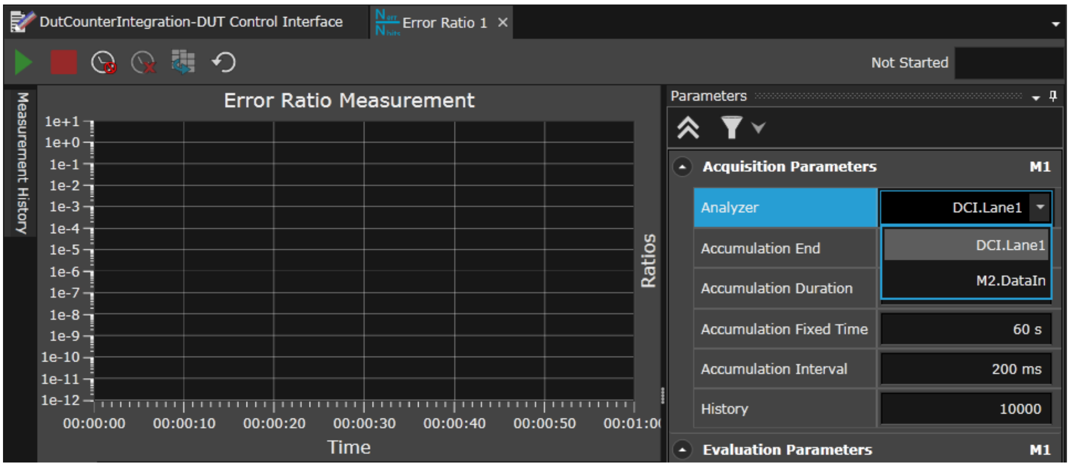
Group View

New locations will be now available to create a group in the **Group View**.



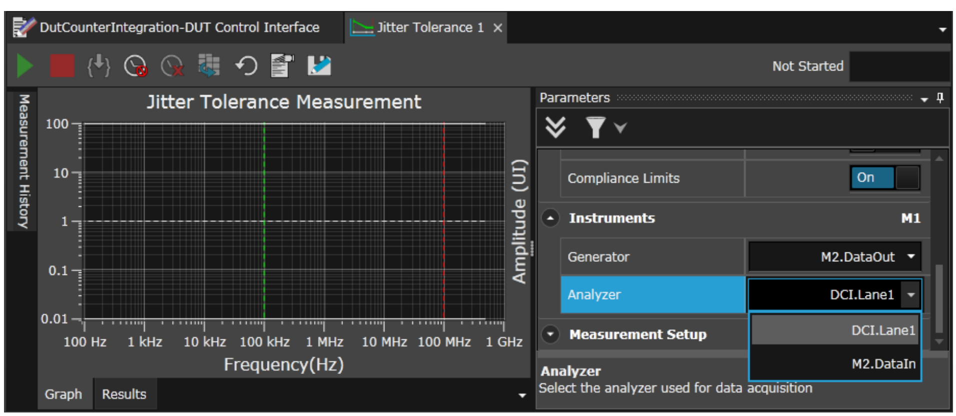
Error Ratio Measurement

A new location will be now available in the **Analyzer** parameter of the **Error Ratio Measurement**.



Jitter Tolerance Measurement

A new location will be now available in the **Analyzer** parameter of the **Jitter Tolerance Measurement**.



Writing a Script Code for DUT Control Interface

Scripting Runtime

- The underlying scripting technology uses IronPython 2.7.5
- The M8070A does not include the full IronPython 2.7.5 installation, only the scripting engine itself. When referencing the modules of an IronPython installation that is available on the computer, it must be ensured that the versions match.
- IronPython makes it very easy to use functionality provided by the Microsoft .NET framework, but it cannot directly access dynamic link libraries that are not based on .NET.
  Functions that are implemented in unmanaged DLLs (usually implemented in C or C++) do require a .NET wrapper library.
  For details see:
  https://msdn.microsoft.com/en-us/library/ms235282.aspx
  Title: Calling Native Functions from Managed Code
- Python functions do not have out parameters. If a function is using out parameters, then these will be returned as additional return values.

  Example:

  - **C# function**

```
bool DoSomething(int arg1, out double resultArg1,
  double arg2, out int resultArg2,
  out string resultArg3)
```

  - **IronPython usage**

```
(returnValue, resultArg1, resultArg2, resultArg3) =
DoSomething(arg1, arg2)
```

- Remote commands to control M8070A cannot be used within the DUT Control Interface scripts.
  While this is possible in principle, it will cause problems when the functions implemented in the DUT Control Interface are being executed in the context of remote commands (e.g. query error counters via :fetc:bco?). In this case the remote interface is busy until the DUT counters have been returned by the script function DUT_getBER, but while the remote interface is busy it cannot process any further remote commands, neither internally issued nor external ones. This can very easily end up in deadlock/timeout scenarios.
- Code outside of function definitions is being executed at load time of the script.
  This can be used to initialize required variables, or establish the communication link with the DUT.

Minimum Requirements to Integrate a DUT's Error Counters

- Implement DUT_getLocations() to return a unique name for each location within the DUT that does provide an error counter (e.g. Lane1, Lane2, Lane3, Lane4)

  Example:

  The following example defines DUT_getLocations to define 4 locations named Lane to Lane 4:

```
def DUT_getLocations():
    return ["Lane1", "Lane2", "Lane3", "Lane4" ]
```

- Implement DUT_getBER(location) to return the error counter for each of the defined locations.

  Example:

  The following example reads BER counter from one of 4 Lanes. Each Lane is assumed to be controlled by an instance of a driver class that implements the IBerReader interface (see code snippet below). The instances of the 4 lanes are held in a dictionary that allows easy lookup by the given location name.

```
# create empty dictionary
myDUTs = {}
# instantiate the driver for each location
myDUTs["Lane1"] = MyDutBerReader(1)
myDUTs["Lane2"] = MyDutBerReader(2)
myDUTs["Lane3"] = MyDutBerReader(3)
myDUTs["Lane4"] = MyDutBerReader(4)

def DUT_getBER(location):
    global myDUTs
    (bits, errors) = myDUTs[location].GetCounter()
    return BitErrorCounter(bits, errors)
```

- Implement DUT_synchPattern(location) to synchronize the error counter to the received data stream.

  Optionally reset the DUT, but be aware that this should not have side effects on other locations.

  The M8070A framework expects the DUT's error counters to be running all the time. If the synchronization occurs on a stopped to running transition, then the sop as well as the start need to be part of DUT_syncPattern(location).

```
def DUT_syncPattern(location):
  global myDUTs
  lane = myDUTs[location]
  lane.Stop()
  lane.ResetDut()
  lane.Start()
```

- Optionally implement DUT_Init(location, initArg) if it is required to configure the DUT to different modes.

  DUT_Init can be called from the remote programming interface, so the test automation is able to execute initialization code on the DUT programmatically.

  The M8070A software will not call DUT_Init unless the script is using it internally, or it is either requested from the remote programming interface, or interactively from within the DUT Control Interface script editor.

NOTE

- Making script code executable from within the remote programming interface can cause a security issue in your computer.
- The M8070A software is not able to limit the user script's functionality. It is the user‘s responsibility to ensure that no unsafe or otherwise critical code is being executed from within DUT_Init.
- Special care should be taken when using the python statements eval or exec, as these allow dynamic execution of python code.
- Keysight recommends not using eval or exec at all, except the DUT or application requires this.
- Keysight is not liable for any kind of damage caused by the use of remote executable script code (see *Limitation of Liability in End User License Agreement*).

Useful Sources of Information

- http://ironpython.net/

  Home of the IronPython project.

  Especially the IronPython .Net Integration documentation is a valuable source of information regarding access to the .Net framework (http://ironpython.net/documentation/dotnet/)
- https://www.python.org/

  Home of the python programming language. This is the preferred souorce of information regarding to the language syntax itself.

  When referring the language documentation make sure to use the Python 2.7 documentation.

  Keep in mind that Python is not the same as IronPython, so Python libraries or modules are not necessarily working with IronPython.
- https://msdn.microsoft.com/en-us/library/gg145045(v=vs.100).aspx

  Title: NET Framework Class Library

  Programming reference of the .NET framework that is accessible to IronPython scripts.

Useful Code Snippets

Importing core .NET functionality

The base .NET functionality is implemented in mscorlib.dll

Making a .NET data type available to the script requires loading the assembly and importing the data type.

Example: Importing and using System.UInt64

```
clr.AddReference("mscorlib")
from System import UInt64
comparedBits = UInt64(0)
comparedBits = comparedBits + UInt64(1e6)
```

Using .NET Generic Types

It is often required to use generic types when calling .NET code.

The C# code to instantiate a List of Strings looks like this:

```
using System;
using System.Collections.Generic;
var stringList = new List<string>()
```

The corresponding IronPython code:

```
clr.AddReference("mscorlib")
from System import String
from System.Collections.Generic import List
stringList = List[String]()
```

Using User Defined .NET Assemblies (e.g. Implementations of IBerReader)

In order to reference user defined assemblies, the search path of the scripting engine needs to be extended. After this is done the assembly reference can added and the data type imported.

```
# Add c:\MyLibraries\MyDUT to the search path import sys
sys.path.append("C:\\MyLibraries\\MyDUT")

# reference MyDutCustomBerReader.dll located in
c:\MyLibraries\MyDUT
clr.AddReferenceToFile("MyDutCustomBerReader")

# from the namespace MyNamespace.MyDutCustomBerReader
# that is defined in the previously referenced assembly,
# import the actual data type.
from MyNamespace.MyDutCustomBerReader import
MyDutBerReader

# create an instance of MyDutBerReader
myDUT = MyDutBerReader()

# call a function defined in the data type MyDutBerReader
myDUT.DoSomething()
```

Using the COM Port to Access the DUT

.NET does provide a dedicated class to work with the computer's COM ports. This class is implemented in the System assembly.

Importing the SerialPort class

```
# import SerialPort from System assembly
clr.AddReference("System")
import System.IO.Ports.SerialPort
```

```
# additional reference assembly mscorlib to have all data
types
# available that are used by SerialPort
clr.AddReference("mscorlib")
```

Open the COM Port

```
# create an instance of SerialPort for COM4
serialPort = System.IO.Ports.SerialPort("COM4")

# configure COM4 to 9600 bit/s 8N1
serialPort.BaudRate = int (9600)
serialPort.DataBits = int(8)
# available choices for Parity are
# None: System.IO.Ports.Parity.None
# Odd   : System.IO.Ports.Parity.Odd
# Even  : System.IO.Ports.Parity.Even
# Mark  : System.IO.Ports.Parity.Mark
# Space : System.IO.Ports.Parity.Space
serialPort.Parity = System.IO.Ports.Parity.None
# available choices for StopBits are
# None : System.IO.Ports.StopBits.None
# One  : System.IO.Ports.StopBits.One
# Two  : System.IO.Ports.StopBits.Two
# OnePointFive : System.IO.Ports.StopBits.OnePointFive
serialPort.StopBits = System.IO.Ports.StopBits.One
# set read and write timeouts to 500ms
serialPort.ReadTimeout = int(500)
serialPort.WriteTimeout = int(500)

# finally open the COM port
serialPort.Open()
```

Reading and Writing Data

```
# write a string
serialPort.Write("Sending string data")

# sending binary data

# prepare the data to be sent. SerialPort expects an Array
of Byte. This requires importing mscorlib!
```

```
buf = System.Array[System.Byte] ([0x5A, 0x02, 0x00,
0xaf])

# send 4 bytes beginning at offset 0
serialPort.Write(buf, 0, 4)

# reading 100 bytes
numBytes = 100
buf = System.Array.CreateInstance(System.Byte, numBytes)
serialPort.Read(buf, 0, numBytes)
```

Closing the COM Port

The following code snippet shows the required code to close the COM port and discard the SerialPort instance. This code should be called in the OnUninstall function of the DUT Control Interface in order to ensure that the COM port becomes unavailable after the currently active script becomes removed from the computer's memory.

```
if (serialPort != None):
  # check if COM port is open
  if (serialPort.IsOpen):
      serialPort.Close()
  # dispose the serialPort instance
  serialPort.Dispose()
  # ensure that the disposed object is not used in future
  serialPort = None
```

Common Pitfalls when Working with COM Ports

A COM port cannot be opened more than once. Therefore it is very important to close the port before the script is unloaded.

This does have several implications:

- If the COM port is not available when the script is being loaded (e.g. when recalling a setting), then the script will fail opening the COM port and the DUT will not be accessible.

- When working with additional tools to communicate with the DUT, e.g. a debug-tool to configure the device, only one process can have control over the COM port at a time. If the M8070A software has opened the COM port in the DUT Control Interface script, then the external tool cannot control the device and vice versa.
- Exceptions in the program flow may not prevent releasing the COM port.

Best Scripting Practices

- Always implement OnUninstall and close the COM port properly.
- Create and open the COM port as late as possible in the script.
  - Import all data types
  - Implement all driver functionality to communicate with the device.
  - Implement the required hook functions and call the device specific functions.
  - Configure and open the COM port.

    The preferred place to do this is in the DUT_connect and DUT_disconnect hooks.
  - Initialize the DUT using the successfully opened COM port.

    If there can be errors during the initialization phase, ensure that the variable that provides access to the COM port is only getting invalid after the COM port has been closed.

Communication with the M8070A Instrument Layer

Overview

The DUT Control Interface provides a functional API to allow interaction with the instrument layer of the M8070A software.

The functional API is provided via the global symbol M8000 within the DUT Control Interface script.

This is similar to the M8000.Scpi API in the Script Editor, but does not allow sending remote commands to the M8070A software. Instead it provides direct access to measurement relevant status and control of the sequencing directly.

This can be used to implement measurement hooks that take the status information of the instrument into consideration and initiate sequence branching in dependency of the reported status.

The API to control the instrument may not be used from within the DUT specific hooks. Doing so will cause error messages when executing the respective hooks at runtime.

If an API call is resulting in an error (e.g. when reading status bits that don't exist for a DUT) then an exception is thrown, which finally results in an error message on the level of the DUT Control Interface script.

Examples

An example of how this can be used to re-train a DUT back into loopback during a Jitter Tolerance measurement can be found in the Tutorials folder of the Factory provided DUT Control Interface scripts.

The script is named Retrain_DUT_in_JTOL.py and demonstrates how the Instrument API can be used to control the sequence in dependency of the reported status information and bit counter values.

API Summary

The following is giving a quick overview over Instrument control API.

Sequencing Control

M8000.Instrument.Sequence.Restart(groupOrLocationName)

Restart the sequence on the given group or location.

M8000.Instrument.Sequence.BreakAll()

Execute a sequence break, which will influence all sequencers in the system that are capable of being controlled by the break command.

M8000.Instrument.Sequence.GetBlockNumber(groupOrLocationName)

Reads the current sequence block number from the addressed locations.

Analyzer Control

M8000.Instrument.Analyzer.AutoAlign(groupOrLocationName)

Execute an auto-alignment on the addressed locations.

M8000.Instrument.Analyzer.DataCenter(groupOrLocationName)

Executes a data-center alignment on the addressed locations.

M8000.Instrument.Analyzer.ThresholdCenter(groupOrLocationName)

Executes a threshold-center alignment on the addressed locations.

M8000.Instrument.Analyzer.SyncNow(groupOrLocationName)

Executes a pattern synchronization on the addressed locations.

M8000.Instrument.Analyzer.GetBitCounters(groupOrLocationName, combineIntoOneResult)

Read the bit counters from the addressed locations and return either the sum of all addressed counters, or each addressed counter individually.

M8000.Instrument.Analyzer.GetDataLoss(groupOrLocationName, combineIntoOneResult)

Read the data loss status bit from the addressed locations and return either the logical OR of all addressed locations, or each location's status individually.

M8000.Instrument.Analyzer.GetSyncLoss(groupOrLocationName,combineIntoOneResult)

Read the sync loss status bit from the addressed locations and return either the logical OR of all addressed locations, or each location's status individually.

M8000.Instrument.Analyzer.GetSymbolLoss(groupOrLocationName,combineIntoOneResult)

Read the symbol loss status bit from the addressed locations and return either the logical OR of all addressed locations, or each location's status individually.

M8000.Instrument.Analyzer.GetCdrUnlock(groupOrLocationName, combineIntoOneResult)

Read the CDR unlock status bit from the addressed locations and return either the logical OR of all addressed locations, or each location's status individually.

API Details

Sequencing Control

M8000.Instrument.Sequence.Restart(groupOrLocationName)

Functionality	Restart the sequence on the given group or location.
Parameters	
groupOrLocationName	Addresses the locations that shall be restarted. The use of .Net regular expression patterns is allowed.
Returns	Nothing
Notes	▪ Initiates the restart, but does not wait until all sequencers have actually executed the request. ▪ Not allowed for "DUT_" hooks. ▪ Not supported for locations defined by the DUT Control Interface script.

M8000.Instrument.Sequence.BreakAll()

Functionality	Execute a sequence break, which will influence all sequencers in the system that are capable of being controlled by the break command.
Parameters	None
Returns	Nothing
Notes	▪ Initiates the break, but does not wait until the sequencers have executed the break. ▪ Not allowed for "DUT_" hooks.

M8000.Instrument.Sequence.GetBlockNumber(groupOrLocationName)

Functionality	Reads the current sequence block number from the addressed locations. The use of .Net regular expression patterns is allowed.
Parameters	
groupOrLocationName	Addresses the locations that shall be read. The use of .Net regular expression patterns is allowed.
Returns	IReadOnlyDictionary<string, int> The dictionary contains one entry for each addressed location.
Notes	▪ Not allowed for "DUT_" hooks. ▪ Not supported for locations defined by the DUT Control Interface script.

Analyzer Control

M8000.Instrument.Analyzer.AutoAlign(groupOrLocationName)

Functionality	Execute an auto-alignment on the addressed locations.
Parameters	
groupOrLocationName	Addresses the locations that shall be aligned. The use of .Net regular expression patterns is allowed.
Returns	Nothing
Notes	▪ Is blocking until the alignment has finished. ▪ Not allowed for "DUT_" hooks. ▪ Not supported for locations defined by the DUT Control Interface script.

M8000.Instrument.Analyzer.DataCenter(groupOrLocationName)

Functionality	Executes a data-center alignment on the addressed locations.
Parameters	
groupOrLocationName	Addresses the locations that shall be aligned. The use of .Net regular expression patterns is allowed.
Returns	Nothing
Notes	▪ Is blocking until the alignment has finished. ▪ Not allowed for "DUT_" hooks. ▪ Not supported for locations defined by the DUT Control Interface script.

M8000.Instrument.Analyzer.ThresholdCenter(groupOrLocationName)

Functionality	Executes a threshold-center alignment on the addressed locations.
Parameters	
groupOrLocationName	Addresses the locations that shall be aligned. The use of .Net regular expression patterns is allowed.
Returns	Nothing
Notes	▪ Is blocking until the alignment has finished. ▪ Not allowed for "DUT_" hooks. ▪ Not supported for locations defined by the DUT Control Interface script.

M8000.Instrument.Analyzer.SyncNow(groupOrLocationName)

Functionality	Executes a pattern synchronization on the addressed locations.
Parameters	
groupOrLocationName	Addresses the locations that shall be aligned. The use of .Net regular expression patterns is allowed.
Returns	Nothing
Notes	▪ Is blocking until the synchronization has finished. ▪ Not allowed for "DUT_" hooks. ▪ Supported for locations defined by the DUT Control Interface script. But requires implementation of DUT_syncPattern.

M8000.Instrument.Analyzer.GetBitCounters(groupOrLocationName, combineIntoOneResult)

Functionality	Read the bit counters from the addressed locations and return either the sum of all addressed counters, or each addressed counter individually.
Parameters	
groupOrLocationName	Addresses the locations that shall be read. The use of .Net regular expression patterns is allowed.
combineIntoOneResult	▪ True Return the sum of all addressed bit counters as a single instance of type BitErrorCounter. ▪ False Return a dictionary of type IReadOnlyDictionary<string, BitErrorCounter> that holds an entry for each addressed location.
Returns	BitErrorCounter or IReadOnlyDictionary<string, BitErrorCounter>
Notes	▪ Not allowed for "DUT_" hooks.

M8000.Instrument.Analyzer.GetDataLoss(groupOrLocationName, combineIntoOneResult)

Functionality	Read the data loss status bit from the addressed locations and return either the logical OR of all addressed locations, or each location's status individually.
Parameters	
groupOrLocationName	Addresses the locations that shall be read. The use of .Net regular expression patterns is allowed.
combineIntoOneResult	▪ True Return the logical OR of all addressed status bits. ▪ False Return a dictionary of type IReadOnlyDictionary<string, bool> that holds an entry for each addressed location.
Returns	bool or IReadOnlyDictionary<string, bool>
Notes	▪ Not allowed for "DUT_" hooks. ▪ Not supported for locations defined by the DUT Control Interface script.

M8000.Instrument.Analyzer.GetSyncLoss(groupOrLocationName,combineIntoOneResult)

Functionality	Read the sync loss status bit from the addressed locations and return either the logical OR of all addressed locations, or each location's status individually.
Parameters	
groupOrLocationName	Addresses the locations that shall be read. The use of .Net regular expression patterns is allowed.

combineIntoOneResult	▪ True Return the logical OR of all addressed status bits. ▪ False Return a dictionary of type IReadOnlyDictionary<string, bool> that holds an entry for each addressed location.
Returns	bool or IReadOnlyDictionary<string, bool>
Notes	▪ Not allowed for "DUT_" hooks. ▪ Not supported for locations defined by the DUT Control Interface script.

M8000.Instrument.Analyzer.GetSymbolLoss(groupOrLocationName,combineIntoOneResult)

Functionality	Read the symbol loss status bit from the addressed locations and return either the logical OR of all addressed locations, or each location's status individually.
Parameters	
groupOrLocationName	Addresses the locations that shall be read. The use of .Net regular expression patterns is allowed.
combineIntoOneResult	▪ True Return the logical OR of all addressed status bits. ▪ False Return a dictionary of type IReadOnlyDictionary<string, bool> that holds an entry for each addressed location.
Returns	bool or IReadOnlyDictionary<string, bool>
Notes	▪ Not allowed for "DUT_" hooks. ▪ Not supported for locations defined by the DUT Control Interface script.

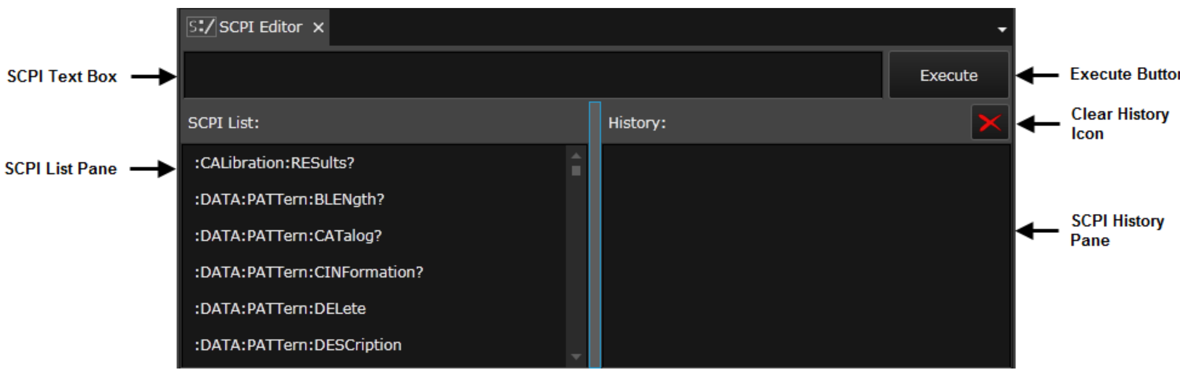
M8000.Instrument.Analyzer.GetCdrUnlock(groupOrLocationName, combineIntoOneResult)

Functionality	Read the CDR unlock status bit from the addressed locations and return either the logical OR of all addressed locations, or each location's status individually.
Parameters	
groupOrLocationName	Addresses the locations that shall be read. The use of .Net regular expression patterns is allowed.
combineIntoOneResult	▪ True Return the logical OR of all addressed status bits. ▪ False Return a dictionary of type IReadOnlyDictionary<string, bool> that holds an entry for each addressed location.
Returns	bool or IReadOnlyDictionary<string, bool>
Notes	▪ Not allowed for "DUT_" hooks. ▪ Not supported for locations defined by the DUT Control Interface script.

SCPI Editor

The **SCPI Editor** lists all SCPI that can be used to program M8020A/M8030A and also provides a platform to execute them.

The following figure depicts the elements of the **SCPI Editor**:



The listed elements are described below:

- **SCPI Text Box** - Allows you to type the SCPI.
- **SCPI List Pane** - Lists all SCPI related to M8020A/M8030A.
- **Execute Button** - Click this button to run the SCPI.
- **SCPI History Pane** - Maintains the history of the executed SCPI commands and also displays the invalid command errors.
- **Clear History Icon** - Clears the contents of the SCPI History Pane.

SCPI Basics

There are a number of key areas to consider when using SCPI for the first time. These are as follows:

- Instrument Model
- Command Syntax
- Optional Command Keywords
- Query Responses
- Command Separators
- SCPI Command Structure
- Invalid Commands

Instrument Model

SCPI guidelines require that the M8020A/M8030A is compatible with an instrument model. This ensures that when using SCPI, functional compatibility is achieved between instruments that perform the same tasks. For example, if two different instruments have a programmable clock frequency setting, then both instruments would use the same SCPI commands to set their frequency. The instrument model is made up of a number of subsystems.

The sub-system defines a group of functions within a module and has a unique identifier under SCPI, which is called the **Root Keyword**.

Command Syntax

Commands may be up to twelve characters long. A short-form version is also available which has a preferred length of four characters or less. In this document the long-form and short-form versions are shown as a single word with the short-form being shown in upper-case letters.

For example, the long-form node command SOURce has the short-form SOUR. Using the short form saves time when entering a program; however, using the long form makes a program more descriptive and easier to understand.

SCPI commands may be commands only, commands and queries, or queries only. A question mark at the end of a command indicates that it is a query. If the question mark appears in brackets ([?]), the command has a command and query form.

Optional Command Keywords

Some layers in the SCPI command structure are optional. These optional keywords are indicated by square brackets ([]). A typical use for these types of keywords is with a command that is unique to one module. In this case, the top layer (Root Keyword) of the command structure may be omitted.

For example, the following command code segments are functionally identical:

```
[:SOURce]:JITTer[:GLOBal][:STATe] <ON|OFF|1|0>
:JITTer <ON|OFF|1|0>
:JITT <ON|OFF|1|0>
:jitt <ON|OFF|1|0>
```

Note that it is not necessary to include the syntax inside the square brackets ([]).

Query Responses

It is possible to interrogate the individual settings and status of a device using query commands. Retrieving data is a two-stage operation.

The query command is sent from the controller using the OUTPUT statement and the data is read from the device using the ENTER statement. A typical example is the SCPI IEEE 488.2 Common Command *IDN? which queries the identity of a device.

NOTE

When sending strings to the instrument, either the double quote (") or the single quote may be used ('), the latter being more suited to PASCAL programs, which make use of a single quote; the former being more suited to use in BASIC programs, which use a double quote as a delimiter.

Command Separators

The SCPI command structure is hierarchical and is governed by commas, semicolons and colons:

- Commas are used to separate parameters in one command.
- Colons are used to separate levels.
- Semicolons are used to send more than one command to the instrument at a time.

It is possible to send several commands in one pass, as long as the commands all belong to the same node in the SCPI tree. The commands have to be separated by semicolons.

The following SCPI commands provide examples of this.

```
SOURce:VOLTage:OFFSet ‘M2.DataOut2’,-0.99
SOURce:VOLTage:AMPLitude ‘M2.DataOut2’,1.11
```

These commands can also be sent as follows:

```
VOLT:OFFS ‘M2.DataOut2’,-0.99; ‘M2.DataOut2’,AMPL
1.11
```

SCPI Command Structure

The SCPI command structure can be best examined by means of an example. For example, the command to set the generator's output amplitude is:

```
[:SOURce]:VOLTage[:AMPLitude] ‘M1.DataOut1’,1.11
```

The structure of this command can be illustrated as follows:

- [:SOURce] - This is the top layer of the command structure and identifies the source subsystem.
- :VOLTage - This is the next layer and defines the subnode for setting a voltage level.
- [:AMPLitude] - This is the command itself for setting the output amplitude level.
- 'M1.DataOut1', 1.11 - This specifies module 1, DataOut port of channel 1, and specifies an amplitude of 1.11.

NOTE

Any optional commands are enclosed in square brackets [] and any optional characters are shown in lower case.

A colon indicates a change of level in the command hierarchy. Commands at the same level in the hierarchy may be included in the same command line, if separated by a semi-colon.

The bar symbol (|) indicates mutually exclusive commands.

To translate this syntax into a command line, follow the convention described above. Remember, however, that the command line can be created in several different ways. It can be created with or without optional keywords, and in a long or short form. The following example gives three possible forms of the command line; all are acceptable.

In long form:

```
:SOURce:VOLTage:AMPLitude 'M1.DataOut1',1.11
```

In short form:

```
:SOUR:VOLT:AMPL 'M1.DataOut1',1.11
```

With the optional commands removed:

```
:VOLT 'M1.DataOut1',1.11
```

The long form is the most descriptive form of programming commands in SCPI.

Invalid Commands

A command is invalid and will be rejected if:

- It contains a syntax error.
- It cannot be identified.
- It has too few or too many parameters.

- A parameter is out of range.
- It is out of context.

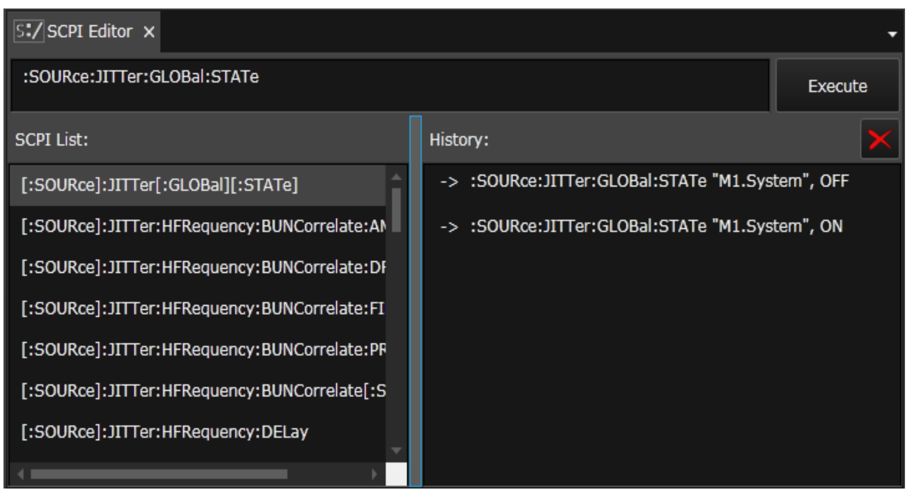
Executing SCPI Commands

To execute a SCPI command, follow the given steps:

1. Select the SCPI from the given list. You can also type the SCPI in the provided text box to expedite the command search.
2. Use the proper SCPI command syntax along with the command separators. For complete details, refer to SCPI Basics on page 411.

   The following example shows a SCPI command to enable Global Jitter State:

   `:SOURce:JITTer:GLOBal:STATe 'M1.SYSTEM', ON`
3. Click **Execute**. The output of the SCPI command will be displayed in the **History** pane as shown in the following figure:



4. Click the  **Clear History** icon to clear the contents of **History** pane.

For complete details on M8020A/M8030A SCPI commands, refer to the *M8070A Programming Guide*.

Self Test Utility

The **Self Test** utility checks the specific system information of the hardware components for basic functionality. On execution, the following results are displayed:

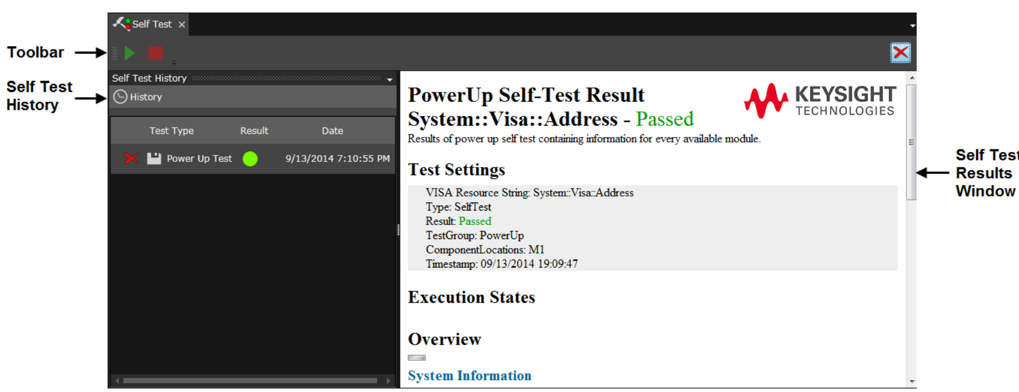
- System related information such as connected modules, serial no. and hardware revision.
- Module related information such as calibration, power supplies and memory controller.

Launching the Self Test Utility

To launch the **Self Test**:

- Go to the **Menu Bar** > **Utilities** and then select **Self Test**.

The **Self Test** utility will appear as shown in the following figure:



The **Self Test** utility consists of the following GUI elements:

- Toolbar
- Self Test History
- Self Test Results Window

Toolbar

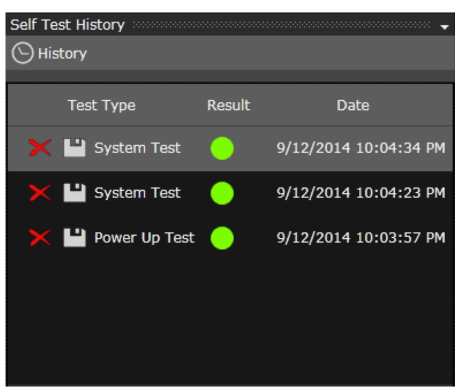
The toolbar provides the following convenient self test functions:

Table 73

Icon	Name	Description
	Execute Self Test	Click this icon to run the self test.
	Abort Self Test	Click this icon abort a self test.
	Save Self Test	Click this icon to save the selected self test from the history. For details, refer to Saving Self Test Results on page 419.
	Delete Self Test	Click this icon to delete the selected self test from the history.
	Close Report	Click this icon to close the self test result.

Self Test History

The **Self Test History** maintains the history of self tests (Passed or Failed) executed by the user. The passed self test results are indicated by green LED while the failed ones are indicated by red LED.



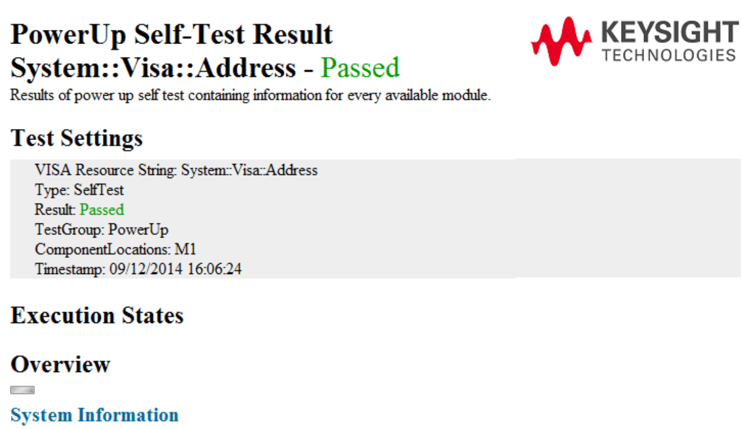
At any point of time, you can double-click on the shown self test entries to view their respective results. Once viewed, you can click on **Close Report** icon to close the self test results.

The **Self Test History** also allows you to save and delete the self test reports. For details, refer to Saving Self Test Results on page 419 and Deleting Self Test Results on page 420.

Self Test Results Window

The **Self Test Results** window displays the results of the executed self test. It display the following results:

- System related information such as connected modules, serial no. and hardware revision.
- Module related information such as calibration, power supplies and memory controller.



Executing Self Test

To execute a self test:

- Terminate each **Data Out** port with 50 Ω.
- Remove any devices from the tested M8000 system as any connected device under test might be damaged.
- Click the **Execute Self Test** icon. The following message will appear:

- Click on **Perform Test** button present on the message box.
- The self test will start and the status will be indicated by the status indicator that appears on the right side of the toolbar. Once the self test is completed, the results will be shown on the **Self Test Results** window.

CAUTION

Before performing a system self-test, terminate each **Data Out** port with 50 Ω. Take care to remove any devices from the tested M8000 system as any connected device under test might be damaged.

Aborting Self Test

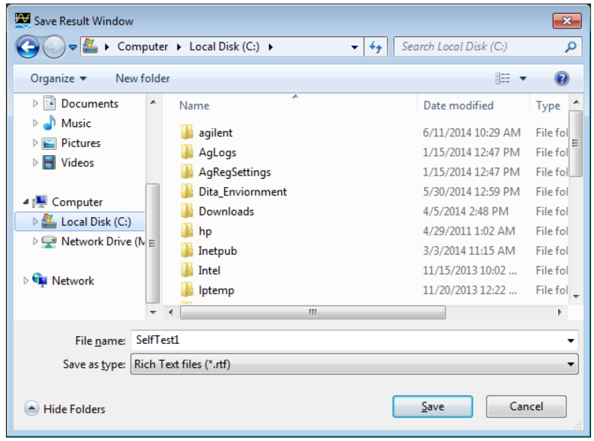
To abort a self test:

- Click **Abort Self Test** icon to abort the self test. The self test will stop and the self test failure entry will be displayed in the **Self Test History**.

Saving Self Test Results

To save the self test results from the **Self Test History**:

- Click the **Save Report** icon. A standard **Save Result** dialog will appear.



- Use the navigation pane to provide a location to save the file.
- Provide a file name and specify the file format. You can save the file in RTF, HTML, XML and TXT formats.
- Click **Save**.

Deleting Self Test Results

To delete a self test entry from the **Self Test History**:

- Click  **Delete Report** icon. The entry will be removed from the **Self Test History**.

Licenses Window

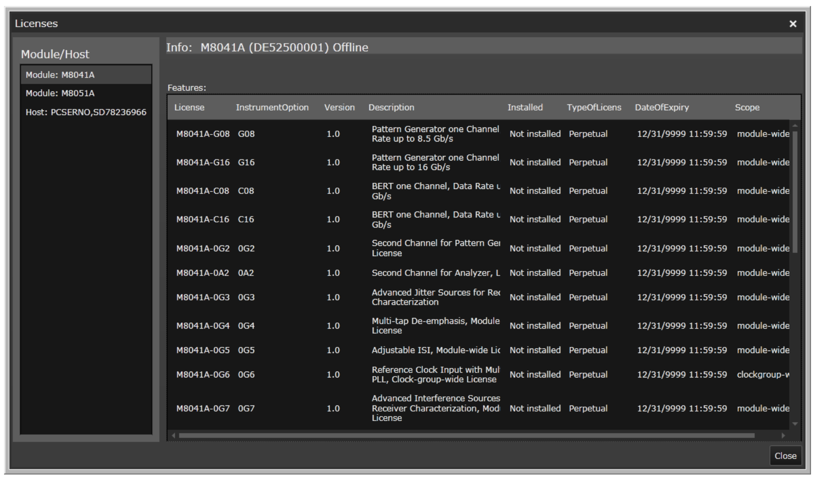
The **Licenses** window displays the license information currently installed in the modules or host.

Launching the Licenses Window

To launch the **Licenses** window:

- Go to the **Menu Bar** > **Utilities** and then select **Licenses...**.

The **Licenses** window is shown in the following figure:



The **Licenses** window is divided into two panels. The left panel shows the modules/host information. Once you select modules/host, the corresponding license information is shown in the right panel.

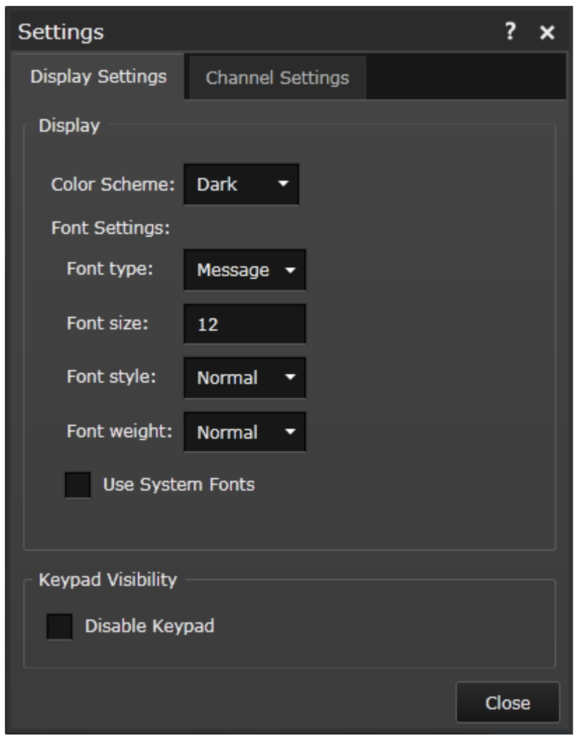
To exit the **Licenses** window, click the **Close** button.

For detailed information on the M8020A/M8030A licenses, refer to Licenses on page 433.

Settings Window

The **Settings** window allows you to set the display and channel settings in the GUI.

To open the **Settings** window, go to **Menu Bar** > **Utilities** and then select **Settings...**. The **Setting** window will appear as shown in the following figure:



The **Settings** window has the following tabs:

Display Settings

The **Display Settings** tab provides the following options:

- **Display** - The **Display** option provides the following choices:

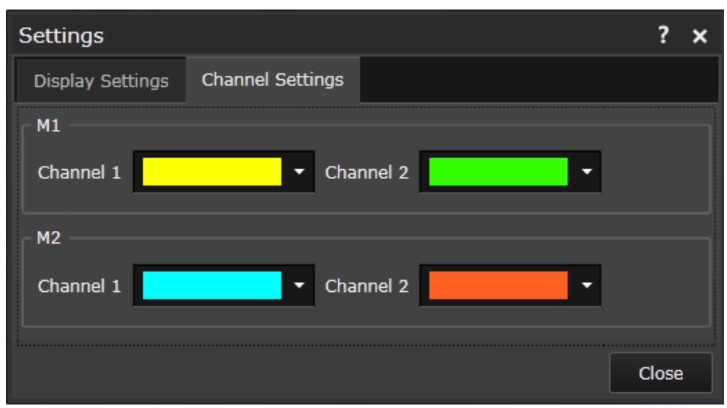
- **Color Scheme** - Use this option to change the color scheme of the GUI. You can choose between the dark or light scheme. The dark scheme is selected by default.
- **Font Setting** - Use this option to change the font settings in the GUI.
- **Use System Fonts** - Select this option to use the system fonts in the GUI.

Keypad Visibility

- **Disable Keypad** - Use this option to disable the on-screen numeric keypad in the GUI. The on-screen numeric keypad is enabled by default.

Channel Settings

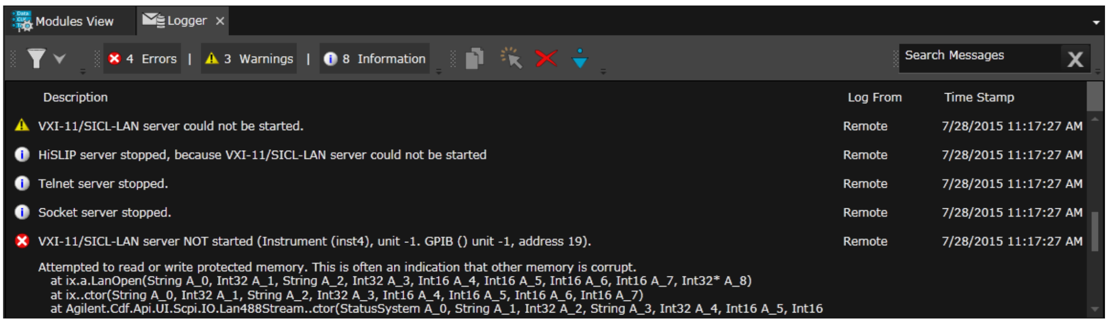
- The **Channel** Settings tab displays the color schemes applied to the various channels of the connected modules. In addition, this tab also allows you to assign your own color schemes to each individual channel of the connected modules. The following figure depicts how the different color schemes are applied to the various channels of the connected modules.



NOTE The color schemes assigned to the channels will change if you switch to light color scheme from the **Display Setting** tab.

Logger Window

The **Logger** window displays errors, warnings and information messages along with their respective descriptions, applications from where they are generated and their time stamps.

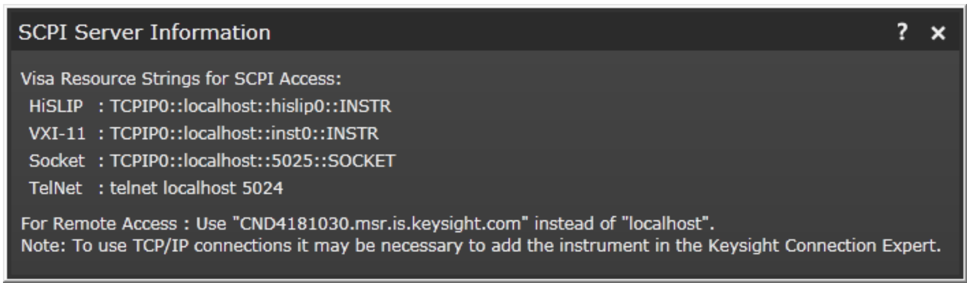


The **Logger** window allows you to:

- **Filter** - Use the filter option to select the loggers from where the generated messages will be displayed.
- **Message Selection** - Use this option to choose whether you want to view errors, warnings or information message.
- **Copy** - Use this option to copy a message. You need to select a message in order to enable copy feature.
- **Select All** - Use this option to select all messages. It also enables copying all messages.
- **Clear Messages** - Use this option to delete all messages.
- **Auto Scroll** - Use this option to enable/disable auto scroll option. When the **Auto Scroll** option is enabled, it will automatically scroll you to the new message without using the scroll bar
- **Search Messages** - Use this option to search messages by providing an input in the **Search Messages** search box.

SCPI Server Information

This dialog lists the VISA resource and remote access strings to connect instruments.



Plug-in Manager

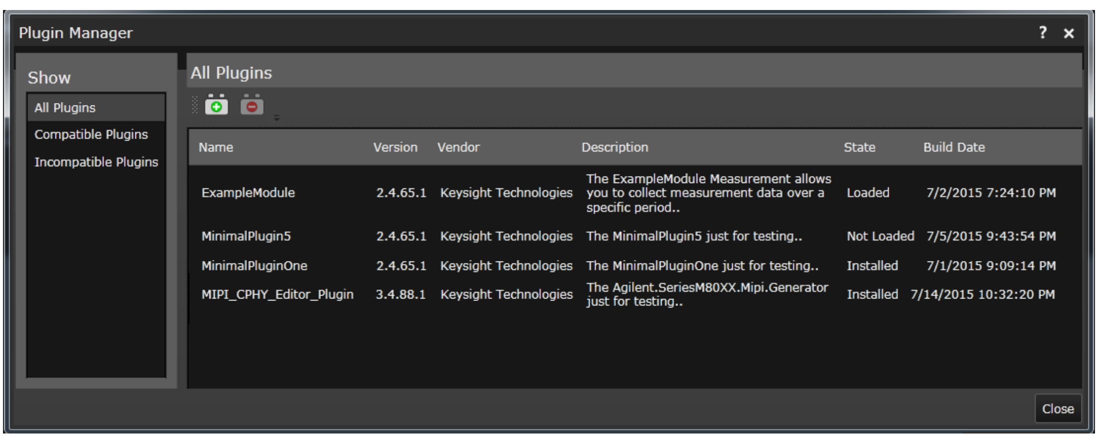
A plug-in is a piece of software application that acts as an add-on to the main software and enhances its capability. Plug-in allows the software to host additional functionality without undergoing a major modification or enhancement. A plug-in is not a permanent part of the software, hence can be installed, uninstalled and updated as and when required.

The M8070A system software for the M8000 Series of BER Test Solutions supports plug-ins. Thus, the present capabilities of M8070A can be further enhanced by simply adding the required plug-ins.

The **Plug-in Manager** simplifies all the tasks related to plug-in management. It displays list of plug-ins that are installed in the software. For each plug-in, it displays the information such as Name, Version, Vendor, Description, State and Build Date. In addition, the **Plug-in Manager** also allows you to install, uninstall and update the plug-ins.

How to Launch Plug-in Manager

To launch **Plug-in Manager**, open the M8070A user interface and then go to **Menu** > **Utilities** and then **Plug-in Manager**. The **Plug-in Manager** window will appear as shown in the following figure:



The **Plug-in Manager** window displays two types of plug-ins:

- **Compatible Plug-ins** – plug-ins which are compatible with the current version of software.
- **In-Compatible Plug-ins** – plug-ins which are not compatible with the current version of software.

The **Plug-in Manager** window displays the following information of each plug-in:

- **Name** - Name of the plug-in
- **Verison** - Version no. of the plug-in
- **Vendor** - Vendor/publisher of the plug-in
- **Description** - Brief description of the plug-in
- **State** - State of the plug-in. For details on different plug-in states, see Plug-in States on page 427.
- **Build Date** - Build date of the plug-in

Plug-in States

There can be the following plug-in states:

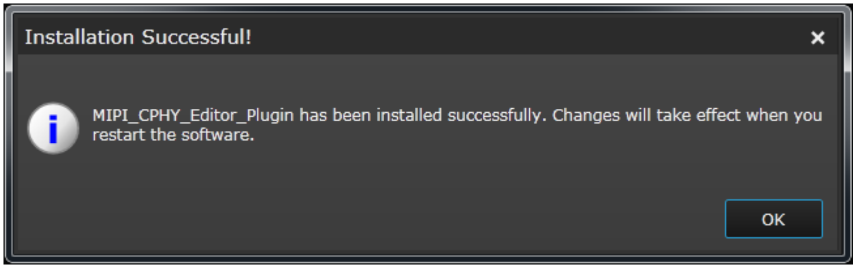
- **Installed** - When the plug-in is installed but the M8070A software is not re-started. In this case, the installed plug-in is not loaded or it cannot be used. You have re-start the M8070A software in order to load the plug-in.
- **Loaded** - When the plug-in is installed and the M8070A software has been re-started. In this case, the installed plug-in is ready to be used. Remember, you have to restart the M8070A software in order to load the plug-in.
- **Not Loaded** - When the plug-in is installed and but failed to load. In this case, the installed plug-in cannot be used.
- **Version In-Compatible** - When the version of the installed plug-in is not compatible with the current version of M8070A software.

How to Install a Plug-in

The **Plug-in Manager** window allows you to install a plug-in.

To do so;

1. Download plug-in file from Keysight webpage: www.keysight.com/find/m8000
2. Click on  **Install Plug-in from File** button. A Window's standard **Open** dialog will appear.
3. Locate the plug-in file (*.M8KP) you want to install and click **OK**.
4. On the successful installation of plug-in, the following message will appear:



5 Restart the software. Once you restart the software, the plug-in state will change to **Loaded**. See Plug-in States on page 427.

NOTE **Ensure to restart the M8070A software for the changes to take effect.**

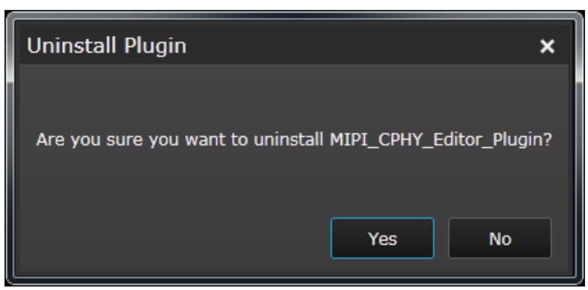
How to Uninstall a Plug-in

The **Plug-in Manager** window allows you to uninstall a plug-in.

To do so;

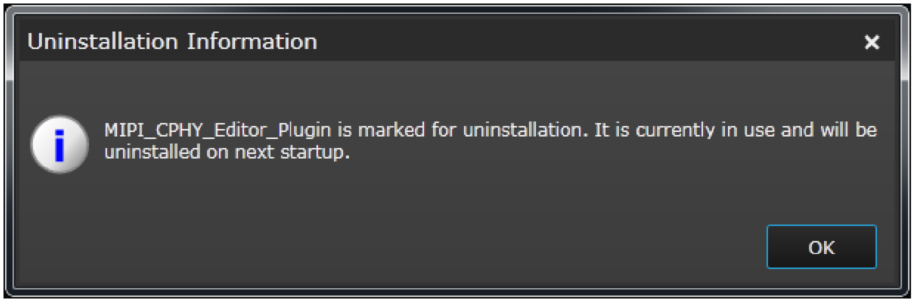
1 Select the plug-in from the list.

2 Click on **Uninstall Selected Plug-in** button or right-click on the selected plug-in. The **Uninstall Plug-in** dialog will open.



3 Click **Yes**. If the state of plug-in is **Installed**, then it will be immediately uninstalled from the software.

4 However, if the plug-in is currently in use (**Loaded or Not Loaded**), then you will receive the following message.



5 Restart the software. The plug-in will be uninstalled on software startup.

How to Update a Plug-in

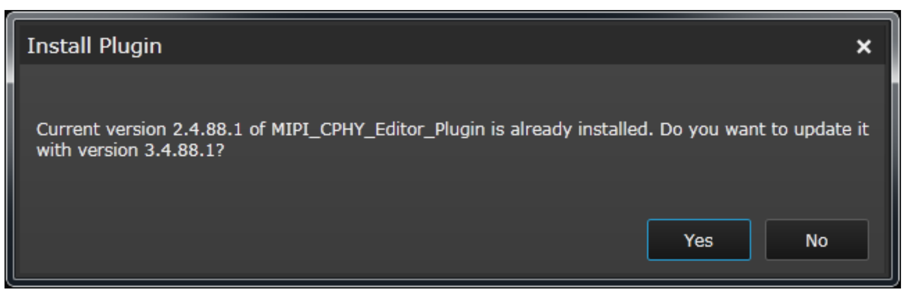
The **Plug-in Manager** window also allows you to update an already installed plug-in.

Following the given steps to update the plug-in with its higher version:

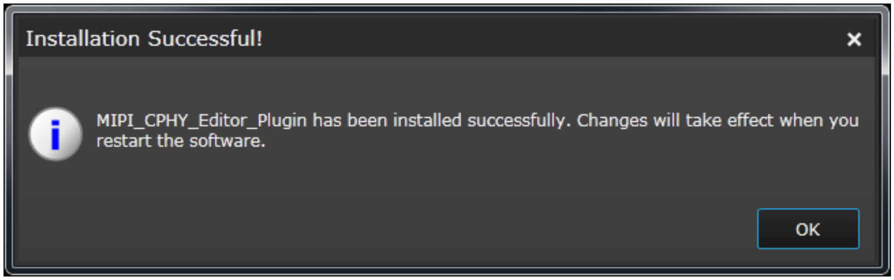
1 Download plug-in file from Keysight webpage: www.keysight.com/find/m8000

2 Click on **Install Plug-in from File** button. A Window's standard **Open** dialog will appear.

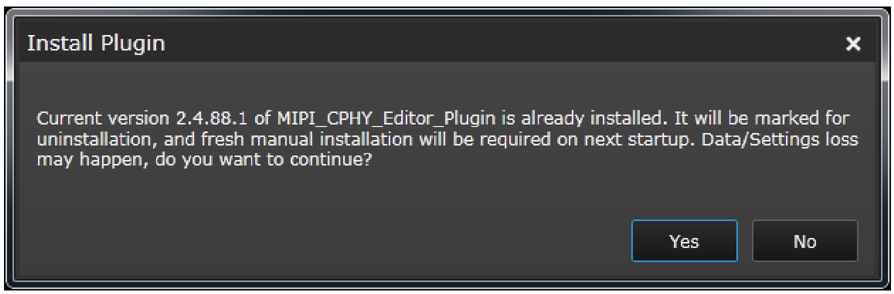
3 Locate the plug-in file (*.M8KP) you want to update and click **OK**. You will see the following message:



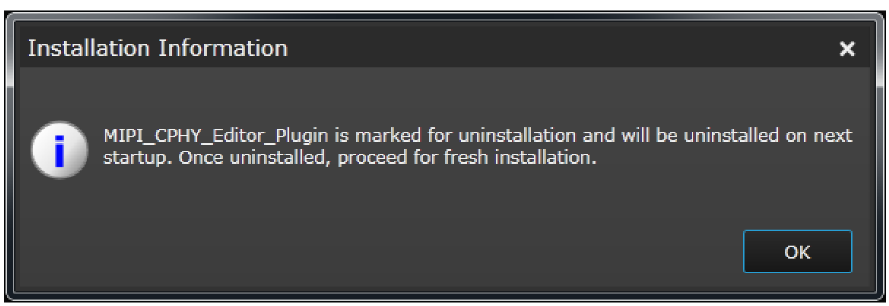
4 Click **Yes**. If the state of plug-in is **Installed**, then it will be immediately updated and the following message will appear:



5 However, if the plug-in is currently in use (**Loaded** or **Not Loaded**), then you will receive the following message.



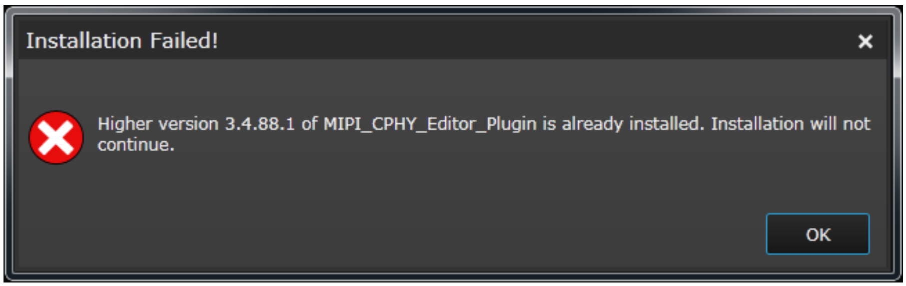
6 Click **Yes**. The following message will appear:



7 Click **OK**. The selected plug-in will be uninstalled from the software. You can now install the new version of that plug-in. For installation procedure, see How to Install a Plug-in on page 427.

NOTE Ensure to restart the M8070A software for the changes to take effect.

The **Plug-in Manager** window does not directly allow you to update an installed plug-in with previous (lower) version. If you try to do so it will give the following error message:



In this case, you have to uninstalled the plug-in and then install the previous (lower) version of that plug-in again.

How to Access an Installed Plug-in Through M8070A User Interface

Follow the steps to access an installed plug-in through M8070A user interface:

1 Launch M8070A software user interface.
2 In the M8070A user interface, go to **Menu Bar** and then click **Application** menu. It will list all installed plug-ins.
3 Select the plug-in.
4 The plug-in user interface will appear in the M8070A software.

For complete details on how to operate plug-in user interface, refer to the respective plug-in's *User Guide*

10 Licenses

KEYSIGHT TECHNOLOGIES

Overview

The M8070A software requires a license to communicate with the M8020A/M8030A hardware. You either purchased an M8070A license to install on a dedicated host computer (M8070A-OTP) or one that can be installed on a network server that will be used as a license server for operation over a company network (M8070A-ONP, floating/networked).

In addition to the M8070A license, the M8020A and M8030A, being a modular product, includes different sets of modules hosted in an M9505A and M9514A AXI chassis, respectively. Each module has its own licenses corresponding to specific features. Therefore, you need to install these licenses in your instrument in order to use the modules/features.

Supported Licenses

The M8020A/M8030A supports the following types of licenses:

- Node-Locked (Uncounted) Licenses
- Floating Licenses
- Module-Specific Licenses

In addition to the binding characteristic (node-locked, floating, or module-specific), licenses have the following characteristics:

- Duration (permanent or time-perishable)
- Version (conventional or date-based)
- Features (an instrument may have one or more features licensed)
- Transportability (the ability to be moved from one instrument to another)

Finally, the following license types are available for legacy systems. Please contact Keysight Technologies support first (http://www.keysight.com) if you are considering using these in new designs:

- Counted Node-Locked License
- Upgrade License

Node-Locked (Uncounted) Licenses

Node-locked licenses are tied to a particular instrument or PC **Host ID**. The **Host ID** consists of the model number and serial number, and is stored in SecureStorage on the instrument or PC.

The node-locked licenses can be of the following type:

- Permanent licenses that do not have expiration date.
- Time-perishable licenses that have expiration date.
- Transportable licenses that can be moved from one instrument to another, and may be either time-perishable or permanent.
- Non-removable licenses that typically provide right-to-use for a hardware component.

Permanent License

A permanent license is the most common type of license. It does not have an expiration date.

Time-Perishable License

A time-perishable license differs from the permanent license in a way that it has an expiration date. The date is in the *dd-mm-yyyy* format. The time-perishable licenses are applied to instruments only and not applied to PC software applications. They are typically used for trial or demo purposes.

Transportable Node-Locked License

A transportable license can be moved (transported) from one instrument to another. It can be either permanent or time-perishable, but it must be node-locked. Once a license has been transported off of a system, that feature is no longer valid on that system. If a user wishes to transport the license back to the original device, he or she is free to do so.

When removed from the device, this license is marked as not reinstallable on the system. The user is notified and a transport confirmation key will be generated so that a new license can be redeemed with a new Host ID.

If the license file is transported back to the original system, a unique serial number ensures that the new license has a different signature from any previously installed and transported licenses (which cannot be reinstalled).

You can acquire a transportable license by moving the license from a system where it was previously installed, or by acquiring the license from the **Keysight Software Manager** web site. You cannot manually create a transportable license. Even if you know the correct syntax, construct the license correctly, and sign it with a license generator, you will not be able to transport the license. The **Keysight Software Manager** which provides the real-time re-hosting does not recognize a transportable license that it did not originate.

Floating Licenses

A floating license entitles anyone on the same network to use the licensed software, up to the numeric limit specified in the license file. You must configure a license server in order to use floating licenses. These licenses require the license server manager daemon (FLEXnet Publisher lmadmin) and the Keysight daemon (*agilent.exe*) to be running in order to count the concurrent usage of the licenses.

Floating licenses are further subdivided into local and remote licenses. Local licenses are stored in the licensing folder and in secure storage; remote licenses are kept in a “Remote Licenses” sub folder, and are not stored in SecureStorage. Floating licenses are installed on two different host machines, the application host and the server host. A license is considered to be local if there is a SERVER line in the license with a host name that refers to the host where the license is installed. Conversely, a license is considered to be remote if there is a SERVER line in the license that has a host name that is different from the host where the license is installed.

Module-Specific Licenses

Module-specific licenses are a specialized type of license that enables a specific module of a modular instrument (such as a PXI or AXIe module). The module-specific license resides on the controller and is bound to both the controller (typically a PC) and the module; the controller by means of the HostID, and the module itself by means of the module serial number which is embedded in the feature name for the license.

Module-specific licenses may be time-perishable.

M8020A/M8030A Licenses

The list of licenses used by the various modules of M8020A/M8030A is listed in the following tables:

M8070A System Software for M8000 Series of BER Test Solutions

M8070 - M8000 Series BER Test Solution License

Table 74 M8070 - M8000 Series BER Test Solution License

Product	Option	Description
M8070A-0TP	0TP	System Software for M8000 Series of BER Test Solutions, Transportable, Perpetual License
M8070A-0NP	0NP	System Software for M8000 Series of BER Test Solutions, Network/Floating, Perpetual License

M8041A - High-Performance BERT Module

M8041A - Basic Selection

Table 75 M8041A - Basic Selection

Product	Option	Description
M8041A-G08	G08	Generator one Channel, Data Rate up to 8.5 Gb/s
M8041A-G16	G16	Generator one Channel, Data Rate up to 16 Gb/s (Upgrade: U16)
M8041A-C08	C08	BERT one Channel, Data Rate up to 8.5 Gb/s
M8041A-C16	C16	BERT one Channel, Data Rate up to 16 Gb/s (Upgrade: UED)

M8041A- Module Functionality

Table 76 M8041A- Module Functionality

Product	Option	Description
M8041A-0G2	0G2	Second Channel for Generator, License
M8041A-0A2	0A2	Second Channel for Analyzer, License
M8041A-0G3	0G3	Advanced Jitter Sources for Receiver Characterization, Module-wide License
M8041A-0G4	0G4	Multi-tap De-emphasis, Module-wide License
M8041A-0G5	0G5	Adjustable ISI, Module-wide License
M8041A-0G6	0G5	Reference Clock Input with Multiplying PLL, Clockgroup-wide License
M8041A-0G7	0G7	Advanced Interference Sources for Receiver Characterization, Module-wide License
M8041A-0S1	0S1	Interactive Link Training for PCI Express, Clockgroup-wide License
M8041A-0S2	0S2	SER/FER Analysis for Coded and Retimed Loopback, Clockgroup-wide License
M8041A-0A3	0A3	Analyzer Equalization, Module-wide License

M8041A - License Upgrades for M8041A High-Performance BERT Module

Table 77 M8041A - License Upgrades for M8041A High-Performance BERT Module

Product	Option	Description
M8041A-U16	U16	Upgrade to 16 Gb/s data rate from M8041A-G08 and M8041A-C08, Module-wide License
M8041A-UED	UED	Upgrade to BERT from M8041A-G08 and M8041A-G16, Module-wide License
M8041A-UG2	UG2	Upgrade to Second Channel for Generator, License
M8041A-UA2	UA2	Upgrade to Second Channel for Analyzer, License

Product	Option	Description
M8041A-UG3	UG3	Upgrade to Advanced Jitter Sources for Receiver Characterization, Module-wide License
M8041A-UG4	UG4	Upgrade to Multi-tap De-emphasis, Module-wide License
M8041A-UG5	UG5	Upgrade to Adjustable ISI, Module-wide License
M8041A-UG6	UG6	Upgrade to Reference Clock Input with Multiplying PLL, Clockgroup-wide License
M8041A-UG7	UG7	Upgrade to Advanced Interference Sources for Receiver Characterization, Module-wide License
M8041A-US1	US1	Upgrade to Interactive Link Training for PCI Express, Clockgroup-wide License
M8041A-US2	US2	Upgrade to SER/FER Analysis for Coded and Retimed Loopback, Clockgroup-wide License
M8041A-UA3	UA3	Upgrade to Analyzer Equalization, Module-wide License

M8051A - High-Performance BERT Module

M8051A- Basic Selection

Table 78 M8051A- Basic Selection

Product	Option	Description
M8051A-G08	G08	Generator one Channel, Data Rate up to 8.5 Gb/s
M8051A-G16	G16	Generator one Channel, Data Rate up to 16 Gb/s (Upgrade: U16)
M8051A-C08	C08	BERT one Channel, Data Rate up to 8.5 Gb/s
M8051A-C16	C16	BERT one Channel, Data Rate up to 16 Gb/s (Upgrade: UED)

M8051A- Module Functionality

Table 79 M8051A- Module Functionality

Product	Option	Description
M8051A-0G2	0G2	Second Channel for Generator, License
M8051A-0A2	0A2	Second Channel for Analyzer, License
M8051A-0G3	0G3	Advanced Jitter Sources for Receiver Characterization, Module-wide License
M8051A-0G4	0G4	Multi-tap De-emphasis, Module-wide License
M8051A-0G5	0G5	Adjustable ISI, Module-wide License
M8051A-0G7	0G7	Advanced Interference Sources for Receiver Characterization, Module-wide License
M8041A-0A3	0A3	Analyzer Equalization, Module-wide License

M8051A - License Upgrades for M8051A High-Performance BERT Module

Table 80 M8051A - License Upgrades for M8051A High-Performance BERT Module

Product	Option	Description
M8051A-U16	U16	Upgrade to 16 Gb/s data rate from M8051A-G08 and M8051A-C08, Module-wide License
M8051A-UED	UED	Upgrade to BERT from M8051A-G08 and M8051A-G16, Module-wide License
M8051A-UG2	UG2	Upgrade to Second Channel for Generator, Module-wide License
M8051A-UA2	UA2	Upgrade to Second Channel for Analyzer, Module-wide License
M8051A-UG3	UG3	Upgrade to Advanced Jitter Sources for Receiver Characterization, Module-wide License
M8051A-UG4	UG4	Upgrade to Multi-tap De-emphasis, Module-wide License
M8051A-UG5	UG5	Upgrade to Adjustable ISI, Module-wide License
M8041A-UG7	UG7	Upgrade to Advanced Interference Sources for Receiver Characterization, Module-wide License
M8051A-UA3	UA3	Upgrade to Analyzer Equalization, Module-wide License
M8051A-U16	U16	Upgrade to 16 Gb/s data rate from M8051A-G08 and M8051A-C08, Module-wide License

M8061A Multiplexer 2:1 with De-emphasis

Table 81 M8061A Licenses

Product	Option	Description
M8061A-001	001	32 Gb/s multiplexer
M8061A-004	004	4-tap de-emphasis
M8061A-008	008	Extension to 8-tap de-emphasis
M8061A-U04	U04	Upgrade to 4-tap de-emphasis
M8061A-U08	U08	Upgrade to 8-tap de-emphasis

M8062A 32Gb/s Front-end for J-BERT M8020A High-Performance BERT

M8062A - Basic Selection

Table 82 M8062A - Basic Selection

Product	Option	Description
M8062A-C32	C32	32 Gb/s BERT front-end
M8062A-G32	G32	32 Gb/s Pattern generator front-end

M8062A - Module Functionality

Table 83 M8062A - Module Functionality

Product	Option	Description
M8062A-0G4	0G4	Multi-tap De-emphasis License
M8062A-0G5	0G5	Adjustable Intersymbol Interference (ISI) License
M8062A-0A3	0A3	Analyzer Equalization License
M8062A-0A4	0A4	Clock Recovery up to 32 Gb/s

M8062A - License Upgrades for M8062A 32Gb/s Front-end for J-BERT M8020A High-Performance BERT

Table 84 M8062A - License Upgrades for M8062A 32Gb/s Front-end for J-BERT M8020A High-Performance BERT

Product	Option	Description
M8062A-UED	UED	Upgrade of M8062A-G32 Pattern Generator to M8062A-C32 BERT License
M8062A-UA3	UA3	Upgrade of M8062A to Analyzer Equalization License

Product	Option	Description
M8062A-UG4	UG4	Upgrade of M8062A to Multi-tap De-emphasis License
M8062A-UG5	UG5	Upgrade of M8062A to Adjustable Intersymbol Interference License
M8062A-UA4	UA4	Upgrade of M8062A to Clock Recovery up to 32 Gb/s

NOTE

In addition to the M8062A-UA4 license, M8062A modules with serial numbers < MY55400300 may also require a hardware upgrade in order to enable the CDR feature.

Keysight License Manager

Keysight License Manager is a software utility used with Full Licensing that enables end users to easily manage right-to-use licenses for software and hardware capabilities on Keysight instruments or systems. The graphical user interface (GUI) gives you a visual representation of the licenses installed on your Keysight Technologies systems and provides access to the following features:

- Install licenses for new capabilities
- Transport licenses from one controller to another
- Remove licenses for capabilities no longer needed

For detailed information on **Keysight License Manager**, refer to the *Keysight License Manager Help*. You can access the *Keysight License Manager Help* from the **Keysight License Manager** web page: http://www.keysight.com/find/LicenseManager

Keysight License Service

The **Keysight License Service** is used in Full Licensing to accomplish installation, listing, validation, removal, and transportation of instrument licenses.

NOTE

The Keysight License Service is a Windows service that runs automatically and appears in the Windows Task Manager and Event Viewer the same as other processes. ACCL code communicates with this service to perform licensing functions, and the service monitors system and user events and take appropriate actions.

NOTE

The Keysight License Service supports certain specific firewalls. These firewalls must be appropriately configured. For details, refer to Firewall Configuration for Licensing on page 448.

Installing the Licenses

Software Requirements

The M8070A uses the following operating system for installation:

- Microsoft Windows 7 (64 bit) SP1
- Windows 8 (64 bit)
- Windows 8.1 (64 bit)

NOTE In case you ordered M8020A-BU1 all module and M8070A licenses have been pre-installed (except for a floating/networked license). All other system configurations require license installation as described in this step.

Installing M8070A-OTP License (not required for M8020A-BU1 and M8030A-BU1)

The following procedure shows how to redeem and install a M8070A license on a dedicated host computer.

1. Locate the **Software License Entitlement Certificate**.
2. Follow the instructions on the **Software License Entitlement Certificate** to redeem your license.
3. You will receive a license file (in an email). The file has the suffix .lic.
4. Follow the instructions in the email to complete the installation of the license file.
5. In the M8070A software interface, verify that the license has been installed by selecting **Help** > **Licenses** and then viewing the license status in the **Installed** column.

Installing M8070A-ONP Floating/Networked License

An M8070A-ONP floating/networked license resides on a network server and is accessible by multiple users. The user checks out a license each time the software is accessed. The server tracks the number of licenses checked out and the number of licenses available for use.

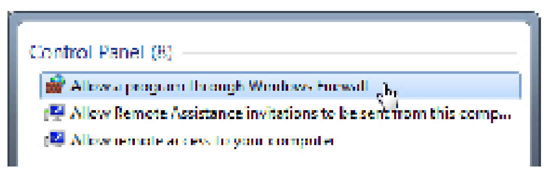
The server must be running license services. This server can be a dedicated computer running the license server or it can run license services and the M8070A software concurrently.

Firewall Configuration for Licensing

If your instrument or PC has a firewall installed and enabled, your system administrator may need to allow certain executables and/or a limited range of port numbers to go through the firewall. This topic discusses those ports and configuration for three common firewalls.

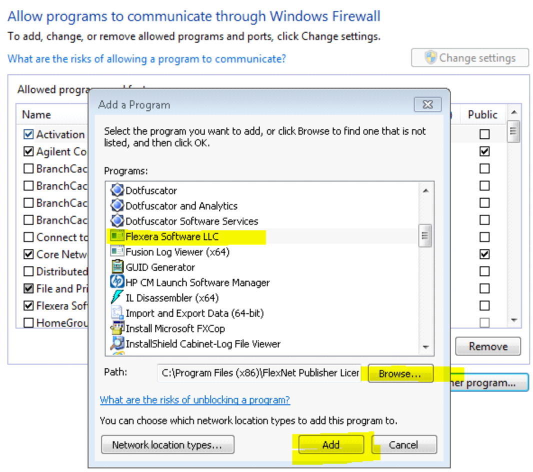
NOTE Typically, the Licensing installer will do the necessary configuration of the Windows firewall. These instructions are provided in case you need to verify or modify the firewall configuration manually.

1. In Windows, click **Start** > **Control Panel** > **System and Security** > **Windows Firewall**.
2. In the left pane, click **Allow a program or feature through Windows Firewall**. If this is not accessible, perform the following alternate procedure:
   1. In Windows, click **Start** and enter **Allow a program through Windows Firewall** in the search field.
   2. In the **Control Panel**, select **Allow a program through Windows Firewall** as shown in the following figure.



   3. Click **Change Settings**. You may need to provide an administrator password or provide confirmation for this action.
   4. Click on the **Allow another program** button.
   5. Select **M8000** then click **Add**.
   6. Add program path:

   C:\Program Files (x86)\FlexNet Publisher License Server Manager\lmadmin.exe.



7 Repeat step 4 to step 6 to add other programs.

8 In the **Allowed Programs** dialog, click **OK** to save the settings.

Installing the FlexNet License Manager

The **FlexNet License Manager** tools associated with floating licenses are available from Flexera at www.keysight.com/find/fnptools. You must use version 11.11.1 or higher.

FlexNet provides two different license server executables: *lmgrd.exe* and *lmadmin.exe*. You can use either one or the other.

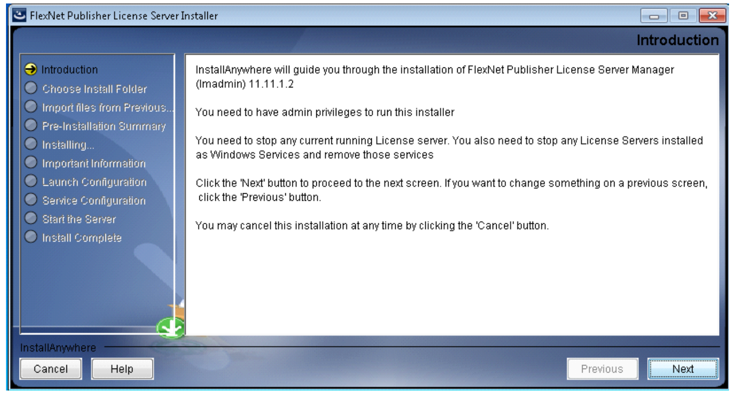
lmadmin Installation

The following procedure describes how to install *lmadmin.exe* (server side):

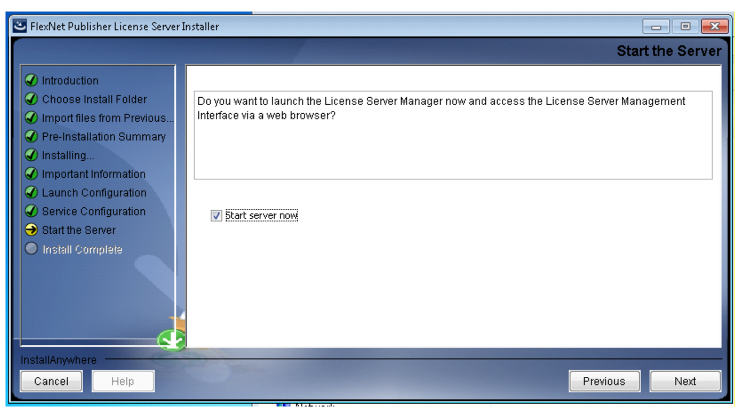
1 Copy the *agilent.lic* file from the Keysight Licensing installation path (typically, *C:\Program Files\Agilent\ACCL\Licensing\bin* or *C:\Program Files (x86)\ACCL\Licensing\bin*) into the licensing folder. (The license filename will change to *agilent0.00.lic* after successful installation.)

2 Determine whether the target system has Java installed, if not, install it from www.java.com. The Java virtual machine is required to run the *lmadmin.exe* installer.

3 Run the *lmadmin.exe*. (Right-click the installer file and **Run as Administrator**.) During the installation:



1 Accept the default installation path.
2 Check the box to **Install Visual C++ 2005 SP1 Redistributable**.
3 On the **Service Configuration** page, enter the **HTTP Port Number**: '8090'.
4 On the **Service Configuration** page, select the check-box to **Run as Service**.
5 On the **Start the Server** page, select the check box to **Start server now**. If you don't do it now, then you will need to start the *lmadmin* service before configuring the license server.

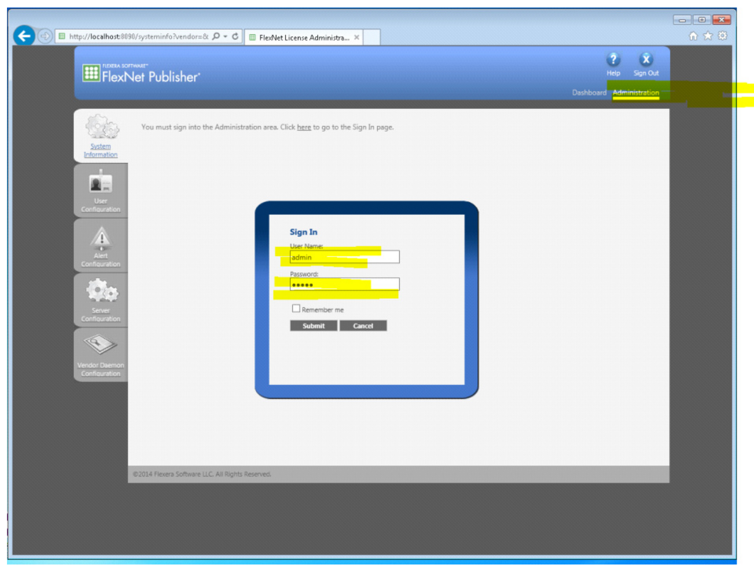


4 If the web browser does not shows the license configuration page, just exit the browser and reboot the PC.

5 After reboot start the browser and use the use following http://localhost:8090/ address again.Now you should see the web configuration page of the *lmadmin*.

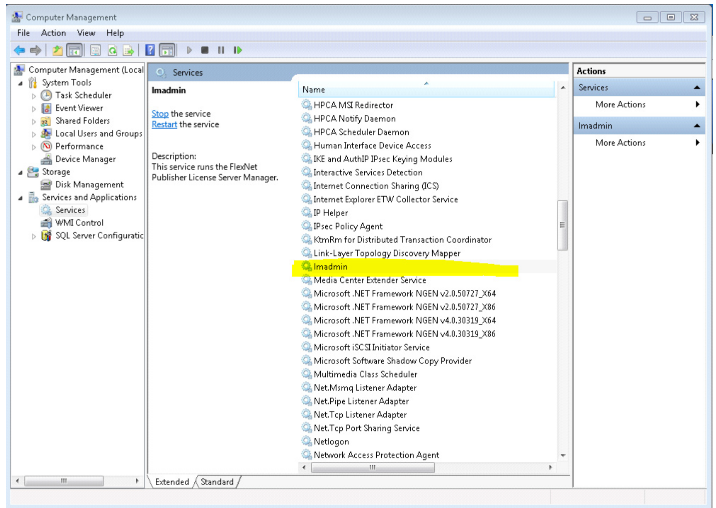
6 However, if you see the message ‘The page can’t be displayed ...’ again the firewall of windows has block the access. You may need to allow certain executable and/or the port number to go through the firewall. For firewall settings, refer to Firewall Configuration for Licensing on page 448.

7 In the login page, if you have not previously set up a login, sign in as “user admin” with password “admin”. You will then be required to change the password.



8 Start the browser and use the use following http://localhost:8090/ address again.

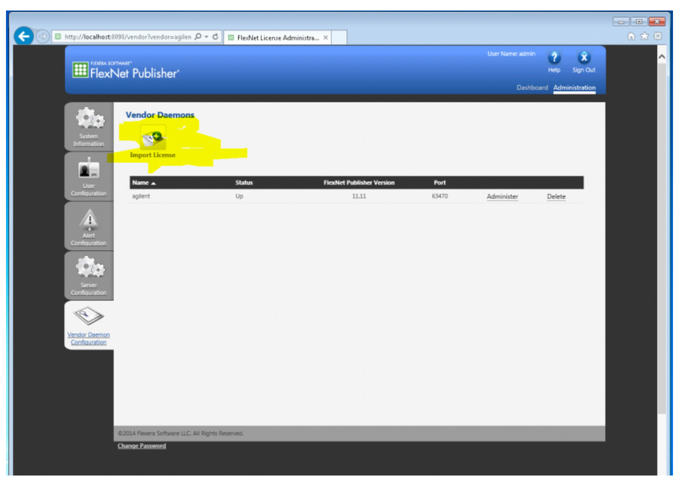
9 The *lmadmin* is installed as service. To check the state of the service, right click on the **Computer** icon on the desktop and then select **Manage**. Click on **Services** and check if service *lmadmin* is started.



10 Login to the *lmadmin* web configuration page. Add the *agilent.lic* and the *M8070A-0NP*.

C:\Program Files (x86)\Agilent\ACCL\Licensing\bin\agilent.lic

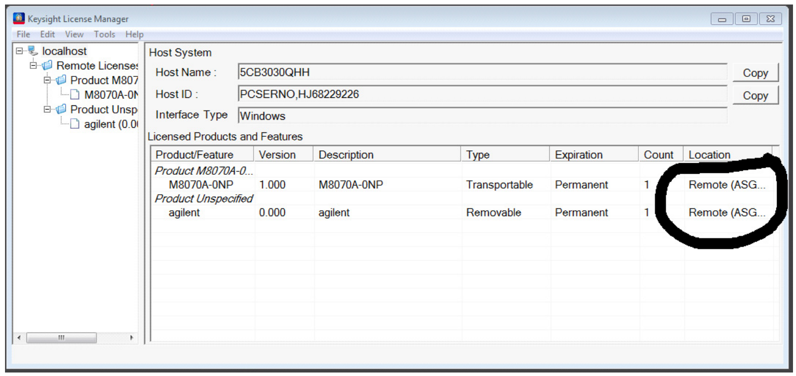
...\1001562893_1166729.lic



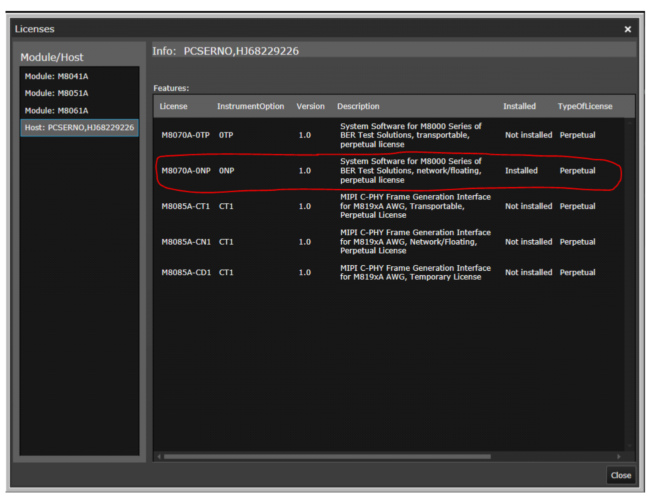
11 On the client side, install the M8070A software on the client PC.

12 Install the edited license M8070A-0NP on the client (import license in the Keysight License Manager).

13 In the **Keysight License Manager**, verify by opening the ‘Licenses list’ and ‘Feature Detail’ view. The following illustration displays the ‘Licenses list’ view.



14 Start the M8070A software. From the menu bar, select **Utilities** > **Licenses**. The M8070A license dialog will display the installed licenses.

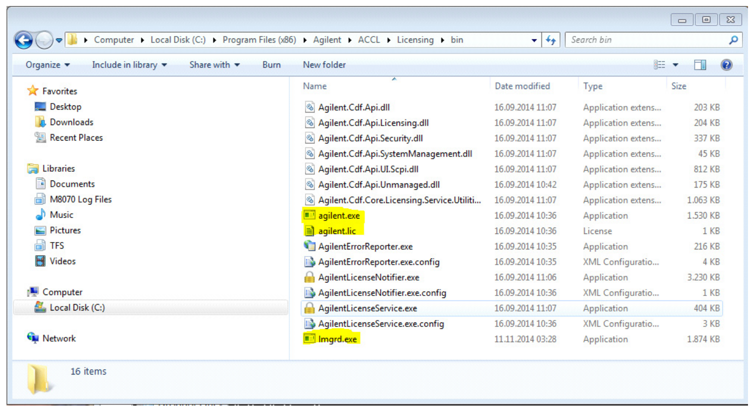


15 Request a floating license. For details, refer to Acquiring Floating Licenses on page 456.

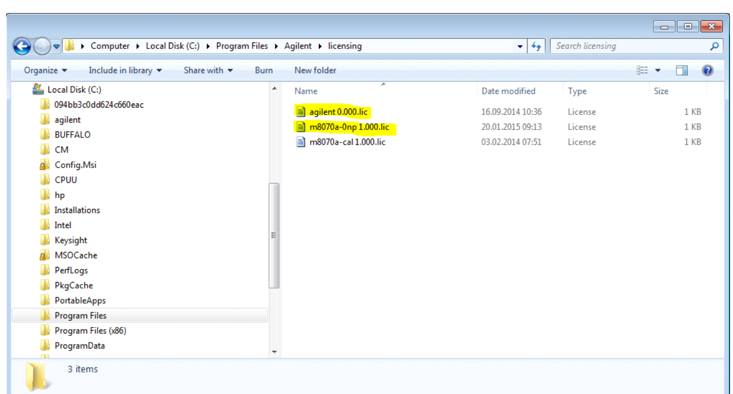
Lmgrd Installation

The following procedure describes how to install *lmgrd.exe (server side)*.

1 Copy the *agilent.lic* file from the Keysight Licensing installation path (typically found in *C:\Program Files\Agilent\ACCL\Licensing\bin* or in *C:\Program Files (x86)\Agilent\ACCL\Licensing\bin* into the licensing folder. (The license file name will change to *agilent0.00.lic* after successful installation).

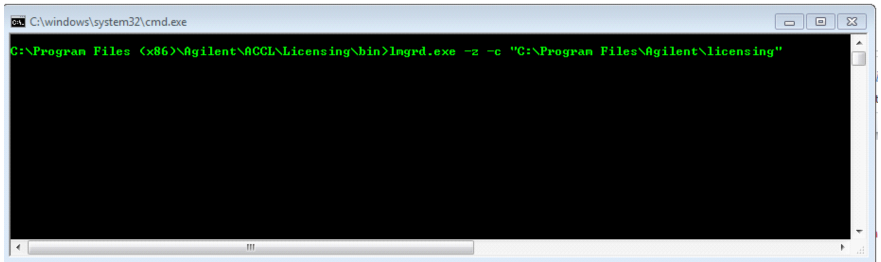


2 Copy the appropriate version of *lmgrd.exe* (32-bit or 64-bit, depending on your operating system) to the target system. It is recommended to put *lmgrd.exe* in the Keysight Licensing installation path (see step 1 for path information).



3 Install floating licenses to the licensing folder, *C:\Program Files\Agilent\Licensing*.

4 Open a DOS prompt and change directory to the location of lmgrd.exe.



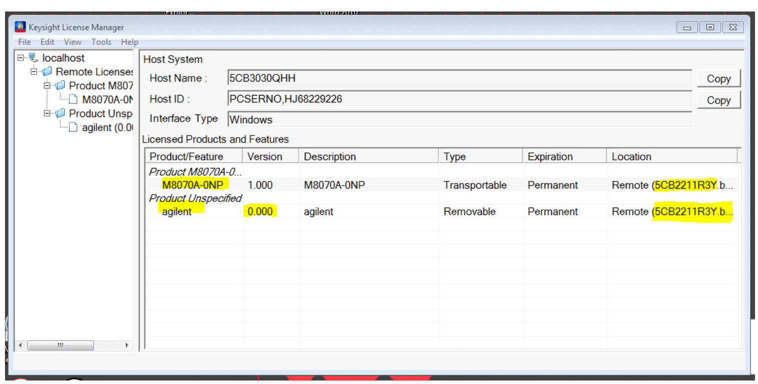
5 Launch lmgrd.exe with the following command:

lmgrd.exe -z -c "*C:\Program Files\Agilent\licensing*"

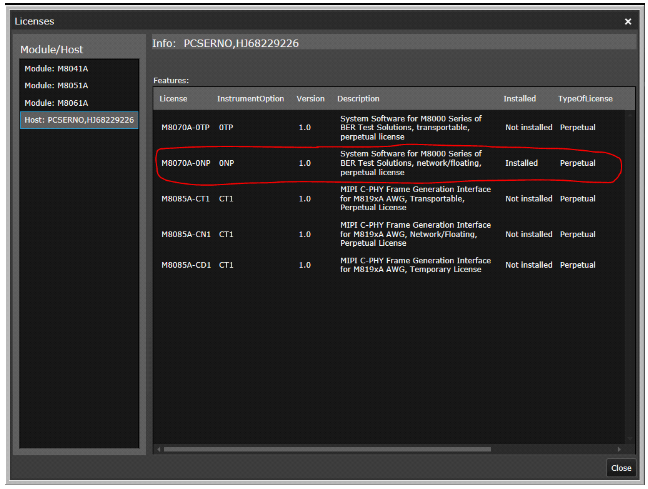
6 If the license server is running (lmgrd.exe) you will see the licenses in the **Keysight License Manager** on the server.

7 On the client side, install the edited license M8070A-0NP (import license in the **Keysight License Manager**)

8 In the **Keysight License Manager**, verify by opening the 'Licenses list' and 'Feature Detail' view. The following illustration displays the 'Licenses list' view.



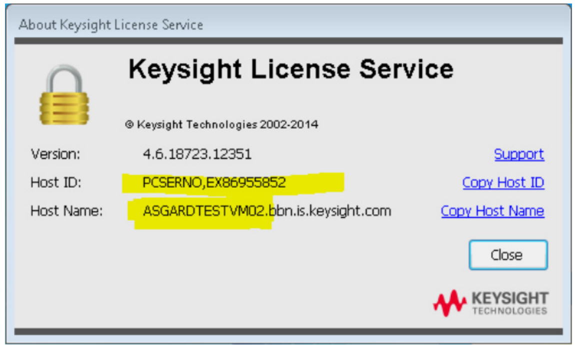
9 Start the M8070A software. From the menu bar, select **Utilities** > **Licenses**. The M8070A license dialog will display the installed licenses.



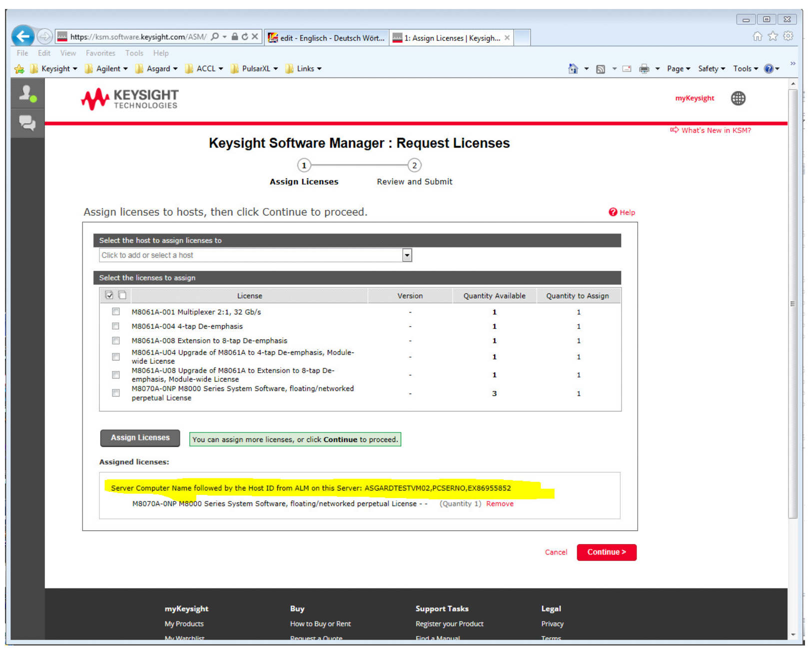
10 Request a floating license. For details, refer to Acquiring Floating Licenses on page 456.

Acquiring Floating Licenses

1. Contact your Keysight Technologies representative to purchase licenses for your software. You will receive an order number and certificate number, which you will use to request a license file in step 3.
2. Select and configure your license server machine as described above. For more information, refer to the *Flexera Software License Administration Guide*. You must install Keysight Licensing on the license server computer, as described in Creating a New License Service using lmadmin.exe on page 458, before requesting a license file in step 3.
3. Request a license file for your license server machine:
    1. Determine the hostname and serial number (10-digit alphanumeric sequence, which is the second component of the host ID) of your license server from **Keysight License Notifier's About** box.
    2. To display the **About** box, right-click the **Keysight License Notifier** icon in the notification area (bottom right of your screen) and click **About Keysight License Notifier**.
    - You will need the highlighted items shown in the following figure:



- Visit the **Keysight Software Manager** website to request a license file.



4 When you receive your license via e-mail from Keysight Technologies, carefully read the instructions sent with the license. If the license is a "served" or "floating" license, put it in the directory *C:\Program Files\Agilent\licensing* on the license server.

Creating a New License Service using lmadmin.exe

Follow the given steps to create a new license service using lmadmin.exe:

1 To access the lmadmin license server management interface, open a web browser and navigate to http://<server>:8080, where <server> is the name of the computer on which the license server is running. For this first-time configuration, you need to be logged on to the license server computer, so that you can simply navigate to http://localhost:8080.

NOTE If the browser cannot find the web page, then the lmadmin service may not be running on the server. Go to **Start** > **Control Panel** > **Administrative Tools** > **Services**, find “lmadmin” and start the service. After starting the service, refresh, or try navigating the browser to the web page again.

2 Select the **Administration** tab.

3 Log in. If you have not previously set up a login, sign in as “user admin” with password “admin”. You will then be required to change the password. Don’t forget this password.

4 Select the **Vendor Daemon Configuration** tab.

5 Select **Import License**.

6 Browse to the new floating license file. You can only choose one file at this step, so if you have more than one license file, then pick one of them, and click **Import License**.

NOTE To add more license files, follow the procedure below.

7 An **Import Error** and an **Import Warning** will be displayed indicating that it can’t find the *Agilent*.exe file. Click **OK**.

8 Select the “Agilent” vendor daemon and click **Administer**.

9 Change the **License File or Directory** to “licenses\agilent” instead of the file you specified above. This will enable you to easily add more license files.

10 Change the **Vendor Daemon Location** to the location of the Agilent.exe file:

   1 For 32-bit Windows, enter “*C:\Program Files\Agilent\ACCL\Licensing\bin\Agilent.exe*”.

   2 For 64-bit Windows, enter “*C:\PROGRA~2\Agilent\ACCL\Licensing\bin\Agilent.exe*”.

NOTE

For 64-bit Windows, the file is in “C:\Program Files (x86)”, but the Web page does not allow you to enter a file path containing an open or close parenthesis, so you must instead enter the short file name “C:\PROGRA~2” of the path.

11 Click **Save**.

12 Start the daemon by pressing **Start**.

13 To verify the status of the server, select the **Dashboard** tab and click the **Concurrent** button. Wait for the Web page to refresh. If the “Agilent” daemon is not selected in the vendor drop-down list in the upper right side of the Licenses page, then select it by clicking **Select**. You will now see the list of floating licenses that are managed by *lmadmin.exe*.

Installing and Setting up Client PC Floating Licenses

The following procedures show how to install and set up client PC floating licenses. Repeat this procedure for each client PC.

1 Ensure that you have the Keysight Licensing Services and the M8000 software to be installed on the client PC.

2 Create the Client floating license files.

   The Client floating license files redirect licensing requests to the license server. Only one client floating license file needs to be created. This client floating license file is copied and re-used by each client PC.

   Make a copy of the original license server floating license file. Remove all line entries except these three lines, which you must modified as shown.

   - “SERVER” line: Keep original text
   - “VENDOR” line: If the optional VENDOR line “port” number is included in the original server license file, remove the optional “Vendor” daemon TCP/IP port number from the line
   - “USE_SERVER” line: Keep original text

3 Install the client floating license files.
   1 Start the **Keysight License Manager** by double clicking the **Keysight License Notifier** icon or click **Start** > **(All) Programs** > **Keysight License Manager** > **Keysight License Manager**.
   2 To install the license file, drag and drop the license file onto the **Keysight License Manager "License Features"** window. Alternatively, you can click the **Keysight License Manager** menu **File**.
   3 Install and follow the prompts to install the license file. The license files are stored in different folder locations depending on whether the license Server and client reside on separate PCs or the same PC.
   - License Server and Client reside on separate PCs: *C:\Program Files\Agilent\licensing*
   - License Server and Client reside on same PC: *C:\Program Files\Agilent\licensing\Remote Licenses*
4 Verify the client PC configuration is correct.

When the license file is successful installed, The client floating licenses that are available on the license server are listed in the **Keysight License Manager "License Products and Features"** window. To view the available licenses, select the Localhost tree, Remote Licenses node. Ensure that your licensed options and features are shown.

Installing Module Licenses (not required for M8020A-BU1 and M8030A-BU1)

Module licenses enable specific features in the modules of the M8020A/M8030A system. Once a module license has been installed using the **Keysight License Manager**, the next time the M8070A software and M8020A/M8030A hardware are started, the license is recognized by the M8070A software and compared to the module's serial number. If the PC Host ID and serial number match, the EEPROM in the module is programmed and the feature is enabled. Even if the M8070A software license is transported to another host computer, the module feature will remain enabled.

The following procedure shows how to redeem and install a module license.

1 Locate the **Software License Entitlement Certificate**.
2 Follow the instructions on the **Software License Entitlement Certificate** to redeem your license.
3 You will receive a license file (in an email). The file has the suffix .lic.
4 Follow the instructions in the email to complete the installation of the license file.

5 In the M8070A software interface, verify that the license has been installed by selecting **Help** > **Licenses** then viewing the license status in the Installed column.

Transporting an M8070A License

Transportable licenses are licenses that can be moved from one host controller to another using the **Keysight License Manager**.

If you need to download and install the **Keysight License Manager**, go to the following Web page:

http://www.keysight.com/find/LicenseManager

1 Start the **Keysight License Manager** by double clicking the **Keysight License Notifier** icon or click **Start** > **(All) Programs** > **Keysight License Manager** > **Keysight License Manager**.

2 In the **Keysight License Manager**, click on **Help** > **Keysight License Manager Help** and perform the procedure in the **Transporting Licenses** help topic.

11 Troubleshooting

Updating software components

Updated versions of the M8020A, M8030A and module specific software components are available on the Keysight website.

These software components are available as .EXE files. To download a software upgrade:

1. Go to http://www.keysight.com.
2. Click the **Technical Support** tab.
3. Click **Drivers and Software**.
4. Type the model number of the instrument module for which software update is needed and click **Find**. Model number is located on the front panel of the module.
5. Click the **Driver & Software** link on the module page.
6. Download the required software update from the list of available updates.

The chassis does not power up

If the chassis or a module does not appear to power up, check the following:

- The circuit breakers at the rear of the chassis are set to the right, which is the **ON** position.
- The AC power cords are connected to a working power source.
- The electrical circuits are not overloaded. Check the combined power requirements of all equipment on the same circuit.
- There are no empty slots in the chassis. Leaving slots empty can overheat the inserted modules, causing them to shut down.

Module is exceptionally hot

Check that the vent holes on the chassis are not blocked.

Check that a filler panel module or an instrument module is installed into empty slots on either side of an instrument module.

Contacting Keysight Technologies

To locate a sales or service office near you, go to www.keysight.com/find/contactus.

Index

Numerics

A

B

C

D

E

F

G

H

I

J

L

M

N

O

P

R

S

T

U

V

KEYSIGHT TECHNOLOGIES

Edition 5.1, March 2016
M8020-91030

www.keysight.com