Tektronix®

TekScope Application User Manual

Offline TekScope

077-1699-00

TekScope Application User Manual

Offline TekScope

Register now!
Click the following link to protect your product.
www.tek.com/register

077-1699-00

Tektronix, Inc.

14150 SW Karl Braun Drive

P.O. Box 500

Beaverton, OR 97077

USA

For product information, sales, service, and technical support:

- In North America, call 1-800-833-9200.
- Worldwide, visit *www.tek.com* to find contacts in your area.

Table of Contents

List of Figures

List of Tables

TEKTRONIX END USER LICENSE AGREEMENT

This End User Agreement ("Agreement") is an agreement between Tektronix, Inc., an Oregon corporation, and its corporate affiliates, subsidiaries, and divisions as applicable (collectively, "**Tektronix**," "**we**," "**us**," or "**our**") and You (including any entity or organization you represent, collectively, "**Customer**" or "**You**"). Please read this Agreement carefully as this Agreement governs the terms and conditions under which You are permitted to use Tektronix's software and services.

THE SOFTWARE, ENCODED OR INCORPORATED WITHIN EQUIPMENT OR ACCOMPANYING THIS AGREEMENT, IS FURNISHED SUBJECT TO THE TERMS AND CONDITIONS OF THIS AGREEMENT. BY INDICATING YOUR ACCEPTANCE OF THESE TERMS BY SELECTING AN "ACCEPT" OR SIMILAR BUTTON IN A SOFTWARE MENU, OR BY RETAINING THE SOFTWARE FOR MORE THAN THIRTY DAYS OR USING THE SOFTWARE IN ANY MANNER YOU (A) ACCEPT THIS AGREEMENT AND AGREE THAT YOU ARE LEGALLY BOUND BY ITS TERMS; AND (B) REPRESENT AND WARRANT THAT: (I) YOU ARE OF LEGAL AGE TO ENTER INTO A BINDING AGREEMENT; AND (II) IF YOU ARE A REPRESENTATIVE FOR A CORPORATION OR OTHER LEGAL ENTITY, YOU HAVE THE RIGHT, POWER, AND AUTHORITY TO ENTER INTO THIS AGREEMENT ON BEHALF OF SUCH ENTITY AND BIND SUCH ENTITY TO ITS TERMS. IF YOU DO NOT AGREE TO THE TERMS OF THIS AGREEMENT, TEKTRONIX WILL NOT AND DOES NOT LICENSE THE SOFTWARE TO YOU AND YOU MUST NOT DOWNLOAD, INSTALL, OR USE THE SOFTWARE. UNITED STATES GOVERNMENT CUSTOMERS OR END-USERS MAY REQUEST A GOVERNMENT ADDENDUM TO THIS AGREEMENT.

NOTWITHSTANDING ANYTHING TO THE CONTRARY IN THIS AGREEMENT OR YOUR ACCEPTANCE OF THE TERMS AND CONDITIONS OF THIS AGREEMENT, NO LICENSE IS GRANTED (WHETHER EXPRESSLY, BY IMPLICATION, OR OTHERWISE) UNDER THIS AGREEMENT TO ANY SOFTWARE THAT YOU DID NOT ACQUIRE LAWFULLY OR THAT IS NOT A LEGITIMATE, AUTHORIZED COPY OF TEKTRONIX'S SOFTWARE. THIS AGREEMENT EXPRESSLY EXCLUDES ANY RIGHTS CONCERNING SUCH ILLEGITIMATE COPIES.

IF THESE TERMS ARE NOT ACCEPTABLE, THE UNUSED SOFTWARE AND ANY ACCOMPANYING DOCUMENTATION SHOULD BE RETURNED PROMPTLY TO TEKTRONIX (WITHIN 30 DAYS OF PURCHASE) FOR A FULL REFUND OF THE LICENSE FEE PAID. (FOR INFORMATION REGARDING THE RETURN OF SOFTWARE ENCODED OR INCORPORATED WITHIN EQUIPMENT, CONTACT THE NEAREST TEKTRONIX SALES OFFICE.)

DEFINITIONS

"Equipment" means Tektronix equipment that the Software is encoded or incorporated within or installed onto.

"Aggregated Statistics" means data and information related to Your use of the Software, including Your Data, that is used by Tektronix in an aggregate and anonymized manner, including to compile statistical and performance information related to the provision and operation of the Software.

"Authorized User" means an employee who has been designated by You as a user within a group of authorized users of the Software.

"Customer Order" means a valid and accepted order, through an online payment portal, purchase order, enterprise agreement, or other written means.

"Documentation" means Tektronix's user manuals, handbooks, and guides relating to the Software provided by Tektronix to You either electronically or in hard copy form.

"Equipment" means Tektronix equipment that the Software is designed to work with.

"Export Laws" means United States export controls and economic sanctions laws, regulations and orders, including the Export Administration Regulations, 15 CFR Parts 730-774 and laws, regulations, and orders administered by the Office of Foreign Assets Control, Department of the Treasury, as well as any similar export control and economic sanctions laws, regulations, and orders issued or administered by governmental authorities with jurisdiction over locations where the Software is provided or used.

"Feedback" means any feedback or suggestions concerning the functionality or performance of the Software.

"Initial Term" means the number of calendar months that You paid for when purchasing Tek Scope.

"Losses" means all losses, damages, liabilities, and costs, including reasonable attorneys' fees.

"Reverse Engineer" means to decompile, decrypt, disassemble, or otherwise attempt to derive source code, techniques, processes, algorithms, know-how, or other information.

“Software” means the software encoded or incorporated within accompanying equipment or accompanying this agreement. Software includes Tek Scope only when Tek Scope is encoded or incorporated within accompanying equipment or accompanies this agreement.

"Term" means the number of months paid for by You.

"Tektronix" means Tektronix, Inc., an Oregon corporation, or local Tektronix’ legal entity that is supplying the equipment.

"Tektronix IP" means the Software, the Documentation, and any and all intellectual property provided to You or any Authorized User in connection with the foregoing. Tektronix IP includes Aggregated Statistics and any information, data, or other content derived from Tektronix's monitoring of Your access to or use of the Software, but does not include Your Data.

“Tek Scope” means the Tek Scope software, specifically.

"Your Data" means, other than Aggregated Statistics, information, data, and other content, in any form or medium, that is submitted, posted, or otherwise transmitted through or submitted to the Software by You or on Your behalf.

LICENSE

Subject to the terms and conditions of this Agreement, Tektronix grants You a non-exclusive, non-transferable license to the Software, as follows

You may:

1. Use the Software with the Equipment, or if the Software is not encoded or incorporated in any Tektronix equipment, on no more than one machine at a time;
2. Activate the Tek Scope software on up to two machines at a time, but only use on one computer at a time;
3. Copy the Software for archival or backup purposes, provided that no more than one (1) such copy is permitted to exist at any one time, and provided that each copy includes a reproduction of any patent or copyright notice or restrictive rights legend that was included with the Software, as received from Tektronix;
4. Fully transfer the Equipment to a third party but only if prominently accompanied by this End User License Agreement, and such third-party recipients agree to be bound by the terms of this Agreement; and
5. Integrate Tektronix products that contain the Software into a system and sell or distribute that system to third parties, provided that those third parties are bound by the terms of this Agreement, and provided that You (i) do not separate the Software from any Equipment it is incorporated into, (ii) do not retain any copies of the Software, and (iii) do not modify the Software.

You may not:

1. Use the Software other than for its intended purpose as provided above in the section “You may,” or in conflict with the terms and restrictions of this Agreement;
2. Distribute or transfer the Software to any person or organization outside of Your organization without Tektronix’s prior written consent, except in connection with a permitted use authorized in “You may” paragraphs 4 or 5 above;
3. Decompile, decrypt, disassemble, or otherwise attempt to derive the source code, techniques, processes, algorithms, know-how, or other information (collectively “Reverse Engineer”) from the Software or permit or induce any third party to do so, except to the limited extent allowed by directly applicable law or third party license (if any), and only to obtain information necessary to achieve interoperability of independently created software with the Software;
4. Modify, translate, adapt, or create derivative works of the Software, or merge the Software with any other software;
5. Copy the documentation accompanying the Software; except as necessary to support an authorized use;
6. Remove any copyright, trademark, or other proprietary notices from the Software or any media relating thereto; or
7. Export or re-export, directly or indirectly, the Software or Equipment, any associated documentation, or systems created in accordance with “You may” section 4 above, to any country to which such export or re-export is restricted by law or regulation of the United States or any foreign government having jurisdiction without the prior authorization,

if required, of the Office of Export Administration, Department of Commerce, Washington, D.C. and the corresponding agency of such foreign government;

8. Use the Software or Equipment in any manner or for any purpose that infringes, misappropriates, or otherwise violates any intellectual property rights or other proprietary rights of any person, or any applicable laws or in the case of Tek Scope, in any way that violates Tektronix's Terms Of Use for tekcloud.com;
9. Use the Software or Equipment in a network or system with other products or services that are incompatible, insecure or not compliant with applicable laws;
10. Bypass, circumvent, damage or otherwise interfere with any security or other features of the Software or Equipment designed to control the manner in which they are used, or harvest or mine Tektronix's proprietary content or information from the Software or Equipment; or
11. Use the Software or Documentation to build a similar or competitive software, product, or service.

THE SOFTWARE MAY NOT BE USED, COPIED, MODIFIED, MERGED, OR TRANSFERRED TO ANOTHER EXCEPT AS EXPRESSLY PERMITTED BY THESE TERMS AND CONDITIONS.

FEEDBACK

If You provide feedback to Tektronix concerning the functionality and performance of the Software or Equipment, including without limitation identifying potential errors and improvements, any comments, questions, suggestions, or the like ("Feedback"), Tektronix is free to use such Feedback without any attribution, compensation, or restriction in any manner to improve or enhance its products, irrespective of any other obligation or limitation between the Parties governing such Feedback. You hereby grant Tektronix an irrevocable, worldwide, perpetual, royalty-free license to use Your Feedback for any purpose whatsoever and waive any moral rights You may have in the Feedback. Tektronix is not obligated to use Your Feedback.

PAYMENT AND DELIVERY

All Software fees and/or support fees are paid up front through Tektronix's online system or through a Customer Order. Except as expressly provided herein, all payment obligations are noncancelable and all sums paid are non-refundable. Tektronix, in its discretion, will deliver all Software and/or Documentation by electronic means, in which case Tektronix's obligation to deliver the Software and/or Documentation is completed at such time as Tektronix makes the Software available on a specific Tektronix server and gives You a method of accessing and downloading the Software and/or Documentation.

You will be responsible for any taxes that may now or hereafter be imposed, levied or assessed with respect to the possession or use of the Software or associated license, including any sales, use, property, and excise taxes, and similar taxes, duties, or charges.

OWNERSHIP

Title to the Software and all copies thereof, but not the media on which the Software or copies may reside, shall remain with Tektronix or others from whom Tektronix has obtained a respective licensing right. Nothing in this Agreement grants, by implication, waiver, estoppel, or otherwise, to You or any third party any intellectual property rights or other right, title, or interest in or to the Software or any intellectual property rights embodied or contained therein.

You retain all right, title, and interest, including all intellectual property rights, in and to Your Data.

In the case of Tek Scope, You hereby grant Tektronix a non-exclusive, royalty-free, worldwide license to reproduce, distribute, and otherwise use and display Your Data and perform all acts with respect to the Your Data as may be necessary for Software functions, and a non-exclusive, perpetual, irrevocable, royalty-free, worldwide license to reproduce, distribute, modify, and otherwise use and display Your Data incorporated within the Aggregated Statistics.

GOVERNMENT NOTICE

If the Software or any related documentation is acquired by or for an agency of the U.S. Government, the Software and documentation will be considered "commercial computer software" or "commercial computer software documentation" respectively, as those terms are used in 48 CFR §12.212, 48 CFR §227.7202, or 48 CFR §252.227-7014, as applicable, and are licensed with only those rights as are granted to all other licensees as set forth in this Agreement. Any use, modification, reproduction, release, performance, display, or disclosure of the Software and Documentation by the U. S. Government will be solely in accordance with the terms of this Agreement.

TERM

The license granted herein is effective until terminated. The license may be terminated by You at any time upon written notice to Tektronix. The license may be terminated by Tektronix if You fail to comply with any term or condition and such failure is not remedied within fifteen (15) days after notice hereof from Tektronix. Upon termination by either party, You shall return to Tektronix or destroy, the Software and all associated documentation, together with all copies in any form.

IF YOU TRANSFER, DISTRIBUTE, OR OTHERWISE MAKE AVAILABLE ANY COPY, MODIFICATION, OR MERGED PORTION OF THE SOFTWARE WITHOUT THE AS EXPRESS PERMISSION OF THESE TERMS AND CONDITIONS OR PRIOR WRITTEN CONSENT OF TEKTRONIX, YOUR LICENSE WILL BE IMMEDIATELY AND AUTOMATICALLY TERMINATED.

LIMITED WARRANTY

Tektronix does not warrant that the functions contained in the Software will meet Your requirements or that the operation of the Software will be uninterrupted, secure, or error-free.

EXCEPT AS SEPARATELY PROVIDED IN A WRITTEN WARRANTY FROM TEKTRONIX, THE SOFTWARE IS PROVIDED “AS IS” WITHOUT ANY WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO, THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, TITLE, QUIET ENJOYMENT, AND NON-INFRINGEMENT.

THE SOFTWARE IS NOT DESIGNED OR INTENDED FOR USE IN HAZARDOUS ENVIRONMENTS REQUIRING FAIL-SAFE PERFORMANCE INCLUDING WITHOUT LIMITATION, IN THE OPERATION OF NUCLEAR FACILITIES, AIRCRAFT NAVIGATION OR COMMUNICATION SYSTEMS, AIR TRAFFIC CONTROL, WEAPONS SYSTEMS, DIRECT LIFE-SUPPORT MACHINES, OR ANY OTHER APPLICATION IN WHICH THE FAILURE OF THE SOFTWARE COULD LEAD TO DEATH, PERSONAL INJURY OR SEVERE PHYSICAL OR PROPERTY DAMAGE (COLLECTIVELY "HAZARDOUS ACTIVITIES"). TEKTRONIX AND ITS AFFILIATES, LICENSORS, AND RESELLERS EXPRESSLY DISCLAIM ANY EXPRESS OR IMPLIED WARRANTY OF FITNESS FOR HAZARDOUS ACTIVITIES.

LIMITATION OF LIABILITY

IN NO EVENT SHALL TEKTRONIX, ITS AFFILIATES, LICENSORS, OR RESELLERS BE LIABLE FOR: (1) ECONOMICAL, INCIDENTAL, CONSEQUENTIAL, INDIRECT, SPECIAL, PUNITIVE OR EXEMPLARY DAMAGES, WHETHER CLAIMED UNDER CONTRACT, TORT OR ANY OTHER LEGAL THEORY, (2) LOSS OF OR DAMAGE TO YOUR DATA OR PROGRAMMING, LOSS OF PROFITS, BUSINESS INTERRUPTION, OR OTHER PECUNIARY LOSS ARISING FROM THE USE OF (OR INABILITY TO USE) THE SOFTWARE, (3) PENALTIES OR PENALTY CLAUSES OF ANY DESCRIPTION, (4) ANY DAMAGE, CLAIMS, OR LOSSES RESULTING FROM THE USE OF THE SOFTWARE IN CONJUNCTION WITH OTHER PRODUCTS OR SERVICES (INCLUDING THIRD-PARTY PRODUCTS OR SERVICES); OR (5) INDEMNIFICATION OF YOU OR OTHERS FOR COSTS, DAMAGES, OR EXPENSES RELATED TO THE GOODS OR SERVICES PROVIDED UNDER THIS LIMITED WARRANTY, EVEN IF TEKTRONIX OR ITS AFFILIATES, LICENSORS, OR RESELLERS HAVE ADVANCE NOTICE OF THE POSSIBILITY OF SUCH DAMAGES. BECAUSE SOME STATES/JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL OR INCIDENTAL DAMAGES, SOME OF THE ABOVE LIMITATIONS MAY NOT APPLY TO YOU, BUT THEY SHALL APPLY TO THE MAXIMUM EXTENT PERMITTED BY LAW. NOTWITHSTANDING ANYTHING HEREIN TO THE CONTRARY, IN NO EVENT SHALL TEKTRONIX’S TOTAL AGGREGATED LIABILITY TO YOU FOR ALL DAMAGES IN ANY ONE OR MORE CAUSES OF ACTION EXCEED THE AMOUNT RECEIVED BY TEKTRONIX FROM YOU FOR THE SOFTWARE OR EQUIPMENT.

You are solely responsible for Your data. You must back up Your data before Tektronix or a third party performs any remedial, upgrade, or other work on Your systems, including any Equipment. If applicable law prohibits exclusion of liability for lost data, then Tektronix will only be liable for the cost of the typical effort to recover the lost data from Your last available back up.

SECURITY DISCLAIMER

This Software and its associated Equipment are not designed or intended to be used with unsecure networks. You acknowledge that use of the Equipment may rely upon certain networks, systems, and data communication mediums that are not controlled by Tektronix and that may be vulnerable to data or security breaches, including, without limitation, internet networks used by Your internet providers and the databases and servers controlled by Your internet providers. Tektronix shall not be liable for any such breaches, including without limitation, damages and/or loss of data related to any security breach, and disclaims all warranties, including any implied or express warranties that any content will be secure or not otherwise lost or altered.

For the avoidance of doubt, if You choose to connect this Software or Equipment to a network, it is Your sole responsibility to provide and continuously ensure a secure connection to that network. You agree to establish and maintain appropriate measures (e.g., firewalls, authentication measures, encryption, anti-virus applications, etc.) to protect the Software and Equipment and any associated data against security breaches including unauthorized access, destruction, use, modification, or disclosure. Notwithstanding the foregoing, You shall not use any Products in a network with other products or services that are incompatible, insecure or not compliant with applicable laws.

THIRD-PARTY DISCLAIMER

The Software may contain software owned by third parties and obtained under a license from those parties ("Third Party Software"). Your use of such Third Party Software is subject to the terms and conditions of this Agreement and the applicable Third Party Software licenses. Except as expressly agreed otherwise, third parties do not warrant the Third Party Software, do not assume any liability with respect to its use, and do not undertake to furnish any support or information relating thereto.

GENERAL

Unless the Customer is the United States Government, this Agreement contains the entire agreement between the parties with respect to the use, reproduction, and transfer of the Software, and shall be governed by the laws of the state of Oregon.

If the Customer is the United States Government, all contract disputes arising out of or relating to this Agreement will be governed by and construed in accordance with the Contract Disputes Act (CDA), 41 U.S.C. §§ 7101-7109. Any legal suit, action, or proceeding arising out of or relating to this Agreement or the transaction contemplated hereby will be instituted in the court or board of jurisdiction under the CDA. If the matter is tortious in nature, the action will be brought under the Federal Tort Claims Act (FTCA), 28 U.S.C. § 1346(b).

In no event will Tektronix be liable to You, or be deemed to have breached this Agreement, for any failure or delay in performing its obligations under this Agreement, if and to the extent such failure or delay is caused by any circumstances beyond Tektronix's reasonable control, including but not limited to acts of God, flood, fire, earthquake, explosion, war, terrorism, invasion, riot or other civil unrest, strikes, labor stoppages or slowdowns or other industrial disturbances, or passage of law or any action taken by a governmental or public authority.

No amendment to or modification of this Agreement is effective unless it is in writing and signed by an authorized representative of each Party.

You shall be responsible for any taxes that may now or hereafter be imposed, levied or assessed with respect to the possession or use of the Software or the rights and licenses granted under this Agreement, including any sales, use, property, value added, and excise taxes, and similar taxes, duties, or charges.

Any waiver by either party of any provision of this Agreement shall not constitute or be deemed a subsequent waiver of that or any other portion.

Except as otherwise set forth in this Agreement, (1) no failure to exercise, or delay in exercising, any rights, remedy, power, or privilege arising from this Agreement will operate or be construed as a waiver thereof; and (2) no single or partial exercise of any right, remedy, power, or privilege under this Agreement will preclude any other or further exercise thereof or the exercise of any other right, remedy, power, or privilege.

If any provision of this Agreement is invalid, illegal, or unenforceable in any jurisdiction, such invalidity, illegality, or unenforceability will not affect any other term or provision of this Agreement or invalidate or render unenforceable such term or provision in any other jurisdiction. Upon such determination that any term or other provision is invalid, illegal, or unenforceable, the Parties will negotiate in good faith to modify this Agreement so as to effect their original intent as closely as possible in a mutually acceptable manner in order that the transactions contemplated in this Agreement be consummated as originally contemplated to the greatest extent possible.

All notices or questions regarding this Agreement should be directed to the nearest Tektronix Sales Office.

Additional Terms for Tek Scope

In addition to the terms above, the use of the Tek Scope software is also governed by the following terms and conditions. In the event of a conflict between these additional terms and the terms above, the additional terms below will control:

YOUR DATA

You are solely responsible for Your Data. You assume all risks associated with use of Your Data, including any reliance on its accuracy, completeness or usefulness by others, or any disclosure of Your Data that personally identifies an Authorized User or a third party. You represent and warrant that Your Data does not violate Tektronix's tekcloud.com Terms of Use. Tektronix reserves the right to review Your Data and to investigate and/or take appropriate action if You violate the tekcloud.com Terms of Use.

You will, to the extent that Your Data contains personal information and/or sensitive personal information about an individual (including an employee or contractor), procure from that individual all necessary consents required by law to enable that information to be used by Tektronix and its agents. Tektronix shall not be obligated to obtain any such consent and shall not be liable to any individual for Your failure to obtain any such consent as required by law.

PRIVACY

Tektronix will handle and store Your Data in compliance with applicable law and the applicable Tektronix privacy statement, available at www.tek.com/privacy-statement.

USE OF DATA

You agree that Tektronix may collect any information associated with your use of the Tek Scope software, including but not limited to information related to your access to and use of Tek Scope, computer or electronic devices, system and application software, and peripherals, that are gathered in connection with Tektronix's provision of Tek Scope, Tek Scope updates, product support, and other services related thereto. You acknowledge that Tektronix may compile Aggregated Statistics based on Your Data or your use of Tek Scope. You agree that Tektronix may use this information and information about you: (a) as necessary to provide and maintain functions of Tek Scope and related services or (b), for other purposes, but only if you opt-in or otherwise agree to those purposes. To facilitate communication with Tek Scope, Tektronix will obtain the IP Address related to the PC on which Tek Scope is running. Tektronix will treat this IP Address as personal identifying information and use it and other personal identifying information only in a manner consistent with our privacy statement.

Tektronix may share personal information with third parties for marketing purposes related to our products and services, with your prior permission or where otherwise permitted by applicable law. To opt out of having your information shared in this way, please contact us at privacy@tektronix.com.

AGGREGATED STATISTICS

Notwithstanding anything to the contrary in this Agreement, as between Tektronix and You, all right, title, and interest in Aggregated Statistics, and all intellectual property rights therein, belong to and are retained solely by Tektronix. You agree that Tektronix may (1) make Aggregated Statistics publicly available in compliance with applicable law, and (2) use Aggregated Statistics with Your permission or to the extent and in the manner permitted under applicable law; provided that such Aggregated Statistics do not identify You or Your confidential information.

RESERVATION OF RIGHTS

Tektronix reserves all rights not expressly granted to You in this Agreement. Except for the limited rights and licenses expressly granted under this Agreement, nothing in this Agreement grants, by implication, waiver, estoppel, or otherwise, to You or any third party any intellectual property rights or other right, title, or interest in or to the Tektronix IP. Tektronix reserves the right to modify, suspend, or discontinue Tek Scope (in whole or in part) at any time. Tektronix will not be liable to You or to any third party for any modification, suspension, or discontinuation of Tek Scope or any part thereof.

NO DATA BACKUP

Tektronix is not obligated to backup Your Data, and Your Data may be deleted at any time without prior notice. You are solely responsible for creating and maintaining Your own backup copies of Your Data. TEKTRONIX HAS NO OBLIGATION OR LIABILITY FOR ANY LOSS, ALTERATION, DESTRUCTION, DAMAGE, CORRUPTION, OR RECOVERY OF YOUR DATA.

SUPPORT

Tektronix will only support the most recent version of Tek Scope.

TERM AND TERMINATION

The license granted herein is effective for the Initial Term, unless terminated earlier pursuant to this Agreement’s express provisions.

Upon expiration of the Initial Term, this Agreement will automatically renew for the subsequent number of months paid for through Tektronix’s online system or a Customer Order (a "Renewal Term") and will be billed at Tektronix’s then-current price. If the Term is renewed for any Renewal Terms pursuant to this Section, Tektronix’s then-current terms and conditions will apply for the Renewal Term. If either Party provides 30 days written notice of its intent to terminate this Agreement, then this Agreement will expire at the end of the current Term.

Termination

IF YOU TRANSFER ANY COPY, MODIFICATION, OR MERGED PORTION OF TEK SCOPE WITHOUT THE EXPRESS PERMISSION OF THESE TERMS AND CONDITIONS OR PRIOR WRITTEN CONSENT OF TEKTRONIX, YOUR LICENSE IS AUTOMATICALLY TERMINATED.

Tektronix may terminate this Agreement if You (1) fail to pay any amount when due under this Agreement and such failure continues more than six days after Tektronix's delivery of written notice thereof, or (2) fail to comply with any term or condition and such failure is not remedied within 30 days after written notice from Tektronix.

Either Party may terminate this Agreement, effective on written notice to the other Party, if the other Party breaches a material provision of this Agreement, and such breach: (A) is incapable of cure; or (B) being capable of cure, remains uncured 30 days after the non-breaching Party provides the breaching Party with written notice of such breach.

Either Party may terminate this Agreement, effective upon written notice to the other Party, if the other Party (1) becomes insolvent or is generally unable to pay, or fails to pay, its debts as they become due; (2) files or has filed against it, a petition for voluntary or involuntary bankruptcy or otherwise becomes subject, voluntarily or involuntarily, to any proceeding under any domestic or foreign bankruptcy or insolvency law; (3) makes or seeks to make a general assignment for the benefit of its creditors; or (4) applies for or has appointed a receiver, trustee, custodian, or similar agent appointed by order of any court of competent jurisdiction to take charge of or sell any material portion of its property or business.

Effect of Expiration or Termination

Upon expiration or termination of this Agreement, You must (1) immediately discontinue use of the Provider IP and Tek Scope; (2) delete, destroy, or return all copies of the Provider IP and; (3) certify in writing to Tektronix that the Provider IP and Tek Scope has been deleted, destroyed, or returned.

INDEMNIFICATION

Indemnification by Tektronix

Tektronix will indemnify, defend, and hold harmless Customer from and against any and all Losses incurred by Customer resulting from any allegation, claim, suit, or proceeding by a third party that Tek Scope or any use of Tek Scope in accordance with this Agreement, infringes or misappropriates such third party's US intellectual property rights, provided that You promptly notify Tektronix in writing of the claim, cooperates with Tektronix, and allows Tektronix sole authority to control the defense and settlement of such claim.

If such a claim is made or appears possible, You agree that Tektronix may, at Tektronix's sole discretion, (1) modify or replace Tek Scope to make it non-infringing; or (2) obtain the right for You to continue use Tek Scope. If Tektronix determines that neither alternative is reasonable, Tektronix may terminate this Agreement, effective immediately on written notice.

This Section does not apply to the extent that any alleged infringement arises from (1) use of Tek Scope in a manner not in accordance with the Documentation; (2) use of Tek Scope in combination with data (including Your Data), software, hardware, equipment, or technology not provided by Tektronix or authorized by Tektronix in writing; (3) modifications to Tek Scope not made by Tektronix; or (4) use of an allegedly infringing version of Tek Scope if the alleged infringement could be avoided by the use of a different version of Tek Scope made available to You.

Indemnification by You

You will indemnify, hold harmless, and, at Tektronix's option, defend Tektronix and its affiliates from and against any Losses resulting from any Third-Party Claim that the Your Data, or any use of the Your Data in accordance with this Agreement, infringes or misappropriates such third party's intellectual property rights and any Third-Party Claims based on Your or any Authorized User's (1) negligence or willful misconduct; (2) use of Tek Scope in a manner not authorized by this Agreement; (3) use of Tek Scope in combination with data, software, hardware, equipment or technology not provided by Tektronix or authorized by Tektronix in writing; or (4) modifications to Tek Scope not made by Tektronix. You may not settle any Third-Party Claim against Tektronix unless Tektronix consents to such settlement, and Tektronix will have the right, at its option, to defend itself against any such Third-Party Claim or to participate in the defense thereof by counsel of its own choice.

Sole Remedy

THIS SECTION SETS FORTH YOUR SOLE REMEDIES AND TEKTRONIX'S SOLE LIABILITY AND OBLIGATION FOR ANY ACTUAL, THREATENED, OR ALLEGED CLAIMS THAT TEK SCOPE INFRINGES, MISAPPROPRIATES, OR OTHERWISE VIOLATES ANY INTELLECTUAL PROPERTY RIGHTS OF ANY THIRD PARTY. IN NO EVENT WILL TEKTRONIX'S LIABILITY UNDER THIS SECTION EXCEED THE TOTAL FEES PAID BY YOU IN THE CURRENT CONTRACT TERM.

LIMITATION OF LIABILITY

EXCEPT FOR THE INDEMNIFICATION OBLIGATIONS ABOVE, IN NO EVENT WILL TEKTRONIX, ITS AFFILIATES, LICENSORS, OR RESELLERS BE LIABLE UNDER OR IN CONNECTION WITH THIS AGREEMENT UNDER ANY LEGAL OR EQUITABLE THEORY, INCLUDING BREACH OF CONTRACT, TORT (INCLUDING NEGLIGENCE), STRICT LIABILITY, AND OTHERWISE, FOR ANY (1) ECONOMICAL, CONSEQUENTIAL, INCIDENTAL, INDIRECT, EXEMPLARY, SPECIAL, ENHANCED, OR PUNITIVE DAMAGES; (2) INCREASED COSTS, DIMINUTION IN VALUE OR LOST BUSINESS, PRODUCTION, REVENUES, OR PROFITS; (3) LOSS OF GOODWILL OR REPUTATION; (4) LOSS OF OR DAMAGE TO YOUR DATA OR PROGRAMMING, LOSS OF PROFITS, BUSINESS INTERRUPTION, OR OTHER PECUNIARY LOSS ARISING FROM THE USE OF (OR INABILITY TO USE) TEK SCOPE, (5) PENALTIES OR PENALTY CLAUSES OF ANY DESCRIPTION, (6) ANY DAMAGE, CLAIMS, OR LOSSES RESULTING FROM THE USE OF TEK SCOPE IN CONJUNCTION WITH OTHER PRODUCTS OR SERVICES (INCLUDING THIRD-PARTY PRODUCTS OR SERVICES); OR (7) COST OF REPLACEMENT GOODS OR SERVICES, IN EACH CASE REGARDLESS OF WHETHER TEKTRONIX WAS ADVISED OF THE POSSIBILITY OF SUCH LOSSES OR DAMAGES OR SUCH LOSSES OR DAMAGES WERE OTHERWISE FORESEEABLE. IN NO EVENT WILL TEKTRONIX'S AGGREGATE LIABILITY ARISING OUT OF OR RELATED TO THIS AGREEMENT UNDER ANY LEGAL OR EQUITABLE THEORY, INCLUDING BREACH OF CONTRACT, TORT (INCLUDING NEGLIGENCE), STRICT LIABILITY, AND OTHERWISE EXCEED THE TOTAL FEES PAID BY YOU IN THE CURRENT CONTRACT YEAR.

EXPORT REGULATION

Tek Scope is subject to Export Laws. You agree to use Tek Scope only in compliance with the Export Laws and will not use Tek Scope in a manner that causes You or Tektronix to violate applicable Export Laws, including without limitation, transfer to, or use of Tek Scope in, any prohibited destination, end-user, or end-use without prior government authorization. You acknowledge that Tek Scope contains encryption functionality, which may be subject to import or use restrictions, and represents that You will not use Tek Scope in a manner that violates any applicable restrictions related to the use of cryptography.

You further warrant that You will not use Tek Scope to store information subject to export control under the International Traffic in Arms Regulations, 22 CFR Parts 120-130.

You agree to indemnify and hold harmless Tektronix and its affiliates against any claim, demand, action, proceeding, judgment, penalty, fine, loss, liability, cost or expense suffered or incurred by Tektronix or its affiliates and arising out of or relating to any violation by You of any Export Control Law, or of any agreement, warranty or representation made in this clause/section, in connection with Tek Scope, including, without limitation, attorney's fees and any and all costs associated with the removal of any information subject to ITAR control from information technology systems owned, operated, leased, or otherwise employed by Tektronix or its affiliates to provide Tek Scope.

Open Source GPL License Notice

For programs licensed under the "GNU General Public License (GPL) or Lesser GNU General Public License (LGPL)" the complete corresponding sources are available. You can order a CD containing the sources from us for a period of three years after download of the software, by sending a written request to:

Chief Intellectual Property Counsel, Tektronix, Inc.

MS 50/LAW

14150 SW Karl Braun Dr.

Beaverton OR, 97077

This offer is valid to anyone in receipt of this information.

Your request should include: (i) the name of the product, (ii) your (company) name, and (iii) your return mailing and email address (if available).

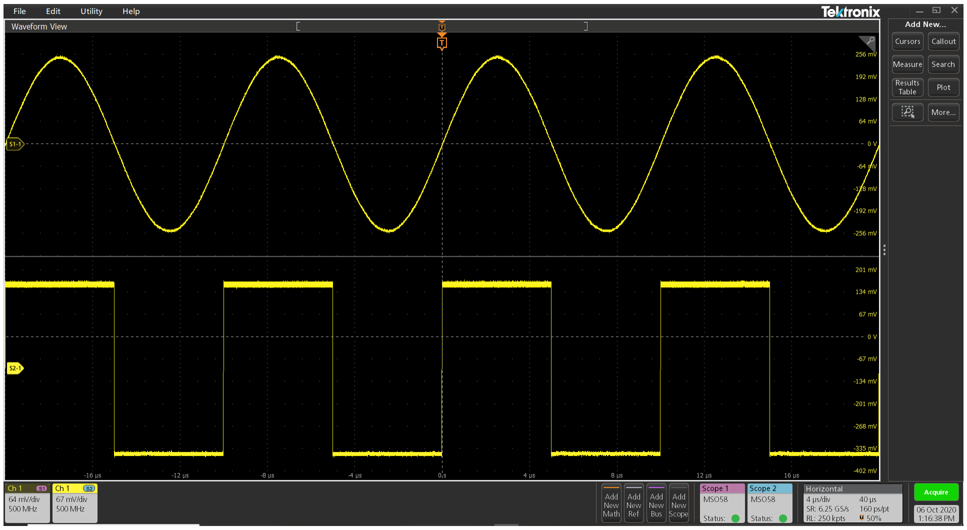
Please note that we may charge you a fee to cover the cost of performing this distribution.

Welcome to the TekScope Software help

This help supports the TekScope Software. Use the tabs on the left to go to specific topics or search for a topic of interest.

The TekScope application supports many features found in our 4/5/6 Series MSO instruments. To learn how to use the user interface most effectively, click *www.tek.com* to refer the 4/5/6 MSO Series User Manual.

This TekScope help document will clarify where on-scope functionality differs from your experience on the PC, as some features available in the 4/5/6 Series MSO instruments are not supported in this application.



Key features and benefits

- Perform your measurement analysis when and where you want to. Tell your data story rather than rely only on screenshots.
- Scalable features based on your needs, including Serial Bus Decode, Power Measurements, and Jitter Analysis.
- Multi-vendor support including simulation data, other oscilloscope vendor file formats, and common waveform file types ensure you can capture data and compare easily.
- Intuitive user interface optimized for touch screen use.
- connect to the remote scopes, acquire waveforms on the instrument, transfer waveforms from the instrument to the tekscope, analyze on tekscope and disconnect the remote instrument.
- Save session files from instrument and open in TekScope for offline analysis.
- Import one or more waveform or acquisitions for analysis (Oscilloscope reference waveform, MATLAB output, simulation output, etc.) for online or offline analysis.
- Emulate a supported instruments to create a setup file on your PC that can then be loaded on the desired scope and have it successfully acquired data with no errors.

Product documents and support

Related documents

Use the related documents for more information on instrument functions, how to remotely program or operate the instrument, understand theory of operation, replace suspected modules, and do other tasks.

MSO44, MSO46 documents

To learn about	Use this document
How to use instrument functions	*4 Series MSO (MSO44, MSO46) Installation and Safety Manual* (This document, Tektronix part number 071-3644-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. A Russian language version is available to download from the Tektronix web site (Tektronix part number 077-1511-xx) *4/5/6 Series MSO Help* (Tektronix part number 077-1303-xx; Printable version of the instrument Help that contains context-sensitive descriptions of all instrument functions; available at www.tek.com/downloads)
How to remotely control the instrument	*4/5/6 Series MSO Programmer Manual* (Tektronix part number 077-1305-xx; available at www.tek.com/downloads)
Instrument specifications and procedures to verify the instrument meets specifications	*4 Series MSO MSO44, MSO46 Specifications and Performance Verification Technical Reference* (Tektronix part number 077-1546-xx; available at www.tek.com/downloads)
Instrument theory of operation, troubleshooting, disassembly, and replaceable parts	*4 Series MSO MSO44, MSO46 Service Manual* (Tektronix part number 077-1547-xx; available at www.tek.com/downloads)
Installing the instrument in a rack	*RM4 Rack Mount Kit Instructions* (Tektronix part number 071-3645-xx; available at www.tek.com/downloads)
Using the TLP058 Logic Probe	*TLP058 FlexChannel™ Logic Probe Instructions* (Tektronix part number 071-3515-xx; available at www.tek.com/downloads)

MSO54, MSO56, MSO58 documents

To learn about	Use this document
How to use instrument functions	*5 Series MSO (MSO54, MSO56, MSO58) Installation and Safety Manual* (This document, Tektronix part number 071-3514-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. A Russian language version is available to download from the Tektronix web site (Tektronix part number 077-1361-xx) *5 Series and 6 Series MSO Help* (Tektronix part number 077-1303-xx; Printable version of the instrument Help that contains context-sensitive descriptions of all instrument functions; available at www.tek.com/downloads)

Table continued…

To learn about	Use this document
How to remotely control the instrument	*5 Series and 6 Series MSO Programmer Manual* (Tektronix part number 077-1305-xx; available at www.tek.com/downloads)
Instrument specifications and procedures to verify the instrument meets specifications	*5 Series MSO MSO54, MSO56, MSO58, MSO58LP Specifications and Performance Verification Technical Reference* (Tektronix part number 077-1306-xx; available at www.tek.com/downloads)
Instrument theory of operation, troubleshooting, disassembly, and replaceable parts	*5 Series MSO MSO54, MSO56, MSO58 Service Manual* (Tektronix part number 077-1307-xx; available at www.tek.com/downloads)
Installing the instrument in a rack	*RM5 Rack Mount Kit Instructions* (Tektronix part number 071-3523-xx; available at www.tek.com/downloads)
Using the TLP058 Logic Probe	*TLP058 FlexChannel® Logic Probe Instructions* (Tektronix part number 071-3515-xx; available at www.tek.com/downloads)

MSO58LP documents

To learn about	Use this document
How to use instrument functions	*5 Series MSO MSO58LP Installation and Safety Manual* (this document, Tektronix part number 071-3568-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. A Russian language version is available to download from the Tektronix web site (Tektronix part number 077-1404-xx) *5 Series and 6 Series MSO Help* (Tektronix part number 077-1303-xx; Printable version of the instrument Help; available at www.tektronix.com/downloads)
How to remotely control the instrument	*5 Series and 6 Series MSO Programmer Manual* (Tektronix part number 077-1305-xx; available at www.tek.com/downloads)
Instrument specifications and procedures to verify the instrument meets specifications	*5 Series MSO MSO54, MSO56, MSO58, MSO58LP Specifications and Performance Verification Technical Reference* (Tektronix part number 077-1306-xx; available at www.tek.com/downloads)
Using the TLP058 Logic Probe	*TLP058 FlexChannel® Logic Probe Instructions* (Tektronix part number 071-3515-xx; available at www.tek.com/downloads)
Converting the instrument for benchtop use	*MSO58LP Bench Conversion Kit Instructions* (Tektronix part number 075-1102-xx; available at www.tek.com/downloads)

6 Series MSO documents

To learn about	Use this document
How to use instrument functions	*5 Series and 6 Series MSO Help* (Tektronix part number 077-1303-xx; Printable version of the instrument Help; available at www.tek.com/downloads) *6 Series B MSO Installation and Safety Manual* (this document, Tektronix part number 071-3579-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. A Russian language version is available to download from the Tektronix web site (Tektronix part number 077-1432-xx)
How to remotely control the instrument	*5 Series and 6 Series MSO Programmer Manual* (Tektronix part number 077-1305-xx; available at www.tek.com/downloads)
Instrument specifications and procedures to verify the instrument meets specifications	*6 Series B MSO Specifications and Performance Verification Technical Reference* (Tektronix part number 077-1461-xx; available at www.tek.com/downloads)
Instrument theory of operation, troubleshooting, disassembly, and replaceable parts	*6 Series MSO Service Manual* (Tektronix part number 077-1462-xx; available at www.tek.com/downloads)
Installing the instrument in a rack	*RM5 Rack Mount Kit Instructions* (Tektronix part number 071-3523-xx; available at www.tek.com/downloads)
Using the TLP058 Logic Probe	*TLP058 FlexChannel® Logic Probe Instructions* (Tektronix part number 071-3515-xx; available at www.tek.com/downloads)

6 Series B MSO documents

To learn about	Use this document
How to use instrument functions	*5 Series and 6 Series MSO Help* (Tektronix part number 077-1303-xx; Printable version of the instrument Help; available at www.tek.com/downloads) *6 Series B MSO Installation and Safety Manual* (this document, Tektronix part number 071-3579-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. A Russian language version is available to download from the Tektronix web site (Tektronix part number 077-1432-xx)
How to remotely control the instrument	*5 Series and 6 Series MSO Programmer Manual* (Tektronix part number 077-1305-xx; available at www.tek.com/downloads)
Instrument specifications and procedures to verify the instrument meets specifications	*6 Series B MSO Specifications and Performance Verification Technical Reference* (Tektronix part number 077-1461-xx; available at www.tek.com/downloads)
Instrument theory of operation, troubleshooting, disassembly, and replaceable parts	*6 Series B MSO Service Manual* (Tektronix part number 077-1462-xx; available at www.tek.com/downloads)
Installing the instrument in a rack	*RM5 Rack Mount Kit Instructions* (Tektronix part number 071-3523-xx; available at www.tek.com/downloads)
Using the TLP058 Logic Probe	*TLP058 FlexChannel® Logic Probe Instructions* (Tektronix part number 071-3515-xx; available at www.tek.com/downloads)

LPD64 documents

To learn about	Use this document
How to use instrument functions	*6 Series Low Profile Digitizer LPD64 Installation and Safety Manual* (this document, Tektronix part number 071-3569-xx); standard accessory with the instrument. Single document with English, French, and German languages. *4/5/6 Series MSO Help* (Tektronix part number 077-1303-xx; Printable version of the instrument Help; available at www.tektronix.com/downloads)
How to remotely control the instrument	*4/5/6 Series MSO Programmer Manual* (Tektronix part number 077-1305-xx; available at www.tek.com/downloads)
Instrument specifications and procedures to verify the instrument meets specifications	*6 Series Low Profile Digitizer LPD64 Specifications and Performance Verification Technical Reference* (Tektronix part number 077-1568-xx; available at www.tek.com/downloads)
Converting the instrument for benchtop use	*MSO58LP/LPD64 Bench Conversion Kit Instructions* (Tektronix part number 075-1102-xx; available at www.tek.com/downloads)

MSO/DPO70000, DPO7000, and MSO/DPO5000 Series

- **Digital Phosphor Oscilloscopes User Manual (071-2980-XX)**: The Quick Start User Manual has information about installingand operating your instrument. The Quick Start User Manual is available in several languages, in addition to English.
- **DigitalPhosphor Oscilloscopes Programmer Online Guide**: The Programmer Guide is provided as online help and as a printable PDF file. See the product software DVD for installation information.
- **Digital Phosphor Oscilloscopes Specifications and Performance Verification Technical Reference Manual (077-0063-XX)**: This is a PDF only manual that includes both the specifications and the performance verification procedure.
- **DigitalPhosphor Oscilloscopes Service Manual (077-0076-XX)**: A service manual is available as a PDF file; it includes procedures to service the instrument to the module level.

TBS1000C Series Oscilloscopes

TBS1000C Series Oscilloscopes User Manual (077-1571-XX)

The Quick Start User Manual has information about installing and operating your instrument. The Quick Start User Manual is available in several languages, in addition to English.

TBS2000B Series Oscilloscopes

TBS2000B Series Oscilloscopes User Manual (077-1525-XX)

The Quick Start User Manual has information about installing and operating your instrument. The Quick Start User Manual is available in several languages, in addition to English

Product support and feedback

Tektronix values your feedback on our products. To help us serve you better, please send us your suggestions, ideas, or comments on your instrument, application, or product documentation.

Contact through mail, telephone, or the Web site. See *Contacting Tektronix* for more information or assistance with your product.

When you contact Tektronix Technical Support, please include the following information (be as specific as possible):

General information

- All instrument model numbers
- Hardware options, if any
- Probes used
- Your name, company, mailing address, phone number, FAX number
- Please indicate if you would like to be contacted by Tektronix about your suggestion or comments.

Application specific information

- Software version number
- Description of the problem such that technical support can duplicate the problem
- If possible, save and send the setup files for all the instruments used and the application
- If possible, save and send status messages text files
- If possible, save and send the waveform on which you are performing the measurement as a .wfm file

Getting started

Minimum requirements

The following table shows the minimum system requirements to install and run the TekScope application.

Table 1: System requirements

Component	Description
Software Version	TekScope Software Version v1.30.x
Operating system	• Windows 10 • Browser: Firefox or Chrome • Processor: Intel® Core™ i5 or AMD Athlon® X4 processor (2GHz or faster) • Memory: 8 GB or higher of RAM recommended • Disk Space: 5 GB of available disk space, 10 GB or higher recommended (exact space is dependent on the number of waveforms and their size) • Display Size: 1920x1080 or greater at 100% scaling recommended • Video Memory: OpenGL® 2.0, 32-bit color, and 1 GB of VRAM **Note:** For the best experience when using advanced features such as Serial Bus Protocol Decode, Power Measurements, Jitter Analysis, or Multi-Scope, or when analyzing large numbers of waveforms and long records, we recommend more capable PC systems with additional RAM, disk space, and processor capability.
Supported Oscilloscopes	• 4 Series MSO (MSO44, MSO46) • 5 Series MSO (MSO54, MSO56, MSO58) • 5 Series Low Profile Digitizer MSO58LPD • 6 Series MSO • 6 Series B MSO • 6 Series Low Profile Digitizer LPD64 • MSO/DPO70000, DPO7000, and MSO/DPO5000 Series • TPS/TDS/TBS 1000 and 2000 Series

Install TekScope application

Prerequisite:

See *Minimum requirements* on page 35 for compatibility.

Complete the following steps to download and install the TekScope application.

1. Go to *tekcloud.com*.

2. Click **Sign up** to create an account for new user OR Click **Sign In** for existing user.
3. After login, select the subcription of your choice.
4. Go to *scope.tekcloud.com/#/help/install*.
5. Click **Install TekScope** to download the ".exe" .
6. Double click on .exe file to install the TekScope.
7. Click **Start > Applications** > **Tektronix** > **TekScope** on your PC to launch the application.

Update Tekscope License

Complete the following steps to add or update the license in the TekScope.

1. Launch TekScope application from your PC, select **Help** > **About** from menu and take a note of the **Host id**.
2. Go to *scope.tekcloud.com/#/help/license* page in browser and enter the host id in the **Host id** field to download the license and click **Get license**.
3. Select **Help** > **About** > **Install License** in the TekScope menu to add the license file.
4. Click browse and navigate to the location and select the .lic file and click **Open**.
5. Click **Help**> **About** > **Installed Licenses** to view the added licenses.

Uninstall TekScope license

Use this process to uninstall a license.

⚠ **Note:** If you uninstall option license, you cannot reinstall the license using the same file used to install it. If you need to reinstall an uninstalled option license, contact Tektronix Customer Support to obtain a new option license file.

Use the below procedure to uninstall option license.

1. Select **Help > About** from TekScope.
2. Click option license in the list that you want to uninstall.
3. Click **Remove License** button.

   The TekScope opens the **Location to Save the Exit Key** menu.
4. Click **Browse** and navigate to the location to save the license key uninstall file.
5. Click **Ok**. The TekScope saves the license key file to the specified location and removes the license from the Installed Options list.

Getting acquainted with the TekScope software

The user interface screen

The touch screen user interface contains waveforms and plots, measurement readouts, and touch-based controls to access all oscilloscope functions.

1. The **Menu bar** provides menus for typical operations including:
   - Saving, loading, and accessing files
   - Undoing or redoing an action
   - Setting oscilloscope display and measurement preferences
   - Configuring network access
   - Loading option licenses
   - Opening a Help viewer
2. The **Waveform View** area displays analog, math, reference, bus, and trend waveforms. The waveforms include waveform handles (identifiers), individual vertical graticule scale labels, and trigger position and level(s) indicators. You can set the Waveform View to stack each waveform vertically in separate graticules, called 'slices' (the default mode, as shown in the previous image), or overlay all the waveforms on the screen (traditional waveform view). See *The user interface elements* on page 39.

   You can add Histogram, Spectral, Eye, and Measurement Results views (plots) for individual measurements. These plot views are separate view windows that you can move on the screen by dragging their title bar to a new position.
3. The **Results Bar** contains controls for displaying cursors, adding callout, plots, and result tables to the screen, and add measurements to the Results bar. The controls are:
   - The **Cursors** button displays on-screen cursors in the selected view. Touch and drag, or use the Multipurpose knobs, to move the cursors. Double-tap on a cursor, or on the cursor readouts, to open a configuration menu to set cursor types and related functions.
   - The **Measure** button opens a configuration menu to select and add measurements to the Results bar. Each measurement you add has a separate badge. Double-tap a measurement badge to open its configuration menu.
   - The **Results Table** button adds a Measurement or Bus Results table to the screen. The Measurement Results table displays all measurements present in the Results bar. The Bus Results table displays bus decode information for

displayed bus waveforms. Each table is contained within its own view window, which can be moved within the display area.

- To remove a measurement, search or other badge from the Results Bar simply flick it off screen.
- The **Callout** button adds a Callout object to the selected view. Double-tap the Callout text to open a configuration menu to change the type of callout, text and font characteristics. Drag any callout other than bookmark to any location on the oscilloscope screen view. Bookmarks callout can only be added to waveform views and spectrum views.
- The **Search** button lets you to detect and mark a waveform where specified events occur. Tap **Search**, to open a Search configuration menu and set the search conditions for analog channels. You can add any number of searches to the same waveform or to different waveforms. Search badges are added to the **Results Bar**.
- The **Plot** button adds an XY, XYZ, or Eye Diagram plot to the display. These plots are contained within their own window and can be moved within the overall display area.
- The **Measurement** and **Search** badges show measurement and search results, and are displayed in the **Results Bar**. See *Badges* on page 40. See *Add a measurement* on page 74. See *Add a Search* on page 91.
- The **Zoom icon** button at the up right of the **Results Bar** lets you to draw a box on the screen to zoom in on an area of interest, drawing segments for mask testing, or drawing areas to define visual trigger conditions.
- The **More...** button at the up right of the **Results Bar** allows you to select Zoom, Visual trigger, or Mask.

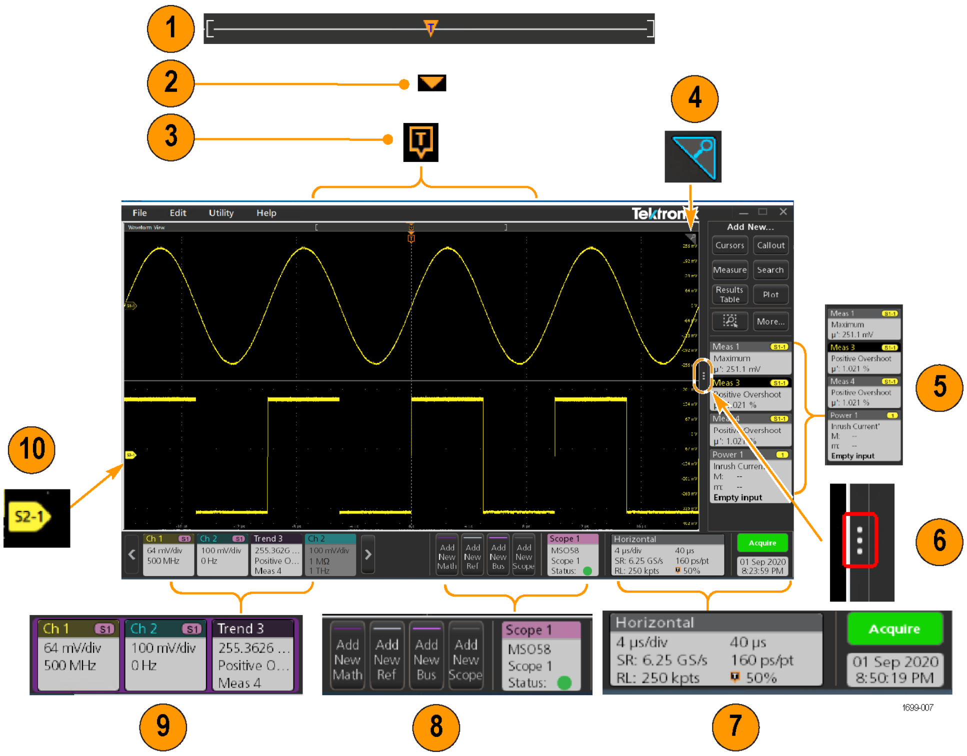
4. The **Settings Bar** contains System badges for setting Horizontal, Trigger, Acquisition, and Date/Time parameters; **Inactive Channel** buttons to turn on channels; **Add New Waveform** buttons to add math, reference, and bus waveforms to the display; **Add New Scope** buttons to add new osciloscope remotely; and Channel and Waveform badges that let you configure the individual waveform parameters. Tap a channel or waveform button to add it to the screen and display a badge. Double-tap a badge to open its configuration menu. See *Badges* on page 40.

5. To remove a measurement, search or other badge from the Results Bar simply flick it off screen.

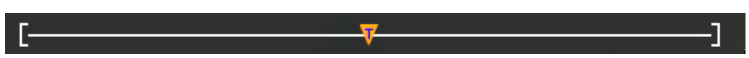
6. **Configuration Menus** let you quickly change the parameters of the selected user interface item. You can open configuration menus by double-tapping on badges, screen objects, or screen areas. See *Remote configuration menus* on page 51.

The user interface elements

Each area of the user interface has a specific function that helps manage information or controls. This topic shows and describes the key user interface elements.



1. The Waveform Record View is a graphical high-level view of the overall waveform record length, how much of the record is on the screen (shown in brackets), the location of key time events including the trigger event, and the current position of waveforms cursors.



If you are displaying a Reference waveform that is shorter than the current acquisition record length, or you are changing the horizontal time scale while the oscilloscope acquisition is stopped, the brackets change position to show the part of the waveform record that is being viewed relative to the current acquisition total record length.

If cursors are active on a waveform, the Waveform Record View shows the relative cursor positions as small vertical dashed lines.

When in Zoom mode, the Waveform Record View is replaced with the Zoom Overview. See *The Zoom user interface elements*.

2. The Expansion Point icon on the waveform view shows the center point around which the waveform expands and compresses when changing horizontal settings.

3. The Trigger Position Indicator shows where the trigger event occurred in the waveform record. The trigger icon is displayed in the waveform slice that is the trigger source.

4. The Zoom icon (in upper right corner of Waveform and Plot views) toggles zoom on and off.

5. Measurement and Search badges show measurement and search results. See *Badges* on page 40. See *Add a measurement* on page 74.
6. The Results Bar Handle opens or closes the **Results bar**, to maximize waveform screen viewing when needed. To reopen the **Results bar**, either tap the handle icon or swipe left from the right side of the display.
7. The System badges show global instrument settings (**Horizontal**, Acquire/Cancel status, and Date/Time). See *Badges* on page 40.
8. The Inactive Channel buttons add channel waveforms to the Waveform view and add an associated Channel badge to the Settings bar.

   The **Add New Math**, **Add New Ref**, and **Add New Bus** buttons add the corresponding signal to the Waveform view, and add an associated Waveform badge to the **Settings** bar. You can add any number of Math, Reference, and Bus waveforms, limited only by system memory.

   The **Add New Scope** button lets you to add new oscilloscope remotely and add an associated Scope badge to the **Settings** bar.
9. Double-tap a badge to open its associated configuration menu. See *Badges* on page 40. See *Remote configuration menus* on page 51.

   If you add more oscilloscope or Waveform badges than can fit in the waveform badge display area, tap the scroll buttons at each end of the waveform badge area to scroll and display hidden badges.
10. The Waveform Handles on each waveform identify the source of that waveform (C*x* for channels, S*x*-*x* for remote channels, M*x* for Math waveforms, R*x* for Reference waveforms, B*x* for bus waveforms). The waveform handles are at the zero-volt level of the waveform by default. The currently selected waveform handle is a solid color; unselected waveform handles are outlined.

    For remotely connected instrument, each waveform identify the source of that waveform as S*x*-*x* (Scope x-Channel) for instrument and channels.

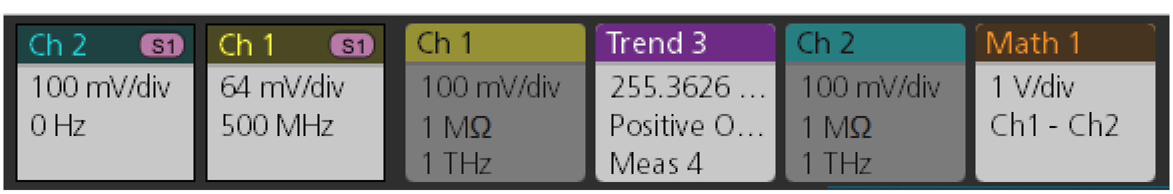
    Double-tapping a waveform handle opens the configuration menu for that waveform.

Badges

Badges are rectangular icons that show waveform, measurement, and instrument settings or readouts. Badges also provide fast access to configuration menus. The badge types are Scope, Channel, Waveform, Measurement, Search, and System.

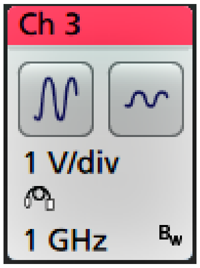
Scope, Channel, and Waveform badges

Scope, Channel, and Waveform (**Scope X**, **Math**, **Ref**, **Bus**) badges are shown in the **Settings** Bar, located along the bottom left of the screen. Each waveform has its own badge. The badges show high-level settings for each displayed channel or waveform. Double-tap a badge to open its configuration menu.



Channel and Waveform badges

Most Channel and Waveform badges also have Scale buttons, shown by single-tapping the badge. Use the Scale buttons to increase or decrease the vertical scale setting for that waveform.



You can drag Channel and Waveform badges to change their position in the **Settings** bar and open the badge right-click menu to access a quick-action menu.

There are two ways to delete Channel and Waveform badges.

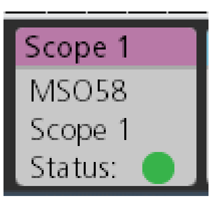
- Right-click the badge and turn it off.
- Flick the badge off the bottom edge of the display to remove it from the **Settings** bar. Flicking upwards from the bottom edge of the **Settings** bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

Channel badges are listed in the channel order unless you have moved them. Channel badges may also display short error or warning messages. For more information double-tap the badge to open its configuration menu, or search the instrument Help.

Waveform badges (**Math**, **Ref**, **Bus**) are listed in the order created (unless they have been moved), and are grouped together by type. Deleting a Waveform badge does not change the order or names of the remaining badges.

Remote Scope badges

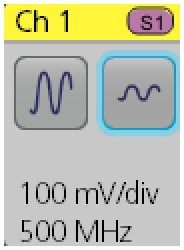
Scope (**Scope X**) badge is shown in the **Settings** Bar, located along the bottom left of the screen. Each connected Scope has its own badge. Double-tap a badge to open its configuration menu.



Badge readout	Description
Scope 1	A label defined in the badge to identify connected oscilloscope. Default is Scope X.
Product type	The Oscilloscope label display which scope is connected. Example: MSO58, MSO64.
Label	A label to oscilloscope for user reference.
Status	The status of the Oscilloscope. Either Online (green) or Offline (red) is displayed.

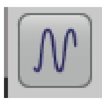
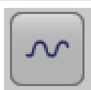
Remote channel badge

Most Channel and Waveform badges also have Scale buttons, shown by single-tapping the badge. Use the Scale buttons to increase or decrease the vertical scale setting for that waveform.

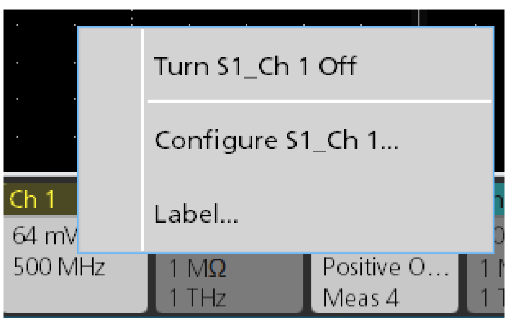


Each remote channel badge shows the instrument label and its channel number in the top right corner of the badge.

Remote channel bage shows the following details.

Badge readout	Description
	V/div is to increase or decrese the source (V/div) size. Increased V/div value gets displayed below the icon.
	V/div is to decrese the source (V/div) size. Decreased V/div value gets displayed below the icon.
Bandwidth	Set the bandwidth for the channel waveform.

Right click menu on channel bage to access a quick-action for the following menu.



Badge readout	Description
Turn ch to On/Off	Tap to turn On or Off the channel depending on the state.
Configure ch	Tap to configure instrument channel. This opens the vertical setting panel.
Label	Enter label text for the channel using virtual keypad or keypad connected to instrument.

You can drag Scope, Channel, and Waveform badges to change their position in the **Settings** bar and open the badge right-click menu to access a quick-action menu.

- Configure Scope: Opens tunneling browser e*Scope connection to scope.
- Connect: Connect a oscilloscope remotely if your not connected.
- Delete: Click to delete the connected oscilloscope.

Scope channel badges are listed in the channel order unless you have moved them. Channel badges may also display short error or warning messages. For more information double-tap the badge to open its configuration menu, or search the instrument Help.

Waveform badges (**Math**, **Ref**, **Bus**) are listed in the order created (unless they have been moved), and are grouped together by type. Deleting a Waveform badge does not change the order or names of the remaining badges.

Measurement badges

Measurement badges are located in the **Results** bar. They show measurements or search results. The badge title also shows the measurement source or sources. To add a Measurement badge, tap the **Add New Measurement** button and select a measurement.

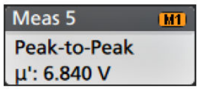


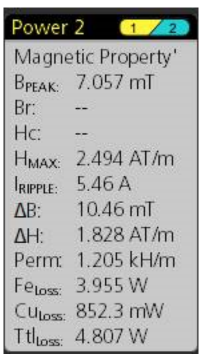
Figure 1: Measurement badges

Figure 2: Remote measurement badges

Each remote measurement badge shows the instrument label and its channel number in the top right corner of the badge.

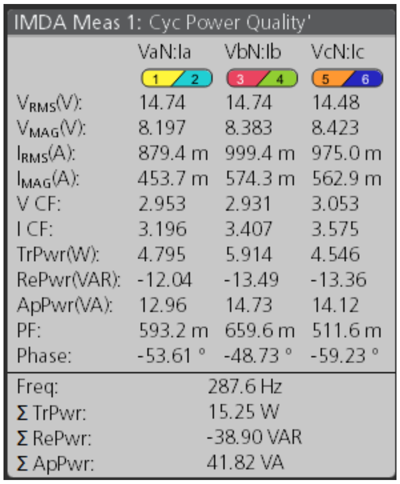
Double-tap a Measurement badge to open its configuration menu to change or refine settings. The default measurement badge readout shows the measurement's mean (µ) value.

Some measurements and their badges are only available as options. For example, Power measurements are only listed in the Add New Measurement menu if the required power option is installed.

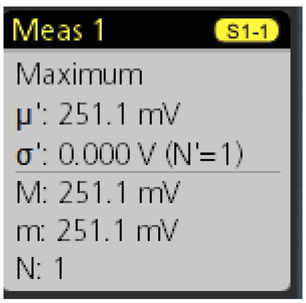


Wide Badge: Wide badge displays all the phases results in a separate column. All sub-measurements are listed in the results badge in the first column. The common result such as Frequency is applicable to all the (3) phases and displayed as single value. The configured sources for each phase are displayed in channel colors.

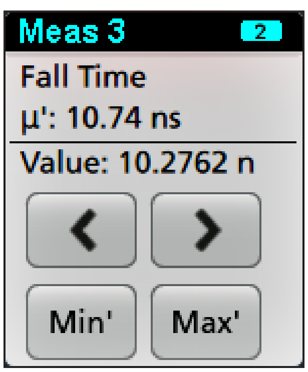
The Wide Badge applies to IMDA measurements only.



To add statistical readouts to individual measurement badges, double-tap a measurement badge to open its configuration menu and select **Show Statistics in Badge**. The measurement badge displays the standard deviation (σ) value. The standard deviation is zero, when the popultion is one.



Some Measurement badges also have Navigation buttons, shown by single-tapping the badge.

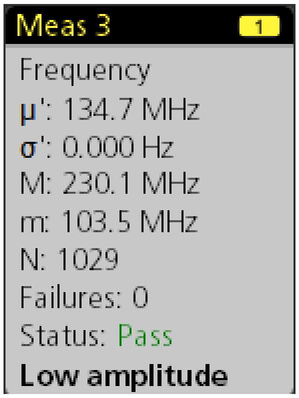


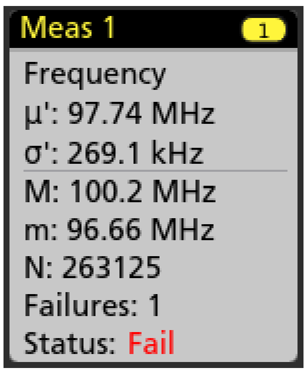
The **<** (Previous) and **>** (Next) buttons center the waveform in the display at the position of the previous or next measurement point in the record (for measurements that take more than one measurement per acquisition).

The **Min'** and **Max'** navigation buttons center the waveform in the display at the minimum or maximum value for that measurement in the current acquisition.

The prime symbol (') shown on measurement readings and Min/Max buttons indicates that the value shown (or moved to in the case of **Min**/**Max** buttons and waveforms) is from the current acquisition. Lack of a prime symbol means the value is from all acquisitions.

The Measurement badge displays **Status** and **Failures** information when pass/fail testing is enabled through the configuration menu. The Status line shows **Pass** (green) or **Fail** (red) according to the conditions defined in the **Pass/Fail Testing** panel. The number of Failures are displayed when statistics are shown in the badge. The Pass/Fail status, number of Failures, and the Limit(s) set in the Pass/Fail Testing panel are available in the Measurement Results table.





Measurement badges are listed in the order created, starting at the top of the Results bar. Deleting a Measurement badge does not change the order or names of the remaining badges.

You can drag Measurement badges to change their position in the **Results** bar and open the badge right-click menu to access a quick-action menu.

There are two ways to delete Scope, Channel, and Waveform badges.

- Right-click the badge and turn it off.
- Flick the badge off the right edge of the display to remove it from the **Results** bar. Flicking left from the right edge of the **Results** bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

Mask Test Badge

The mask test results and measurement statistics are displayed in the **Mask Test** badge in the Results bar. The badge is created when the first segment of a mask is defined.

Badge readout	Description
Label	A label defined in the badge configuration menu.
Wfms	The total number of waveforms tested against the mask.
Failed	The number of waveforms that contained one or more samples that violated the mask.
Hits (optional readout)	A row is created for each segment that makes up the mask. The number displayed is the number of times that segment has been hit.
Total	The total number of hits on all segments.
Status	The status of the mask test. Either Pass (green) or Fail (red) is displayed.

Double-tap a Mask Test badge to open its configuration menu to change or refine settings.

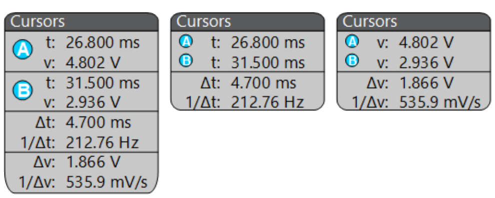
You can drag the badge to change its position in the **Results** bar and open the badge right-click menu to access a quick-action menu.

There are two ways to delete Channel and Waveform badges.

- Right-click the badge and turn it off.
- Flick the badge off the right edge of the display to remove it from the **Results** bar. Flicking left from the right edge of the **Results** bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

Cursor Badges

You can display the cursor readouts in a **Cursors** badge in the Results bar. The badge contents depend on the cursor in use.



Each remote curser badge shows the instrument label and its channel number in the top right corner of the badge.

To create a cursor readouts badge, turn on **Cursors**, double-tap a cursor readout to open its configuration menu, and set the **Readouts** mode to **Badge**.

Note: You can only view cursor readouts in one location at a time; either on the waveform or in a Cursors badge.

Note: You cannot move cursor readouts to a badge for Spectrum View cursors.

You can drag the badge to change its position in the **Results** bar and open the badge right-click menu to access a quick-action menu.

There are two ways to delete Channel and Waveform badges.

- Right-click the badge and turn it off.
- Flick the badge off the right edge of the display to remove it from the **Results** bar. Flicking left from the right edge of the **Results** bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

Search badges

Search badges are also shown in the Results Bar, below the Measurement badges. A search badge lists the search source, search type, and the number of search event occurrences in the current acquisition. The instrument marks the waveform where those events occur with small down-pointing triangles along the top of the waveform graticule. Double-tap a search badge to open its configuration menu to change or refine search settings.

Figure 3: Search badge

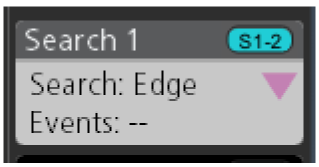
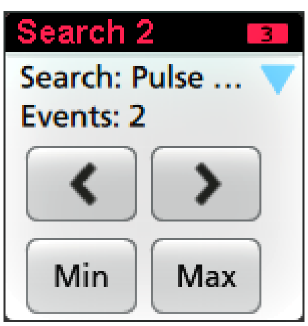


Figure 4: Remote search badge

Each remote search badge shows the instrument label and its channel number in the top right corner of the badge.

Search badges are created by tapping the **Add New... Search** button. Use the displayed configuration menu to set the search criteria.

Search badges have **<** (Previous) and **>** (Next) Navigation buttons that open the Zoom mode and center the waveform in the display at the position of the previous or next search mark in the waveform record. Search badge Navigation buttons are only usable when the oscilloscope is in single acquisition mode. Single-tap a badge to close the Navigation buttons.



Some searches also provide **Min** and **Max** navigation buttons that open the Zoom mode and center the waveform in the display at the minimum or maximum value for that search event in the current acquisition.

Search badges are listed in the order created. Deleting a **Search** badge does not change the order or names of the remaining badges.

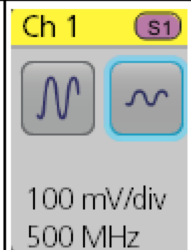
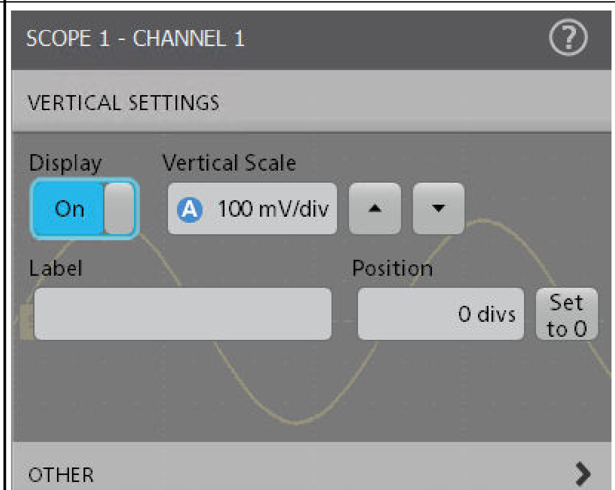
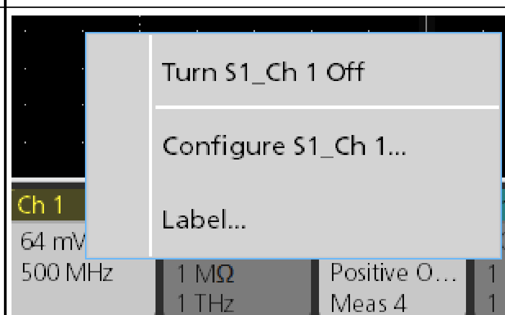
You can drag Search badges to change their position in the **Results** bar and open the badge right-click menu to access a quick-action menu.

There are two ways to delete Channel and Waveform badges.

- Drag a the badge into the Trash Can icon.

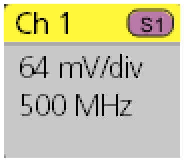
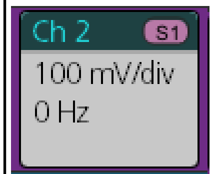
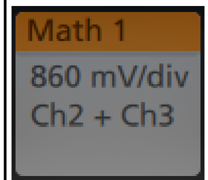
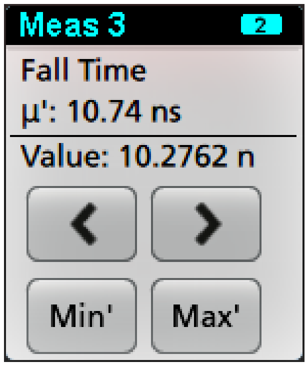
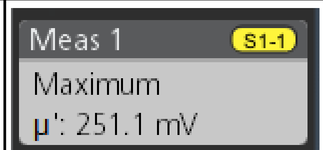
- Right-click the badge and turn it off.
- Flick the badge off the right edge of the display to remove it from the **Results** bar. Flicking left from the right edge of the **Results** bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

Common remote badge actions

Action	Result	Example
Single tap	Immediate access controls (Scale, Navigation).	
Double tap	Configuration menu with access to all settings for the badge.	
Touch and hold	Right-click menu with single tap access to common actions. Typical actions include turning off a channel and deleting a measurement or search badge.	
Flick	Flick the badge off the bottom edge of the display, to remove it from the **Settings** bar. Flick the badge off the right edge of the display, to remove it from the **Results** bar. Flick from the right or bottom edge, to recover a removed badge. This action is only possible within 10 seconds of badge removal.	

Badge selection status

The appearance of a badge indicates its selection status (selected or unselected), or if a measurement needs to be deleted to close a channel or waveform badge.

Badge type	Selected	Unselected	Turned off or in use [1]
Channel or Waveform			
Measurement			N/A

Moving waveform and measurement badges

You can move waveform and measurement badges within their display bars to meet your measurement needs. Simply touch and drag the badge to a new position.

Moving waveform badges in the Settings Bar

- A waveform badge can only be moved within the Settings Bar.
- Dragging a waveform badge to a new location selects that waveform.
- Dragging a badge to a new location causes the non-selected badges to move slightly to create the position at which to insert the badge.
- To move a badge to a position that is off screen of the displayed bar badges (scroll buttons are present), drag the badge that you're moving onto one of the scroll buttons. The badges that are off screen shift onscreen one at a time until you move the badge off of the scroll button and position it in the displayed badges.
- Changing the order of waveform badges or badge groups also changes the order of displayed waveforms in the Waveform view. The order of badges or badge groups (left to right) in the Settings bar determines the order of slices in the display (top to bottom).
- Once you have moved any waveform badge (Channel, Math, Ref, Bus, Trend) on the Settings bar to a new position, adding a new badge adds that badge to the right of the existing badges. This differs from the default badge addition action, which listed badges together by category (Channels, Math, Ref, Bus) and arranged the badges in numerical order within each badge category. To restore the default badge addition method, tap **File** > **Default Setup** and add the waveform badges back onto the Settings bar.
- Changing the order of bus waveform badges changes the order of the tabs in a bus decode Results table.

Moving badges in the measurement Results Bar

- A Measurement or Search badge can only be moved within the Results Bar.
- Dragging a badge to a new location causes the non-selected badges to move to create the position at which to insert the badge.
- Once you have moved any Results badge to a new position, adding a new Measurement or Search badge adds that badge to the bottom of the Results bar badges.

1 A dimmed Channel badge means the screen waveform is turned off (but not deleted). A dimmed Waveform badge means that the waveform display is turned off, or it is being used as a source by a measurement and cannot be deleted until the measurement is deleted.

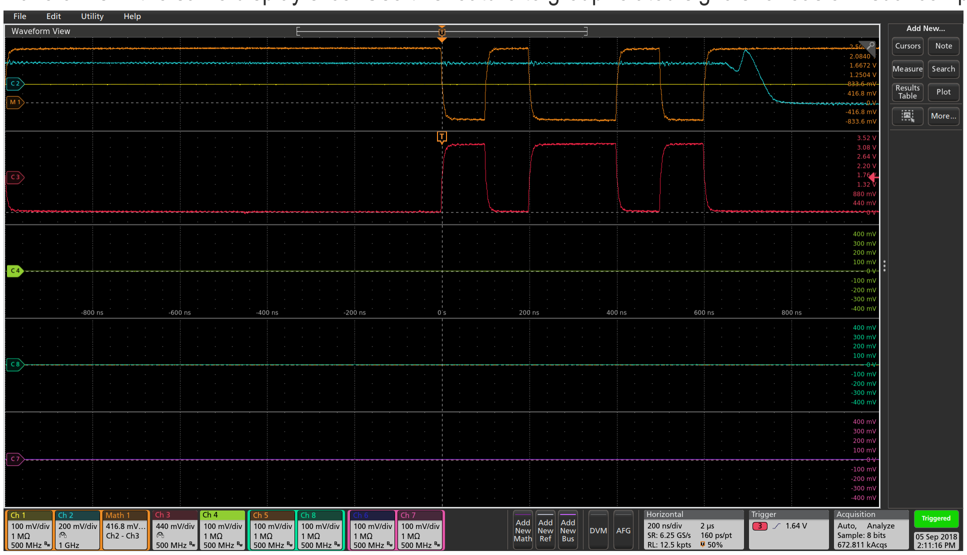
- To move a badge to a position that is off screen of the displayed bar badges (scroll buttons are present), drag the badge that you're moving onto one of the scroll buttons. The badges that are off screen shift onscreen one at a time until you move the badge off of the scroll button and position it in the displayed badges.
- Changing the order of Measurement badges in the Results bar changes the order of measurements shown in a Results table.
- Changing the order of Search badges in the Results Bar changes the order of the tabs shown in a Search Results table.
- Changing the order of harmonics badges in the Results bar changes the order of the tabs shown in a Harmonics Results table.

Badge grouping

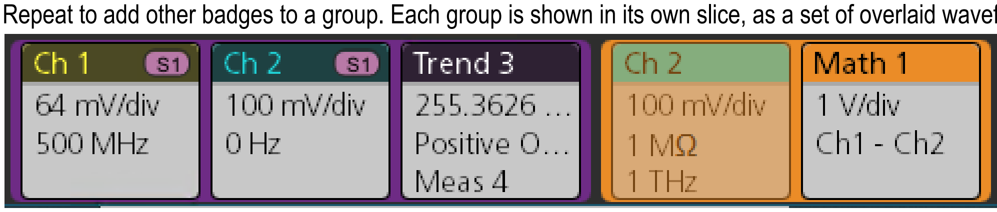
You can group signal badges in the Settings bar to display multiple waveforms in a single slice. See *Group signal badges in the Settings bar* on page 50 for more information.

Group signal badges in the Settings bar

You can group signal badges in any combination of channel, waveform, math, reference and bus badges to display related waveforms in the same display slice. Use this feature to group related signals for easier visual comparisons.



To group one or more signal badges, drag a badge onto another badge until the background badge turns red, then release. Repeat to add other badges to a group. Each group is shown in its own slice, as a set of overlaid waveforms.



Badge group characteristics

- Badges in a group are listed left to right in the order they were added to the group.
- The badge group color is the color of the last selected badge in the group.
- Use standard touch and menu interactions to select and change individual badge settings.
- You can drag and move badge groups the same way as individual badges.
- The order of badges or badge groups (left to right) determines the order of slices in the display (top to bottom).
- To ungroup a single badge, drag it vertically and release. You can also ungroup a single badge by touching and holding a badge in the group to open its right-click menu, and select Ungroup Badge(x) to ungroup the selected badge.

- To ungroup the entire group, touch and hold any badge in the group to open its right-click menu. Select to ungroup the selected badge or ungroup all of the badges.

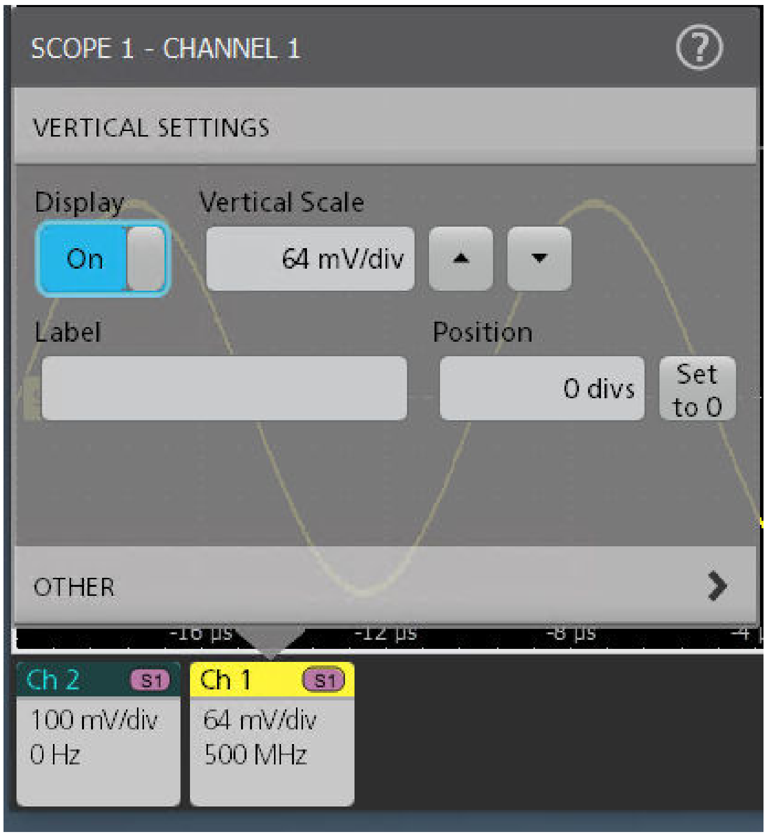
Badge group caveats

- You cannot drag badges within a group to change their order.
- You cannot add one badge group to another.
- You cannot drag a badge group onto an individual badge to group those badges.
- You cannot drag a badge group onto the Trash icon to ungroup or delete those badges.

Remote configuration menus

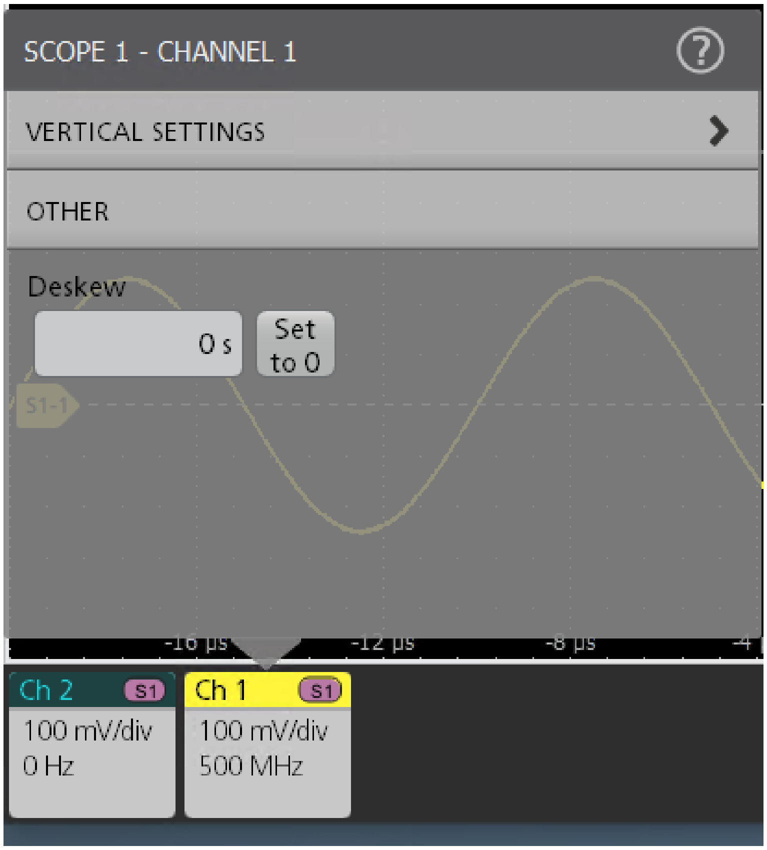
Configuration menus let you quickly set the parameters for channels, system settings (Horizontal, Trigger), measurements, cursor readouts, Waveform and Plot views, callout text, and so on.

Double-tap an item (remote badge, **Waveform View** or **Plot View**, cursor readouts, callout text, and so on) to open its configuration menu. For example, double-tap a remote channel badge in the **Settings Bar** to open its configuration menu.



The effect of entered values or selections, depends on remote control and on pressing Acquire button. Menu contents are dynamic, and can change depending on your selections, instrument options, or attached probes.

Related settings are grouped into 'panels.' Tap the panel name to show those settings. Changes to panel settings can change the values and/or fields shown in that panel and other panels.



Tap anywhere outside a configuration menu to close it.

To open Help content for a configuration menu, tap the question mark icon in the upper right corner of the menu.

Configure channel or waveform settings

Use the channel and waveform configuration menus to set parameters such as vertical scale and offset, coupling, bandwidth, probe settings, deskew values, external attenuation values, and other settings.

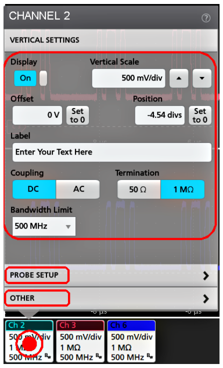
Before you begin

Prerequisite: There is a channel or waveform badge in the Settings bar.

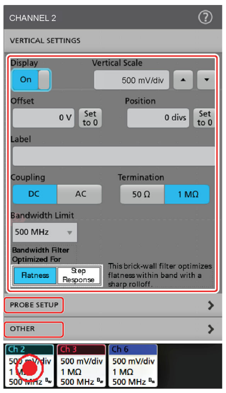
Procedure

1. Double-tap a **Channel** or **Waveform** badge to open a configuration menu for that item.

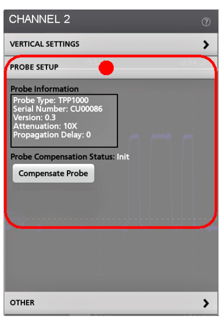
   For example, in a Channel menu, use the **Vertical Settings** panel to set basic probe parameters such as vertical scale and position, offset, coupling, termination, and bandwidth limit.



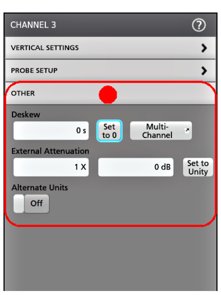
Available settings depend on the probe.



2. Tap the **Probe Setup** panel to confirm probe settings and run configuration or compensation on supported probes.



3. Tap the **Other** panel to set probe deskew, external attenuation, and alternate units parameters.



4. Tap the Help icon on the menu title to open the help topic for more information.
5. Tap outside the menu to close the menu.

The Zoom user interface elements

Use the zoom tools to magnify waveforms to view signal details.

1. The **Zoom Overview** shows the entire waveform record. All waveforms are shown in Overlay mode in the Zoom Overview area.

   Note: Using pinch and expand gestures on the Zoom Overview waveforms changes the horizontal time base settings.

2. The **Zoom Box** shows the area of the Zoom Overview to display in the Zoom View (see 5). You can touch and drag the box to move the area to view. You can also use the zoom **Pan** knob to move the Zoom Box left or right.

   Note: Moving the Zoom Box, or changing its position, does not change the horizontal time base settings.

3. The **Zoom** icon (in the upper right corner of the Waveform View) switches zoom mode on and off.
4. The **Draw-a-Box** button toggles between drawing a zoom box (default mode), drawing areas for the **Visual Trigger** function, and drawing segments for **Mask Testing**. The button is located at the bottom of the **Results Bar**.

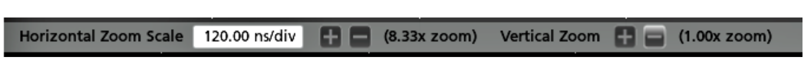
   A zoom box lets you quickly draw a box around an area of interest in the Waveform or Zoom Overview. Drawing a box immediately puts the oscilloscope into zoom mode. To draw a zoom box, tap the Draw-a-Box button (while in Zoom mode), then touch and drag on the waveform to draw a box waveform. You can continue to draw zoom boxes until you single tap anywhere on the screen or open a menu.

   To toggle between **Zoom** mode, **Visual Trigger** mode, and **Mask** mode, double-tap the Draw-a-Box button and select one of the three options. Search for the **Visual Trigger** and **Mask Testing** topics in the oscilloscope embedded Help for more information.

5. The **Zoom View** shows the zoomed waveforms, as marked by the Zoom Box, in the Zoom Waveform Record View. Use pinch and/or drag options in the zoom view to change the zoomed area of interest.

   Note: Pinch, expand, and drag gestures in the Zoom View only change zoom magnification settings and Zoom Box position.

6. Use the **Zoom Title Bar** controls to adjust the vertical and horizontal size of the zoom area. Click or tap the + or - buttons.



Accessing application help

Use the instrument online help to quickly get information about a function or assistance in performing a task.

To access the user interface on a Low Profile instrument, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

Using context-sensitive help

To open help on a particular menu or item, tap the Help button (question mark symbol) in the title bar. The browser opens with content relevant to the menu or item.

Navigating the online help

Select **Help > Help**. The Help Browser is similar in look and feel to PC-based Help tools.

Note: The Help tool does not have a virtual keyboard. Connect a keyboard to your instrument to enter text in the search fields of the Search or Index tabs.

From the help browser, select one of the following tabs:

- **Contents Tab**. Click any entry to display information on the subject.
- **Index Tab**. Double-click an entry to display information on the subject. Or, enter a keyword you are looking for (the list scrolls to that topic). Click Display to open the topic.
- **Search Tab**. Type in the keyword you are looking for; then click List Topics. Every topic that contains the keyword will be displayed. Select a topic, and then click Display to open the topic.
- **Bookmarks Tab**. To save the current topic to a list, click the Bookmarks tab; then right-click and select Add. Double-click to return to a topic at any time.

Other help features

- Zoom on help text. Use the magnifying glass icons to zoom in or out of the help text.

TekScope operating basics

Launch TekScope

To launch the TekScope application, select **Start** >**Tektronix** > **TekScope** in your PC.

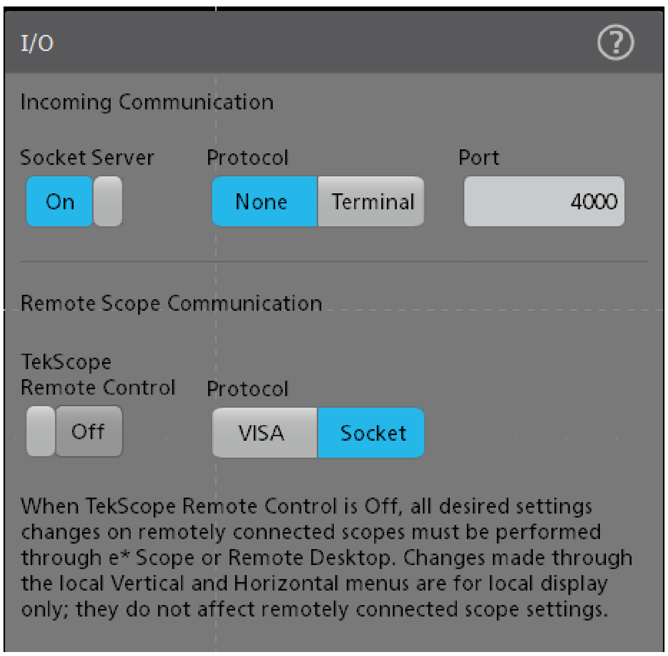
Remote I/O configuration

Pre-requisites

Connecting to a network allows you to remotely access the instrument. To add new instrument remotely, the instrument must be connected to, and accessible from the network to which the PC is connected.

Follow the procedure to set the remote I/O configuration settings.

1. Select **Utility** > **I/O** on the menu bar to open the I/O configuration menu.



2. Set **Socket server** to **On**.
3. Select the **Protocol** to **None** or **Terminal**.
4. Set the **TekScope Remote Control** to **On**.
5. Select the **Protocol** to **VISA** for USB or Network connection or to **Socket** for connecting using Socket communication.

Add new scope

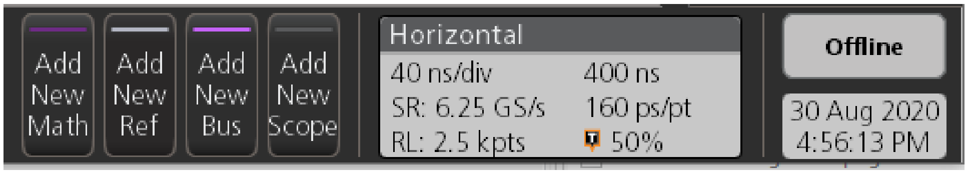
This procedure describes how to remotely connect standard instruments.

Pre-requisites

- To add new Scope, the instrument must be connected to, and accessible from, the network to which the PC is connected.
- Set the Direct remote control settings.
- The IP address of the instrument that you want to access. To determine the instrument's IP address, select **Utility** > **I/O** in the instrument menu bar and see the network settings in the LAN panel.

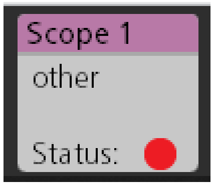
Add new scope remotely

1. Tap on **Add New Scope** on setting bar, to add new oscilloscope remotely.

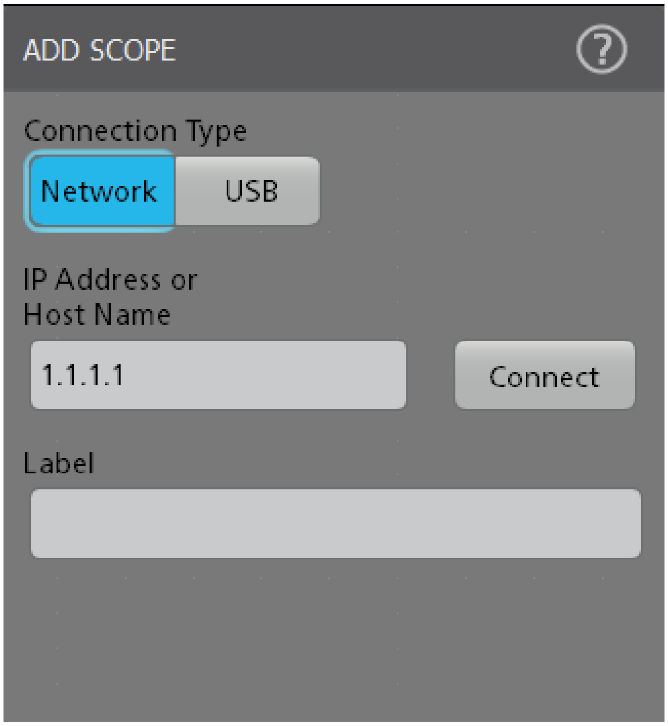


The badge gets added to the right.

2. Double-tap on badge to open its associated configuration menu, Scope badge shows **Add Scope** panel.



3. Select Connection type as **Network**.
4. Enter the **IP Address** or **Host Name**. This input box accepts an IP address (Enter periods as well as numbers) or an instrument host name.



5. Use the **Label** fields to enter labels for the individual instrument.
6. Tap **Connect**.

   If connection is established, “Connected” message is displayed. Status turns green when connected.

   The error message displays if connection fails, verify the access information is correct and try again.

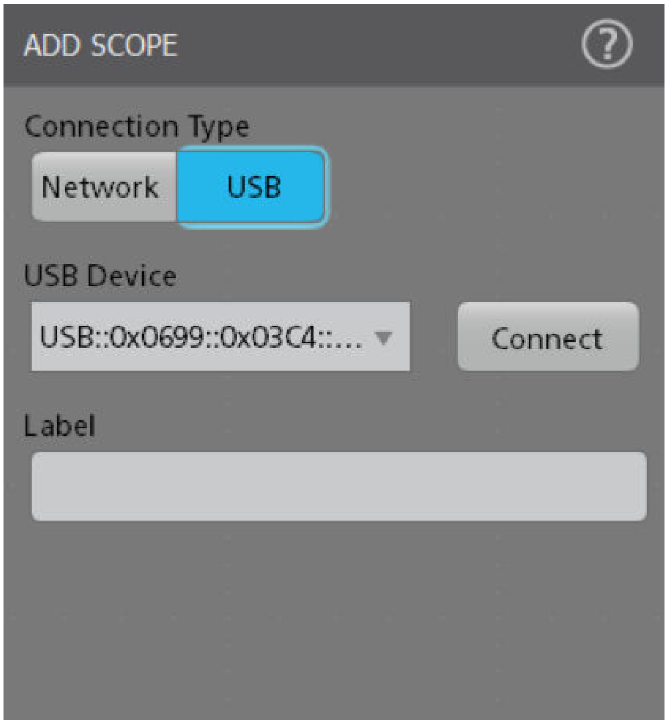
   You can add oscilloscopes numerically left to right (assuming that you cannot change the order of the badges, they would naturally increment from left to right).

You can establish a connection with any 4/5/6 Series MSO and DPO70000 Series oscilloscope.

Add new scope via USB

1. Select **Utility** > **I/O** > **VISA** to set the Remote scope communication settings.
2. Tap on **Add New Scope** on setting bar, to add new oscilloscope.

   The badge gets added to the right.
3. Double-tap on badge to open its associated configuration menu, Scope badge shows **Add Scope** panel.
4. Select Connection type as **USB**.
5. Select the connected device from the **USB device** drop down list.



6. Tap **Connect**.

   If connection is established, “Connected” message is displayed. Status turns green when connected.

   The error message displays if connection fails, check the USB connection or Remote scope communication settings and try again.

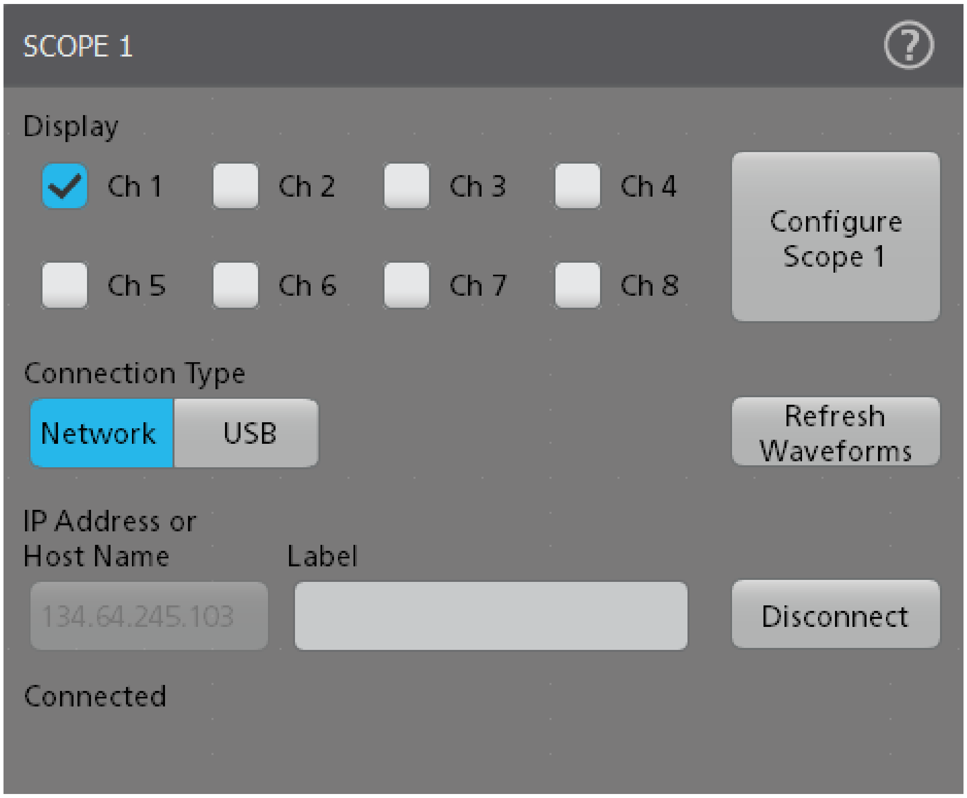
   Use the **Label** fields to enter labels for the individual instrument.

For TBS2000B and TBS1000C Series oscilloscopes: Use USB 2.0 Full Speed Device port to connect to a PC for direct remote control of the oscilloscope using TekVISA connectivity, and other remote connectivity tools that support USBTMC.

Configure Scope X

Double-tap on Scope X badge to opens the following menu.

⚠ **Note:** This option gets enabled only when the oscilloscope is remotely connected.



1. Configure the oscilloscope channels using checkboxes.

   If a channel box is checked, it turns on the display of the channel locally.

   If a channel box is unchecked, it turns off display of the channel locally.
2. Changing display checkbox state affects only tekscope channels display, it does not affect the remote instrument display unless control mode is on and acquire is done.
3. **Connection Type** shows the **Network** or **USB** as connection type for added instrument.
4. Tap **Refresh waveforms** button to refresh the waveforms when all channel data is transferred from the scope without taking a new acquisition.
5. Tap **Disconnect** to disconnect the oscilloscope.
6. Tap **Configure Scope X** opens a browser or remote desktop to indirectly view and configure the instrument settings.

   For Windows 10 instrument, Enter IP address, User name, and Password of the instrument. Tap **Connect**.

   For standard instrument follow the procedure *Configure e*Scope (standard instrument)* on page 59
7. Drag the Scope X badge off, to delete the Scope X.

   ⚠ **Note:** If you delete Scope X, all the wavefroms and anaylsis results related to Scope X will also get deleted.

Configure e*Scope (standard instrument)

This procedure describes how to remotely access the UI controls and screen for standard (non Windows 10) instruments. To remotely access the UI controls and screen for Windows 10 instruments, see the Remote access to a Windows 10 instrument topic in the Help.

Prerequisites:

You can remotely access your network-connected standard instrument (not running Windows) from a Web browser to display the instrument user interface on a PC.

- The instrument must be connected to, and accessible from, the network to which the PC is connected.
- The IP address of the instrument that you want to access. To determine the instrument's IP address, select **Utility** > **I/O** in the instrument menu bar and view the network settings in the LAN panel.
- You are accessing a standard (instrument that does not have the Windows OS option installed).

1. Open a Web browser on a PC connected to the same network as the instrument.
2. Enter just the instrument IP address on the URL line of the browser and press **Enter**. For example: 135.62.88.157. The browser searches for and opens the Web page for the instrument.
3. Select **Instrument Control** (e*Scope®). The browser displays the instrument screen.
4. Use a mouse to select and interact with the instrument controls shown in the Web browser. If your remote PC or laptop has a touch screen monitor, you can use the remote touchscreen monitor to access the instrument controls.

   Note: When you access the instrument from an e*Scope browser, you cannot directly paste text (such as path, IP address information, and so on) from the PC to an instrument menu field. You must use an intermediate clipboard function that is available in the e*Scope application.

5. Use the following steps to copy text from an e*Scope-connected PC to the instrument:
   a. Open a connection to the instrument using e*Scope.
   b. Select and copy the text on your PC.
   c. In e*Scope, press Ctrl-Alt-Shift to open the Clipboard menu.
   d. Paste the text into the Clipboard field.
   e. Press Ctrl-Alt-Shift to close the browser Clipboard menu.
   f. Use e*Scope to open the instrument menu to which to paste content, and position the cursor in the field where you want to paste the text.
   g. Press Ctrl-V (on real keyboard or from virtual keyboard) to paste the text from the e*Scope browser clipboard to the menu field.
   h. Repeat steps 4.b through 4.g to copy and paste other text from the PC to the instrument.

Acquiring waveforms from TekScope

When instrument is connected remotely, the status on setting bar turns to **Acquire**.

1. Tap **Acquire** button on setting bar to transfer the waveform data to TekScope.
2. The channels which are selected in the scope context panel gets transferred in TekScope.
3. Previous waveforms associated with all connected instrument gets trasferred regardless of whether the connection(s) are still present .
4. The status changes from **Acquire**> **arming** (oscilloscope starts arming) > **Cancel** (triggering on oscilloscope) > and again return to **Acquire** state.
5. If you tap on **Cancel** button it cancels acquisition on the instrument and set instrument to stop mode.
6. The Progress bar shows current waveform transfer status in %.
7. Assuming the connected instrument(s) is triggered, it begins transferring one waveform at a time to TekScope.
8. While transferring waveforms, the UI is locked out except the **Cancel** button.
9. If an error or disconnection happens while transferring waveforms, the error message with options to Cancel or Retry gets displayed.
10. Tap **Retry** to resume transfer of waveforms.
11. Once the waveforms are transferred, the status turns to **Acquire**.

Note: While acquiring data, if you open an e*Scope window into a connected instrument and then proceeds to make changes to a connected instrument's settings through e*Scope, it will affect the acquired data.

Configure remote channel

Vertical settings panel for remote channel (Remote control On)

- Double-tap a **Channel** or **Waveform** badge to open the configuration menu to the Vertical settings panel. Displayed fields and controls can change depending on menu selections.

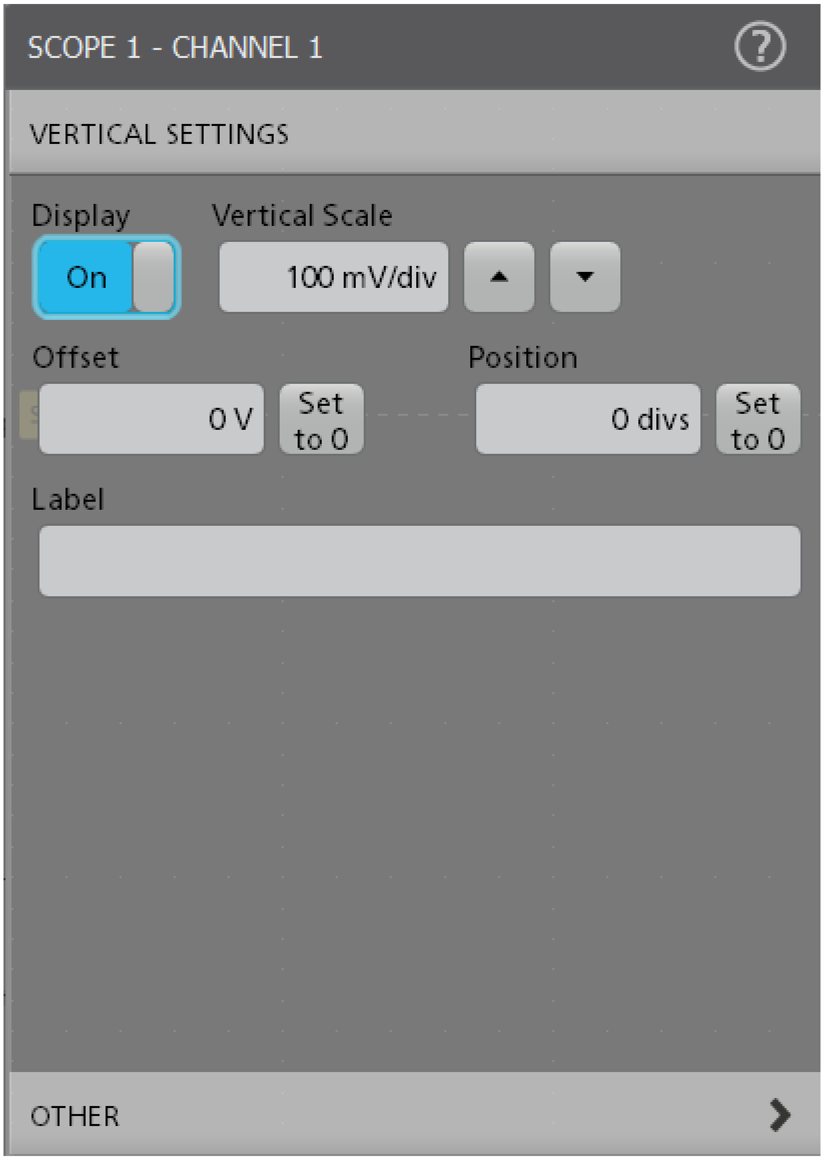


Figure 5: Vertical settings panel for remote channel whenremote control on

Field or control	Description
Display	Enables (On) or disables (Off) displaying the channel waveform in the Waveform View window.
Vertical Scale	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Offset	Sets the channel signal vertical offset.
Set to 0	Sets the channel signal vertical offset to 0 units.
Position	Double-tap the field to set the vertical position using the virtual keypad.

Table continued…

Field or control	Description
Set to 0	Tap to set the waveform zero-volt level to the center of the slice or waveform view.
Label	Double-tap the field to add a label to the channel display using the virtual keypad.

Other panel for remote channel

Use the Other panel to set the channel deskew, external attenuation, and alternate vertical scale units. To open the analog Channel configuration menu Other panel:

1. Double-tap a Channel badge on the Settings bar to open the Channel configuration menu.
2. Tap the Other panel.

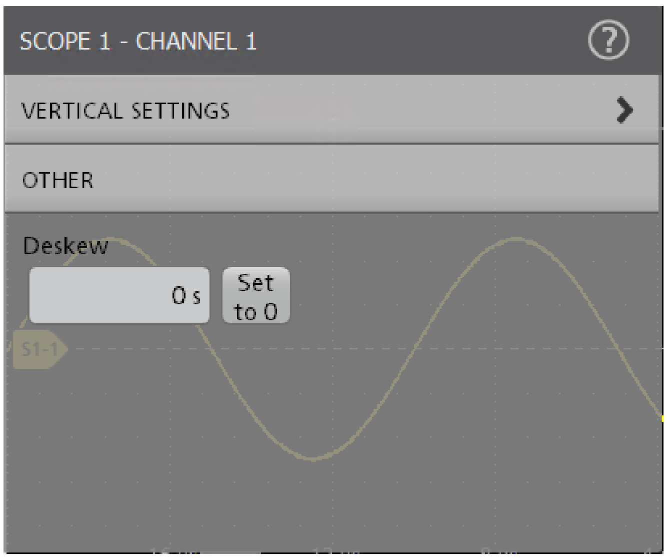


Figure 6: Other panel

Table 2: Other panel fields and controls

Field or control	Description
Deskew	Sets or displays the probe deskew value.
Set to 0	Sets the probe deskew value to zero (0) seconds.

Vertical settings panel for remote channel (Remote Control Off)

- Double-tap a **Channel** or **Waveform** badge to open the configuration menu to the Vertical settings panel. Displayed fields and controls can change depending on menu selections.

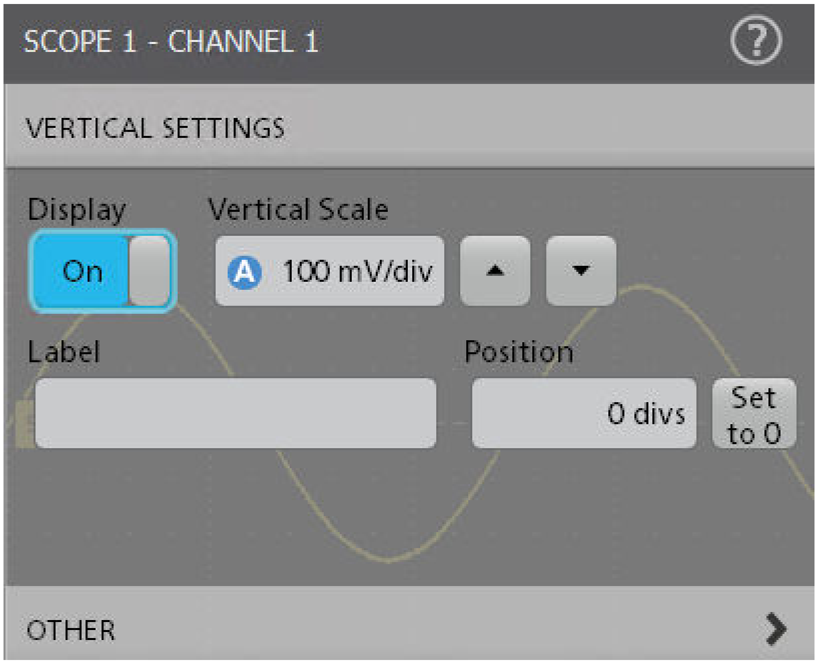


Figure 7: Vertical settings panel for remote channel when remote control off

Field or control	Description
Display	Enables (On) or disables (Off) displaying the channel waveform in the Waveform View window.
Vertical Scale	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Position	Double-tap the field to set the vertical position using the virtual keypad.
Set to 0	Tap to set the waveform zero-volt level to the center of the slice or waveform view.
Label	Double-tap the field to add a label to the channel display using the virtual keypad.

Other panel

Use the Other panel to set the channel deskew, external attenuation, and alternate vertical scale units. To open the analog Channel configuration menu Other panel:

1. Double-tap a Channel badge on the Settings bar to open the Channel configuration menu.
2. Tap the Other panel.

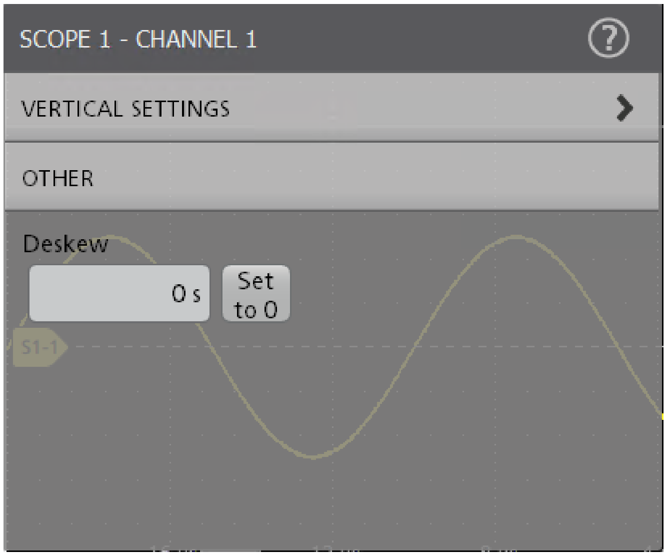


Figure 8: Other panel

Table 3: Other panel fields and controls

Field or control	Description
Deskew	Sets or displays the probe deskew value.
Set to 0	Sets the probe deskew value to zero (0) seconds.

Configure channel and waveform settings

Use the channel and waveform configuration menus to set parameters such as vertical scale and offset, coupling, bandwidth, probe settings, deskew values, external attenuation values, and other settings.

Use this procedure to set up analog signal input.

Before you begin

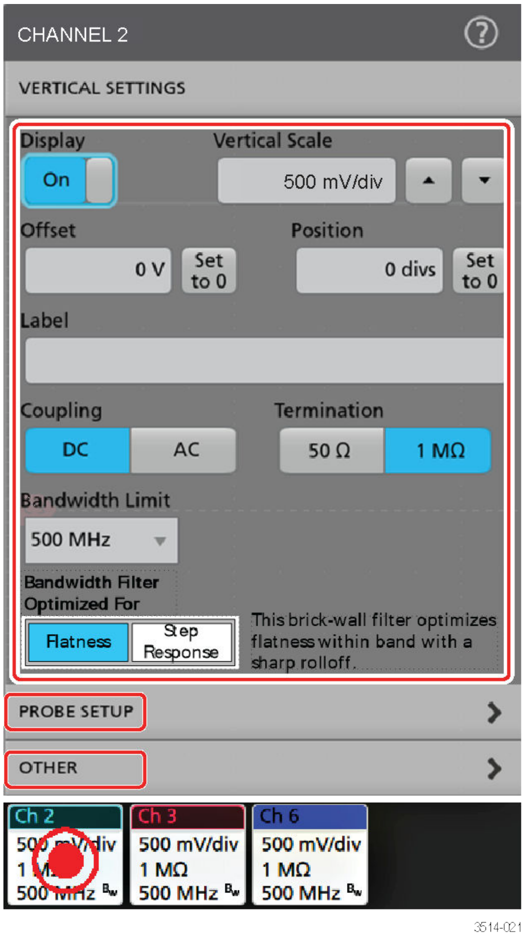
Prerequisites:

- An analog probe or cable is connected to the channel before setting up the channel.
- There is a channel or waveform badge in the Settings bar.

About this task

Procedure

1. Double-tap a **Channel** or **Waveform** badge to open the configuration menu to the Vertical Settings panel. Displayed fields and controls can change depending on menu selections.

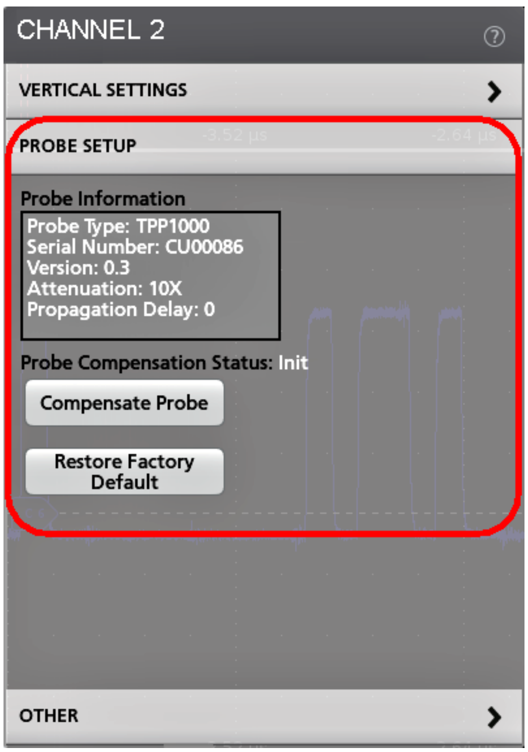


Field or control	Description
Display	Enables **(On)** or disables **(Off)** displaying the channel waveform in the Waveform View window.
Invert	Reverses the amplitude polarity of each sample point. When On, Invert adds a down arrow symbol to the channel badge.
Vertical Scale	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Offset	Sets the channel signal vertical offset.
Set to 0	Sets the channel signal vertical offset to 0 units.
Position	Double-tap the field to set the vertical position using the virtual keypad.
Set to 0	Tap to set the waveform zero volt level to the center of the slice or waveform view.
Label	Double-tap the field to add a label to the channel display using the virtual keypad.

Table continued…

Field or control	Description
Coupling	Tap to set the input coupling to DC or AC.
Termination	Tap to set the input termination to 1 MΩ or 50 Ω. If you are using a supported TekVPI probe, this value is automatically set by the probe and these controls are not available.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Bandwidth Filter Optimized For	Tap to select a bandwidth filter that is optimized for flatness or step response. This selection is only available on 6 Series MSO instruments. **Flatness** selects a brick-wall filter optimized for flatness within band with a sharp rolloff. Flatness filtering is not compatible with Peak Detect and Envelope acquisition modes. **Step Response** selects a Bessel-Thompson filter that minimizes overshoot with a gradual rolloff. **High Res** acquisition mode requires Flatness filtering.

2. To set up your probe, tap the **PROBE SETUP** panel. The fields and controls in the Probe Setup panel vary with the type of probe connected to the channel. A TPP probe is shown in this example, for other probes, see the probe user documentation.

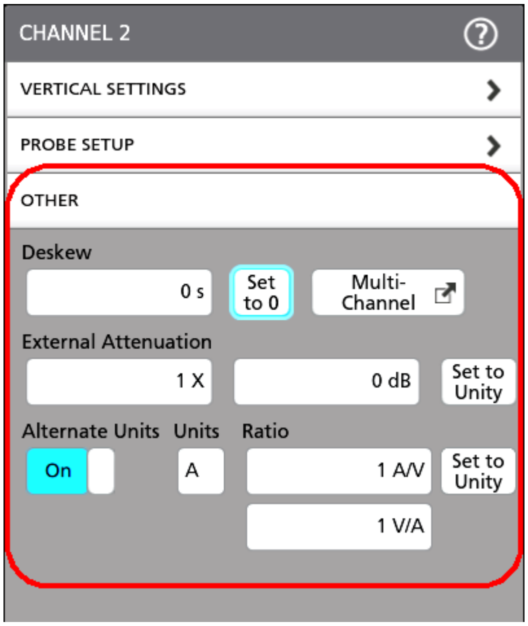


Field or control	Description
Probe Information	Displays the probe type, serial number, and attenuation, if it is available.
Probe Compensation Status	Displays the compensation status of the attached probe.
Compensate Probe	Starts the probe compensation procedure.

Table continued…

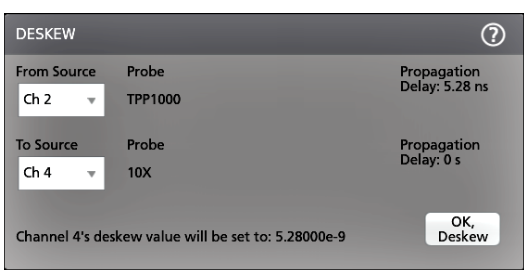
Field or control	Description
Restore Factory Default	Returns the compensation to the factory default settings.

3. To set up deskew, external attenuation, or alternate units, tap the **OTHER** setup panel.



Field or control	Description
Deskew	Allows setting the channel deskew value.
Set to 0	Sets the channel deskew to 0.
Multi-Channel	Brings up the Deskew configuration menu.
External Attenuation	Allows setting an external attenuation for the channel. As one field is edited, the other field changes to reflect the corresponding value.
Set to Unity	Sets left numeric input field to 1X and right input to 0.0 dB.
Alternate Units	Toggles alternate units on or off.
Units	Sets the ratio units. This field is disabled when the Alternate Units switch is off.
Ratio	Sets the desired ratio value. As one field is edited, the other field changes to reflect the corresponding value. This button is disabled when the Alternate Units switch is off.
Set to Unity	Sets the ratio to 1. This button is disabled when the Alternate Units switch is off.

4. When **Multi-Channel** in the **Other** panel is selected, the DESKEW configuration menu is displayed.



Field or control	Description
From Source	Selects the From Source to Deskew.
Probe	Displays the probe name or a drop-down list to select the probe connected to the From Source.
Propagation Delay	Displays the propagation delay of the probe shown in the Probe control.
To Source	Selects the To Source to Deskew.
Probe	Displays the probe name or a drop-down list to select the probe connected to the To Source.
Propagation Delay	Displays the propagation delay of the probe shown in the Probe control.
OK, Deskew	Uses the deskew values to adjust the horizontal delay between channels

5. Tap outside the menu to close the menu.

Horizontal setting panel

Use this menu to select the horizontal mode, set horizontal parameters, and enable trigger delay.

To open the Horizontal configuration menu, double-tap the Horizontal badge in the Settings bar.

Note: TekScope Remote Control is off, there is no Horizontal configuration menu.

Field or control	Description
Horizontal Mode	Tap to select either Automatic or Manual horizontal mode. In Automatic mode you can set the Minimum Sample Rate and Horizontal Scale. In Manual mode you can set the Sample Rate and Record Length. Horizontal Scale is a dependent variable calculated from the sample rate and record length.
Minimum Sample Rate	Sets the minimum sample rate for acquisitions. Changing horizontal scale will not reduce the sample rate below this value. This can result in partial waveform records at lower frequencies, but at the same time would provide more sample points for a few cycles of the signal. Available when Horizontal Mode = Automatic = Off. This setting can be overridden if Allow Horizontal Scale to Override Min Sample Rate is selected.
Allow Horizontal Scale to Override Min Sample Rate	Select to set the oscilloscope to automatically change the sample rate to acquire a full waveform record. Changes to the sample rate are shown in the Horizontal badge readout, not in the Minimum Sample rate field of the menu. Only available when Horizontal Mode = Automatic= Off.

Table continued…

Field or control	Description
Horizontal Scale	Tap to set the Horizontal Scale using the assigned multipurpose knob, double-tap to set the scale using the virtual keypad, or tap the up and down arrows. You can also use the front-panel Horizontal Scale knob to change this value. The horizontal scale determines the size of the acquisition window relative to the waveform. You can scale the window to contain a single waveform edge, a single cycle, several cycles, or thousands of cycles.
Delay	Delay positions the trigger event to a specified time relative to the center of the waveform record. Use delay to focus on events that occur before (pretrigger) or after the trigger point (posttrigger).
Position	Tap to set the trigger Position using the assigned multipurpose knob or double-tap to set the Position using the virtual keypad. When horizontal Delay is on, the time from the trigger point to the horizontal reference (center of waveform record) is the horizontal delay. The horizontal position determines the number of pretrigger and posttrigger samples in the waveform record. When horizontal delay is off, the trigger point and the horizontal reference are at the same time in the middle of the waveform record.
Set to 10%	Tap to set the trigger delay to 10% of the waveform record. Only available when Delay = Off.

Set Horizontal parameters

Use this procedure to set the horizontal time base parameters such as mode, minimum sample rate, horizontal scale, delay, and trigger delay time (relative to the center of the waveform record.

1. Double-tap the **Horizontal** badge on the Settings bar to open the Horizontal configuration menu.

Field or control	Description
Horizontal Mode	Tap to select either Automatic or Manual horizontal mode. In Automatic mode you can set the Minimum Sample Rate and Horizontal Scale. In Manual mode you can set the Sample Rate and Record Length. Horizontal Scale is a dependent variable calculated from the sample rate and record length.

Table continued…

Field or control	Description
Minimum Sample Rate	Sets the minimum sample rate for acquisitions. Changing horizontal scale will not reduce the sample rate below this value. This can result in partial waveform records at lower frequencies, but at the same time would provide more sample points for a few cycles of the signal. Available when Horizontal Mode = Automatic = Off. This setting can be overridden if Allow Horizontal Scale to Override Min Sample Rate is selected.
Allow Horizontal Scale to Override Min Sample Rate	Select to set the oscilloscope to automatically change the sample rate to acquire a full waveform record. Changes to the sample rate are shown in the Horizontal badge readout, not in the Minimum Sample rate field of the menu. Only available when Horizontal Mode = Automatic= Off.
Horizontal Scale	Tap to set the Horizontal Scale using the assigned multipurpose knob, double-tap to set the scale using the virtual keypad, or tap the up and down arrows. You can also use the front-panel Horizontal Scale knob to change this value. The horizontal scale determines the size of the acquisition window relative to the waveform. You can scale the window to contain a single waveform edge, a single cycle, several cycles, or thousands of cycles.
Delay	Delay positions the trigger event to a specified time relative to the center of the waveform record. Use delay to focus on events that occur before (pretrigger) or after the trigger point (posttrigger).
Position	Tap to set the trigger Position using the assigned multipurpose knob or double-tap to set the Position using the virtual keypad. When horizontal Delay is on, the time from the trigger point to the horizontal reference (center of waveform record) is the horizontal delay. The horizontal position determines the number of pretrigger and posttrigger samples in the waveform record. When horizontal delay is off, the trigger point and the horizontal reference are at the same time in the middle of the waveform record.
Set to 10%	Tap to set the trigger delay to 10% of the waveform record. Only available when Delay = Off.

2. Use the menu selections to set horizontal parameters.

3. Tap the Help icon on the menu title for more information on these settings.

How to trigger on a signal

Use this procedure to open the Trigger menu to select and configure the trigger event type and conditions.

Note: When TekScope Remote Control is off, there is no Trigger badge or Trigger configuration menu. For TekScope trigger input only supports Edge triggering.

1. Double-tap the Trigger badge on the Settings bar to open the Trigger configuration menu.
2. Select a trigger from the Trigger Type list. The trigger type sets what fields are available in the menu and also updates the illustration to show a graphic of the trigger type.
3. Select the other fields and panels to refine the trigger conditions. The menu fields and trigger graphic update as you make changes to the trigger settings. Displayed fields depend on the selected trigger type. Selection changes take effect immediately.
4. Tap the Help icon on the menu for more information on these settings.
5. Tap outside the menu to close the menu.
6. When TekScope Remote Control is on, the Trigger badge is shown in the GSRB and the Trigger configuration menu is the same as a 4/5/6 Series Trigger menu with the following exceptions:

Field or control	Description
Trigger Type	Set to Edge.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control. AC Line source (Edge trigger): The instrument uses the AC line to generate a trigger based on the power line frequency. The trigger level is fixed at zero volts. This source is often used to look at signals related to the power line frequency (for example, signals from devices such as lighting equipment and power supplies). Because the instrument generates the trigger from the power line, you do not have to use a channel input. AUX In: The MSO58LP and 6 Series MSO also have an AUX In (auxiliary) edge trigger source. AUX In provides an extra trigger signal source when you need to use the input channels for other signals. For example, you might want to trigger on a clock while displaying related logic signals. The AUX In input connector is not compatible with most probes, and you cannot display the auxiliary trigger signal in the Waveform view.

Table continued…

Field or control	Description
Coupling	Set the conditioning to apply to the source signal trigger circuit from the source signal. DC coupling passes all input signals directly to the trigger circuitry. HF Reject coupling attenuates signals above 50 kHz before passing the signal to the trigger circuitry. LF Reject coupling attenuates signals below 50 kHz before passing the signal to the trigger circuitry. Noise Reject coupling provides stable triggering by increasing the trigger hysteresis. Increased hysteresis reduces the trigger sensitivity to noise so may require greater signal amplitude.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Slope	Sets the signal transition direction to detect. (rising, falling, or either direction).

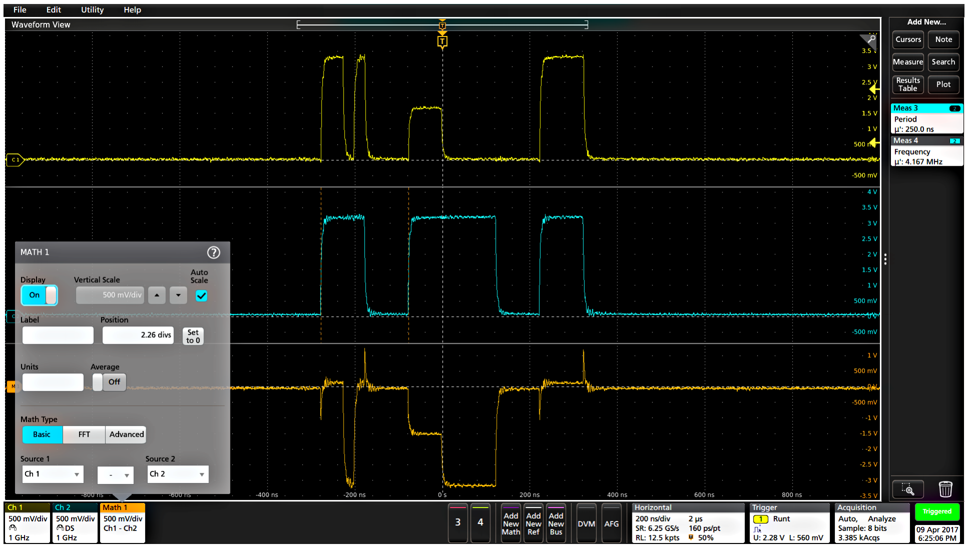
Add a math, reference, or bus waveform

Math waveforms let you create new waveforms based on operations between two or more waveforms or by applying equations to waveform data. A reference waveform is a static waveform record displayed for comparison. Bus waveforms let you view and analyze serial or parallel data.

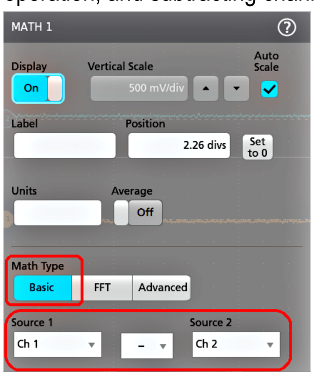
There is no set limit to the number of Math, Reference, or Bus waveforms you can add to the Waveform View, other than system physical memory constraints.

1. Tap the **Add New Math**, **Add New Ref**, or **Add New Bus** button in the Settings bar.

2. The instrument adds the waveform to the Waveform view, adds a Waveform badge to the Settings bar, and opens a configuration menu. This example shows adding a Math waveform.



3. Use the configuration menus to refine the waveform parameters. Displayed fields depend on the waveform and selections made in the menu. Selection changes take effect immediately. This example shows adding a Math waveform, using the Math **Source** fields to select Ch 1 and Ch 2 as the waveform sources, set the math type to **Basic** math operation, and subtracting channel 2 from channel 1.



4. When adding a Reference waveform, the instrument displays a **Recall** configuration menu. Navigate to and select the reference waveform file (*.wfm) to recall, then tap the **Recall** button. The instrument displays the Reference waveform.
5. Double-tap a math, reference, or bus badge to check or change that waveform's settings.
6. Tap the Help icon on a configuration menu title for more information on math, reference, and bus waveform settings.
7. Tap outside the menu to close the menu.

Add a measurement

Use this procedure to select and add measurements.

Procedure

1. Acquire the channel(s) and/or waveform(s) on which you want to take measurements.

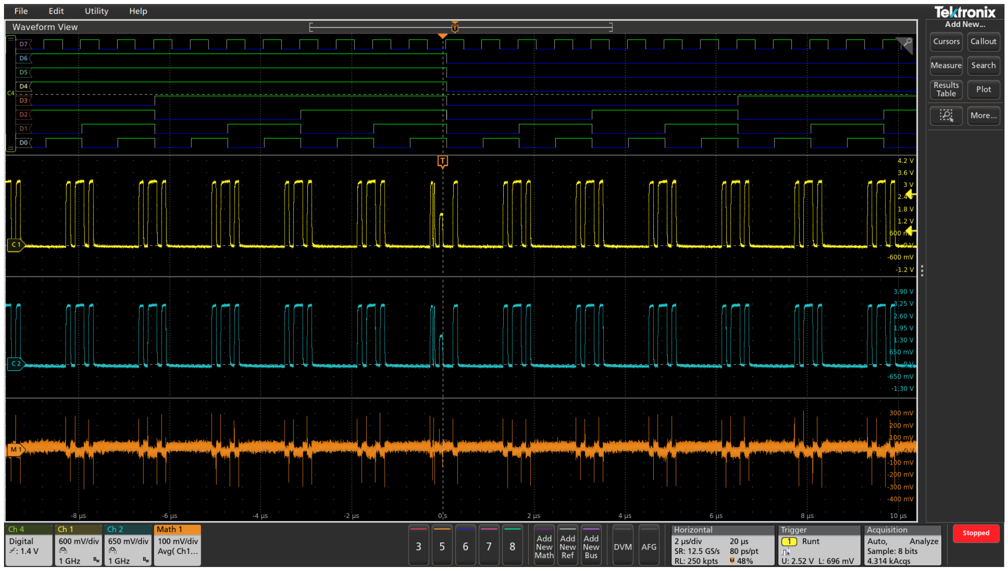


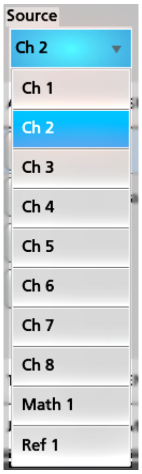
Figure 9: Acquire the channel waveform

Note: Waveforms do not need to be displayed to be used for measurements, as long as the channel or waveform badge is on the **Settings** bar and is acquiring the signal to measure.

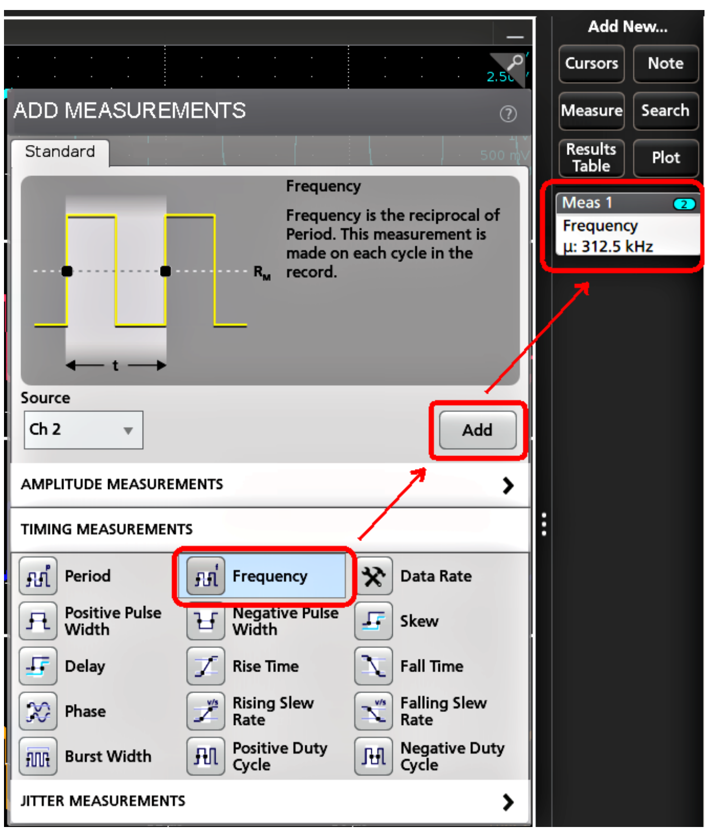
2. Tap the **Add New...Measure** button to open the **Add Measurements** configuration menu or drag the Measure button onto a waveform in the waveform display area to automatically set the source.

⚠ **Note:** If the menu shows tabs other than **Standard**, then optional measurement types have been installed on the instrument. Select a tab to show the measurements for that option.

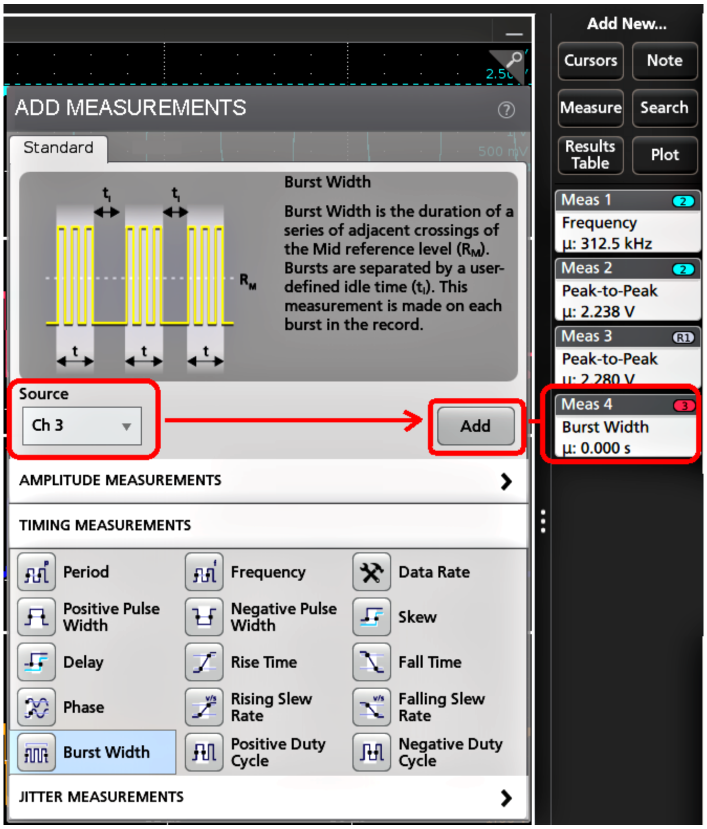
3. Tap the **Source** field and select the measurement source. The list shows all available sources that are valid for the measurement.



4. Select a measurement category panel, such as **Amplitude Measurements** or **Timing Measurements**, to display measurements for those categories.
5. Some optional measurement packages are available in this menu as well like PWR, DJA, DBDDR3, DPM, IMDA.
6. Select a measurement and tap **Add** to add the measurement to the **Results** bar. You can also double-tap a measurement to add it to the **Results** bar.



7. Select and add other measurements for the current source. Tap the measurement category panels to display and select other measurements to add.
8. To add measurements for other sources, select a different source, select a measurement, and add the measurement.



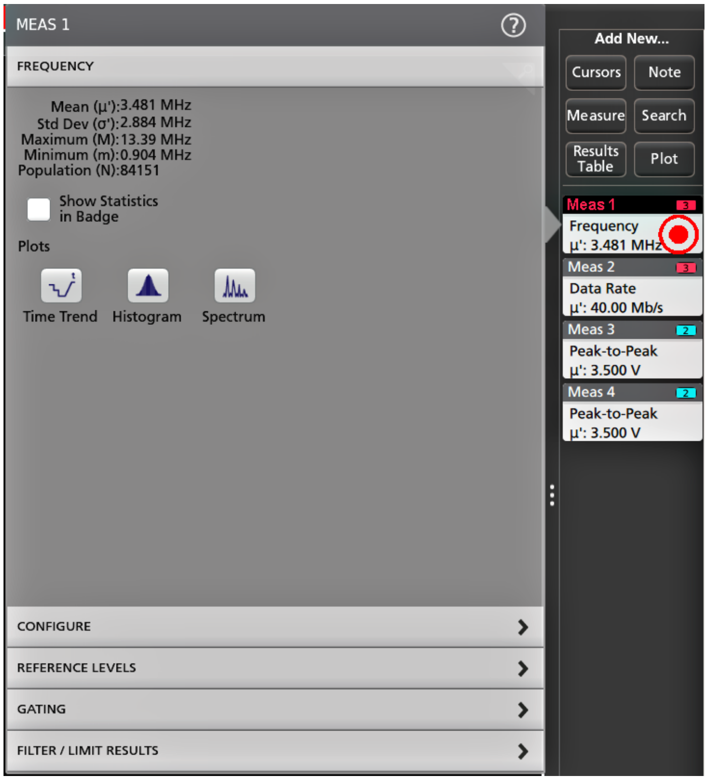
9. Tap outside the **Add Measurements** menu to close the menu.
10. To further adjust a measurement's settings, double-tap a measurement badge to open a configuration menu for that measurement. See *Configure a measurement* on page 77.
11. Tap the Help icon on the menu title for more information on settings.

Configure a measurement

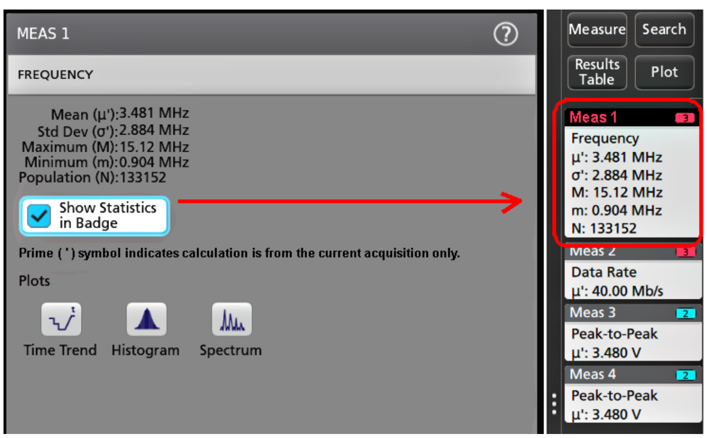
Use this procedure to add statistical readouts to the measurement badge, display plots for the measurement, and refine measurement parameters (configuration, global versus local scope of settings, gating, filtering, and so on).

Procedure

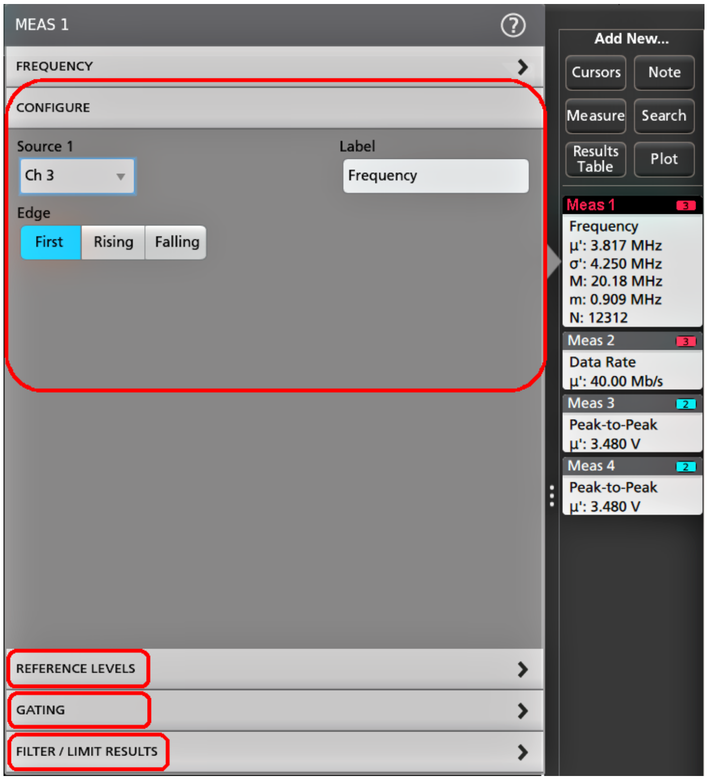
1. Double-tap a measurement badge to open its **Measurement** configuration menu.



2. Tap **Show Statistics in Badge** to add statistical readouts to the measurement badge.



3. Tap available panel titles to make changes for those categories.



4. Use the available fields to refine the measurement conditions. Displayed fields depend on the measurement. Selection changes take effect immediately. Selection changes can also change fields in other panels.
5. Tap the Help button on the menu title for more information on this menu's settings.
6. Tap outside the menu to close the menu.

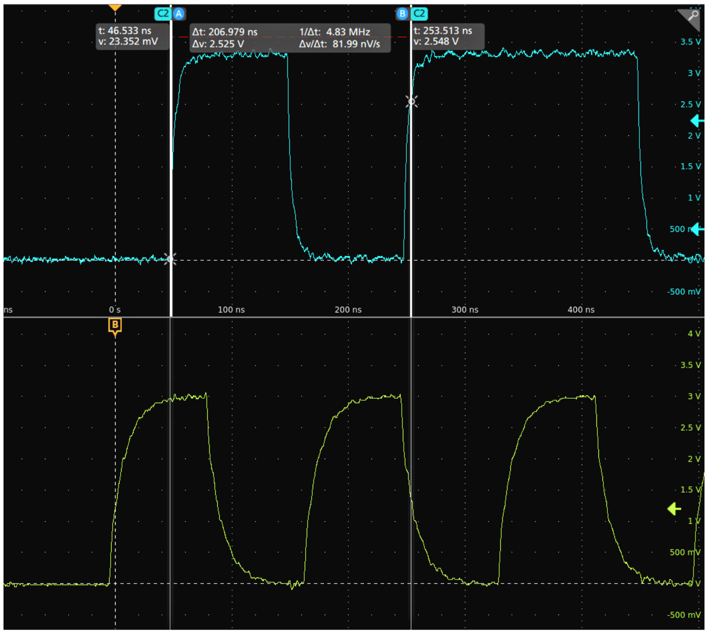
Display and configure cursors

Cursors are on-screen lines that you can move to take measurements on specific parts of a waveform or plot, or between two different waveforms. Cursor readouts show both current position values and the difference (delta) between cursors. Polar cursor readouts are available through the cursor configuration menu for XY and XYZ plots.

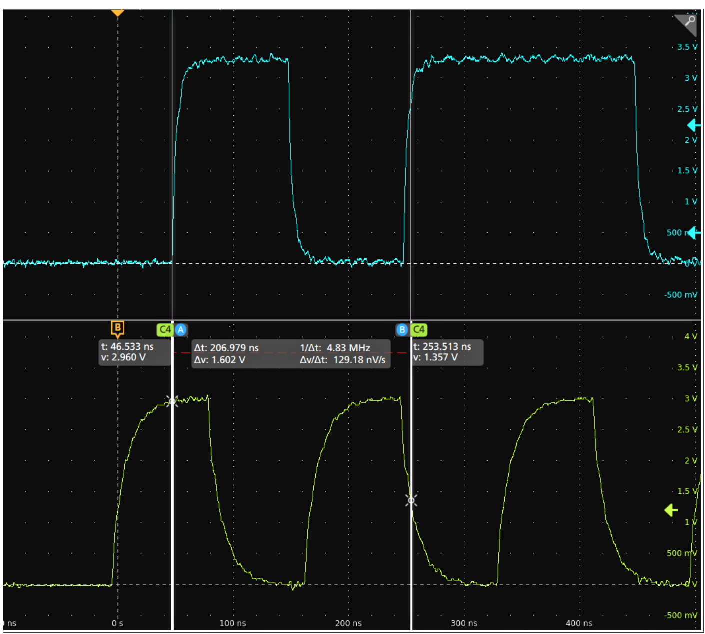
Procedure

1. Tap the waveform slice (in Stacked mode), or the channel or waveform badge (in Overlay mode) to which you want to add cursors.
2. Tap the **Add New...Cursors** button, or push the front-panel **Cursors** button.

   The cursors are added to the display.

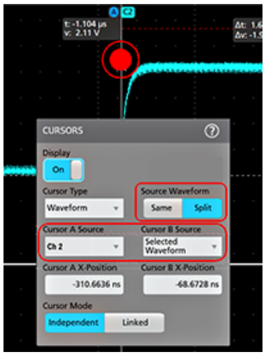


3. Use Multipurpose Knobs **A** and **B** to move the cursors, or touch and drag a cursor. Cursors show readouts that show position and difference measurements between the cursors.
4. To move the cursors to a different channel or waveform, just tap in that waveform graticule.

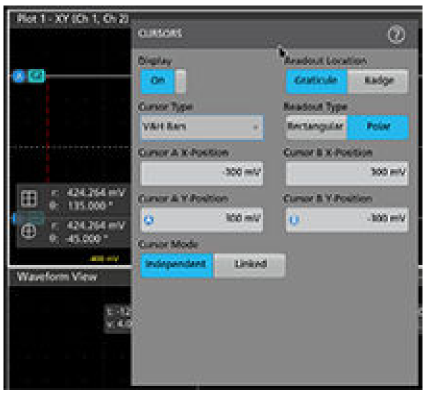


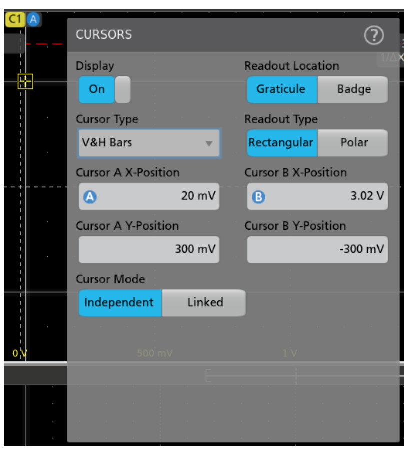
5. To further configure cursors, double-tap on either cursor line or the cursor readouts to open the **Cursors** configuration menu. For example, tap the Cursor type to select the cursors to display, such as Waveform, V Bars, H Bars, and V&H Bars. See *Cursor configuration menu* on page 82.

   The cursor configuration menu in the waveform view.

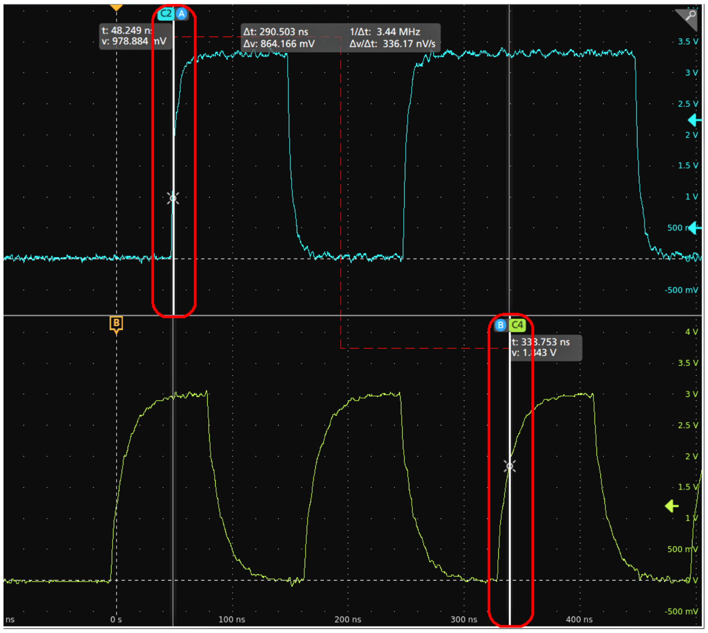


The cursor configuration menu in an XY plot.





6. To split the cursors between two waveforms, tap the **Source** field and select **Split** and select the source for each cursor.



The cursors are moved to the specified waveforms.

7. Tap the Help icon on the menu title for more information on the menu settings.
8. To stop showing cursors, push the front panel **Cursor** button, press and hold to open the right-click menu and turn cursors off, or open the Cursors configuration menu and set Display to **Off**.

Cursor configuration menu

Cursors are on-screen lines (bars) that you position in a Waveform or Plot views to take manual measurements on signals. They appear as horizontal and/or as vertical lines.

To open the Cursors configuration menu:

1. Double-tap a cursor readout or cursor line, or
2. Touch and hold a cursor readout or cursor line and select **Configure Cursors** from the right-click menu.

To display cursors on the screen:

1. Tap the **Add New... Cursors** button of the display, or
2. Push the **Cursors** front-panel button to toggle cursors on or off.

Cursor configuration menu fields and controls

Some fields or controls are only available when certain other controls are selected.

Field or control	Description
Display	Tap to toggle the cursor display **On** or **Off**.
Readout Location	Tap to set where to display cursor readouts. **Graticule** displays the cursor readouts on the screen cursor bars (default method). You can move the readouts on cursors by touching and dragging them along the cursor bar. **Badge** removes the screen cursor readouts and displays the cursor information in a **Cursors** badge in the Results bar. The **Cursors** badge content changes as you change the cursor type.

Table continued…

Field or control	Description
Cursor Type	Tap to select the cursor type from the drop-down list. **Waveform** cursors measure vertical amplitude and horizontal time parameters simultaneously at the point the cursor intersects a waveform. The cursor intersect point tracks waveform amplitude changes. **V Bars** are vertical cursors that measure horizontal parameters (typically time). They are not associated with the waveform, but show the time position of the cursor in the waveform record. **H Bars** are horizontal cursors that measure amplitude (typically in volts or amperes). They are not associated with the waveform, but show the amplitude position of the cursor. **V&H Bars** cursors measure vertical and horizontal parameters simultaneously. They are not associated with the waveform, but show the time and amplitude position of the cursors.
Source Waveform	Tap to select the source waveform (the waveform on which to display the cursors). **Same** places both cursors on the same waveform. **Split** allows each cursor to be on a different waveform. Only available when **Cursor Type** = **Waveform**.
Readout Type	Tap to select the type of readout to display for a XY or XYZ plot. This control is only available when the Cursor Type is set to Waveform or V&H Bars. **Rectangular** displays the position values and the difference (delta) between cursors. **Polar** displays the polar information for each cursor. The top value is the magnitude of the vector drawn from the origin of the plot to the location of the cursor. The bottom value is the angle of the vector drawn from the origin of the plot to the location of the cursor.
Source	Tap to select the source waveform from the drop-down list. **Selected Waveform** automatically moves the waveform cursors to the selected source. Or select a specific source from the drop-down list. Only available when **Source Waveform** = **Same**.
Cursor A Source, Cursor B Source	Tap to select the waveform sources for Cursor A and B. Only available when **Source Waveform** = **Split**.
Cursor A X-Position	Tap to set a specific x-axis position for Cursor A using the multipurpose knob, or double-tap to set the position using the keypad.
Cursor B X-Position	Tap to set a specific x-axis position for Cursor B using the multipurpose knob, or double-tap to set the X-Position using the keypad.
Cursor A Y-Position	Tap to set a specific Y-axis position for Cursor A using the multipurpose knob, or double-tap to set the position using the keypad. Only available when **Cursor Type** = **H Bars** or **V&H Bars**.
Cursor B Y-Position	Tap to set a specific Y-axis position for Cursor B using the multipurpose knob, or double-tap to set the X-Position using the keypad. Only available when **Cursor Type** = **H Bars** or **V&H Bars**.

Table continued…

Field or control	Description
Cursor Mode	Tap to select the cursor mode. **Independent** mode sets multipurpose knobs A and B to move each cursor separately. **Linked** mode sets multipurpose knob A to move both cursors at the same time. Knob B will still move cursor B independently of knob A.

Display a Histogram plot

Use this procedure to display a histogram plot.

Before you begin

To display a histogram plot you must be taking a measurement.

Procedure

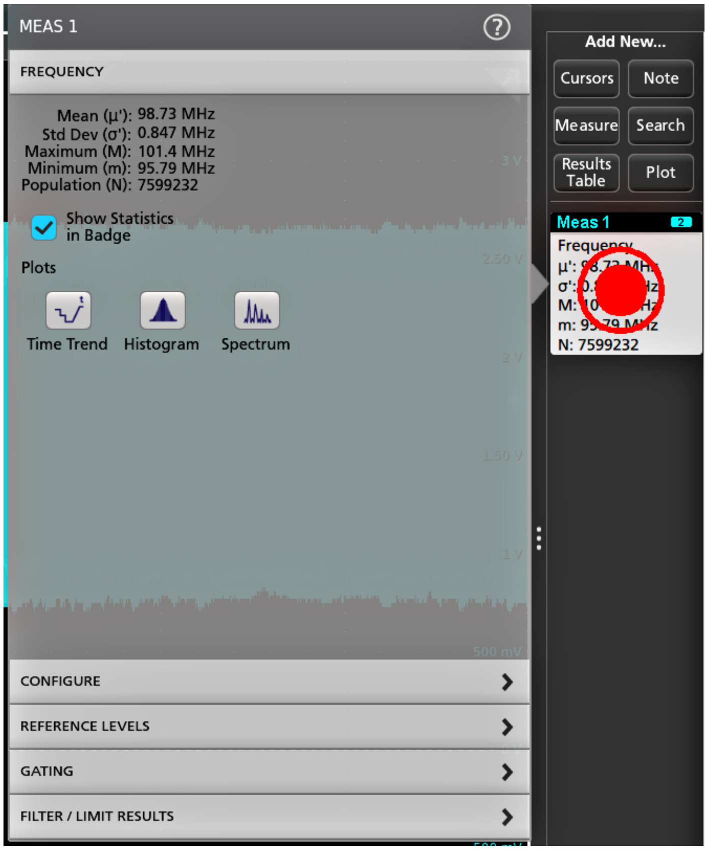
1. Double-tap a measurement badge.
   The Measurement configuration menu is displayed.
2. Tap the **Histogram** plot button.
   The Histogram plot is displayed in a separate Plot view.
3. Double-tap in the plot view to open a configuration menu for that plot.

Add a plot of a measurement

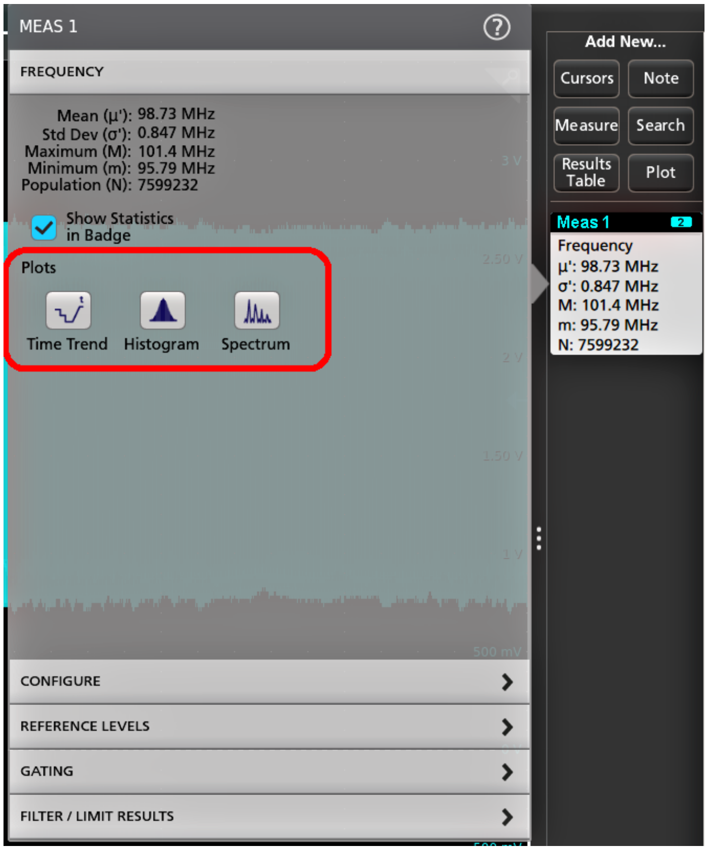
Measurement plots let you graph the distribution of waveform data point occurrences (histogram), plot the frequency components (spectrum) of a waveform, show the time trend of a measurement, display an eye diagram, and other supported plots. Available plots depend on the measurement.

Procedure

1. Double-tap a Measurement badge to open the **Meas** configuration menu.



2. Tap a **Plots** button to add that plot for the measurement to the screen.



The following shows adding a Histogram plot.

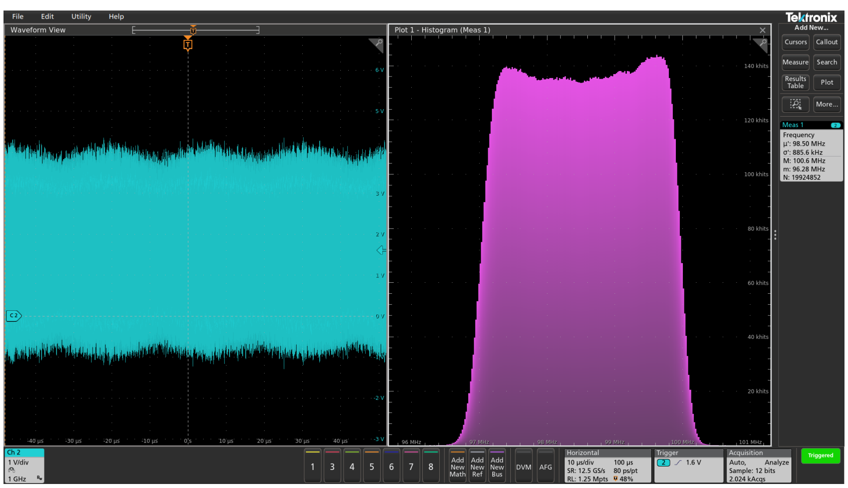
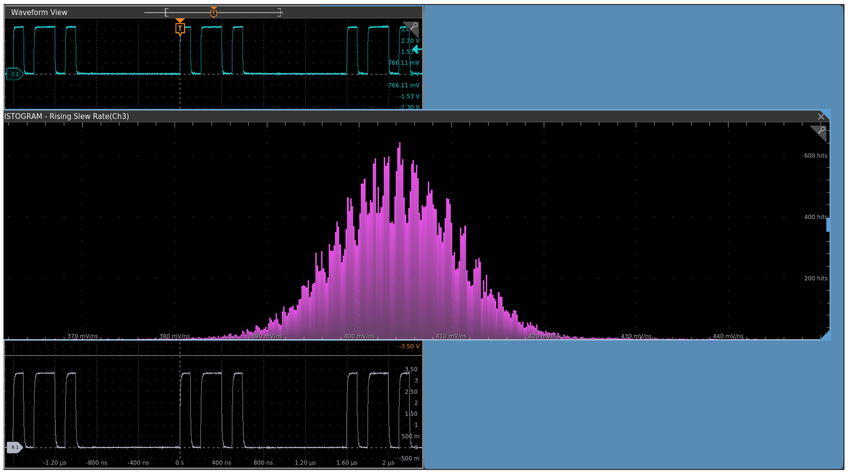


Figure 10: Histogram plot

You can add more than one plot to measurements (to different measurements or the same measurement). For example, you can add two histogram plots for the same measurement, set one to display the X-Axis with a Logarithmic scale, and the other plot to display the X-Axis with a Linear scale.

3. You can move plot windows by dragging the Plot view title bar to a new position. The blue background area moves to show where the plot will be located when you remove your finger from the title bar. You can also resize plot windows by

selecting and dragging the Plot view border. You should use a mouse to do these operations, as it is easier to select and drag plots with a mouse.



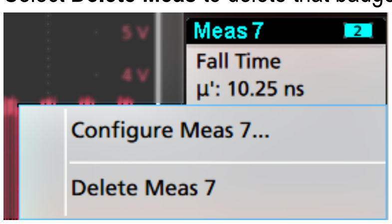
4. Double-tap within a Plot view to open a configuration menu to set display characteristics. Tap the Help icon on the configuration menu title for more information on that menu's settings. Tap outside the menu to close the menu.

Delete a Measurement or Search badge

Use this procedure to remove a Measurement or Search badge from the Results bar.

Procedure

1. Touch and hold the Measurement or Search badge that you want to delete. The instrument opens a right-click menu.
2. Select **Delete Meas** to delete that badge from the Results bar.



Note: You can undo a measurement delete.

3. The second way to delete a Measurement or Search badge is by dragging it to the **Trash Can** icon, located at the bottom of the **Results** bar. When the Trash can icon and the badge turns red, lift your finger to delete the badge. You can use the mouse to drag and delete a badge.
4. The third way to delete a Measurement or Search badge is by flicking it off the right edge of the display. Flicking to the left from the right edge of the display recovers the badge.

Note: Badge recovery is only possible within 10 seconds of removal.

Display a Time Trend plot

Use this procedure to display a time trend plot.

Before you begin

To display a time trend plot you must be taking a measurement.

Procedure

1. Double-tap a measurement badge.
   The Measurement configuration menu is displayed.
2. Tap the **Time Trend** plot button.
   The Time Trend plot is displayed. Time Trend plots are shown in the Waveform View rather than in a separate Plot view.

Display a Acq Trend plot

Use this procedure to display a acq trend plot (IMDA-Power Quality and Phasor Diagram).

Before you begin

To display a acq trend plot you must be taking a measurement.

Procedure

1. Double-tap a measurement badge.
   The Measurement configuration menu is displayed.
2. Tap the **Acq Trend** plot button.

   Note:

   - Plot rendering resets when there is change in the oscilloscope settings such as the vertical settings, horizontal, acquisition and trigger etc.
   - When oscilloscope acquisition is stopped and restarted again, the plot will continue to render without resetting.
   - Acq trend plot does not support Save/Recall session feature.
   - Cursors can be enabled on the plot.

   The Acq Trend plot is displayed in a separate Plot view.
3. Double-tap in the plot view to open a configuration menu for that plot.

Display an Eye Diagram plot

Use this procedure to display an eye diagram of a waveform.

Before you begin

Prerequisite: An active signal for which you want to plot an eye diagram.

Procedure

1. Tap **Add New... Plot.** to open the Plot configuration menu.
2. Tap **Eye Diagram**.
3. Tap the **Source** drop down and select the signal source from the list.
4. Tap **Add.** The instrument adds the eye diagram in a separate plot window and adds a TIE measurement badge to the Results bar.
5. Double-tap in the Plot view to open a configuration menu for that plot.

Display a Spectrum plot

Use this procedure to display a spectrum plot.

Before you begin

To display a time spectrum plot you must be taking a measurement.

Procedure

1. Double-tap a measurement badge.
   The Measurement configuration menu is displayed.
2. Tap the **Spectrum** plot button.
   The Spectrum plot is displayed in a separate Plot view.
3. Double-tap in the plot view to open a configuration menu for that plot.

Display an XY or XYZ plot

Use this procedure to display an XY or XYZ plot.

Procedure

1. Tap **Add New... Plot.**
   The Plot configuration menu is displayed.
2. If creating an XY plot, tap **XY**.
3. If creating an XYZ plot, tap **XYZ**.
4. Tap the **X Source** drop down and select the X Source from the list.
5. Tap the **Y Source** drop down and select the Y Source from the list.
6. If creating an XYZ plot, tap the **Z Source** drop down and select the Z Source from the list.
7. Tap **Add.**
   The plot is displayed in a separate Plot view.
8. Double-tap in the Plot view to open a configuration menu for that plot.

Display a Harmonics Bar Graph plot for the Power Harmonics measurement

Use this procedure to display a bar graph.

Before you begin

To display a harmonics bar graph, you must be taking a Harmonics measurement (**Add New**... **Measurement** > **Power** tab > **Electrical Analysis** panel > **Harmonics** measurement)

Procedure

1. Double-tap a **Harmonics** measurement badge.
   The Measurement configuration menu is displayed.
2. Tap the **Bar graph** button.
   The Bar graph is displayed in a separate Plot view.
3. Double-tap in the Plot view to open the configuration menu for that plot.

Display a Power Supply Rejection Ratio plot

Use this procedure to display a Power Supply Rejection Ratio plot for a power measurement.

About this task

To display a Power Supply Rejection Ratio power measurement and plot:

Procedure

1. Tap the **Add New... Measurement** button.
2. Tap the **Power** tab.
3. Set the voltage and current sources for the measurement.
4. Tap the **Power Supply Rejection Ratio** panel.
5. Select the **Power Supply Rejection Ratio** measurement and tap **Add**.
   The Power Supply Rejection Ratio measurement adds a Power measurement badge to the Results bar, and automatically adds the Power Supply Rejection Ratio plot to the screen.
6. Double-tap in the Plot view to open a configuration menu for that plot.

Display a power Switching Loss (SWL) Trajectory plot

Use this procedure to display a trajectory plot for a power measurement.

Before you begin

To display a trajectory power plot, you must be taking a Switching Loss measurement (**Add New**... **Measurement** > **Power** tab > **Switching Analysis** panel > **Switching Loss** measurement).

Procedure

1. Double-tap the **Switching Loss** measurement badge to open the configuration menu.
2. Tap **Trajectory Plot** to add a new Trajectory Plot view to the screen.
3. Double-tap in the Plot view to open a configuration menu for that plot.

Display a Control Loop Response (Bode) plot

Use this procedure to display a Control Loop Response (Bode) plot for a power measurement.

About this task

To display an Control Loop Response power measurement and plot:

Procedure

1. Tap the **Add New... Measurement** button.
2. Tap the **Power** tab.
3. Set the voltage and current sources for the measurement.
4. Tap the **Frequency Response Analysis** panel.
5. Select the **Control Loop Response** measurement and tap **Add**.
   The Control Loop Response measurement adds a Power measurement badge to the Results bar, and automatically adds the Control Loop Response plot to the screen.
6. Double-tap in the Plot view to open a configuration menu for that plot.

> **Note:** Gain Margin (GM) represents the margin value and not the actual value annotated in the Bode plot. The GM value indicates distance (basically how far) from the reference value (zero dB line). The margin is

computed as zero minus the measured value. The measured value becomes negative when it is below 0 dB, then margin is positive and vice versa.

Add a Search

Use this procedure to set search criteria and mark a waveform where those events occur.

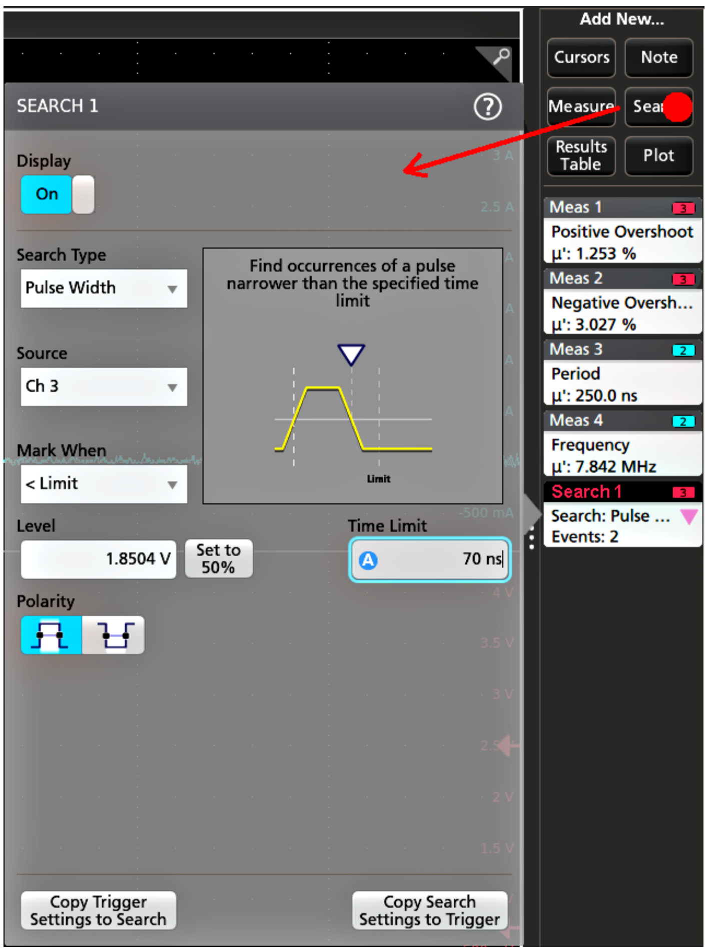
About this task

You can search on analog and digital signals, math waveforms, and reference waveforms. You can add searches to different waveforms and multiple searches to the same waveform.

Prerequisite: Display the channel or waveform signal on which to search. The waveform must be displayed to create a search for it.

Procedure

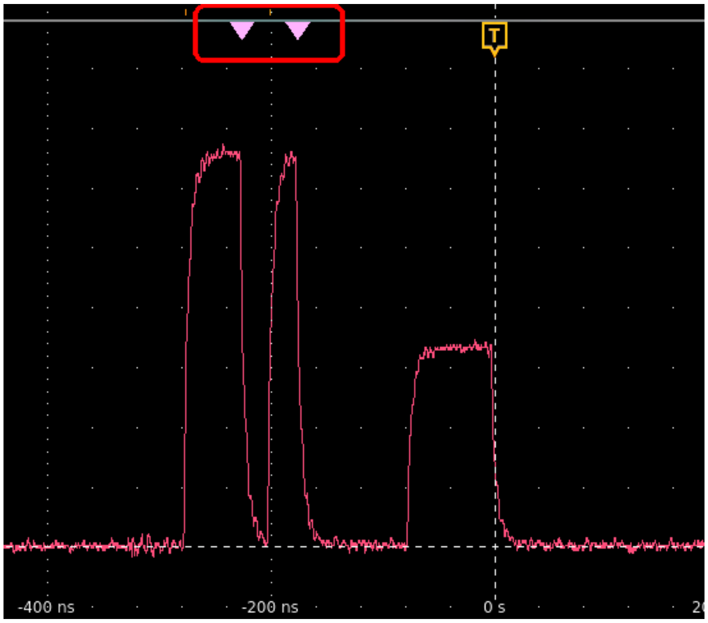
1. Display the channel or waveform signal on which to search. The waveform must be displayed to create a search for it.
2. Tap the **Add New...Search** button to open the Search configuration menu.



3. Use the configuration menu fields to set the search criteria in the same way that you would set for a trigger condition (select the **Search Type**, **Source**, and conditions on which to search).

Note: You cannot search for sequential events (there is no Sequence search type).

4. The searched waveform is marked with one or more triangles as soon as the search criteria becomes true. Each search uses a different color for its markers. The example image shows search criteria set to find positive pulse widths that are less than 70 ns wide.



5. To stop showing marks on a waveform, double-tap the **Search** badge and tap **Display** to **Off.**
6. To move the waveform to center marks on the display, push the **Run/Stop** front panel button to stop acquisition, single-tap a **Search** badge, and tap the < or > Navigation button.

Note: Navigation buttons are only functional when the oscilloscope acquisition mode is set to **Stop**.

This opens the **Zoom** mode and moves the waveform to the previous or next event mark on the waveform.

7. If available for a search, tap the **Min** or **Max** button to center the waveform in the display at the minimum or maximum value of the search events in the waveform record.
8. To return the instrument to normal acquisition mode, tap the **Zoom** icon in the upper right corner of the Waveform View to turn off **Zoom** mode, and push the **Run/Stop** front-panel button to set it to Run mode.

Display a B-H curve plot for Magnetic Property measurement

Use this procedure to display a B-H curve plot for a power measurement.

Before you begin

To display a B-H Curve plot, you must be taking a Magnetic Property measurement (**Add New**... **Measurement** > **Power** tab > **Magnetic Analysis** panel > **Magnetic Property** measurement).

Procedure

1. Double-tap the **Magnetic Property** measurement badge to open the configuration menu.

2. Tap **B-H Curve** to add a new B-H Curve Plot view to the screen.
3. Double-tap in the Plot view to open a configuration menu for that plot.

Display instantaneous power and energy plots from the Power Quality measurement

Use this procedure to display a power/energy plot from the Power Quality measurement.

Before you begin

To display the power and energy plots you must be taking the Power Quality measurement.

Procedure

1. Tap the **Add New... Measurement** button.
2. Tap the **Power** tab.
3. Set the Voltage and Current sources for the measurement.
4. Tap the **Electrical Analysis** panel.
5. Select the **Power Quality** measurement and tap **Add**.
   The measurement is added to the Results bar, and automatically adds **PQ: Power** and **PQ: Energy** math waveforms (plots) to the Waveform view. Math plots are shown in the Waveform View rather than in a separate Plot view.
   - PQ: Power displays the power waveform computed from the Power Quality input sources.
   - PQ: Energy displays the energy waveform computed from the Power Quality input sources. See *Waveform View configuration menu* on page 402.

Display a Safe Operating Area (SOA) measurement and plot

Use this procedure to display a SOA plot. This is an XY plot with a mask.

About this task

To display an SOA power measurement and plot:

Procedure

1. Tap the **Add New... Measurement** button.
2. Tap the **Power** tab.
3. Set the voltage and current sources for the measurement.
4. Tap the **Switching Analysis** panel.
5. Select the **SOA** measurement and tap **Add**.
   The SOA measurement adds a Power measurement badge to the Results bar, and automatically adds the SOA plot to the screen.
6. Double-tap in the Plot view to open a configuration menu for that plot.

Display an Inductance Curve plot for Inductance measurement

Use this procedure to display an inductance curve plot.

Before you begin

To display an inductance curve plot, you must be taking an Inductance measurement (**Add New**...**Measurement** > **Power** tab > **Magnetic Analysis** panel > **Inductance** measurement).

Procedure

1. Double-tap an **Inductance** measurement badge.

The Measurement configuration menu is displayed.

2. Tap the **Inductance** button.
   The Inductance Curve plot is displayed in a separate Plot view.
3. Double-tap in the Plot view to open the configuration menu for that plot.

Display an I vs (integral of) V plot for I vs V measurement

Use this procedure to display an I vs. ∫V plot for I vs. ∫V measurement.

Before you begin

To display an I vs. ∫V plot, you must be taking a I vs. ∫V measurement (**Add New**... **Measurement** > **Power** tab > **Magnetic Analysis** > **I vs.** **∫V** measurement).

Procedure

1. Double-tap the **I vs.** **∫V** measurement badge to open the configuration menu.
2. Tap **I vs.** **∫V** to add a new I vs **I vs.** **∫V** Plot view to the screen.
3. Double-tap in the Plot view to open a configuration menu for that plot.

Display an FFT math waveform

Use this procedure to display an FFT math waveform.

About this task

The FFT process mathematically converts the standard time-domain signal (repetitive or single-shot acquisition) into its frequency components. The FFT function processes the waveform record and displays the FFT frequency domain record, which contains the input signal frequency components from DC (0 Hz) to ½ the sample rate (also called the Nyquist frequency).

Procedure

1. Tap **Add New Math** to create a math waveform and open the Math configuration menu.
2. Tap **Source** and select the signal source from the list.
3. Set Math Type to **FFT**.
   The FFT of the waveform is displayed in an FFT Math waveform view.
4. Double-tap on the FFT waveform display to open a configuration menu to further refine the FFT display. See *Math FFT plot configuration menu (Math waveform)* on page 335.

Add Mask Testing to an Eye Diagram plot

Use this procedure to add mask test segments to an eye diagram plot and display the mask segment hits information in the Results bar.

Before you begin

Prerequisites:

An eye diagram plot is on the screen.

A mask configuration file (.msk) that matches the time scale (unit interval) and vertical settings of the eye diagram.

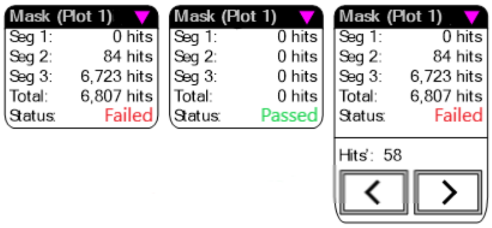
Procedure

1. Double-tap in the eye diagram plot to open the **Eye Diagram** configuration menu.
2. Tap the **Mask** panel.

3. Use the **Mask** panel fields to select and load a mask configuration file and set the bit type for testing. See *Eye Diagram plot configuration menu* on page 333.

   Note: The mask configuration file that you load must match the time scale (unit interval) and vertical settings of the active eye diagram.

   This step also adds a **Mask** badge to the Results bar.

   The **Mask** badge shows the number of segments in the mask, the hits per segment, and the pass/fail status. You can expand the Mask badge when acquisition is stopped to show the Navigation buttons.

   Use the Navigation buttons to center the source waveform on the previous or next mask search mark where a mask hit occurred. In the case of mask hit groups (see the following text), the Navigation buttons center the source waveform on the beginning mask hit group search mark where the successive mask hits started.

   The mask hit search marks on the source waveform are not precise due to the interpolated waveform used to create the eye diagram.

   When a set of successive waveform sample points (for example, 50 sequential sample points) touch a mask, then the whole series of successive sample points is considered as one mask hit group. The successive mask hit groups are marked with a beginning and ending search mark, instead of individual search marks for each hit.
4. Tap the **Mask Test** button to set it to **On** and display the mask segments in the eye diagram.
5. Tap outside of the menu to close it.
6. To set the number of hits required to fail a mask test, double-tap the **Mask** badge in the Results bar to open the **Mask Hits** menu and set the hit fail value. See *Mask Badge configuration menu* on page 327.

Display a Harmonics Bar Graph plot for IMDA Harmonics measurement

Use this procedure to display a bar graph.

Before you begin

To display a harmonics bar graph, you must be taking a Harmonics measurement (**Add New**... **Measurement** > **IMDA** tab > **Electrical Analysis** panel > **Harmonics** measurement)

Procedure

1. Double-tap a **Harmonics** measurement badge.
   The Measurement configuration menu is displayed.
2. Tap the **Bar graph** button.
   The Bar graph is displayed in a separate Plot view.
3. Double-tap in the Plot view to open the configuration menu for that plot.

Display a Phasor Diagram for DQ0 measurement

Use this procedure to display a phasor diagram plot for DQ0 measurement.

Before you begin

To display a phasor diagram plot, you must be adding a DQ0 measurement (**Add New**...**Measurement** > **IMDA** tab > **Electrical Analysis** > **DQ0** measurement).

Procedure

1. Double-tap an **DQ0** measurement badge.
   The Measurement configuration menu is displayed.
2. Tap the **Phasor Diagram** button.
   The Phasor Diagram plot is displayed in a separate Plot view.
3. Double-tap in the Plot view to open the configuration menu for that plot.

Display a Phasor Diagram for Power Quality measurement

Use this procedure to display a phasor diagram plot.

Before you begin

To display a phasor diagram plot, you must be taking an Power Quality measurement (**Add New**...**Measurement** > **IMDA** tab > **Electrical Analysis** panel > **Power Quality** measurement).

Procedure

1. Double-tap an **Power Quality** measurement badge.
   The Measurement configuration menu is displayed.
2. Tap the **Phasor Diagram** button.
   The Phasor Diagram plot is displayed in a separate Plot view.
3. Double-tap in the Plot view to open the configuration menu for that plot.

Using Default Setup

Use Default Setup to restore TekScope settings to their factory defaults.

Select **File** > **Default Setup** to restore default settings.

Add a callout to a view

Tap the **Add New... Callout** button to add callouts at the default position, but you can relocate it later.

Procedure

1. Tap the **Add New... Callout** button on the upper right of the results bar. The oscilloscope adds a text placeholder with default text as *Double tap to edit* at the center of the screen. However, you can move it around to place it at your choice.
2. The default callout field wraps text at about 15 characters. Use a mouse to select the callout and select any point from the available 8 touch points to resize the callout field to a larger size, to autowrap the callout text for the longer callout.

3. Double-tap or Double-click on the callout text top to open the callout Settings configuration menu.

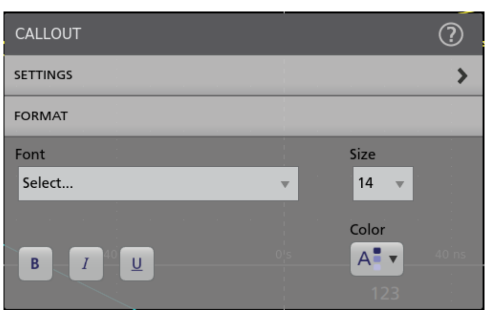
4. Select the type of callout from the drop-down list. The types of callout available are as follow:

Callout Type	Description
Note	Adds a callout as a text.
Arrow	Adds a callout text with an arrow.
Rectangle	Adds a callout text with an arrow pointing to a rectangle. The rectangle can be dragged to any position.
Bookmark	Adds a callout text with a bookmark added for either Waveform view (ch, math, ref), Spectrum view (freq domain traces, rf vs time domain traces), or Time Trends. Specify the source channel from the drop-down list and X-position of the callout.

5. Tap in the **Text** field and use a keyboard to enter the callout text, or double-tap in the Text field and use the on-screen keyboard to enter the callout text.

   Note: You can enter only one row of text in the Text field of the Text Settings configuration menu (using an attached keyboard). You can enter multiple rows using the on-screen keyboard entry field.

6. If you used the on-screen keyboard to enter text, tap the **Enter** button on the keyboard to close the keyboard and show the callout on the screen.
7. Use the format menu controls to set font type, size, color and other characteristics. See *Text Settings configuration* for details.



8. Tap outside the callout Settings configuration menu to close it.
9. To move a callout, touch and drag the callout text to a new position.

Delete a Callout

Tap and hold on a callout text to open a menu to delete that Callout.

Procedure

1. Touch and hold on the callout text you want to delete (or right-click with the mouse). The oscilloscope opens the right-click menu.
2. Select **Delete.** The callout is deleted immediately.

Add a serial bus to the Waveform view

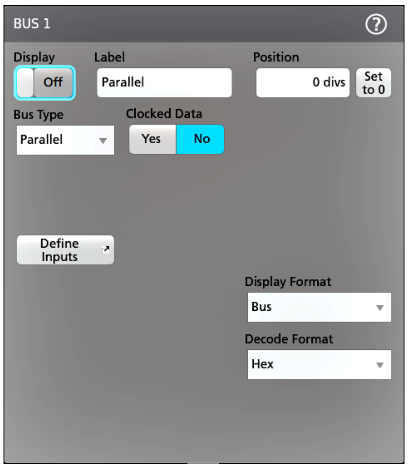
Use this procedure to add a serial bus to the Waveform view.

This instrument supports decoding parallel buses (standard with the instrument) and several serial buses options. All serial bus functions are options that must be purchased and installed before they are available in the instrument menus.

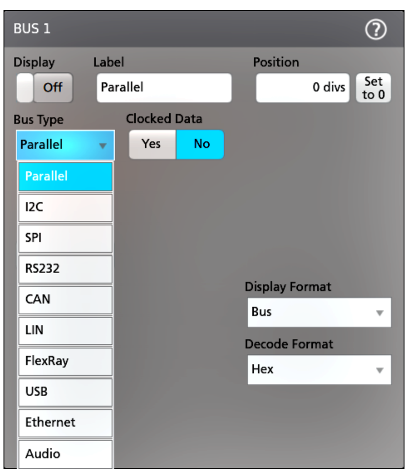
Use a Bus configuration menu to define a bus from which to acquire, decode, and display data.

Procedure

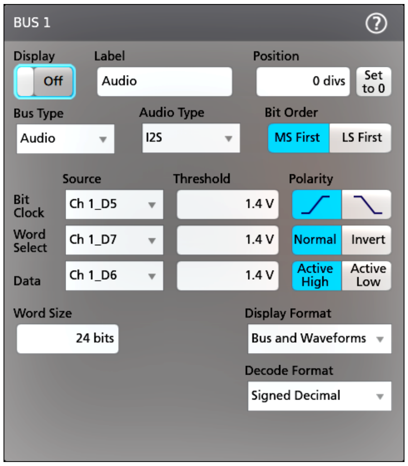
1. Tap the **Add New Bus** button on the Settings bar to add a Bus badge to the Settings bar, add a bus waveform to the screen, and open the Bus configuration menu. The default bus type is parallel.



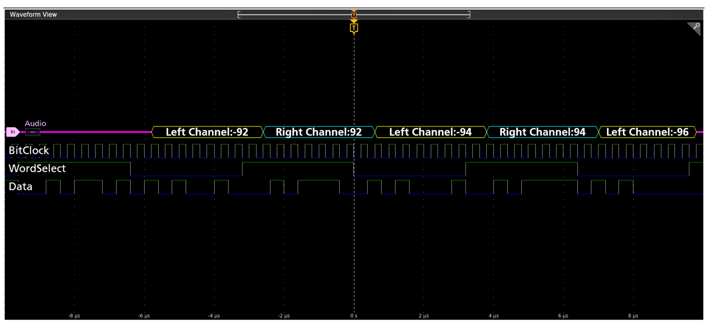
2. Tap **Bus Type** and select the bus type from the drop-down list.



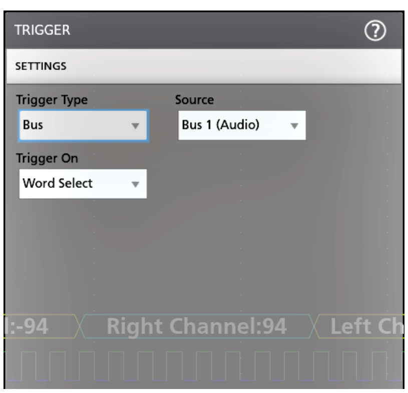
3. Use the fields and controls to select the bus signal sources, thresholds, other parameters, and the output format. The following image shows the settings for an Audio I2S serial bus.



The decoded bus is updated on the screen as you make changes to the settings.



4. Tap outside of the Bus configuration menu to close it.
5. Double-tap the **Trigger** badge and use the Trigger configuration menu to trigger on a specific condition in the bus.



6. For more information on serial bus settings, tap the Help button on the Bus configuration menu.

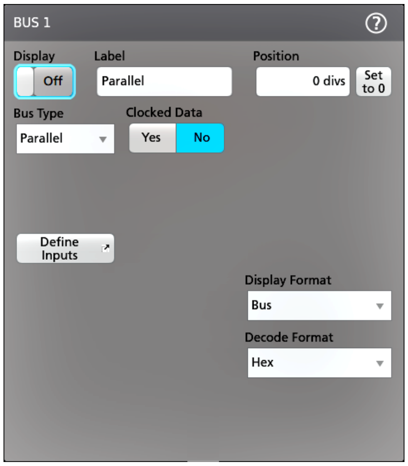
Add a parallel bus to the Waveform view

Use this procedure to add a parallel bus to the Waveform view.

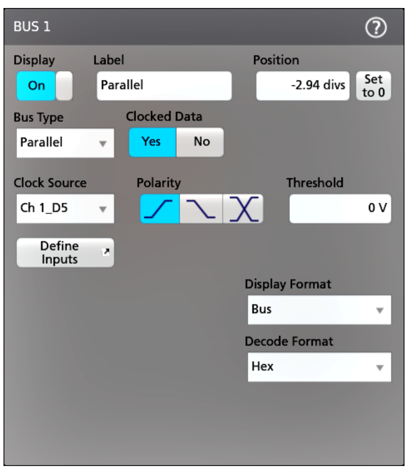
When you acquire data from a Parallel bus, you can set up the bus to be clocked or unclocked. If the bus is not clocked, the instrument acquires all data from the parallel bus at the sample rate of the instrument.

Procedure

1. Tap the **Add New Bus** button on the Settings bar. This adds a Bus badge bus to the Settings bar, adds a bus waveform to the screen, and opens the Bus configuration menu. The default bus type is parallel.

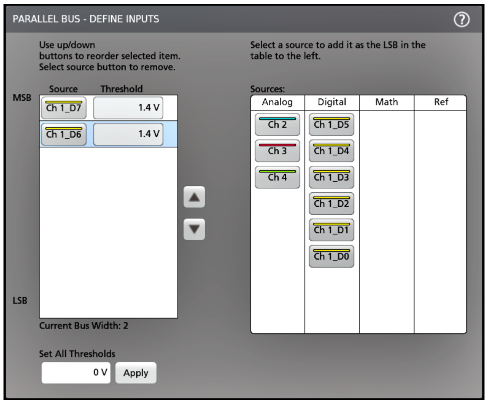


2. If setting up a clocked bus:

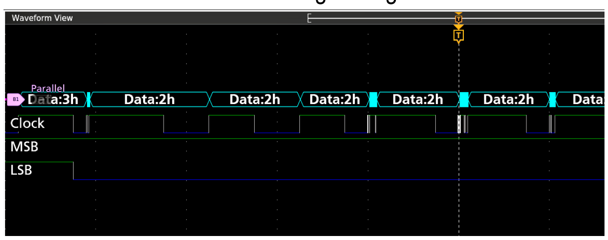


   a) Set Clocked Data to **Yes**.
   b) Tap the **Clock Source** field and select the source for the parallel bus clock signal.
   c) Tap the **Polarity** and **Threshold** controls and set the clock signal transition to detect and threshold level, respectively.

3. Tap **Define Inputs** and select the signal sources for the parallel bus. Signal sources can be analog, digital, math, or reference. Tap a signal in the Sources list to add it to the bus list on the left.



The bus waveform updates as you make changes on the configuration menu. Tap the + symbol next to the waveform handle to turn on and off showing the signals associated with the bus waveform.



4. Use the rest of the fields and controls in the configuration menu to set up the parallel bus parameters (label, position, display and decode formats).
5. Tap outside of the Bus configuration menu to close it.
6. To get a stable triggered waveform, double-tap the **Trigger** badge, set the Trigger Type to **Bus**, select the bus Source to the parallel bus you just set up, and enter the data condition on which to trigger in the **Data** field.
7. For information on parallel bus menu settings, tap the Help button on the Bus configuration menu.

Saving and recalling information remotely

Use these procedures to save or recall waveforms, setups, or sessions.

Save a screen image

Use this procedure to save a screen image.

Procedure

1. Tap the **File** menu and select **Save As**.
   The Save As configuration menu opens.
2. Tap **Screen Capture** to open the Screen Capture tab.
3. Tap **Save Location** or **Browse** to select the location to save the file.
   a) Tap the drop down arrow in the **Save Location** field and select the location to save the file from a list of recent save locations. Or double-tap the field and use the virtual keyboard to enter a path to the save location.
   b) Tap **Browse** to open the *Browse Save As Location* configuration menu to navigate to and select a location at which to save the file.
4. **File Name** shows the name last used to save a file. The default name is Tek000. To change the file name, double-tap the file name and enter a new file name using the virtual keyboard.
5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.
   - If Auto Increment File Name is enabled the **Count** defaults to 000 if there are no files at the specified location and file name that already use incremented file names. If there are files at the save location that already use the specified file name, and have already been saved using count increments, the Count field shows the next count value that will be added to the file name when the file is saved.
   - To change the starting count value, tap the Count field and use the assigned knob to change the value, or double-tap the field and use the virtual keypad to change the value.
6. Tap **Save As Type** and select the desired graphic image file type from the list.
7. Tap **Save** to save the screen image to the specified file name, location, and type.

Note: Once you have saved a file using the Save As configuration menu, you can push the front-panel **User** button to immediately save the same type file again, without opening any menus.

Save a waveform or spectrum trace to a file

Use this procedure to save channel waveform (analog or digital) data to a comma-separated values (csv) or Tektronix waveform data (wfm) file, for later analysis or inclusion in reports. You can also save Spectrum View traces to files.

1. Tap the **File** menu and select **Save As**.

   The Save As configuration menu opens.
2. Tap **Waveform** to open the Waveform tab.
3. Tap **Save Location** or **Browse** to select the location to save the file.
   a. Tap the **Save Location** drop down arrow and select the location to save the file from a list of recent save locations. Or double-tap the field and use the virtual keyboard to enter a path to the save location.
   b. Tap **Browse** to open the Browse Save As Location configuration menu to navigate to and select a location at which to save the file.
4. **File Name** shows the name last used to save a file. The default name is Tek000. To change the file name, double-tap the file name and enter a new file name using the virtual keyboard.

5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.
   - If Auto Increment File Name is enabled the **Count** defaults to 000 if there are no files at the specified location and file name that already use incremented file names. If there are files at the save location that already use the specified file name, and have already been saved using count increments, the Count field shows the next count value that will be added to the file name when the file is saved.
   - To change the starting count value, tap the Count field and use the assigned knob to change the value, or double-tap the field and use the virtual keypad to change the value.
6. Tap **Save As Type** and select the desired waveform data type from the list.
7. Tap **Source** and select the source of the waveform to save from the list. You can save a single waveform or all waveforms.

The **Source** menu shows a right arrow in the source list if a source has more than one waveform available to save, such as an analog channel and its corresponding spectrum trace(s). Tap the source menu item to expand that item to show available waveform sources.

Save instrument settings to a file

Use this procedure to save instrument settings to a Tektronix setup (.set) file.

Procedure

1. Tap the **File** menu and select **Save As**.
   The Save As configuration menu opens.
2. Tap **Setup** to open the Setup tab.
3. Tap **Save Location** or **Browse** to select the location to save the file.
   a) Tap the **Save Location** drop down arrow and select the location to save the file from a list of recent save locations. Or double-tap the field and use the virtual keyboard to enter a path to the save location.
   b) Tap **Browse** to open the *Browse Save As Location* configuration menu to navigate to and select a location at which to save the file.
4. **File Name** shows the name last used to save a file. The default name is Tek000. To change the file name, double-tap the file name and enter a new file name using the virtual keyboard.
5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.
   - If Auto Increment File Name is enabled the **Count** defaults to 000 if there are no files at the specified location and file name that already use incremented file names. If there are files at the save location that already use the specified file name, and have already been saved using count increments, the Count field shows the next count value that will be added to the file name when the file is saved.
   - To change the starting count value, tap the Count field and use the assigned knob to change the value, or double-tap the field and use the virtual keypad to change the value.
6. Enable **Include Reference Waveforms** to include waveform files for all active waveforms in the setup file.
7. Tap **Save** to save the setup information to the specified file name and location.

⚠ **Note:** Once you have saved a file using the Save As configuration menu, you can push the front-panel **User** button to immediately save the same type file again, without opening any menus.

Save reports

Use this procedure to save reports.

1. Tap the **File** menu and select **Save As**.
2. The Save As configuration menu opens.
3. Tap **Report** to open the Report tab.
    a. Tap the **Save Location** drop down arrow and select the location to save the file from a list of recent save locations. Or double-tap the field and use the virtual keyboard to enter a path to the save location.
    b. Tap **Browse** to open the Browse Save As Location configuration menu to navigate to and select a location at which to save the file.
4. **File Name** shows the name last used to save a file. The default name is Tek000. To change the file name, double-tap the file name and enter a new file name using the virtual keyboard.
5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.
    - If Auto Increment File Name is enabled the **Count** defaults to 000 if there are no files at the specified location and file name that already use incremented file names. If there are files at the save location that already use the specified file name, and have already been saved using count increments, the Count field shows the next count value that will be added to the file name when the file is save.
    - To change the starting count value, tap the Count field and use the assigned knob to change the value, or double-tap the field and use the virtual keypad to change the value.
6. Tap **Save As Type** and select the report file format from the list.
7. Tap **Append Report** to enable appending this report to a previous report that uses the same file name.
8. Double-tap **Comments** and use the virtual keyboard to add descriptive comments to the report.
9. Tap **Include Images and Annotations** to include screen images and annotations in the report.
10. Tap **Include Setup Configuration** to include the instrument settings data in the report.
11. Tap **Save** to save the report file to the specified file name, location, and type.

Note: Once you have saved a file using the Save As configuration menu, you can push the front-panel **User** button to immediately save the same type file again, without opening any menus.

Save sessions

Use this procedure to save a session file. A session file contains instrument setup information, measurements, reference waveforms of active signals, plots, and eye diagram masks if enabled.

Procedure

1. Tap the **File** menu and select **Save As**.
   The Save As configuration menu opens.
2. Tap **Session** to open the Session tab.
3. Tap **Save Location** or **Browse** to select the location to save the file.
    a) Tap the **Save Location** drop down arrow and select the location to save the file from a list of recent save locations. Or double-tap the field and use the virtual keyboard to enter a path to the save location.
    b) Tap **Browse** to open the *Browse Save As Location* configuration menu to navigate to and select a location at which to save the file.
4. **File Name** shows the name last used to save a file. The default name is Tek000. To change the file name, double-tap the file name and enter a new file name using the virtual keyboard.
5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.
    - If Auto Increment File Name is enabled the **Count** defaults to 000 if there are no files at the specified location and file name that already use incremented file names. If there are files at the save location that already use the

specified file name, and have already been saved using count increments, the Count field shows the next count value that will be added to the file name when the file is saved.

- To change the starting count value, tap the Count field and use the assigned knob to change the value, or double-tap the field and use the virtual keypad to change the value.

6. Tap **Save** to save the session data to the specified file name, location, and type.

⚠ **Note:** Once you have saved a file using the Save As configuration menu, you can push the front-panel **User** button to immediately save the same type file again, without opening any menus.

Recall a reference waveform or spectrum trace

Use this procedure to recall (load) and display a saved waveform or spectrum trace as a Reference waveform. There is no set limit to the number of reference waveforms that you can load and display.

Procedure

1. Tap the **Add New Ref** button on the Settings bar.
   The *Recall configuration menu* opens to the last location used to recall a waveform or spectrum trace file. If the location is not available, the menu defaults to the instrument C drive.
2. Tap **Files of Type:** and select the file type from the list.
   The files list updates to show all files that match the selected file type.
3. Tap the file name in the list to add the file to the **File Name** field.
4. Tap **Open** to exit the menu and add the reference waveform to the screen.
5. If the file location shown in the **Look In** field does not contain the file to recall, navigate to the folder that contains the file to recall, using one of the following methods:
   - To recall files from recently accessed file locations, tap the drop down arrow in the **Look in** field and select from the list of recently accessed locations.
   - Tap in the **Drive** column and use the displayed contents to navigate to locations on the internal storage location C, a connected USB drive (E -K), or a network-connected drive.
   - Double-tap the **Look in** field and use the virtual keyboard to manually enter a path to the file location (folder).

   As you navigate the folders, the files list area shows all files that match the file type selected in the Files of Type field.
6. Select the file to recall, using one of the following methods:
   - If the file was recently recalled, tap the drop down arrow in the **File Name:** field and select from a drop-down list of recently recalled files.
   - Select a file name in the main files list. The file **Name** field updates to show the selected file name.

   ⚠ **Note:** You can double-tap on a file name to immediately recall the file and close the menu.
7. Tap **Recall**.

   The reference waveform is loaded and displayed, and a **Ref** badge is added to the Settings bar. If you recalled a spectrum trace waveform, that waveform is shown in the **Spectrum View** window, with the handle name of R1, R2, and so on.

Recall a Setup file

Use this procedure to recall (load) and configure instrument settings from a Setup file.

1. Select **File** > **Recall** from the Menu bar to open the Recall configuration menu.
2. Tap **Setup** to open the Setup tab.

   The Recall configuration menu opens to show the settings last used to recall a setup file.

3. Tap **Files of Type**: and select the file type from the list.

   The files list updates to show any files that match the selected file type.
4. Navigate to the folder that contains the file to recall, using one of the following methods:
   - To recall files from recently accessed file locations, tap the drop down arrow in the **Look in** field and select from the list of recently accessed locations.
   - Tap in the **Drive** column and use the displayed contents to navigate to locations on the internal storage location C, or on a connected USB drive (E -K).
   - Double-tap the **Look in** field and use the virtual keyboard to manually enter a path to the file location (folder).

   As you navigate the folders, the files list area shows all files that match the file type selected in the Files of Type field.
5. Select the file to recall, using one of the following methods:
   - If the file was recently recalled, tap the drop down arrow in the **File Name:** field and select from a drop-down list of recently recalled files.
   - Select a file name in the main files list. The File Name field updates to show the selected file name.

   ⚠ **Note:** You can double-tap on a file name to immediately recall the file and close the menu.
6. Tap **Recall**.

   The instrument loads the setup file and reconfigures the oscilloscope to the setup file settings.

Recall a Session file

Use this procedure to recall (load) instrument settings and associated waveforms (as Reference waveforms) from a Session file.

Procedure

1. Select **File > Recall** from the Menu bar to open the *Recall configuration* menu.
2. Tap **Session** to open the Session tab.
   The Recall configuration menu opens to show the settings last used to recall a session file.
3. Tap **Files of Type:** and select the file type from the list.
   The files list updates to show any files that match the selected file type.
4. Navigate to the folder that contains the file to recall, using one of the following methods:
   - To recall files from recently accessed file locations, tap the drop down arrow in the **Look in** field and select from the list of recently accessed locations.
   - Tap in the **Drive** column and use the displayed contents to navigate to locations on the internal storage location C, or on a connected USB drive (E -K).
   - Double-tap the **Look in** field and use the virtual keyboard to manually enter a path to the file location (folder).

   As you navigate the folders, the files list area shows all files that match the file type selected in the Files of Type field.
5. Select the file to recall, using one of the following methods:
   - If the file was recently recalled, tap the drop down arrow in the **File Name:** field and select from a drop-down list of recently recalled files.
   - Select a file name in the main files list. The File Name field updates to show the selected file name.

   ⚠ **Note:** You can double-tap on a file name to immediately recall the file and close the menu.
6. Tap **Recall**.
   The instrument loads the session file and reconfigures the oscilloscope to the session file settings.

Using Spectrum View

Spectrum View provides a time-domain view of the frequency components of a signal.

Spectrum View concepts

Typical oscilloscope displays show electrical signals as a graph of time on the x-axis versus amplitude on the y-axis (time domain). A spectrum trace shows the same electrical signals as a graph of frequency on the x-axis versus amplitude on the y-axis (frequency domain).

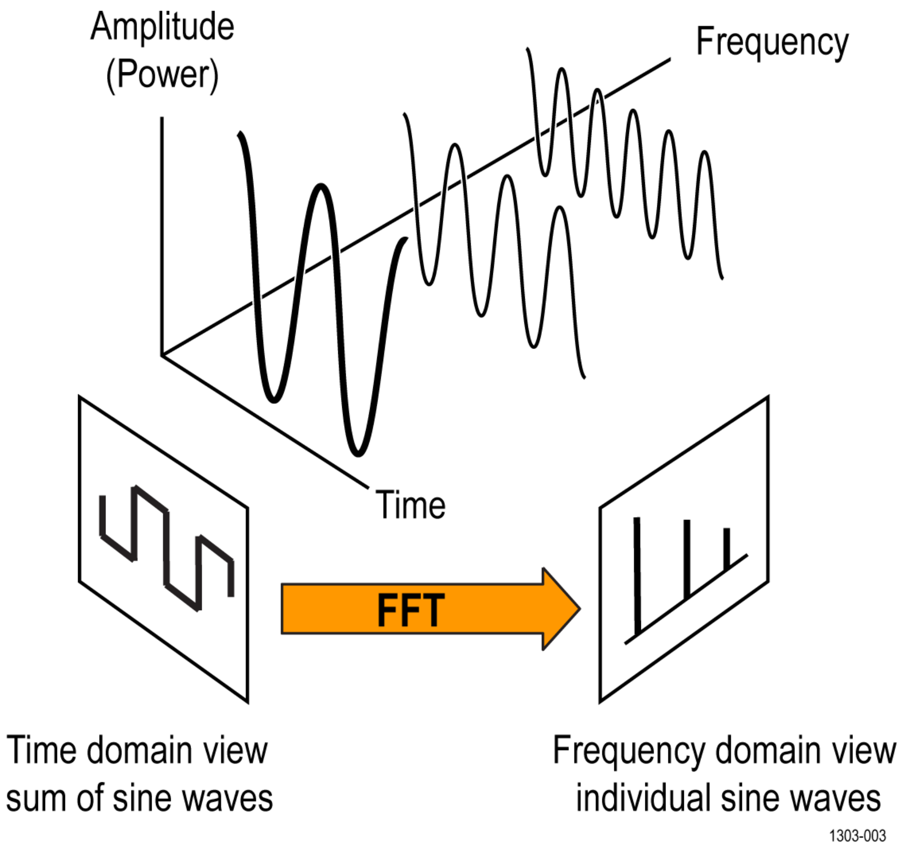


Figure 11: The time and frequency domain components of a waveform.

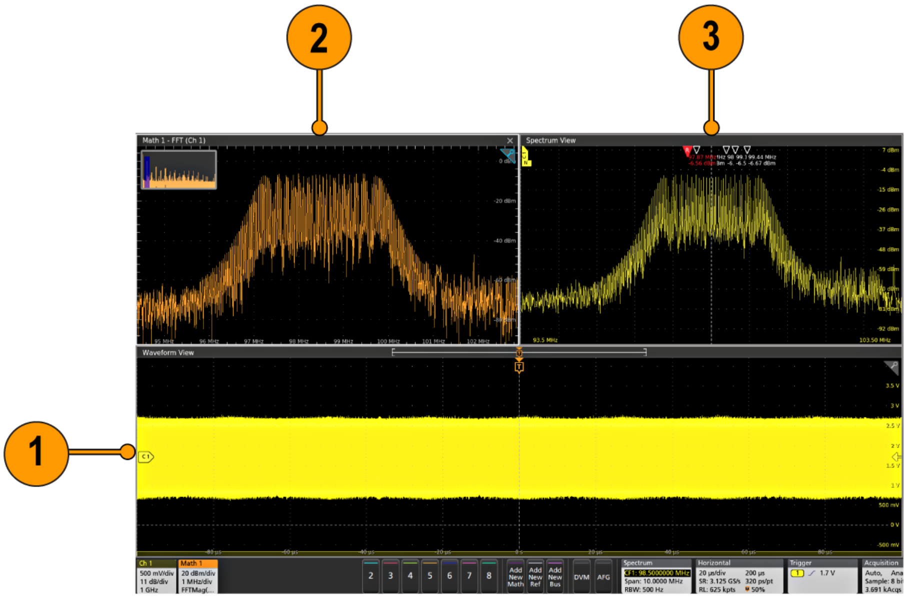
All time-domain signals can be defined as a composite of discrete sine waves, each with their own frequency, magnitude, and phase. The frequency-domain waveform (or spectrum trace) is a Fast Fourier Transform (FFT) decomposition of the time-domain signal into its constituent sine wave frequency components.

There are several advantages to examining signals in the frequency domain:

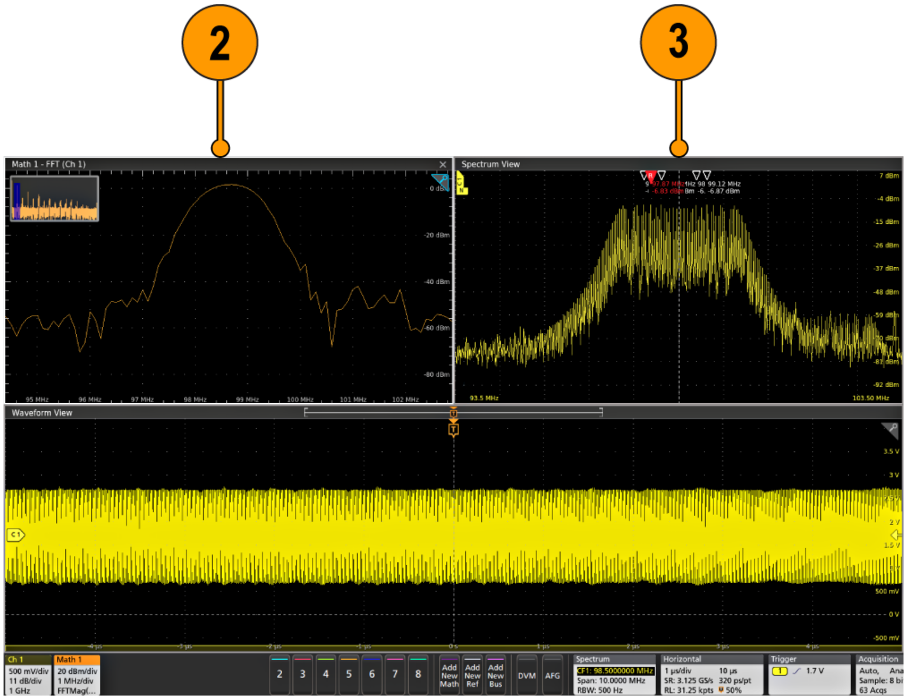
- It is easier to analyze signal noise characteristics. In the time domain, noise can appear as a fuzzy waveform, but in the frequency domain you can determine if it is broadband, random noise, or perhaps cross-talk from another signal on your DUT.
- It is easier to analyze and measure the distortions on signals. Although a 1% distortion on a sine wave may not be discernable in the time domain, it will be very obvious in the frequency domain as harmonics of the signal.
- Wireless communication measurements are described in the frequency domain in terms of frequency bands and channels. A spectrum trace lets you focus analysis on the relevant frequency range and make specialized measurements such as occupied bandwidth and modulation quality.

FFT and Spectrum View

Most modern oscilloscopes provide a Math FFT function to display the frequency-domain spectrum of a signal. The typical approach to creating an FFT is to use the same data acquisition used for the time-domain waveform (1), process it through an FFT, and display that data in a separate area or window (2).



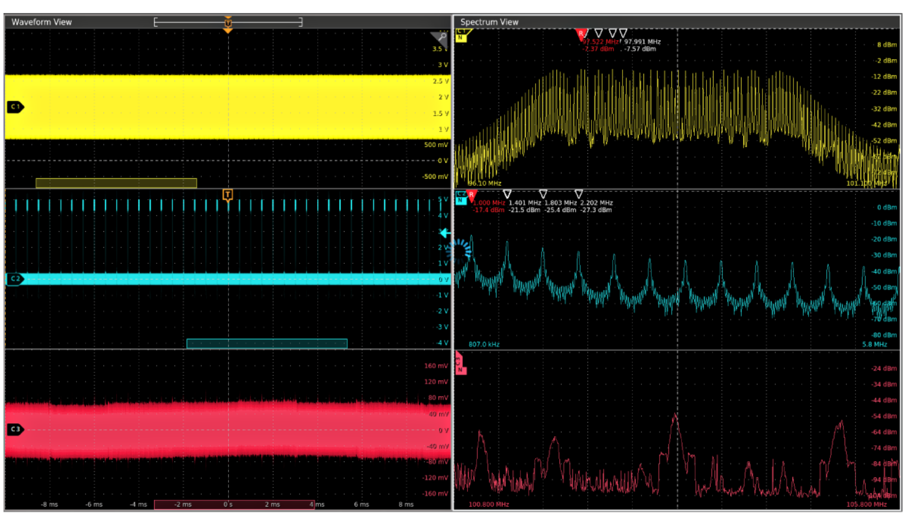
The disadvantage of a standard **Math FFT** is that the instrument uses the same acquisition data for both the time- and frequency-domain views. Therefore, the instrument settings for sample rate and record length affect both the time-domain and **Math FFT** acquisitions. Displaying an optimized FFT trace often involves manual calculations of the time-domain parameters like sample rate and record length. This also means that the instrument settings often cannot be optimized to show relevant information in both the waveform and **Math FFT** views at the same time. The following image shows a change in the Horizontal Scale setting that quickly changes the **Math FFT** plot.



1303-012

The **Spectrum View** feature in 4/5/6 Series instruments solves this problem by using a separate hardware path in the acquisition engine of each channel to acquire the frequency-domain data in parallel with the time-domain data. This frequency-domain data is processed, transformed, and displayed in a separate **Spectrum View** window (3) that is optimized to display and adjust spectrum traces independently of the time-domain waveform. Notice that the **Spectrum View** trace has not changed in the above images with changes to the time-domain signal, but the **Math FFT** signal is not usable for measurements in the second image.

Another key feature of **Spectrum View** is that you can display a spectrum trace for each **FlexChannel** input, along with their corresponding time-domain waveforms. The **Spectrum View** user interface is optimized to display and control frequency-domain parameters such as reference level, center frequency, span, and resolution bandwidth. Changing settings when the **Spectrum View** window is active (selected) does not change settings of the associated time-domain waveform in the **Waveform View**.



1303-016

Time Domain Traces concepts

Time Domain Traces let you view and analyze Magnitude, Frequency, and/or Phase versus time traces for each Spectrum View trace.

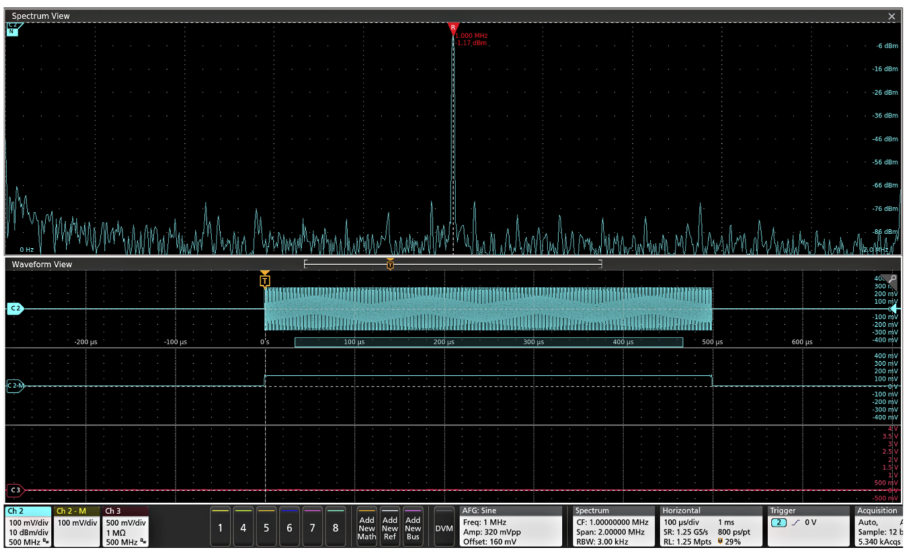
Overview

Time Domain Traces display information collected from Spectrum View alongside other time domain waveforms. This allows for analysis of spectral content as it changes over time, providing some features of a real-time spectrum analyzer. Because the Spectrum View controls are not tied to the horizontal settings of the oscilloscope, these traces can be examined in detail without sacrificing Span or RBW.

The data in these traces is derived from the IQ data of the acquired signal. This is the same source that is used to calculate the spectrum trace.

Magnitude Vs. Time

The **Magnitude Vs. Time** trace shows the magnitude (amplitude) of the Spectrum View signal span versus time. Typical uses include viewing and analyzing RF pulse power and transmission/channel power.



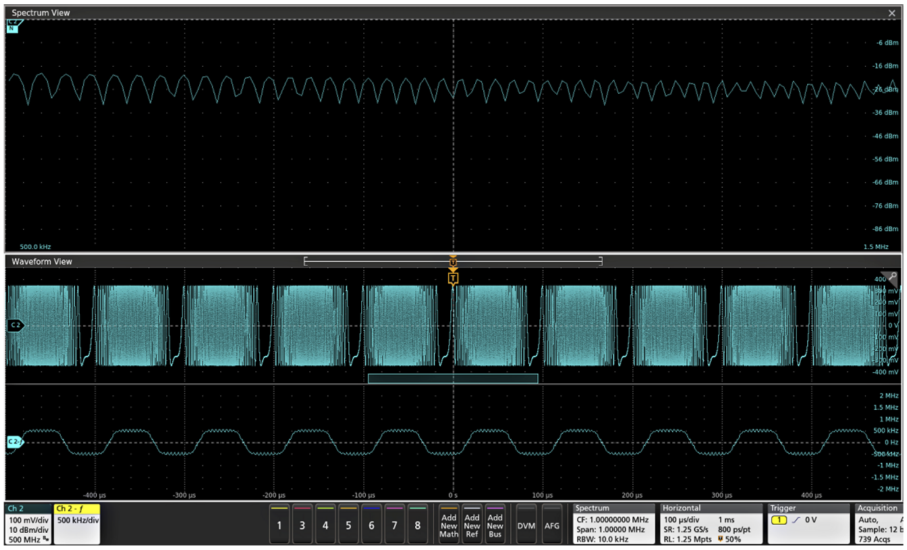
1303-017

This trace provides three different ways to calculate and present RF magnitude:

- Linear Power: Direct use of linear power
- Linear Amplitude: Square root of the linear power values
- Log Power (dB)

Frequency Vs. Time

The **Frequency Vs. Time** trace shows the dominant frequency of the Spectrum View signal and the change in frequency over time.

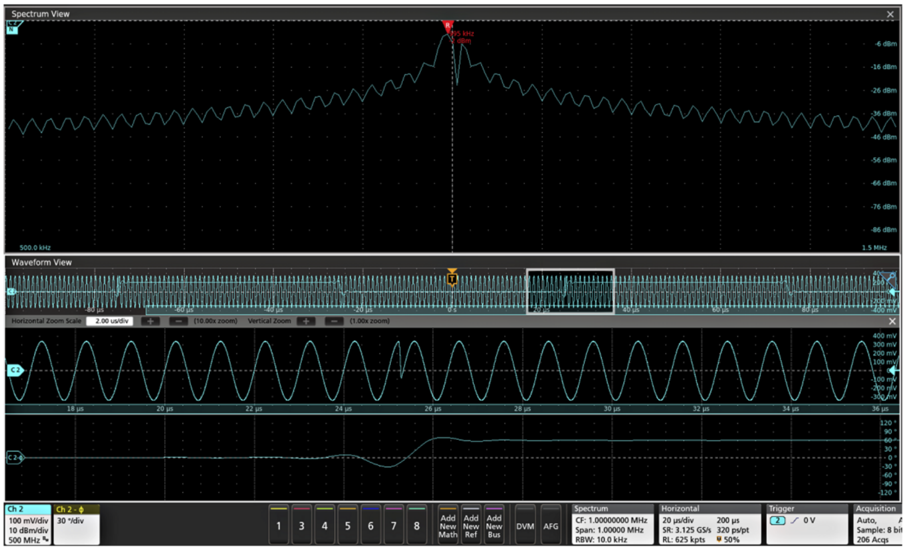


1303-018

This trace can help characterize frequency modulation, frequency drift, and brief changes in frequency that might not be obvious in the spectrum trace.

Phase Vs. Time

The **Phase Vs. Time** trace shows the change in the signal phase over time, where the instrument measures the incoming signal's dominant or carrier frequency to derive a reference signal to use as the phase reference.



1303-019

This trace can help characterize phase modulation and phase drift. Changes in phase are not visible in the spectrum trace, and can be difficult to locate in the time-domain signal.

Display a spectrum trace

Spectrum traces are initially set up from each channel's badge menu.

Procedure

1. Display the channel signal for which you want a spectrum trace.
2. Double-tap the channel badge.

3. Tap the **Spectrum View** panel.
4. Set **Display** to **On**. The instrument opens the **Spectrum View** window and adds a **Spectrum** badge to the Settings bar.
5. Select the **Frequency Domain Traces** to show in the **Spectrum View** (**Normal**, **Max Hold**, **Min Hold**, and/or **Average**).
6. Select the vertical **Units** (default is dBm).
7. Select which RF vs. Time waveforms to display.
8. Double-tap the **Spectrum** badge. See *Spectrum badge menu*.
9. Use the **Center Frequency**, **Span**, FFT **Window**, and **RBW Mode** controls to set these parameters to display the spectrum trace for the channel waveform. This menu also controls the **Spectrum Time Bar** (see *Spectrum View and Spectrum Time* on page 114).
10. To change the **Span**, use a horizontal pinch or expand motion on the touchscreen, or use the front-panel **Horizontal Scale** knob.

    Note: Setting Span sets the same span value across all spectrum traces, regardless of the center frequency. For example, the trace 1 center frequency is set to 40 MHz, and span is set to 20 MHz, so trace 1 shows frequencies from 30 MHz to 50 MHz. If you add a spectrum trace for channel 2 with a center frequency of 60 MHz, trace 2 uses the same 20 MHz to show frequencies from 50 MHz to 70 MHz.
11. To change the vertical **Scale**, use a vertical pinch or expand motion on the touchscreen, or use the front-panel **Vertical Scale** knob.
12. To add peak markers to the trace, double-tap in a spectrum slice to open the **Spectrum View** menu and tap the **Markers** panel. Use the controls to set the number of markers to display, the threshold level at which to detect peaks, and the excursion range required by the peaks. See *Spectrum View window configuration menu*

The Spectrum View user interface

A spectrum trace shows the magnitudes of the frequency components that make up the time-domain waveform. Use spectrum traces in the **Spectrum View** to view noise, distortion, and crosstalk information of time-domain signals.

The Spectrum View elements

Predominant frequencies (with greater magnitude) are shown as peaks in the view. The vertical graticule represents the signal magnitude, and the horizontal axis represents frequency, with the starting (lowest) frequency at the left side of the graticule.

Each spectrum trace can display up to four spectrum trace types for each source channel: **Normal**, **Max Hold**, **Min Hold**, and **Average**. You can turn each of these traces on and off independently. You can display all or some of them simultaneously.

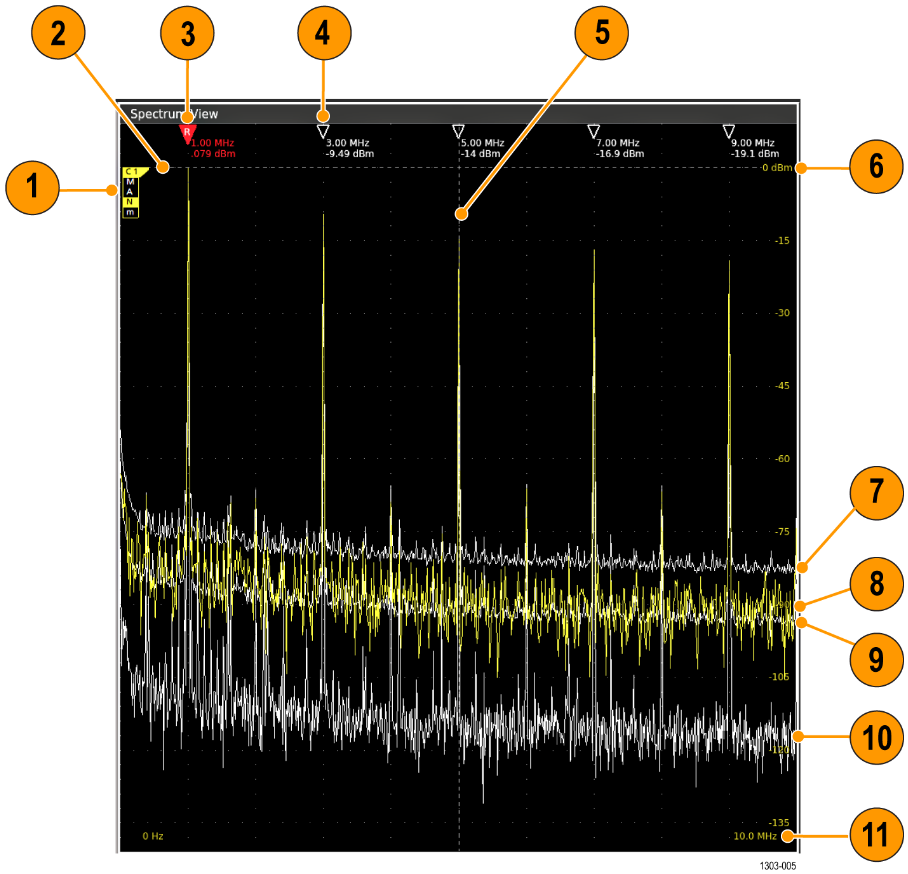


Figure 12: The Spectrum View window elements

1. The trace handle. See *Spectrum trace handle and trace types* on page 117.
2. The Reference level. The vertical scale (V/div) setting of the source time-domain waveform sets the spectrum trace reference level as follows:

Source vertical scale	Trace reference level	Source vertical scale	Trace reference level
500 µV/div	-42 dBm	100 mV/div	4 dBm
1 mV/div	-36 dBm	200 mV/div	10 dBm
2 mV/div	-30 dBm	500 mV/div	18 dBm
5 mV/div	-22 dBm	1 V/div	24 dBm
10 mV/div	-16 dBm	2 V/div	30 dBm
20 mV/div	-10 dBm	5 V/div	38 dBm
50 mV/div	-2 dBm	10 V/div	44 dBm

3. The Reference marker labels and measures the highest-magnitude peak of the visible spectrum trace.

4. The Automatic markers label and measure the adjacent next-highest-magnitude peaks of the spectrum trace. The frequency and magnitude readouts can be absolute values or values relative to the Reference marker. See *Spectrum peak markers* on page 118.
5. The **Center Frequency** is a specified frequency at the center of the spectrum display. For many measurements it is the frequency of interest to measure, such as a carrier frequency. It may or may not be the highest magnitude peak in the trace. Each trace can have a separate center frequency setting. Changing the horizontal position of a trace in the **Spectrum View** window changes the center frequency of that trace and repositions the trace as needed.
6. The Amplitude (magnitude) scale. The vertical scale setting of the spectrum trace (pinch/expand actions on the touch screen, or the Vertical Scale knob on the front panel) sets the scale values.
7. The **Max Hold** trace shows the maximum data values that are accumulated and displayed over multiple acquisitions of the Normal trace.
8. The **Normal** trace shows each acquisition as new data is acquired. This is the default trace mode. The trace is shown in the same color as the source channel.
9. **Average** trace shows data that is averaged over multiple acquisitions. This is true power averaging, which occurs before the log conversion. Each power of 2 averaging reduces the displayed noise by 3 dB.
10. **Min Hold** trace shows the minimum data values accumulated over a specified number of acquisitions.
11. The **Span** is the range of frequencies (lowest to highest) to show on the spectrum trace. All spectrum traces have the same span regardless of the center frequency setting.

Spectrum View guidelines

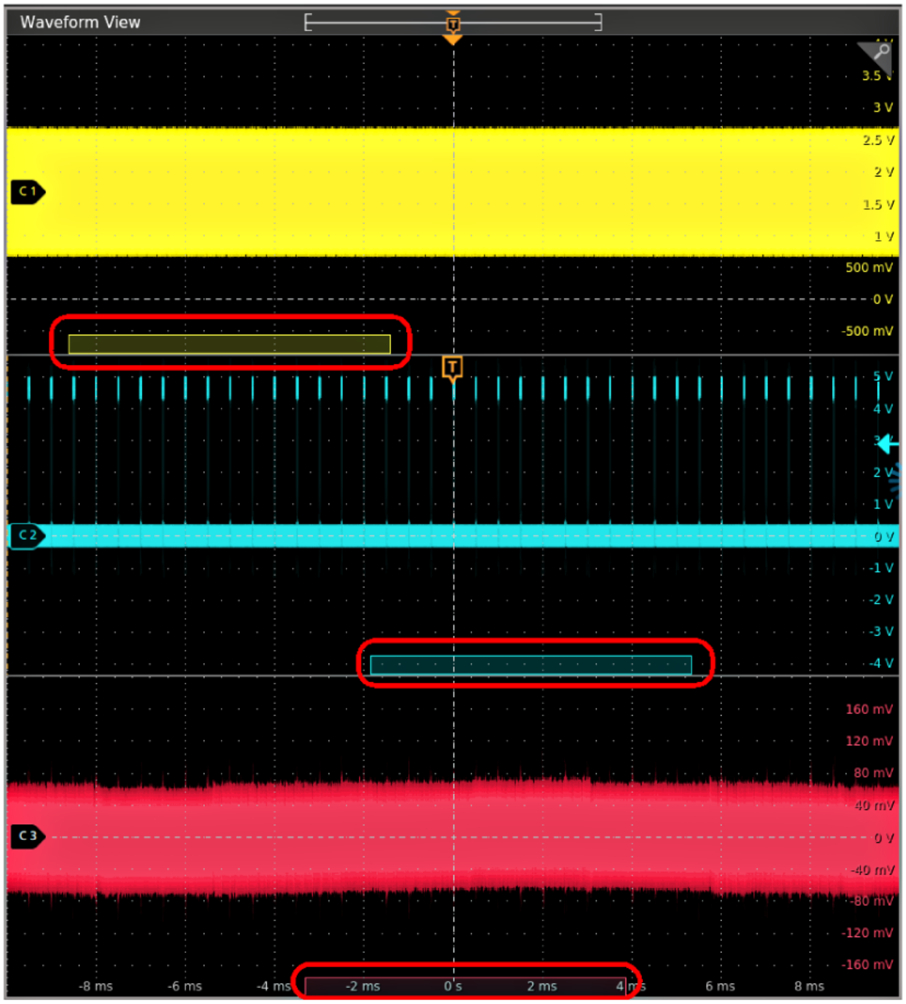
- The **Spectrum View** window supports stacked and overlay display modes. When in Overlay display mode:
  - The vertical and horizontal in-graticule labels apply to the currently selected channel.
  - The MANm spectrum trace handle (see *Spectrum trace handle and trace types* on page 117) for the currently selected channel is shown highlighted, just like with analog channels.
  - The MANm trace handle for non-selected traces are shown in the same fashion as non-selected analog channels.
- The **Spectrum View** does not support zoom.
- Horizontal pinch and expand touch gestures adjust the spectrum span in the same style as a horizontal pinch/expand in the **Waveform View**.
- A horizontal pan adjusts the center frequency. While panning, the pan motion will pause momentarily when the Reference Marker reaches the middle of the graticule. This provides a very simple Marker to Center functionality.
- Vertical pinch and expand touch gestures adjust the spectrum vertical scale in the same style as a vertical pinch/expand in the **Waveform View**.
- A vertical pan adjusts the vertical position. While panning, the pan motion will pause momentarily when the trace handle reference level reaches the top of the graticule. This provides a very simple reference level to top functionality.

Spectrum View and Spectrum Time

Spectrum Time controls the area in the time-domain acquisition used to create the **Spectrum View** trace. Conceptually like gating, use Spectrum Time to examine the spectrum of specific areas of interest in the acquisition record.

Spectrum Time is the part of the time-domain waveform acquisition used to create the **Spectrum View** trace for that channel.

Spectrum Time is shown by a shaded bar along the bottom of a Channel graticule in the **Waveform View**. This bar is called the **Spectrum Time Bar**. The Spectrum Time Bar visually indicates both the range and time position in the acquisition record used to create the spectrum trace.



1303-015

The Spectrum Time width (amount of the acquisition record) is determined by the FFT Window factor divided by Resolution Bandwidth (RBW). Changing the FFT window type and/or the RBW parameters changes the Spectrum time width.

The Spectrum Time position can be moved through the time domain acquisition using a variety of methods including tapping and dragging it, using the **Wave Inspector Pan** knob, or with the **Spectrum Time** position control in the **Spectrum** configuration menu.

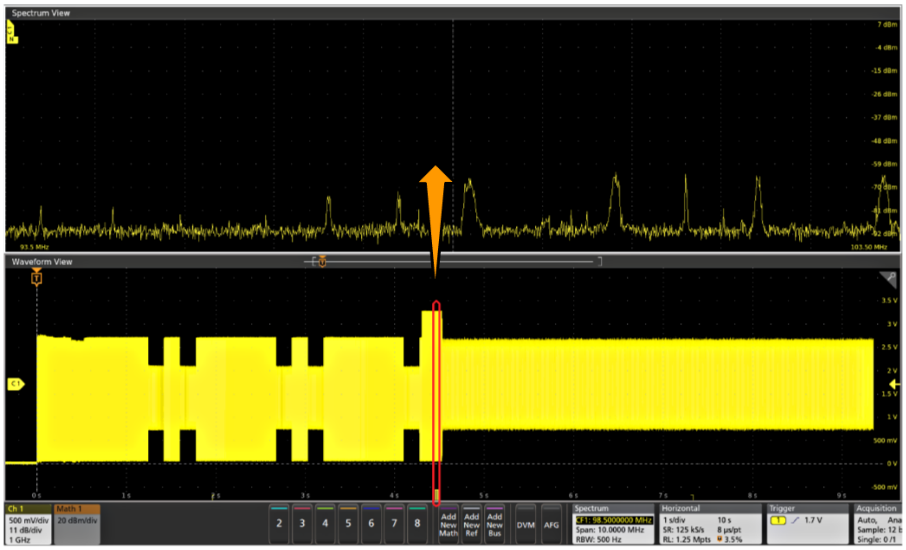
The FFT Window, **RBW**, and **Spectrum Time** settings are in the **Spectrum** menu (see *Spectrum badge menu*).

The Spectrum Time Bar is shown whenever a **Spectrum View** trace is enabled for a channel.

Example use of Spectrum Time to determine the time to a stable signal.

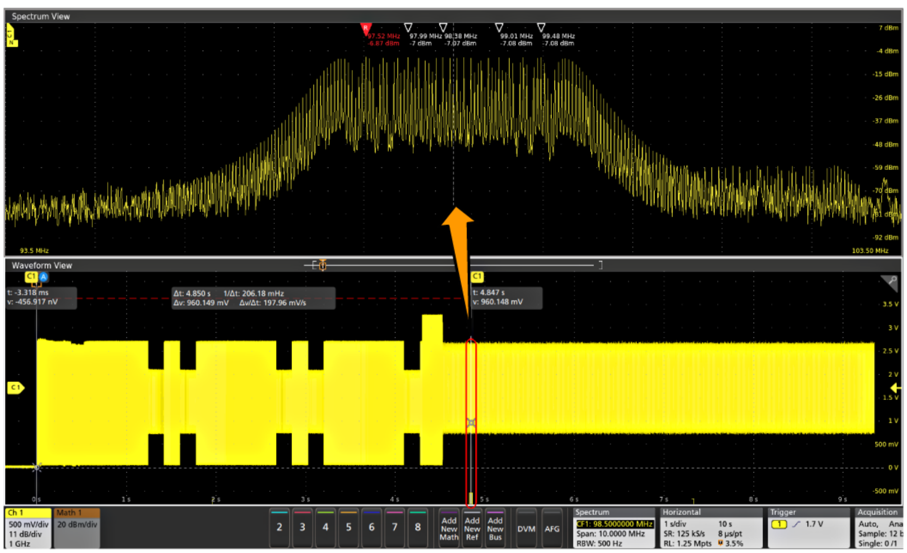
This example shows the use of Spectrum Time to determine the time for a spread spectrum (SS) clock to become stable after power-up. The clock signal is centered around 98.5 MHz.

An acquisition was taken with a long enough time setting to ensure that the SS clock signal had stabilized. The RBW was set to 500 Hz, and the Spectrum Time position was set to just before the SS clock signal became stable. See the following image.



1303-013

The Spectrum Time Bar position was slowly moved to the right until a valid SS clock signal was shown in the Spectrum view, as shown in the following image.



1303-014

Using cursors in the **Waveform View**, the measured time from power-up to when the SS clock signal is stable was 4.847 seconds.

Spectrum Time guidelines

- Spectrum Time can be as long as, but never longer than, the RF acquisition time.
- To position the Spectrum Time Bar to areas of interest, tap and drag the bar, or use the front-panel **Pan** knob to move the bar of the selected Waveform View channel.
- You can set the Spectrum Time Bars to lock together across all input channels, so that moving one moves all Spectrum Time bars. The Spectrum Time Bars can also be unlocked and moved to show the Spectrums of different portions of different channels.
- When **Spectrum Time** is not locked across all channels (**Lock Spectrum Time Across All Channels** is not selected), and zoom is **On**, the **Pan** front-panel knob controls the **Zoom Box** of the selected channel.

- When **Spectrum Time** is not locked across all channels (**Lock Spectrum Time Across All Channels** is not selected), and zoom is **Off**, the **Pan** front-panel knob controls the **Spectrum Time Bar** of the selected channel.

Spectrum trace handle and trace types

The spectrum trace handle provides details on the trace reference level, the source channel for the trace, and which trace types are displayed and selected.

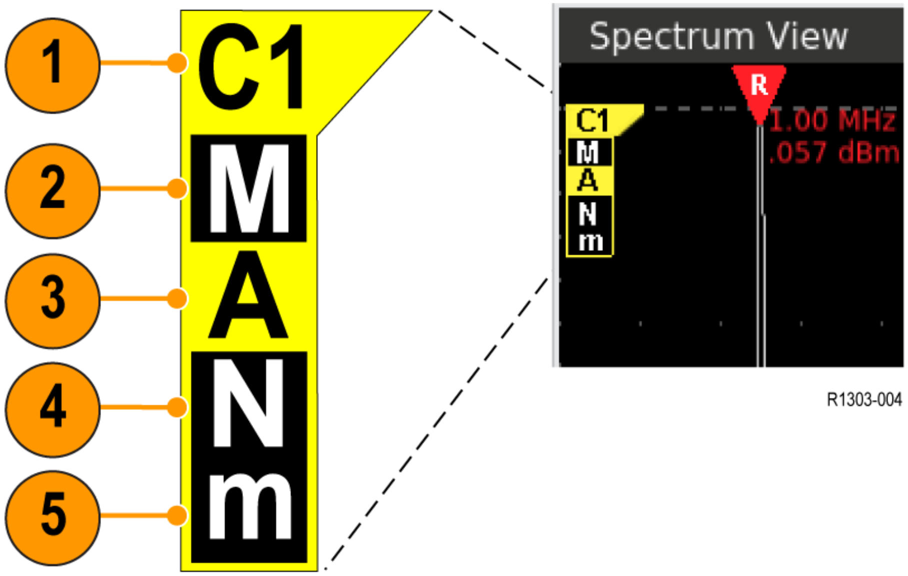


Figure 13: The Spectrum View trace handle elements

1. The trace's source channel (at top of the handle) and trace reference level (top edge of the handle, with dashed line extending across the screen). Spectrum View shows the spectrum traces relative to a Reference Level reference point. The color of the handle is the same as the source channel.

   If the Reference level is outside of the displayed trace (above or below the graticule), the handle is drawn with the channel source text pointing toward the Reference level.
2. A capital **M** indicates that the maximum trace is enabled. If the maximum trace is not enabled, this letter is not shown.
3. A capital **A** indicates that the average trace is enabled. If the average trace is not enabled, this letter is not shown.
4. A capital **N** indicates that the normal trace is enabled. If the normal trace is not enabled, this letter is not shown.
5. A small **m** indicates that the minimum trace is enabled. If the maximum trace is not enabled, this letter is not shown.

The above image shows all traces enabled and the **A** (**Average**) trace selected. Highlighting around a letter indicates that that trace type is selected.

Trace handle and trace type guidelines

There are important distinctions between enabled (displayed) and selected traces:

- An enabled trace is a trace type that is enabled in the **Spectrum View** panel of the source channel badge and is shown in the spectrum view window. If the trace type is not enabled, its corresponding letter is not shown in the trace handle and there is no corresponding trace shown on the screen.
- A selected trace (highlighted around the letter) is the trace that is used for marker and cursor readouts. Only one trace type can be selected at a time. Double-tap the trace handle to change the selected trace type.
- You can enable and display any combination of trace types for each spectrum trace.
- Disabling all trace types causes the spectrum view for that channel to close.
- If there is only one spectrum trace shown in the **Spectrum View**, and you disable all trace types or turn off the spectrum function for that channel, the **Spectrum View** window closes.

Spectrum peak markers

Automatic peak markers assist with quickly identifying the frequency and amplitude of peaks in the spectrum trace. Access the marker settings by double-tapping in a spectrum trace slice window (Stacked mode) or trace (Overlay mode) to open the **Spectrum View** menu.

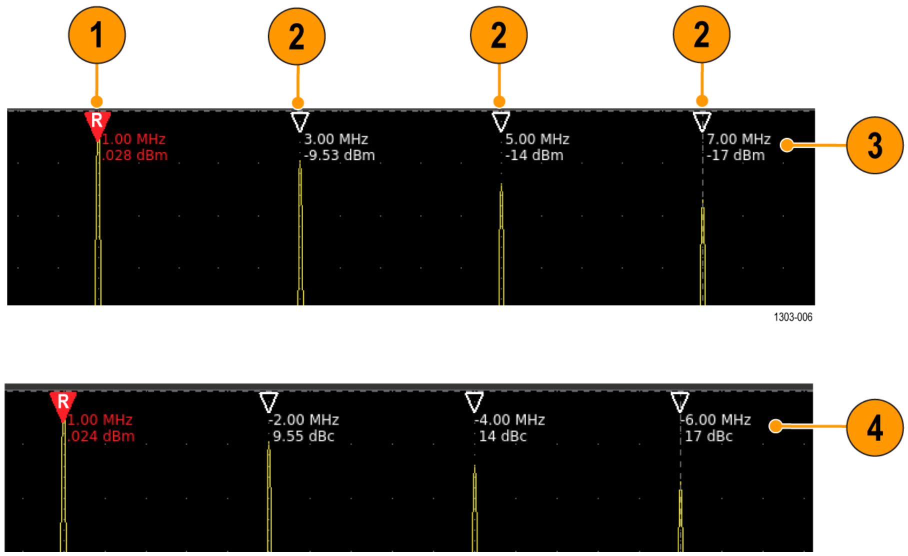


Figure 14: The spectrum peak markers

1. The highest peak shown is considered the **Reference** peak, and is labeled with a red Reference marker triangle. The **Reference** marker readout (also in red) always shows the absolute amplitude and frequency of that peak, regardless of the Readout type setting (**Absolute** or **Delta**).
2. The **Automatic** peak markers are hollow white triangles that mark the next highest peaks in the trace, depending on the marker parameters.
3. **Absolute** readouts on markers show the actual frequency and amplitude values of the marked peaks of the selected trace.
4. **Delta** readouts on Automatic markers show the delta frequency and delta amplitude of the marked peaks, of the selected trace, relative to the Reference marker readout.

Marker parameters

To set marker parameters, double-tap in the **Spectrum View** window and tap the **Markers** panel (see *Spectrum View window configuration menu*). Marker parameters include:

- Turning marker display **On** or **Off**.
- The number of markers to display (1 to 11). Selecting 1 shows just the **Reference** marker.
- The peak **Threshold** value, which is the minimum absolute peak level that peaks must equal or exceed to display a mark on that peak.
- The peak **Excursion** value, which is the minimum magnitude (excursion range) that peaks must equal or exceed to display a mark on a peak.
- The type of readout values to display (**Absolute** or **Delta**).

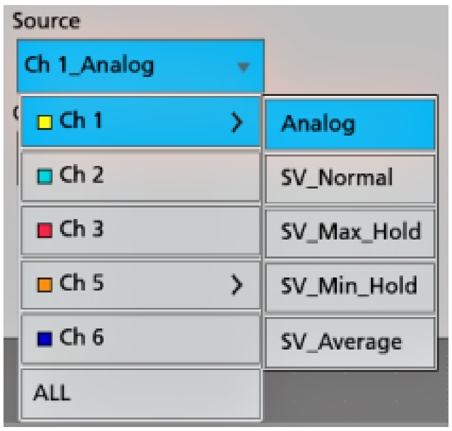
Save and recall spectrum traces

You can save spectrum trace data to a file, and recall the trace to use as a reference trace or for other analysis purposes. Recalled spectrum traces are shown in the **Spectrum View** window.

Save a spectrum trace

Use the **File** > **Save As** menu to save spectrum trace data to a specified location. See *Save a waveform or spectrum trace to a file*.

In the **Source** field, select the spectrum trace source channel and select the spectrum trace type from the expanded list. The **Source** list shows all available sources (**Analog**, **SV_Normal**, **Normal**, **Max Hold**, and so on) that are enabled in the **Spectrum View** panel of the Channel badge.



Recall a spectrum trace

Use the **File** > **Recall** menu to navigate to and select a spectrum trace file to recall. See *Recall a reference waveform or spectrum trace*. Select the spectrum trace file to recall In the **Name** field.

Spectrum trace file names include the name of the source channel (rf1 = Ch1, rf2 = Ch2, and so on) and the spectrum trace type (normal, average, and so on). For example, the file name `SpreadSpectrum_022_rf5_average.wfm` indicates that the spectrum trace was from channel 5, and is an averaged spectrum trace.

Setting waveform display parameters

Use waveform display controls to set the display mode, persistence, style, and intensity display parameters, and graticule style and intensity.

Use the following topics for more information on setting display parameters.

Set waveform display mode (Stacked or Overlay)

Use this procedure to change the waveform display mode (Stacked or Overlay).

Procedure

1. Double-tap on an open graticule area to open the Waveform View configuration menu.
2. Tap the **Display Mode** buttons to toggle between **Overlay** and **Stacked** modes.

   In stacked display mode, each waveform is stacked vertically in separate graticule slices. This is the default display mode.

   In overlay display mode, all waveforms are displayed in a single graticule (traditional waveform view).

Set the waveform persistence, style, and intensity

Use the Waveform View configuration menu to set waveform persistence, style, and intensity.

Procedure

1. Double-tap on an open graticule area to open the Waveform View menu.
2. Tap the **Persistence** field to select the persistence option.
   - **Off** disables display persistence.
   - **Auto** lets the oscilloscope automatically determine a persistence time for you.
   - **Infinite** persistence continuously accumulates record points until you change one of the acquisition display settings. Use infinite persistence for displaying unique signal anomalies, such as glitches.
   - **Variable** persistence accumulates record points for a specified time interval. Each record point decays independently according to the time interval. Use variable persistence for displaying infrequently appearing signal anomalies, such as glitches.

     If you select Variable persistence, tap **Variable Persistence Time** and set the time using the multipurpose knob, or double-tap the field and use the virtual keypad to enter the time value.
3. Tap the Waveform Style buttons to set waveforms to draw as vectors (continuous lines) or dots.
   - **Vectors** displays the waveform with the waveform sample values connected using the selected interpolation method.
   - **Dots** displays the individual waveform sample values with no interpolation.
4. Tap the **Waveform Intensity** field and use the multipurpose knob to set the brightness of all waveforms.

Set the Waveform Interpolation mode

Use this procedure to set the waveform interpolation mode, which sets how waveform data points are calculated between record data points.

Procedure

1. Double-tap on an open graticule area to open the Waveform View configuration menu.
2. Tap the buttons under **Interpolation** to select either Sin(x)/x or Linear.

Sin(x)/x calculates record points along a curve between the actual acquired samples. This form of interpolation is useful when acquiring rounded waveforms such as sine waves. It is good for general-purpose uses but may introduce overshoot or undershoot in signals with fast rise times. This interpolation is also useful for looking at high-frequency signals, especially where the frequency components are just below the Nyquist frequency.

Linear calculates record points between actual acquired samples using a straight-line fit. This interpolation is useful for measuring waveforms with fast rise times, such as pulse trains.

Set the graticule style and intensity

Use this procedure to set the graticule (display grid) style and intensity.

Procedure

1. Double-tap on an open graticule area to open the Waveform View configuration menu.
2. Tap the **Graticule Style** field to select a graticule style from the list.

   Grid provides a grid, cross hairs, and frame on the instrument display.

   Time provides a vertical grid of time marks, cross hairs, and frame on the instrument display.

   Full shows a frame and a grid on the instrument display. This style is useful for making quick, full-screen measurements with cursors and automatic readouts when cross hairs are not needed.

   None provides a frame without a grid and cross hairs.
3. Tap the Graticule Intensity field and use the multipurpose knob to set the brightness of all graticules.

   Note: Changing the graticule intensity also changes the intensity of vertical scale readouts and horizontal time readouts on the screen. Graticule intensity does not change the intensity of callout text on the screen.

Waveform acquisition concepts

Acquisition concepts

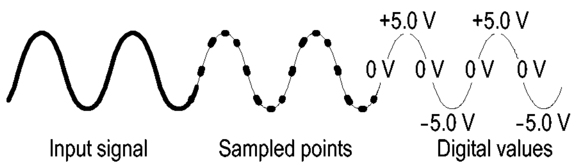
The Acquisition system sets which data points are used to acquire waveforms.

Acquisition hardware

Before a signal is displayed, it must pass through the input channel where it is scaled and digitized. Each channel has a dedicated input amplifier and digitizer. Each channel produces a stream of digital data from which the instrument extracts waveform records.

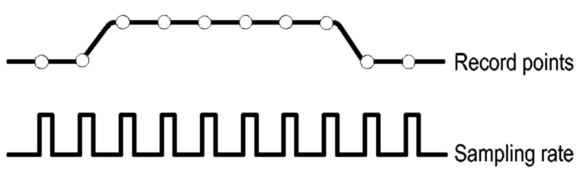
Sampling process

Acquisition is the process of sampling an analog signal, converting it into digital data, and assembling it into a waveform record, which is then stored in acquisition memory.



Real-Time sampling

In real-time sampling, the instrument digitizes all of the points it acquires using one trigger event. Use real-time sampling to capture single-shot or transient events.



Interpolated Real-Time sampling

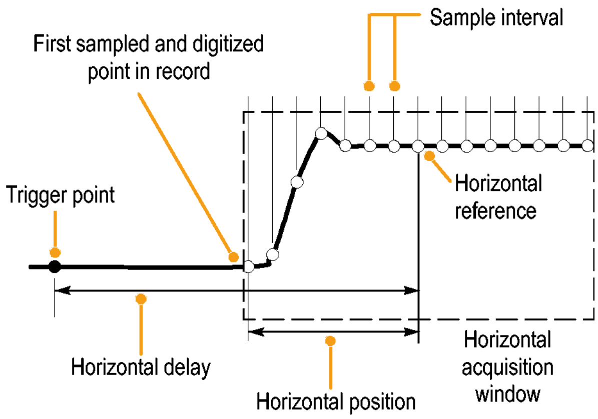
In interpolated real-time sampling, the instrument digitizes all of the points it acquires using one trigger event. If the instrument cannot acquire enough samples for a complete waveform at the maximum real-time sample rate, it interpolates. Use interpolated real-time sampling to capture single-shot or transient events.

Waveform record

The instrument builds the waveform record through use of the following parameters:

- Sample interval: The time between sample points.
- Record length: The number of samples required to fill a waveform record.
- Trigger point: The zero time reference in a waveform record.
- Horizontal position: When horizontal delay is off, the horizontal position is a percentage of the waveform record between 0 and 99.9 percent. The trigger point and the horizontal reference are at the same time in the waveform record. For

example, if the horizontal position is 50 percent, then the trigger point is in the middle of the waveform record. When horizontal delay is on, the time from the trigger point to the horizontal reference is the horizontal delay.



Interpolation

Your instrument can interpolate between the samples it acquires when it does not have all of the actual samples it needs to fill the waveform record. Linear interpolation computes record points between actual acquired samples by using a straight line fit.

Sin(x)/x interpolation computes record points using a curve fit between the actual values acquired. Sin(x)/x interpolation is the default interpolation mode because it requires fewer actual sample points than linear interpolation to accurately represent the waveform.

Acquisition modes

Acquisition is the process of sampling an analog signal, converting it into digital data, and assembling it into a waveform record, which is then stored in acquisition memory. The acquisition mode determines how the waveform record points are calculated from the sampled waveform data.

How the acquisition modes work

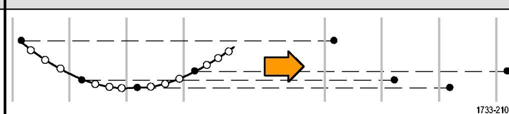
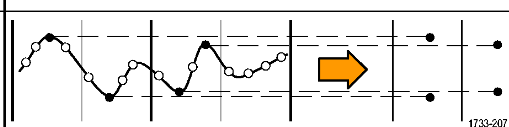
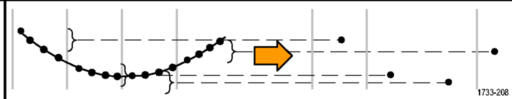
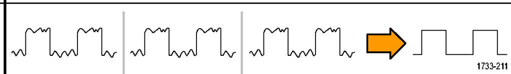
Acquisition mode	
Sample mode retains the first sampled point from each acquisition interval. Sample is the default mode. The instrument does no post processing of the acquired samples in this mode.	
Peak Detect mode retains the highest and lowest values of all the samples in each acquisition interval. This mode only works with real-time, noninterpolated sampling and is useful for catching high frequency glitches.	

Table continued…

Acquisition mode	
High Res mode applies unique FIR filtering based on the current sample rate. This FIR filter maintains maximum bandwidth possible for that sample rate while rejecting aliasing. The filter removes noise from the oscilloscope amplifiers and ADC above the usable bandwidth for the selected sample rate. Implementation of the filter in hardware, ahead of the trigger and storage, reduces trigger jitter and enables Fast Acq while in High Res mode.	
Envelope mode finds the highest and lowest record points over many acquisitions. Envelope uses Peak Detect for each individual acquisition.	
Average mode calculates the average value for each record point over many acquisitions. Average uses Sample mode for each individual acquisition. Use Average mode to reduce random noise.	
FastAcq™ mode is helpful in finding elusive signal anomalies. Fast acquisition mode reduces the dead time between waveform acquisitions, enabling the capture and display of transient events such as glitches and runt pulses. Fast acquisition mode can also display waveform phenomena at an intensity that reflects their rate of occurrence.	
FastFrame™ segmented memory acquisition uses multiple trigger events to capture widely spaced events of interest at high sample rates while conserving acquisition memory. Capturing thousands of frames is possible, allowing analysis of long-term trends and changes in the bursting signal.	
Roll Mode scrolls sequential waveform points across the display in a right-to-left rolling motion. Roll mode starts automatically when the timebase is set to ≥40 ms/div. Roll mode works at sample rates up to **10** MS/s.	

Fast Frame concepts

FastFrame™ lets you to capture only the waveform, or waveform segment of interest, eliminating the dead space between conditions of interest. Each captured event is stored in its own numbered memory segment. Multiple memory segments, or frames, can then be viewed individually in the order they were captured, or layered to show their similarity and contrast.

Advantages of using Fast Frame include:

- The high waveform capture rate increases the probability of capturing infrequent events
- The waveform detail is preserved by using high sample rates
- The events are captured while ignoring the dead time between them, ensuring efficient use of the record length memory
- The segments can be quickly and visually compared to determine if an anomaly "sticks out" of the overlaid stack

Each frame can be viewed individually and you can scroll through them by selecting frame numbers with a mouse, virtual keypad or the multi-purpose knob on the instrument's main console. When a particular frame of interest is identified, you can use the instrument's features to characterize, measure, zoom and analyze the waveform in detail.

To quickly see anomalies that stand out from the common shape of the waveform, multiple frames can be overlaid to show common and outlying points. Enabling Overlay Frames in the Fast Frame configuration menu overlays all frames in the current acquisition using color to highlight how frequently the points are overlaid on each other.

You can also view multiple signal input frames simultaneously by using stacked waveform display mode. When you scroll through the frames, the oscilloscope displays the acquisition frame for all waveforms captured.

Debugging signal errors with time stamps

Fast Frame mode provides a different type of functionality for digital designers. For example, if your processor system is being infrequently interrupted, it can be difficult to gather timing information with an oscilloscope. If you don't know when or how frequently the event occurs, you can't set up the instrument in normal acquisition mode and be assured of capturing the information you need.

Fast Frame mode can do this easily by providing information on the interval between frames. For example, the active high interrupt strobe on a microprocessor system is measured to be roughly 100 nanoseconds wide, so we set up the oscilloscope to capture 100 frames of 1250 points. In this example, the shape of the pulse is not of particular interest. We are, however, interested in the time of the pulses' rising edges.

After turning Fast Frame on and selecting "Single Acquisition" to capture 100 frames, you use the readouts in the Fast Frame results badge to compare the time stamp data at the trigger point. The "Reference Frame" was chosen to be the first interrupt pulse and the "Selected Frame" is the fourth pulse. The time difference between these pulses is shown in the Delta readout on the badge.

The time stamps of all the frames can be output in tabular form for in-depth analysis using Excel or a wide variety of other popular software tools that read comma-delimited files (.CSV).

⚠ **Note:** Enabling Fast Frame mode disables FastAcq mode (if it was enabled). Likewise, enabling FastAcq mode disables Fast Frame mode (if it was enabled).

Waveform sample interpolation

When the sample density falls to less than one sample per display column, the instrument must calculate intermediate points to display a waveform. This process is called interpolation.

There are three options for interpolation:

- Sin(x)/x interpolation. Computes record points using a curve fit between the actual values acquired. It assumes that all the interpolated points fall along that curve.
- Linear interpolation. Computes record points between actual acquired samples by using a straight line fit. It assumes that all the interpolated points fall in their appropriate point in time on that straight line.
- Auto interpolation. Select the best interpolation method.

Coupling

All instruments and probes specify a maximum signal level. Do not exceed the limit, even momentarily, as the input channel or probe may be damaged. Use external attenuators if necessary to prevent exceeding the limits.

Coupling determines whether an input signal is directly connected to the input channel (DC coupling), connected through a DC blocking capacitor (AC coupling), or not connected at all (GND coupling).

The input resistance of each input channel can be 1 MΩ [2] or 50 Ω. To properly terminate signals when using coaxial cables, or to support active probes with different termination requirements, select the termination in the Channel menu Vertical Settings panel.

All probes expect a specific coupling and input termination. Both coupling and input termination are displayed on the screen. If the instrument determines the coupling and termination required by the probe, either implicitly because of the TekProbe/

2 The LPD64 does not have a 1 MΩ input impedance setting.

TekVPI interface or through performing a probe compensation, the instrument sets the required coupling and input termination.

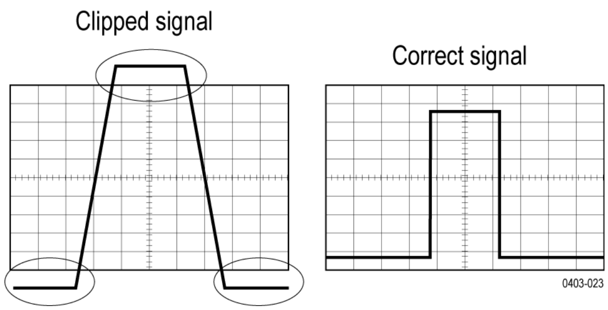
Consider the following when you use 50 Ω termination with any channel:

- The instrument does not accurately display frequencies under 200 kHz if AC coupling is selected.
- The instrument reduces the maximum volts per division setting for the channel, since input amplitudes appropriate for the higher settings would overload the 50 Ω input.

Scaling and positioning

The scaling and positioning controls determine the portion of the input signal received by the acquisition system. Set vertical scaling, positioning, and DC offsets to display the features of interest on your waveform and to avoid clipping.

Each waveform Slice or Graticule contains ten major divisions. This represents the maximum digitizing range of the instrument for any given vertical scale. Vertical waveform data that is outside (above and/or below) of the waveform Slice or Graticule is clipped; that is, the data values exceed the digitizing capability of the ADC at the current settings. This causes inaccuracies in amplitude-related measurements. For more information see *Vertical acquisition considerations* on page 126 .



Set the horizontal scale, position, and resolution (record length) to include the acquired waveform record waveform attributes of interest with good sampling density on the waveform. These settings define the horizontal acquisition window, described in *Horizontal acquisition considerations* on page 127 .

Note: The terms vertical acquisition window and horizontal acquisition window refer to the vertical and horizontal range of the segment of the input signal that the acquisition system acquires. The terms do not refer to any display windows on screen.

Vertical acquisition considerations

You can set the vertical scale, position, and offset of each channel independently of other channels. Vertical scale and offset specify the vertical parameters of the waveform display for each channel. The oscilloscope only acquires signals that fall within these parameters.

The offset control subtracts a constant DC level from the input signal before the vertical scale factor is applied, and the vertical position control adds a constant number of divisions of signal after the scale factor is applied to the resulting difference.

The vertical scale and position controls have the following effects on the waveform display and the displayed waveform:

The vertical volts per division you set determines the vertical size of the waveform display, allowing you to scale it to contain all of a waveform amplitude or only part.

Note: Amplitude-related automatic measurements (for example, peak-to-peak and RMS) will be accurate for vertical windows if the waveform is not clipped (that is, the waveforms are acquired). But if signal amplitude were to extend outside the vertical acquisition window, the data acquired is clipped. Clipped data causes inaccurate results if used in amplitude-related automatic measurements. Clipping also causes inaccurate amplitude values in waveforms that are stored or exported for use in other programs.

If the scale of a math waveform is changed so that the math waveform is clipped, it will affect the amplitude measurements on that math waveform as follows:

- The vertical position adjusts the display of the graticule relative to the waveform display (position is a display control). That is all position does; it does not determine what data is acquired as do vertical scale and offset.
- As you vary vertical offset, the middle voltage level moves relative to zero. This moves the waveform display up or down relative to the acquired waveform. With input signals that are smaller than the window, it appears the waveform moves in the waveform view. Actually, the offset moves the middle of the waveform display up and down on the input signal. Offset moves the waveform display to control the portion of the waveform amplitude the display captures.
- Applying a negative offset moves the vertical range down relative to the DC level of the input signal. Likewise, applying a positive offset moves the vertical range up.

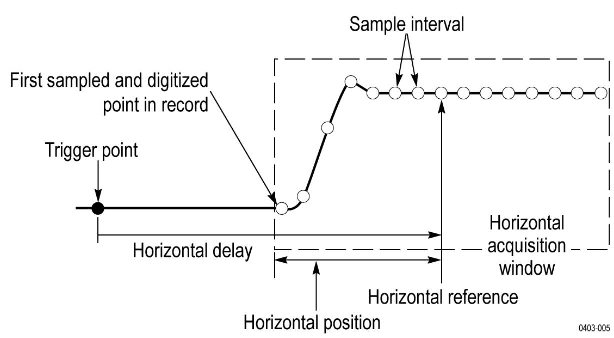
Horizontal acquisition considerations

The instrument lets you define the horizontal waveform display parameters that determine which segment of an incoming signal becomes the waveform record, following acquisition.

These common parameters specify a horizontal scale and position that is applied to all channels simultaneously.

These parameters are shown in the next figure (horizontal window with delay on):

- The trigger position determines where the trigger event will be located in the waveform record. To see more pretrigger data move your trigger position to the right on the graticule.
- The horizontal position determines the number of pretrigger and posttrigger samples. Samples before the trigger point are pretrigger samples and those after the trigger point are posttrigger samples. When Delay is off, the horizontal position is the same as the trigger position.
- The horizontal delay determines the time from the trigger point to the Horizontal Reference.
- The horizontal scale determines the horizontal size of the display relative to any waveform, allowing you to scale it to contain a waveform edge, a cycle, or several cycles.



Menus and dialog boxes

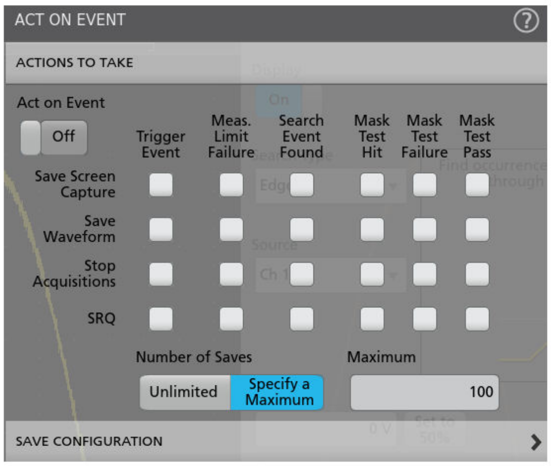
Act On Event configuration menu

Use this configuration menu to set the actions the instrument takes when specified conditions are met (trigger event, measurement limit failure, mask hit failure, and so on).

To open the **Act On Event** menu, select the **Act On Event** button in menus where this button is available.

Act On Event menu fields and controls

Displayed fields and controls can change depending on menu selections.



Field or control	Description
Actions To Take panel	
Act on Event on/off switch	Allows you to configure AOE before enabling. If the maximum number of saves is hit, this control turns off.
Save Screen Capture	Saves the screen image at the time of the event, to the format designated in the **Save Configuration** panel.
Save Waveform	Saves the waveform data at the time of the event, to the format designated in the **Save Configuration** panel.
Stop Acquisitions	Stops the instrument from acquiring any more data.
SRQ	This is a signal sent via the VISA connection to alert listeners that an event has occurred. Enabling this will send such an event when the action has occurred and if the status registers are configured correctly.
Number of Saves	Allows you to toggle between *Unlimited* or *Specify a Maximum* for number of saves. Default option selected is Unlimited.
Maximum	Enter the maximum number of saves in the text field. This is enabled when Number of Saves is set to Specify a Maximum.
Save Configuration panel	

Table continued…

Field or control	Description
Save Location	Choose the location of the saved files when an action triggers a save event.
Browse	Browse to the save location through a visual file tree.
File Name	Name the saved file. When a file is saved, the file name is followed by a time stamp.
Screen Capture Format	Choose format of the screen capture save files.
Waveform Format	Choose waveform format for the screen capture save files.
Source	Choose the source for the waveform save files.

Actions guidelines

- All actions are taken on a per acquisition basis. This means that if Save Screen Capture is checked in multiple columns and, during a single acquisition the criteria is met in multiple columns, only a single screen capture is saved.

Add Measurements configuration menu overview

Use this configuration menu to select measurements you want to take on waveforms and add the measurements to the Results bar.

To open the **Add Measurements** configuration menu, tap the **Add New... Measure** button in the **Analysis** controls area.

The **Add Measurements** configuration menu always opens on the **Standard** measurement tab. The listed tabs and measurements depend on the installed measurement options and the selected signal source.

To add a measurement, select the measurement type tab, select the input source or sources, select the measurement, and either tap the **Add** button or double-tap the measurement. The measurement is added to the Results bar.

To change individual measurement settings, double-tap the Measurement badge to open a configuration menu for that measurement. See *Measurement configuration menu overview* on page 148.

Add Measurements menu fields and controls

Field or control	Description
Measurement tabs	The tabs along the top organize measurements by their type. The Standard tab is the default set of measurements that are built in to the instrument. Other tabs are shown when you install measurement options.
Measurement description (graphic and text)	Shows a graphic and short description of the selected measurement. Use this information to verify that the selected measurement is correct for what you want to measure.
Source	Selects the measurement source. If the measurement requires more than one source (for example, Skew, Phase, or many Power measurements), the menu shows two source fields from which to select.
Add button	Adds the selected measurement as a measurement badge to the Results bar. You can also double-tap a listed measurement to add it to the Results bar.

Standard tab measurement panels

Panel	Description
Amplitude Measurements panel	Tap the panel bar to list the available amplitude measurements. Touch and drag the list to scroll through all measurements.
Timing Measurements panel	Tap the panel bar to list the available time measurements. Touch and drag the list to scroll through all measurements.
Jitter Measurements panel	Tap the panel bar to list the available standard jitter measurements. ⚠ **Note:** If you have installed the Advanced Jitter and Eye Analysis option, the **Jitter Measurements** panel is removed from the **Standard** measurement tab and is replaced with a **Jitter** tab at the top of the **Add Measurements** menu.

Other tab measurements (require optional licenses)

Add Measurement tab	Description
Jitter	Advanced Jitter and Eye Analysis measurements (option). Provides triggers and measurements for advanced jitter and eye analysis.
Power	Advanced Power Analysis (option). Provides measurements for electrical analysis, output analysis, amplitude analysis, timing analysis, and switching analysis.
DPM	Digital Power Management Analysis (option). Provides measurements for ripple analysis, transient analysis, power sequence analysis, jitter and eye analysis.
DDR	DDR (option). Provides measurements for DDR amplitude and timing analysis .
IMDA	IMDA (Inverter Motors and Drive Analysis) (option). 3 phase measurement is provided for **Electrical Analysis**.

Use the following links to learn more about the measurements.

The Standard measurements tab

The Standard tab lists the default set of amplitude, timing, and jitter measurements that are included with the instrument.

To open the Standard measurements tab, tap the **Add New...Measure** button. The Add Measurements configuration menu opens on the Standard tab by default.

To add a measurement to the Results bar:

1. Select the signal source.
2. Tap a measurement panel:

   Amplitude Measurements panel on page 131

   Timing Measurements panel on page 132

   Jitter Measurements panel on page 134
3. Select a measurement. If a measurement requires two signal sources, select the sources in the Source 1 and Source 2 fields.
4. Tap **Add**. You can also double-tap a measurement to add it immediately to the Results bar.

Note: If you have installed Advanced Jitter and Eye Analysis option, all jitter measurements are in the **Jitter** tab. See *The Jitter tab (Advanced Jitter and Eye Analysis)* on page 135.

Amplitude Measurements panel

The Amplitude Measurements panel lists available amplitude-related measurements that you can take on analog channel signals, math waveforms (time-domain), and reference waveforms. Amplitude measurements are not available for digital signals.

To open the Amplitude Measurements panel:

1. Tap **Add New... Measure** button.
2. Tap the **Amplitude Measurements** panel.

To add a measurement to the Results bar:

1. Select the signal source.
2. Select a measurement.
3. Tap **Add**. You can also double-tap a measurement to add it immediately to the Results bar.

The Amplitude Measurements panel measurements

Measurement	Description
AC RMS	The true Root Mean Square voltage, minus any DC component, of the waveform data points that are above the Mean signal level. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Amplitude	The difference between the Top value and the Base value. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Area	The area between the waveform and the Mean value, calculated by integrating the data points. The area above ground (0 V) is positive, while the area below ground is negative. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Base [3]	The most common data point value below the midpoint of the waveform, over the measurement region. Base is used as the 0% value whenever low reference values are calculated, such as in rise time or fall time measurements. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Maximum	The maximum data point value. You can take this measurement on each cycle in the waveform record or across the entire waveform record.
Mean	The arithmetic mean of all data points over the measurement region. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Minimum	The minimum data point value. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Negative Overshoot[3]	The difference between the Minimum and Base values, divided by the Amplitude, and multiplied by 100 to express the measurement as a percentage of amplitude. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Peak-To-Peak	The absolute difference between the Maximum and Minimum amplitudes in the measurement region. You can take this measurement on each cycle in the waveform record or on the entire waveform record.

Table continued…

[3] Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated.

Measurement	Description
Positive Overshoot [3]	The difference between the Maximum value and the Top value, divided by the Amplitude, and multiplied by 100 to express the measurement as a percentage of amplitude. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
RMS	The true Root Mean Square (The square root of the mean (average) value of the squared function of the waveform data points.) You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Top [3]	The most common data point value above the midpoint of the waveform over the measurement region. Base is used as the 100% reference value whenever high reference values are calculated, such as in rise time or fall time measurements. You can take this measurement on each cycle in the waveform record or on the entire waveform record.

See also

Measurement configuration menu overview on page 148

Timing Measurements panel

Use the Timing Measurements panel to add timing-related measurements to the Results bar. Timing measurements can be taken on time-domain analog, math, and reference waveforms. Timing measurements can also be taken on some digital channel signals.

To open the Timing Measurements panel:

1. Tap the **Add New... Measure** button.
2. Tap the **Timing Measurements** panel.

To add a measurement to the Results bar:

1. Select the signal source.
2. Select a measurement. If a measurement requires two signal sources, select the sources in the Source 1 and Source 2 fields.
3. Tap **Add**. You can also double-tap a measurement to add it immediately to the Results bar.

Timing Measurements panel

Measurement	Description
Burst Width [4]	The duration of a series of adjacent crossings of the Mid reference level. Bursts are separated by a specified idle time. The measurement is taken on each burst in a waveform record.
Data Rate [4]	Data Rate is the reciprocal of Unit Interval. The measurement is taken on each bit of the waveform record.
Delay	The time difference between the mid reference (default 50%) amplitude point of two different waveforms. You can specify the signal edges to measure in the measurement's Configuration menu. The measurement requires two sources.

Table continued…

4 This measurement can also be taken on digital signals.

Measurement	Description
Duration N-Periods [4]	The time required to complete N cycles. A cycle is the time between two adjacent (same direction) crossings of the mid reference level. The measurement is taken on each cycle in the waveform record.
Fall Time	The time required for the trailing edge of the first pulse in the measurement region to fall from the high reference value (default = 90%) to the low reference value (default = 10%). The measurement is taken on each cycle of the waveform record.
Falling Slew Rate	The rate of change (in volts/second) as an edge transitions from a high reference level to a low reference level. The measurement is taken on each cycle of the record in the measurement region.
Frequency [4]	The frequency of the waveform. Frequency is the reciprocal of Period (Frequency = 1/ Period).
High Time	The time the signal remains above the Top reference level. The measurement is taken on each cycle in the waveform record.
Hold Time [4]	The time between the specified Mid reference level crossing on the clock signal to the closest Mid reference level crossing on the specified data signal. The measurement is taken on each specified clock edge in the waveform record.
Low Time	The time the signal remains below the Base reference level. The measurement is taken on each cycle in the waveform record.
Negative Duty Cycle [4]	The ratio of the negative pulse width to the signal period, expressed as a percentage. The duty cycle is measured on the first cycle in the measurement region.
Negative Pulse Width [4]	The distance (time) between the mid reference (default 50%) amplitude points of a negative pulse. The measurement is taken on each cycle in the waveform record or measurement region.
Period [4]	The time between two adjacent crossings of the Mid reference level (one cycle) of the waveform. The measurement is taken on each cycle of the waveform record or measurement region.
Phase [4]	The time difference (phase shift) between the specified signal edges of waveform source 1 and waveform source 2. The measurement is expressed in degrees, where 360° comprise one waveform cycle. This measurement requires two sources. The measurement is taken on each cycle of the waveform record.
Positive Duty Cycle [4]	The ratio of the positive pulse width to the signal period, expressed as a percentage. The duty cycle is measured on the first cycle in the measurement region.
Positive Pulse Width [4]	The distance (time) between the mid reference (default 50%) amplitude points of a positive pulse. The measurement is made on the first pulse in the measurement region.

Table continued…

Measurement	Description
Rise Time	The time required for the leading edge of the first pulse in the measurement region to rise from the low reference value (default = 10%) to the high reference value (default = 90%). The measurement is taken on each cycle of the waveform record.
Rising Slew Rate	The rate of change (in volts/second) as an edge transitions from a low reference level to a high reference level. The measurement is taken on each cycle of the record in the measurement region.
Setup Time [4]	The time between the specified Mid reference level crossing on the data signal to the closest Mid reference level crossing on the specified clock signal. The measurement is made on each specified clock edge in the waveform record.
Skew [4]	The time between the specified Mid reference level crossing on one source to the closest Mid reference level crossing on the second source signal. The measurement is made on each cycle in the waveform record.
Time Outside Level	The time the specified signal remains above the Top reference level and/or below the Base reference level. The measurement is made on each occurrence in the waveform record.
Time to Min	Time to Min is the amount of time from the trigger point to the minimum data point. This measurement can be made across the entire record or on each cycle in the record.
Time to Max	Time to Max is the amount of time from the trigger point to the maximum data point. This measurement can be made across the entire record or on each cycle in the record.
Unit Interval [4]	The time difference between two successive bits. The measurement is taken on each bit of the waveform record.

See also

Measurement configuration menu overview on page 148

Jitter Measurements panel

The Jitter Measurements panel lists the standard jitter-related measurements that you can add to the Results bar. These jitter measurements are part of the Standard measurements that are provided by default.

To open the Jitter Measurements panel:

1. Tap the **Add New...Measure** button.
2. Tap the **Jitter Measurements** panel.

To add a measurement to the Results bar:

1. Select the signal source.
2. Select a measurement.
3. Tap **Add**. You can also double-tap a measurement to add it immediately to the Results bar.

Note: If you have installed Advanced Jitter and Eye Analysis option, the jitter measurements are moved to the **Jitter** tab of the **Add Measurements** menu. See *The Jitter tab (Advanced Jitter and Eye Analysis)* on page 135.

Jitter Measurements panel measurements

Measurement	Description
Phase Noise	The RMS magnitude of all integrated jitter falling within a specified offset range of the fundamental clock frequency. This measurement is taken on the entire waveform record.
TIE	The difference in time between an edge in the source waveform and the corresponding edge in a recovered reference clock signal. The measurement is made on each waveform edge in the measurement region.

See also

Measurement configuration menu overview on page 148

The Jitter tab (Advanced Jitter and Eye Analysis)

The Jitter tab lists advanced jitter, eye, amplitude, and timing measurements that you can add to the Results bar. The Jitter tab is only shown if you have purchased and installed the Advanced Jitter and Eye Analysis option.

To open the Jitter measurements tab:

1. Tap **Add New... Measure** button.
2. Tap the **Jitter** tab.

To add a measurement to the Results bar:

1. Select the signal source.
2. Select a measurement panel.
3. Select a measurement.
4. Tap **Add**. You can also double-tap a measurement to add it to the Results bar.

Adding eye diagram plots for a jitter measurement

To display an eye diagram plot for a jitter measurement, double-tap the jitter measurement badge for which you want to show an eye diagram. If available, tap the **Eye Diagram** plot button.

Jitter Measurements panel

Measurement	Description
DCD	Duty cycle distortion. The peak-to-peak amplitude for the component of the deterministic jitter correlated with the signal polarity. The measurement is taken on the entire record.
DDJ	Data dependent jitter. The peak-to-peak amplitude for the component of the deterministic jitter correlated with the date pattern in the waveform. The measurement is taken on the entire record.
DJ	Deterministic jitter. The peak-to-peak amplitude of all timing errors that exhibit deterministic behavior. The measurement is taken across the entire record.

Table continued…

Measurement	Description
DJ-δδ	Dual-Dirac deterministic jitter. The deterministic jitter on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal-magnitude Dirac functions. The measurement is taken on the entire record.
F/2	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (date rate) divided by two. The measurement is taken on the entire record.
F/4	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (date rate) divided by four. The measurement is taken across the entire record.
F/8	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (date rate) divided by eight. The measurement is taken on the entire record.
J2	The total jitter at a bit error rate of 2.5e-3 (TJ@2.5e-3). The measurement is taken on the entire record.
J9	The total jitter at a bit error rate of 2.5e-3 (TJ@2.5e-3). The measurement is taken on the entire record.
Jitter Summary	Adds multiple jitter measurements to the measurement badge, and displays Bathtub, TIE Spectrum trend plot, TIE Histogram, Eye Diagram plots on the screen.
NPJ	Nonperiodic jitter. The portion of the BUJ (Bounded uncorrelated jitter) that is random. BUJ excludes DDJ, DCD, and RJ. The measurement is taken on the entire record.
Phase Noise	The RMS magnitude of all integrated jitter falling within a specified offset range of the fundamental clock frequency. The measurement is taken on the entire record.
PJ	Periodic jitter. The peak-to-peak amplitude of the uncorrelated sinusoidal components of the deterministic jitter. The measurement is taken on the entire record.
RJ	Random jitter. The RMS magnitude of all random timing errors following a Gaussian distribution. The measurement is taken on the entire record.

Table continued…

Measurement	Description
RJ-δδ	Dual-Dirac random jitter. The random jitter on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal-magnitude Dirac functions. The measurement is taken on the entire record.
SRJ	Sub-rate jitter. The composite jitter due to periodic components at ½, 1/4, and 1/8 of the data rate. The measurement is taken on the entire record.
TIE	Time Interval Error. The difference in time between an edge in the source waveform and the corresponding edge in a recovered reference clock signal. The measurement is taken on each waveform edge.
TJ@BER	Total error at a specified bit error rate. The predicated peak-to-peak amplitude of jitter that will only be exceeded with a probability equal to the bit error rate. The measurement is taken on the entire record.

Eye Measurements panel

Measurement	Description
Eye Height	The minimum vertical eye opening at the center of the recovered unit interval. The measurement is taken on the entire waveform record.
Eye High	The amplitude of a high (one) bit measured at a specified location within the recovered unit interval. The measurement is taken on each high bit in the waveform record.
Eye Low	The amplitude of a low (zero) bit measured at a specified location within the recovered unit interval. The measurement is taken on each low bit in the waveform record.
Eye Width	The minimum horizontal eye opening at the center of the recovered unit interval. The measurement is taken on the entire waveform record.
EyeHeight@BER	The predicted vertical eye opening that will be violated with a probability equal to the bit error rate. The measurement is taken on the entire waveform record.
EyeWidth@BER	The predicted horizontal eye opening that will be violated with a probability equal to the bit error rate. The measurement is taken on the entire waveform record.

Table continued…

Measurement	Description
Q-Factor	The ratio of the vertical eye opening to RMS vertical noise measured at a specified location in the recovered unit interval. The measurement is taken on the entire waveform record.

Amplitude Measurements panel

Measurement	Description
AC Common Mode (Pk-Pk)	The peak-to-peak of the common mode voltage of the two specified sources. The measurement is taken on the entire record.
Bit Amplitude	The difference between the amplitudes of the 1 and 0 bits surrounding a transition. The amplitude is measured over a specified part of at the center of the recovered time interval. The measurement is taken on each transition bit of the entire record (Mean) or on the entire record (Mode).
Bit High	The amplitude of a 1 bit, measured over a specified percent of the center of the recovered time interval. The measurement is taken on each high bit of the entire record (Mean) or on the entire record (Mode).
Bit Low	The amplitude of a 0 bit, measured over a specified percent of the center of the recovered time interval. The measurement is taken on each low bit of the entire record (Mean) or on the entire record (Mode).
DC Common Mode	The arithmetic mean of the common mode voltage of two sources. The measurement is taken on the entire record.
Differential Crossover	The voltage level of a differential signal pair at the crossover point(s). The measurement is taken at each crossover point in the record.
T/nT Ratio	The ratio of a nontransition bit voltage (second and subsequent bit voltage after a transition) to its nearest preceding transition bit voltage (first bit voltage after the transition). Bit voltage is measured at the interpolated midpoint of the recovered unit interval. The measurement is taken on each nontransition bit in the record.

Timing Measurements panel

Measurement	Description
SSC Freq Dev	The spread spectrum clock frequency deviation. This measurement enables a time trend plot of the spread spectrum clock modulation profile. The measurement is taken on each cycle of the entire record.

Table continued…

Measurement	Description
SSC Modulation Rate	The modulation frequency of a spread spectrum clock. The measurement is taken on each cycle of the entire record.

See also

Measurement configuration menu overview on page 148

The Power tab

The Power tab lists the optional power-related measurements that you can add to the Results bar. Power measurements include input analysis, amplitude analysis, timing analysis, switching analysis, and output analysis.

To open the Power Measurements tab:

1. Tap the **Add New...Measure** button.
2. Tap the **Power** tab.

To add a measurement to the Results bar:

1. Select the signal source/s.
2. Tap a measurement panel:

   Input Analysis Measurements panel on page 139

   Amplitude Analysis Measurements panel on page 140

   Timing Analysis Measurements panel on page 140

   Switching Analysis Measurements panel on page 141

   Output Analysis Measurements panel on page 142

   Magnetic Analysis Measurements panel on page 141

   Frequency Response Analysis Measurements panel on page 142
3. Select the measurement. If the measurement requires two signal sources, select the sources in the Source 1 and Source 2 fields.
4. Tap **Add**. You can also double-tap a measurement to add it to the Results bar.

Input Analysis Measurements panel

Measurement	Description
Power Quality	Measures the Frequency, RMS values of the voltage and current, Crest Factors of the voltage and current, Real Power, Reactive Power, Apparent Power, Power Factor, and Phase Angle of the AC signal.
Harmonics	Plots the signal amplitudes at the fundamental line frequency and its harmonics. Measures the RMS amplitude and Total Harmonic Distortion of the signal. Plots the Harmonics Bar Graph.
Input Capacitance	Measures the DUT input capacitance when powered on.
Inrush Current	Measures the positive and negative peak input current during DUT power-on.

Note: You should take the Input Capacitance and Inrush Current measurements independently. Input Capacitance and Inrush Current measurements use **Power Preset** to optimize the settings and trigger to measure the DUT power-on signal, which may result in inaccurate measurements for other measurement types.

Amplitude Analysis Measurements panel

Measurement	Description
Cycle Amplitude	The difference between the Top value and the Base value. Measurement can be made across the entire record or on each cycle in the record.
Cycle Top	The most common data point value above the midpoint of the waveform over the measurement region. This measurement is made on each cycle in the record.
Cycle Base	The most common data value below the midpoint of the waveform. This measurement is made on each cycle in the record.
Cycle Peak-to-Peak	The difference between the Maximum and Minimum values in the measurement region. This measurement is made on each cycle in the record.
Cycle Maximum	The maximum data point. This measurement is made on each cycle in the record.
Cycle Minimum	The minimum data point. This measurement is made on each cycle in the record.

Timing Analysis Measurements panel

Measurement	Description
Period	The time between two adjacent crossings of the Mid reference level (one cycle) of the waveform. This measurement is taken on each cycle of the waveform record or measurement region.
Frequency	Frequency is the reciprocal of Period (Frequency = 1/Period). This measurement is made on each cycle in the record.
Positive Duty Cycle	The ratio of the positive-pulse width to the signal period. This measurement is taken on each cycle of the waveform record or measurement region.
Negative Duty Cycle	The ratio of the negative-pulse width to the signal period. This measurement is taken on each cycle of the waveform record or measurement region.
Positive Pulse Width	The time the signal remains above the Mid-reference level. This measurement is taken on each cycle of the waveform record or measurement region.

Table continued…

Measurement	Description
Negative Pulse Width	The time the signal remains below the Mid-reference level. This measurement is taken on each cycle of the waveform record or measurement region.

Switching Analysis Measurements panel

Measurement	Description
Switching Loss	The mean instantaneous power and energy in the turn-on, turn-off, and conduction regions of a switching device. Provides the SWL trajectory plot.
dv/dt	The rate of change (slew rate) of the voltage, as it rises from the Base reference level (R_B) to the Top reference level (R_T), and as it falls from the Top reference level (R_T) to the Base reference level (R_B).
di/dt	The rate of change (slew rate) of the current, as it rises from the Base reference level (R_B) to the Top reference level (R_T), and as it falls from the Top reference level (R_T) to the Base reference level (R_B).
SOA	An X-Y plot of switching device voltage and current. The SOA mask testing provides pass/fail testing to component specifications.
$R_{DS(on)}$	The resistance (the slope of the V-I curve) when the switching device is conducting.

Magnetic Analysis Measurements panel

Measurement	Description
Inductance	The integral of the voltage divided by the current of a magnetic component during circuit operation.
Magnetic Property	The B-H curve for a magnetic component during circuit operation.
Magnetic Loss	The average value of the product of the voltage and current through the inductor. This represents the total loss of the magnetic device and consists of resistive and eddy current losses during circuit operation.
I vs. ∫V	The integral of voltage against current.

Output Analysis Measurements panel

Measurement	Description
Line Ripple	The RMS and peak-to-peak values of the line frequency portion of the AC signal. **Note:** RMS is measured at the configured ripple frequency
Switching Ripple	The RMS and peak-to-peak values of the input signal. **Note:** RMS is measured at the configured ripple frequency
Efficiency	The ratio of output power to input power for a power conversion circuit.
Turn On Time	Measures the time from when the input voltage is applied to the DUT to the time when the output voltage reaches a steady level.
Turn Off Time	Measures the time from when the input voltage is removed from the DUT to the time when the output voltage reaches a zero level.

Note: You should take the Turn On Time and Turn Off Time measurements independently. Turn On Time and Turn Off Time measurements use the **Power Preset** function to optimize the settings and trigger to measure the DUT power-on signal, which may result in inaccurate measurements for other measurement types.

Note: If you save a session file that includes power measurements, make sure to recall (load) the session file into an instrument with the same number of channels that was used to create the session file.

Frequency Response Analysis Measurements panel

Measurement	Description
Control Loop Response	Plots the gain as 20 log (V_{out}/V_{in}) and phase difference between V_{in} and V_{out} at each frequency within the swept band. The resulting plot is commonly referred to as a Bode Plot.
Power Supply Rejection Ratio	Measures both the modulated input and output AC voltage levels and plots the rejection ratio as 20 log (V_{in}/V_{out}) at each frequency within the specified band.
Impedance	Computes and plots the channel ratio (V_{out}/V_{in}) at each frequency within the swept band. The impedance curve displays maximum annotations of 3 peaks, if they are available only. **Note:** It is recommended to use a splitter with the built-in AFG source for Impedance measurements. If you use an external AFG, use a single channel with a splitter.

See also

Power measurement configuration menu overview on page 160

The IMDA tab

The IMDA tab lists the IMDA-related measurements that you can add to the Results bar. IMDA measurements include electrical analysis, switching analysis, and output analysis.

To open the IMDA Measurements tab:

1. Tap the **Add New...Measure** button.
2. Tap the **IMDA** tab.

⚠ **Note:** IMDA tab is available only for 5 and 6 Series OScilloscopes. The 3-Phase tab is available for 4 Series Oscilloscopes.

To add a measurement to the Results bar:

1. Tap a measurement panel:
2. Select the measurement. *Electrical Analysis Measurements panel* on page 143
3. Tap **Add**. You can also double-tap a measurement to add it to the Results bar.

Electrical Analysis Measurements panel

Measurement	Description
Power Quality	Measures the operating Frequency, RMS values of the voltage and current, Crest Factors of the voltage and current, Real Power, Reactive Power, Apparent Power, Power Factor, and Phase Angle of the AC signal. Plots the Phasor diagram.
Harmonics	Measures the signal amplitudes at the fundamental line frequency and its harmonics. Measures the RMS amplitude and Total Harmonic Distortion of the signal. Plots the Harmonics Bar Graph.
Ripple	The RMS and peak-to-peak values of the 3-Phase signal.
Efficiency	Measures the ratio of output power to input power per phase for a AC/DC input and AC Drive output. Measures the total efficiency of the system.
DQ0	Transforms three-phase AC (Voltage or Current) time domain waveforms into DC signals and are graphically represented as vectors on a phasor plot. ⚠ **Note:** Supported only when 3V3I wiring is selected.

See also

IMDA measurement configuration menu overview on page 184

The DPM tab

The DPM tab lists the optional DPM-related (Digital Power Management) measurements that you can add to the Results bar. DPM measurements include ripple analysis, transient analysis, power sequence analysis, jitter and eye analysis.

Power rail probes are recommended for DPM measurements. For turn on time and turn off time measurements, optional to use passive probe at the input voltage and recommended to power rail probes at the output side. The power rail probe is recommended because it takes care of the DC rail output voltage offset and measurement results will be precise.

To open the DPM Measurements tab:

1. Tap the **Add New...Measure** button.

2. Tap the **DPM** tab.

To add a measurement to the Results bar:

1. Tap a measurement panel:

   Ripple Analysis Measurements panel on page 144

   Transient Analysis Measurements panel on page 144

   Power Sequence Analysis Measurements panel on page 144
2. Select the measurement.
3. Select the number of rails in the **Power Rails** field.
4. Tap **Add**. You can also double-tap a measurement to add it to the Results bar.

Ripple Analysis Measurements panel

Measurement	Description
Ripple	Measures RMS and peak-to-peak values of the 3-Phase signal on the DC Rail. RMS is measured at the configured ripple frequency.

Transient Analysis Measurements panel

Measurement	Description
Overshoot	Overshoot is the difference between Maximum and Top, divided by the Amplitude. This measurement can be made across the entire record or on each cycle in the record.
Undershoot	Undershoot is the difference between Minimum and Base, divided by the Amplitude. This measurement can be made across the entire record or on each cycle in the record.
Turn on Overshoot	Turn on Overshoot computes the maximum amplitude value of the dc rail output voltage during turn ON of the DUT. This supports multiple rail outputs. User has to provide input signal level to trigger properly and measurement happens on output rail. Annotation of max value on the output is shown as cross hair per rail output.
DC Rail Voltage	DC Rail Voltage is the mean value of each rail output. This can be performed per cycle or for the entire record. In cycle mode, the user can specify Edge qualifier as the source to find edges. Annotation and navigation available in cycle mode. One can traverse in navigation mode using Prev and Next buttons.

Power Sequence Analysis Measurements panel

Measurement	Description
Turn On Time	Measures the time delay difference between the input voltage applied to the system and the time to develop the steady state output voltage.

Table continued…

Measurement	Description
Turn Off Time	Measures the time delay difference between the input voltage removed from the system and the time for the output voltage to become zero.

Jitter Analysis Measurements panel

Measurement	Description
TIE	TIE (Time Interval Error) is the difference in time between an edge in the source waveform and the corresponding edge in a reference clock. The reference clock is usually determined by a clock recovery process performed on the source waveform. For Explicit-Clock clock recovery, the process is performed on an explicitly identified source. User can specify the jitter frequency and BW limit for each rail in the measurement configuration. The measurement supports Spectrum, Eye diagram and Histogram plots per rail.
RJ	Random Jitter (RJ) is the rms magnitude of all timing errors not exhibiting deterministic behavior. A single RJ value is determined for each acquisition, by means of RJ-DJ separation analysis. User can specify the jitter frequency and BW limit for each rail in the measurement configuration. The measurement supports Eye diagram and Histogram plots per rail.
DJ	Deterministic Jitter (DJ) is the peak-to-peak amplitude for all timing errors that follow deterministic behavior. A single DJ value is determined for each acquisition, by means of RJ-DJ separation analysis. User can specify the jitter frequency and BW limit for each rail in the measurement configuration. The measurement supports Eye diagram and Histogram plots per rail.
PJ	Periodic Jitter (PJ) is the peak-to-peak amplitude for that portion of the deterministic jitter which is periodic, but for which the period is not correlated with any data pattern in the waveform. A single PJ value is determined for each acquisition, by means of RJ-DJ separation analysis. User can specify the jitter frequency and BW limit for each rail in the measurement configuration. The measurement supports Eye diagram and Histogram plots per rail.
EYEHEIGHT	Eye Height is the minimum vertical eye opening at the mid of the unit interval. This measurement is made across the entire record. User can specify the jitter frequency for each rail in the measurement configuration. The measurement supports Eye diagram and Histogram plots per rail.
EYEWIDTH	Eye Width is the minimum horizontal eye opening at the user-specified reference level. This measurement is made across the entire record. User can specify the jitter frequency and BW limit for each rail in the measurement configuration. The measurement supports Eye diagram and Histogram plots per rail.

Table continued…

Measurement	Description
EYEHIGH	Eye High calculates the voltage at a selected horizontal position across the unit interval, for all High bits in the waveform. Configure the measurement to include all bits, only transition bits, or only non-transition bits. A histogram of the Eye High measurement corresponds to a vertical slice through the upper half of a three-dimensional eye diagram. User can specify the jitter frequency and BW limit for each rail in the measurement configuration. The measurement supports Eye diagram and Histogram plots per rail.
EYELOW	Eye Low calculates the voltage at the selected horizontal position across the unit interval, for all Low bits in the waveform. A histogram of the Eye Low measurement corresponds to a vertical slice through the lower half of a three-dimensional eye diagram. User can specify the jitter frequency and BW limit for each rail in the measurement configuration. The measurement supports Eye diagram and Histogram plots per rail.

See also

Configure panel (DPM measurement configuration menu) on page 198

The DDR measurements tab

The DDR tab lists the optional double data rate (DDR) measurements that you can add to the Results bar. DDR measurements include amplitude and timing analysis.

To open the DDR Measurements tab and add a measurement to the Results bar:

1. Tap the **Add New... Measure** button.
2. Tap the **DDR** tab.

To add a measurement to the Results bar:

1. Select the signal source/s.
2. Tap a measurement panel:

   Amplitude Analysis measurement panel

   Timing Analysis measurement panel
3. Select a measurement. If the measurement requires two signal sources, select the sources in the **Source 1** and **Source 2** fields.
4. Tap **Add**. You can also double-tap a measurement to add it to the Results bar.

Amplitude Measurements panel

Measurement	Description
AOS	The total area of the signal above the specified reference level.
AUS	The total area of the signal below the specified reference level.
Vix(ac)	The differential input cross-point voltage measured from the actual crossover voltage and its complement signal to a designated reference voltage. This is measured on a single ended signal.

Table continued…

Measurement	Description
AOS Per tCK	The total area of the signal that crosses the specified reference level calculated over consecutive periods. It is applicable to clock and address/command waveforms only.
AUS Per tCK	The total area of the signal that crosses the specified reference level calculated over consecutive periods. It is applicable to clock and address/command waveforms only.
AOS Per UI	The total area of the signal that crosses the specified reference level calculated over consecutive unit intervals. It is applicable to data and data strobe waveforms only.
AUS Per UI	The total area of the signal that crosses the specified reference level calculated over consecutive unit intervals. It is applicable to data and data strobe waveforms only.

Timing Measurements panel

Measurement	Description
tRPRE	The width of the Read burst preamble. This is measured from the exit of tristate to the first driving edge of the differential strobe.
tWPRE	The width of the Write burst preamble. It is measured from the exit of tristate to the first driving edge of the differential strobe.
tPST	The width of Read or Write burst' postamble. It is measured from the last falling edge crossing the mid reference level to the start of an undriven state (as measured by a rising trend per JEDEC specification).
Hold Diff	The elapsed time between the designated edge of the single ended waveform and the designated edge of a differential waveform. The measurement uses the closest single ended waveform edge to the differential waveform edge that falls within the range limits.
Setup Diff	The elapsed time between the designated edge of a single ended waveform and when the differential waveform crosses its own voltage reference level. The measurement uses the closest single ended waveform edge to the differential waveform edge that falls within the range limits.
tCH(avg)	The average high pulse width calculated across a sliding 200 cycle window of consecutive high pulses.
tCK(avg)	The average clock period across a sliding 200-cycle window.
tCL(avg)	The average low pulse width calculated across a sliding 200 cycle window of consecutive low pulses.

Table continued…

Measurement	Description
tCH(abs)	The high pulse width of the differential clock signal. It is the amount of time the waveform remains above the mid reference voltage level.
tCL(abs)	The low pulse width of the differential clock signal. It is the amount of time the waveform remains below the mid reference voltage level.
tJIT(duty)	The largest elapsed time between tCH and tCH(avg) or tCL and tCL(avg) for a 200-cycle window.
tJIT(per)	The largest elapsed time between tCK and tCK(avg) for a 200-cycle sliding window.
tJIT(cc)	The absolute difference in clock period between two consecutive clock cycles.
tERR(n)	The cumulative error across multiple consecutive cycles from tCK(avg). It measures time difference between the sum of clock period for a 200-cycle window to n times tCK(avg).
tERR(m-n)	The cumulative error across multiple consecutive cycles from tCK(avg). It measures the time difference between the sum of clock periods for a 200-cycle window to n times tCK(avg).
tDQSCK	The strobe output access time from the differential clock. It is measured between the rising edge of the clock before or after the differential strobe Read preamble time. The edge locations are determined by the mid-reference voltage levels.
tCMD-CMD	The elapsed time between two logic states.
tCKSRE	The valid clock cycles required after Self Refresh Entry (SRE) command. Changing the input clock frequency or the supply voltage is permissible only after tCKSRE time when the SRE command is registered.
tCKSRX	The valid clock cycles required before the Self Refresh Exit (SRX) command. Changing the input clock frequency or the supply voltage is permissible provided the new clock frequency or supply voltage is stable for the tCKSRX time prior to SRX command.

See also

DDR measurement configuration menu overview on page 205

Measurement configuration menu overview

Use this configuration menu to add statistics to a measurement badge readout, plot a measurement, and change measurement settings including source, scope (global or local), reference levels, gating, clock recovery, bandwidth filters, results limits, and pass/fail testing.

To open a Measurement configuration menu for a measurement, double-tap a Measurement badge in the Results bar. The configuration menu and panels only show fields and controls relevant to the selected measurement.

The menu opens on the measurement name panel (the name of the measurement), which provides controls to display additional statistics to the measurement badge, display plots of the measurement, and so on. The content of the measurement name panel depends on the measurement. The most common Measurement Name fields are listed in the following table.

Measurement configuration menu fields, controls, and panels

Field, control, or panel	Description
Measurement Statistics (Measurement name panel)	A list of measurement statistics related to the measurement. You can add these statistics to a measurement badge by selecting the **Show Statistics in Badge** control.
Show Statistics in Badge (Measurement name panel)	Adds the listed statistical measurement readouts to the measurement badge readout.
Plots (Measurement name panel)	Buttons that open Plot views of the measurement. Available plots depend on the measurement. Plot types include Time Trend, Histogram, Spectrum, and Eye Diagram (for jitter measurements). To add a plot to the screen, tap the plot button.
Configure panel	Sets the source, label text, and other fields that are specific to each measurement type.
Reference Levels panel	Sets the reference levels and units used to take measurements, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values.
Clock Recovery panel (jitter measurements)	Sets the clock recovery settings for some jitter measurements.
Gating panel	Sets the measurement region (gate) used to take measurements. Select the scope of the gate setting (global or local), and the type of gating to use.
Filter/Limit Results panel	Sets the scope of the filtering setting (global or local), high and low pass filter settings, the range of measurement result limits, and the limit measurement population size.
Pass/Fail Testing panel	Sets the conditions for pass/fail testing for measurements and sets actions to take when a measurement fails.

Use the following links to access information on measurement panel contents.

Measurement Name panel (Measurement configuration menu)

The Measurement Name panel (the name of the measurement) provides controls for adding display statistics to the measurement badge and opening plots of the measurement.

To open the measurement name panel, double-tap a Measurement badge. This is the default panel shown when you open a Measurement settings menu.

The content of the Measurement Name panel depends on the measurement.

Field or control	Description
Measurement Statistics	A list of measurement statistics. You can add these statistics to a measurement badge by selecting the **Show Statistics in Badge** control.
Show Statistics in Badge	Select to add the listed statistical measurement readouts to the measurement badge.
Plots	Adds a Plot view of the measurement value to the screen. Available plots depend on the measurement. Plot types include Time Trend, Histogram, Spectrum, and Eye Diagram (for jitter measurements). Trend adds the trend plot to the Waveform view.

See also

Configure panel (Measurement configuration menu) on page 150

Reference Levels panel (Measurement configuration menu) on page 151

Clock Recovery panel (Measurement configuration menu) on page 152

Gating panel (Measurement configuration menu) on page 157

Filter/Limit Results panel (Measurement Settings menu) on page 158

Pass/Fail Testing panel (Measurement Settings menu) on page 159

Configure panel (Measurement configuration menu)

Use the Configure panel to set the measurement source(s), add a custom name (label) for the measurement, and other parameters.

To open the Configure panel:

1. Double-tap a Measurement badge to open the Measurement configuration menu.
2. Tap the **Configure** panel.

Not all items listed are shown for all measurements; The panel only shows fields and controls relevant to the selected measurement.

Field or control	Description
Source	Sets the signal source used to take the measurement. Tap the field to show the list of available sources. If the measurement requires more than one source, multiple Source fields are displayed.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Signal Type	Sets the signal type (Clock, Data, Auto) of the source signal for some measurements.
Edge, Clock edge	Sets the edge of the signal to use for starting the measurement.
From Edge	Sets the Source 1 waveform edge on which to start the measurement, for two-source measurements.
To Edge	Sets the Source 2 waveform edge on which to stop the measurement, for two-source measurements.
Calculate One Measurement Per	Sets the amount of waveform data to use to calculate one measurement; one measurement across the entire waveform record or one measurement for each cycle of the waveform in the record.
Pattern Detection	Auto attempts to detect the type of pattern and set
Pattern Type	Sets whether the source signal data is a Repeating pattern or an Arbitrary pattern type.
Pattern Length	Sets the pattern length to use when Pattern Type = Repeating.
Window Length	Sets the overall window length to use when Pattern Type = Arbitrary.

Other measurement panels

Measurement Name panel (Measurement configuration menu) on page 149

Reference Levels panel (Measurement configuration menu) on page 151

Clock Recovery panel (Measurement configuration menu) on page 152

Clock Recovery- Advanced Settings configuration menu on page 156

Gating panel (Measurement configuration menu) on page 157

Filter/Limit Results panel (Measurement Settings menu) on page 158

Pass/Fail Testing panel (Measurement Settings menu) on page 159

Reference Levels panel (Measurement configuration menu)

Use the Reference Levels panel to set the scope of the reference level settings (global or local), the reference levels (High, Mid, and Low), the units used to take measurements, and the method used to calculate the Top and Base waveform values. You can set the levels to be the same or different for rising and falling edges.

To open the Reference Levels panel:

1. Double-tap a Measurement badge.
2. Tap the **Reference Levels** panel.

Reference Levels panel- fields and controls

Not all items listed in the table are shown for all measurements; The panel only shows fields and controls relevant to the selected measurement.

Field or control	Description
Reference Levels	Global sets whether the reference levels defined in this measurement apply to all measurements that are set to global (the default setting). Local sets the Reference Level parameters to apply to just this measurement.
Source	Lists the source signals used for each edge of the measurement.
Set Levels In	Sets the method used to set or calculate the High, Mid, and Low reference levels. Select % or Units and use the Multipurpose Knob to set custom reference values.
Levels	Sets the reference levels as specified percentages of the Top and Base waveform measurement. To set custom reference values, tap **Custom**, tap a setting field, and use the Multipurpose Knob to set the different % (relative) or absolute values. **High** and **Low** references are used to calculate rise and fall times. The default High reference is 90% and Low reference is 10%. **Mid** reference is primarily used for measurements between edges such as pulse widths. The default level is 50%.

Table continued…

Field or control	Description
Base Top Method	Sets the method to calculate the waveform Base and Top values, which is then used to calculate the High, Mid, and Low reference levels. **Auto** is the default method, and automatically determines the best Base Top method to use. Most commonly sets the Top Base method to Histogram Mode. **MinMax** Uses the minimum and maximum values in the waveform record to determine the base and top amplitude. Useful on a waveform with low noise and free from excessive overshoot. **Histogram Mean** uses histogram analysis to calculate the mean or average value using all values above and below the waveform midpoint. Top is set to the mean high value, and Base is set to the mean low value. **Histogram Mode** uses histogram analysis to calculate the most common values above and below the waveform midpoint. Top is set to the common high value, and Base is set to the common low value. **Histogram Eye Center** uses histogram analysis to determine the base top amplitude. Creates a histogram of the amplitudes in the center of each bit (unit interval) while ignoring the waveform during bit transitions. The histogram should have a peak at the nominal high level and another peak at the nominal low level. ⚠ **Note:** If you set the Base Top Method to other than Auto, and do not change the Reference Levels mode to Local, many existing measurements, as well as measurements that you add to the Results bar, will use the new Base Top Method values for taking measurements. This may result in measurement values that you are not expecting.
Hysteresis	Sets the threshold margin to the reference level which the signal must cross to be recognized as changing; the margin is the relative reference level plus or minus half the hysteresis. Use hysteresis to filter out spurious events. Tap the field and use the Multipurpose Knob to change the value.

Other measurement settings panels

Measurement Name panel (Measurement configuration menu) on page 149

Configure panel (Measurement configuration menu) on page 150

Clock Recovery panel (Measurement configuration menu) on page 152

Clock Recovery- Advanced Settings configuration menu on page 156

Gating panel (Measurement configuration menu) on page 157

Filter/Limit Results panel (Measurement Settings menu) on page 158

Pass/Fail Testing panel (Measurement Settings menu) on page 159

Clock Recovery panel (Measurement configuration menu)

Clock recovery refers to the process of establishing a reference clock, the edges of which can be used as a basis for timing comparisons. Use the Clock Recovery panel to configure the clock recovery settings for measurements that require a clock signal.

To access the Clock Recovery panel:

1. Double-tap a Measurement badge on the Results bar that uses clock recovery (such as jitter measurements), to open the Meas configuration menu.
2. Tap the **Clock Recovery** panel.

Measurement configuration menu, Clock Recovery panel

Field or control	Description
Clock Recovery	**Global** sets whether the Clock Recovery settings defined in this panel apply to all measurements with clock recovery settings that are set to global (the default setting). **Local** sets the Clock Recovery parameters to apply to just this measurement.
Method	**PLL** simulates the behavior of the specified hardware Phase Locked Loop clock recovery circuit to derive the clock signal. **Constant Clock** uses linear regression so that the recovered clock minimizes the mean squared sum of the Time Interval Error (TIE) for that waveform. **Explicit Clock** derives the reference clock from a specified channel other than the one upon which the measurement is defined.
Standard	Sets the standard to use for the PLL model. Information for a selected standard is listed under the drop-down list. Only available when Method = PLL.
Mode	**Mean** chooses both the frequency and the phase to minimize the mean squared error. Available when Method = Constant Clock. **Median** chooses the phase so that the median error between the recovered and measured edges is zero. Only available when Method = Constant Clock. **Fixed** uses the specified frequency but chooses the phase so that the median error between the recovered and measured edges is zero. Only available when Method = Constant Clock. **Select Explicit Clock-Edge** method if you want to use the edges found in the selected clock source (possibly multiplied up by an integral number). If the Clock Multiplier is set to 1 (the default), only these edges will be used. If the Clock Multiplier is set to a number N other than 1, linear interpolation will be used between each pair of actual edges to create N-1 additional reference edges. The interpolated edge times, combined with the actual edges, give a total of N reference edge times per actual edge. Only available when Method = Explicit Clock. **Select Explicit Clock-PLL** as the clock recovery method if you want to feed the edges from the selected clock source through a PLL rather than using them directly. The actual edges from the clock source will be used to drive a software PLL model, and the edge times coming out of the PLL will be used as the reference edges for the target measurement. If the Clock Multiplier is set to a number N other than 1, the output of the PLL will have N edges per actual edge. Only available when Method = Explicit Clock.

Table continued…

Field or control	Description
Calculate On	**First Acq.** sets the clock-recovery algorithm to choose a new best-fit clock frequency and phase on just the first acquisition. Subsequent acquisitions will choose a best fit on clock phase but retain the clock frequency found in the first acquisition. **Every Acq.** the clock-recovery algorithm to choose a new best-fit clock frequency and phase for each new oscilloscope acquisition. Clearing the measurement results will reset the clock recovery so that both frequency and phase are optimized on the subsequent acquisition. Only available when Method = Constant Clock and Mode = Mean or Median.
Clock Frequency	Sets the clock frequency to use in Fixed mode. The clock in the waveform is ignored; the instrument uses the specified frequency, with the clock phase determined by best fit. **Note:** This method typically results in a closed eye. Only available when Method = Constant Clock and Mode = Fixed.
Loop BW	Sets the PLL loop bandwidth. Displays the Closed Loop bandwidth that has been configured based on the current standard. Only available when Method = PLL.
Clock Source	Sets the source for the explicit clock. Only available when Method = Explicit Clock.
Clock Edge	Set whether the rising, falling or both edges of the clock source should be considered. Only available when Method = Explicit Clock.
Clock Multiplier	Set the number of edges to be used. If the Clock Multiplier is set to 1 (the default), only these edges will be used. If the Clock Multiplier is set to a number N other than 1, linear interpolation will be used between each pair of actual edges to create N-1 additional reference edges. The interpolated edge times, combined with the actual edges, give a total of N reference edge times per actual edge. Only available when Method = Explicit Clock.
Clock Offset	Set to Auto or Manual.
Offset	Sets the clock offset amount relative to data. To compare the reference clock times to the edge times from the data source, some assumptions must be made about how they align. The default assumption is that each data source edge is associated with the reference clock edge to which it is nearest in time. This assumption may not be optimum, for example if the probes for the reference clock and data signal have different cable lengths. Only available when Clock Offset = Manual.

Table continued…

Field or control	Description
PLL Model	Select the PLL model type. The PLL control area provides control over the phase-locked loop used for clock recovery. You can choose the loop bandwidth and the loop order, and if a Type II loop is chosen, you can specify the damping factor. To set the loop bandwidth automatically, based on a serial standard, select PLL: Standard BW as the clock recovery method. From the Standard: b/s list box, select the standard that matches your data link. For example, choose “PCI-E: 2.5” to test a 2.5 Gbit/second PCI Express link. In this case, the PLL bandwidth will be set to 1.5 MHz, which is 1/1667 of the baud rate as specified in PCI Express standard. You can use the PLL Model list box to choose between Type I and Type II loop. A Type I loop has a transfer function that approaches zero frequency with a slope of 1/s and a Type II loop approaches zero frequency with a 1/s2 slope (In much of the PLL literature, these terms are used interchangeably with First-Order and Second-Order loops. For a thorough discussion of loop type versus order, see Frequency Synthesis by Phase Lock, by William Egan). Only available when Method = Explicit Clock and Mode = Explicit Clock - PLL.
JTF BW	Displays or sets the Jitter Transfer Function bandwidth that has been configured based on the current standard. Only available when Method = Explicit Clock and Mode = Explicit Clock - PLL and PLL Model = Type II.
Damping	Sets the damping factor for the PLL. It is enabled only for Type II phase-locked loop. Only available when Method = Explicit Clock and Mode = Explicit Clock - PLL and PLL Model = Type II.
Advanced	Opens the Clock Recovery-Advanced Settings configuration menu to refine the recovered clock signal. See *Clock Recovery- Advanced Settings configuration menu* on page 156. Only available when Method = PLL.

About constant clock recovery

In Constant Clock Recovery, the clock is assumed to be of the form A*sin (2π ft +phase), where the frequency (f) and phase are treated as unknown constants. Once a source waveform has been acquired and the edges extracted, one or both of these constants are determined using linear regression, so that the recovered clock minimizes the mean squared sum of the Time Interval Error (TIE) for that waveform.

About PLL loop BW versus JTF BW

Phase locked loops are characterized according to their bandwidth (BW), and several different bandwidths are commonly used. The terminology used for these bandwidths is described here, since it varies somewhat across different industries.

- Loop BW (or Closed Loop BW) is the frequency at which the closed-loop gain has fallen to -3 dB (half power) relative to unity-gain. The closed-loop gain function has the character of a low-pass filter.
- JTF BW (Jitter Transfer Function BW or Error Function BW) is the frequency below which input jitter to a tracking loop is removed. The JTF BW has a high-pass filter characteristic.

For Type I loops, the Loop BW and the JTF BW are always equal. For Type II loops, these two bandwidths are different, and their ratio depends on the PLL damping factor. You can choose to specify either bandwidth, and the other is displayed for reference.

PLL-based clock recovery

PLL-based clock recovery is implemented using a software model of a hardware PLL circuit, sequentially processing waveform transitions and adjusting the clock period in a feedback loop. This approach means that the transition density of the input signal has subtle effects on the effective bandwidth and damping factor of the feedback loop, just as it does with actual hardware PLLs. The influence of transition density is only relevant for data signals, since clock signals (or data signals with a two bit pattern) have 100% transition density.

Other measurement settings panels

Measurement Name panel (Measurement configuration menu) on page 149

Configure panel (Measurement configuration menu) on page 150

Reference Levels panel (Measurement configuration menu) on page 151

Clock Recovery- Advanced Settings configuration menu on page 156

Gating panel (Measurement configuration menu) on page 157

Filter/Limit Results panel (Measurement Settings menu) on page 158

Pass/Fail Testing panel (Measurement Settings menu) on page 159

Clock Recovery- Advanced Settings configuration menu

Sets advanced clock recovery settings such as defining a nominal data rate or basing clock recovery on a known data pattern.

To open the Clock Recovery-Advanced Settings configuration menu:

1. Double-tap a measurement that requires clock recovery information (such as jitter measurements).
2. Tap the **Clock Recovery** panel
3. Tap the **Advanced** button.

Clock Recovery-Advanced Settings configuration menu, fields and controls

Field or control	Description
Advanced Clock Recovery Method	Sets the method used to recover a clock from the measured signal. Available selections are **None** (default), **Nominal Data Rate** and **Known Data Pattern**.
Bit Rate	Sets the clock bit rate. The **Bit Rate** field is only present when **Advanced Clock Recovery Method** is set to **Nominal Data Rate**.
Pattern File	Drop-down list with 20 most recent pattern files loaded onto the oscilloscope. Available when Advanced Clock Recovery = Known Data Pattern.
Browse	Opens a standard file navigation window. Use to navigate to and select a pattern file.

See also

Measurement Name panel (Measurement configuration menu) on page 149

Configure panel (Measurement configuration menu) on page 150

Reference Levels panel (Measurement configuration menu) on page 151

Clock Recovery panel (Measurement configuration menu) on page 152

Gating panel (Measurement configuration menu) on page 157

Filter/Limit Results panel (Measurement Settings menu) on page 158

Pass/Fail Testing panel (Measurement Settings menu) on page 159

Gating panel (Measurement configuration menu)

Use Gating to confine a measurement to a certain part of a waveform.

To open the **Gating** panel:

1. Double-tap a Measurement badge in the Results bar to open the Measurement configuration menu.
2. Tap the **Gating** panel.

Gating panel, fields and controls

Not all items listed in the table are shown for all measurements; The panel only shows fields and controls relevant to the selected measurement.

Field or control	Description
Gating	Sets whether this measurement's gating settings are **Global** or **Local**. When **Global** is selected, changing anything in this panel causes the same change to all other measurements that also have **Global** selected. When **Local** is selected, settings in this panel only effect this measurement.
Gating Type	Sets the gate type used to take measurements. **None**: Measurements are taken across the entire record. **Screen**: Measurements are taken on the portion of the waveform shown on the display. When zoom is on, the 'display' on which to measure is the zoom window. **Cursors**: Measurements are taken on the portion of the waveform between the cursors. **Logic**: Measurements are taken only when the logical state of a specified waveform is true. **Search**: Measurements are taken between the start and stop times defined by a specified search. Search gating is only valid for DDR3 measurements on a 6 Series MSO instrument. **Time**: Measurements are taken between the times specified in the **Start Gate Time** and **End Gate Time** fields, relative to the trigger point (0 s).
Source	Sets the signal source to use for Logic or Search gates. If **Gating Type** = **Logic**, the **Source** field lists all available sources. If **Gating Type** = **Search**, the **Source** field lists all available searches.
Threshold	Sets the threshold value for the Logic gate source to be considered a logic 1 value.
Hysteresis	Sets the Hysteresis value for the Logic gate source.
Active	Sets the logic state value for the Logic gate source.
Start Gate Time **End Gate Time**	Sets the starting time (Start Gate) and ending time (End Gate) between which to take a measurement. The starting and ending gate times are relative to the trigger time (0 s).

Guidelines

- Only one gated region is allowed per measurement. You can create duplicate measurements and define different gates for each one.
- Tap on a gating-enabled measurement badge to display the vertical gate bars on the waveform for that measurement.
- Gating is indicated by two vertical bars; one at the start of the gated region and one at the end.
- If the measurement has more than one source, gate bars are drawn on both sources.
- Time gates can be placed anywhere. They can precede acquisition start value and extend beyond the acquisition value.
- The Time Gating bars are shown in both the overview and zoomed view when zoom is on.

See also

Measurement Name panel (Measurement configuration menu) on page 149

Configure panel (Measurement configuration menu) on page 150

Reference Levels panel (Measurement configuration menu) on page 151

Clock Recovery panel (Measurement configuration menu) on page 152

Clock Recovery- Advanced Settings configuration menu on page 156

Filter/Limit Results panel (Measurement Settings menu) on page 158

Pass/Fail Testing panel (Measurement Settings menu) on page 159

Filter/Limit Results panel (Measurement Settings menu)

Use these settings to apply a High Pass and/or Low Pass filter to block specified frequency band components when taking measurements. Use the limit controls to set range of measurement values to measure, and the number of measurements to take (population).

To open the **Filter/Limit Results** panel:

1. Double-tap a Measurement badge.
2. Tap the **Filter/Limit Results** panel.

Filter/Limit Results panel fields and controls

Not all items listed in the table may be shown for all measurements; The panel only shows fields and controls relevant to the selected measurement.

Field or control	
Filter Measurement Results	Sets whether this measurement's filter and limits settings are **Global** or Local. When set to **Global** (the default), changing anything in this panel causes the same change to all other measurement filter and limit settings that also have Global selected. When set to **Local**, changing anything in this panel only effects this measurement.
High Pass Filter	Blocks the low frequency band and passes only the high frequency band of the waveform. Select a Butterworth filter order (No Filter (default), 1st, 2nd, or 3rd) and enter the roll-off frequency in the field.
FP Freq (F1)	High Pass filter cut-off frequency at which the filter magnitude falls by 3 dB.

Table continued…

Field or control	
Low Pass Filter	Blocks the high frequency band and passes only the low frequency band of the waveform. Select a Butterworth filter order (No Filter (default), 1st, 2nd, or 3rd) and enter the roll-off frequency in the field.
LP Freq (F2)	Low Pass filter cut-off frequency at which the filter magnitude falls by 3 dB.
Limit Measurement Results	Limit taking measurements results to those that are within the specified Min Value and Max value range.
Limit Measurement Population	Limit measurements to the specified number of measurements.

Other measurement settings panels

Measurement Name panel (Measurement configuration menu) on page 149

Configure panel (Measurement configuration menu) on page 150

Reference Levels panel (Measurement configuration menu) on page 151

Clock Recovery panel (Measurement configuration menu) on page 152

Clock Recovery- Advanced Settings configuration menu on page 156

Gating panel (Measurement configuration menu) on page 157

Pass/Fail Testing panel (Measurement Settings menu) on page 159

Pass/Fail Testing panel (Measurement Settings menu)

Use these settings to test measurement values against a specified limit and set actions for the instrument to take on failure.

To open the **PASS/FAIL TESTING** panel:

1. Double-tap a Measurement badge.
2. Tap the **Pass/Fail Testing** panel.

Pass/Fail Testing panel fields and controls

Not all items listed in the table may be shown for all measurements; The panel only shows fields and controls relevant to the selected measurement.

Field or control	
Pass/Fail Test	Turns pass/fail testing on or off. Default is off.
Fail When	Sets the condition for when the test on the measurement will fail. Default is **< Limit**.
Limit	Sets the measurement limit value. This control is present when is not set to **Inside Range** or **Outside Range**.
Low Limit	Sets the low limit for the measurement value. This control is only present when Fail When is set to **Inside Range** or **Outside Range**.
High Limit	Sets the high limit for the measurement value. This control is only present when Fail When is set to **Inside Range** or **Outside Range**.
Actions on Failure	Opens a configuration menu to set the actions the instrument takes when the test fails. See *Act On Event configuration menu* on page 128.

Table continued…

Field or control	
Badge Navigation	Sets the navigation in the results badge. Tap to toggle to view Failures or All Occurrences. When Failure is selected the navigation buttons move from failure to failure. When All Occurrences is selected, the navigation button moves through each occurrence.

Other measurement settings panels

Measurement Name panel (Measurement configuration menu) on page 149

Configure panel (Measurement configuration menu) on page 150

Reference Levels panel (Measurement configuration menu) on page 151

Clock Recovery panel (Measurement configuration menu) on page 152

Clock Recovery- Advanced Settings configuration menu on page 156

Gating panel (Measurement configuration menu) on page 157

Filter/Limit Results panel (Measurement Settings menu) on page 158

Power measurement configuration menu overview

Use this configuration menu to add statistics to a Power measurement badge readout, plot a measurement, and change measurement settings including source, scope (global or local), reference levels, and gating.

To open the Power measurement configuration menu for a measurement, double-tap a Power measurement badge in the Results bar. The configuration menu and panels only show fields and controls relevant to the selected measurement.

The menu opens on the measurement name panel (the name of the measurement), which provides controls to display additional statistics to the measurement badge, display plots of the measurement, and so on. The measurement name panel only shows fields and controls relevant to the selected measurement. The most common Power measurement Name fields are listed in the following table.

Power Measurement configuration menu fields, controls, and panels

Field, control, or panel	Description
Measurement Statistics (Measurement name panel)	A list of measurement statistics related to the measurement. You can add these to a measurement badge by selecting the **Show Statistics in Badge** control.
Show Statistics in Badge (Measurement name panel)	Adds the listed statistical measurement readouts to the measurement badge readout.
Plots (Measurement name panel)	Buttons that open Plot views of the measurement. Available plots depend on the measurement. Plot types include Time Trend, Histogram, Harmonic bar graph, SOA, Switching Loss Trajectory, B-H curve, I vs. ∫V, Inductance, Power Supply Rejection Ratio, Control Loop Response, and Instantaneous Math. To add a plot to the screen, tap the plot button. See *Add Plot configuration menu* on page 241.

Table continued…

Field, control, or panel	Description
Configure panel	Sets the source, label text, and other fields that are specific to each measurement type. See *Configure panel (Power measurement configuration menu)* on page 162.
Reference Levels panel	Sets the reference levels and units used to take measurements, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values. See *Reference Levels panel (Measurement configuration menu)* on page 151.
Gating panel	Sets the measurement region (gate) used to take measurements. Select the scope of the gate setting (global or local), and the type of gating to use. See *Gating panel (Measurement configuration menu)* on page 157.

Use the following links to access information on measurement panel contents.

Power Measurement Name panel (Measurement configuration menu)

The Power Measurement Name panel (the name of the measurement) provides controls for adding display statistics to the measurement badge and opening plots of the measurement.

To open the power measurement name panel, double-tap a Power Measurement badge. This is the default panel shown when you open a Power Measurement settings menu.

The contents of the Measurement Name panel depends on the measurement.

Note: The Power Measurement Name panel and the Configure panel are merged together for Input Capacitance, Inrush Current, Turn On Time, Turn Off Time, Control Loop Response, Power Supply Rejection Ratio (PSRR), and Impedance measurements.

Field or control	Description
Power Autoset	Sets the oscilloscope acquisition system for optimal results for all active power measurements except Inrush Current, Input Capacitance, Turn-on Time, and Turn-off time. See *Power Autoset*.
Power Preset	Sets the oscilloscope acquisition system for optimal results for Turn On Time, Turn Off Time, Input Capacitance, and Inrush Current power measurements. See *Power Preset* on page 178. Available only for Turn On Time, Turn Off Time, Input Capacitance, Inrush Current, Power Supply Rejection Ratio, and Control Loop Response power measurements.
Measurement Statistics	Shows a list of measurement statistics. You can add these statistics to a measurement badge by selecting the **Show Statistics in Badge** control.
Show Statistics in Badge	Adds the listed statistical measurement readouts to the measurement badge.
Plots	Adds a Plot view of the measurement value to the screen. Available plots depend on the measurement. Plot types include SWL Trajectory, Bar Graph, Time Trend, Histogram, Control Loop Response, PSRR, and SOA. Trend adds the trend plot to the Waveform view.

See also

Configure panel (Measurement configuration menu) on page 150

Reference Levels panel (Measurement configuration menu) on page 151

Gating panel (Measurement configuration menu) on page 157

Configure panel (Power measurement configuration menu)

Use the configure panel to set the measurements source(s), add a custom name (label) for the measurement and other parameters.

To open the Configure panel for a power measurement:

1. Double-tap a power measurement badge to open the Power measurement configuration menu.
2. Tap the **Configure** panel.

di/dt and dv/dt measurement: Configure panel

Field or control	Description
Source	Sets the signal source used to take the measurement. Tap the field to show the list of available sources. Select the current source.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Edge	Sets the signal edges to detect (rise or fall).

Harmonics measurement: Configure panel

Field or control	Description
Standard	Sets the standard to use for measurements. None (no Standard), IEC 61000-3-2, MIL-STD-1399, AM14, or D0-160G (Standard for Airborne equipment. Supported for harmonics measurement for single phase DUT). **Note:** When standard is set as None with the current source, it displays an error as Low Amplitude. Change reflevels method to MinMax manually. This error is displayed when you recall the demo setup files for this measurement.
Harmonics	Sets the harmonics order (number of harmonics) for the selected standard. Ranges from 40 to 100.
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Line Frequency	Sets the line frequency of the power signal source. In Auto mode, the application measures the frequency of the input signal automatically. Input signal is used to compute harmonics in the current source. Default is Auto.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

Table continued…

Field or control	Description
Harmonics Source	Selects the source of harmonics calculation source (Voltage or Current) for computation of the Harmonics.
Power Level (MIL-STD-1399)	Selects the required power level, High or Low. It is used to compute limit values for MIL-STD-1399.
Current (MIL-STD-1399)	Selects either rated or measured. Specify the value for rated or measure the Input current signal.
Start Frequency	Specify the fundamental frequency to be measured for each standard. • **None**: The values are **Auto**, 50, 60, 400, or **Custom** (specify the start Frequency) • **IEC-61000-3-2**: The values are **Auto**, 50, 60, or **Custom** • **MIL-STD-1399**: The values are **Auto**, 400, or **Custom** • **AM-14**: The values are **Auto**, 50, 60, or **Custom** • **D0-160G**: The values are **Auto**, 360, 400, 800, or **Custom** Default is **Auto**. The measured frequency value is displayed in the measurement badge.

Power Quality measurement: Configure panel

Field or control	Description
Voltage Source	Select the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Select the current source used to take the measurement. Tap the field to show the list of available sources.
Label	Set the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Calculate Over Full Cycles	Enables calculating the measurement over the entire acquisition.
Frequency Reference	Select the signal source (Voltage or Current) to use to determine the measurement frequency.

Input Capacitance measurement: Configure panel

Field or control	Description
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Peak Voltage	Sets the peak voltage value of the DUT. **Power Preset** uses the specified peak voltage value to compute the vertical scale.
Peak Current	Sets the peak current value of the DUT. **Power Preset** uses the specified peak current value to compute the vertical scale.

Inrush Current measurement: Configure panel

Field or control	Description
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Peak Current	Sets the peak current value of the DUT. **Power Preset** uses the specified peak current value to compute the vertical scale.

SOA measurement: Configure panel

Field or control	Description
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Define Mask	Defines the linear mask for SOA measurements. See *Define Mask*

Switching Loss measurement: Configure panel

Field or control	Description
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Type	Select SMPS / PFC/ Flyback: • **SMPS**: Select this option in case of signals without noise and ringing. The Vg source is not required. Select Vg source (Source 3), in case of noisy signal. • **PFC**: Select this option when input DUT signals are from Power Factor Correction Circuit. For this case, Vg source is mandatory. • **Flyback**: This option does not need a Vg source and works with switching voltage and current sources. **Note:** For SMPS amd PFC settings to correctly calculate results, set REF Level to "Units" instead of "Percentage".
Gate Voltage (Vg)	Sets the Vg input source, which is a clean signal. Available when Type = **SMPS** or **PFC**.

Table continued…

Field or control	Description
Vg Level Ton - Start	Selects the source of computation of the harmonic. In the voltage source, standard is always **None**. Available when Gate Voltage (Vg) ≠ **None**.
PWM Type	Select **Fixed** or **Variable** based on the varying pulse width of the switching signal.
Conduction calculation	Select **MOSFET** or **BJT/IGBT** semiconductor types.
R_{DS} (on)	If **MOSFET** is selected, then R_{DS}(on) is used to compute for conduction Loss. Available when Conduction Calculation = **MOSFET**
V_{CE} (sat)	If **BJT/IGBT** is selected, then V_{CE}(sat) is used to compute for conduction loss. Available when Conduction Calculation = **BJT/IGBT**.
Set On/Off Levels In:	Sets the REF levels for computation of the T_{ON} and T_{OFF} regions. Levels can be set in % or absolute values.
T_{off} -Stop Current Level	Sets the T_{on}-Start and T_{off}-Stop of the max switch current.
T_{on} -Stop & T_{off} - Start Voltage Level	Sets the voltage level value for T_{off}-Stop and T_{on}. Can be entered as a percent or as a voltage, depending on the setting of the **Set On/Off Levels In:** control.

Inductance measurement: Configure panel

Field or control	Description
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Edge Source	Selects the voltage or current source for computing edges.

Note: When you are measuring the inductance of a transformer, do not load the secondary winding. The measurement of the inductance at the primary winding under no load condition is as good as measuring the inductance for a single winding. When you are measuring the inductance of the coupled inductor with multiple windings on the same core, the measured value of the inductance will deviate from the actual value due to the influence of the current on other windings. You can use this measured value to calculate the Ripple current.

Magnetic Loss and I vs. ∫V measurement: Configure panel

Field or control	Description
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

Efficiency measurement: Configure panel

Field or control	Description
Voltage Source	Selects the input voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the input current source used to take the measurement. Tap the field to show the list of available sources.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
No of Outputs (1, 2, or 3)	Sets the number of outputs on the SMPS DUT, up to a maximum of three. You can configure these outputs in the application, and set up the sources (V and I), so that the efficiency per output and total efficiency for all available outputs are displayed.
Output 1 Voltage	Selects the voltage source for output one when the number of outputs is set to one.
Output 1 Current	Selects the current source for output one when the number of outputs is set to one.

Turn On Time measurement: Configure panel

Field or control	Description
Input Source	Selects the channel connected to the input side of the DUT.
Type	Selects the input to output power conversion type. Default is **DC-DC**.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Maximum Voltage	Sets the maximum input voltage, in the range of 1 V to 500 V.
Input Trigger	Sets the input trigger level, in the range of 1 V to 500 V.
Frequency	Sets the DUT input AC signal frequency, in the range of 1 Hz to 1 MHz. Available when **Type** = **AC-DC**
Maximum Time	Sets the maximum time window in which to capture the turn on time measurement (from the initial state to a steady state), in the range of 1 μs to 500 s, with a resolution of 0.1 μs.
Number of Outputs (1, 2, 3, 4, 5, 6, or 7)	Sets the number of outputs to measure on the DUT (maximum of seven).
Output Source	Selects the channel source to use for the listed output.
Output Voltage	Sets the expected maximum voltage for each listed output, in the range of -6 kV to +6 kV, with a resolution of 0.001 V. The measurement uses this value to set the vertical scale units for each output waveform. The instrument measures the actual maximum voltage on each output, and uses this value to determine the output On/Off state levels, where On is ≥90% of the measured value, and Off is ≤10% of the measured value.

Note: The configuration value for maximum voltage and trigger level will not be remembered when it changes from AC-AC, AC-DC, DC-AC, and DC-DC modes. It displays the default values.

Turn Off Time measurement: Configure panel

Field or control	Description
Input Source	Selects the channel connected to the input side of the DUT.
Type	Selects the input to output power conversion type. Default is **AC-DC**.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Maximum Voltage	Specify the maximum input voltage, in the range of 1 V to 500 V.
Input Trigger	Specify the input trigger level, in the range of 1 V to 500 V.
Frequency	Specify the DUT input AC signal frequency, in the range of 1 Hz to 1 MHz. Available when **Type** = **AC-DC**
Maximum Time	Sets the maximum time window to capture the turn off time measurement (from the initial state to a steady state), in the range of 1 μs to 5 s, with a resolution of 0.1 μs.
Number of Outputs (1, 2, 3, 4, 5, 6, or 7)	Sets the number of outputs to measure on the DUT (maximum of seven).
Output Source	Selects the channel source to use for the listed output.
Output Voltage	Sets the expected maximum voltage for each listed output, in the range of -6 kV to +6 kV, with a resolution of 0.001 V. The measurement uses this value to set the vertical scale units for each output waveform. The instrument measures the actual maximum voltage on each output, and uses this value to determine the output On/Off state, where ≥90% of the measured value = On, and ≤10% of the measured value = Off.

$RDS_{(on)}$ measurement: Configure panel

Field or control	Description
Voltage Source	Selects the source used to take the voltage measurement. Tap the field to show the list of available sources.
Current Source	Selects the source used to take the current measurement. Tap the field to show the list of available sources.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Device Type	Sets the semiconductor (transistor device) type. **Switching (v/i)** is the ratio of voltage to current in Ohms. **PN Junction / Diode (dv/di)** is the ratio of rate of change of voltage to current in Ohms.

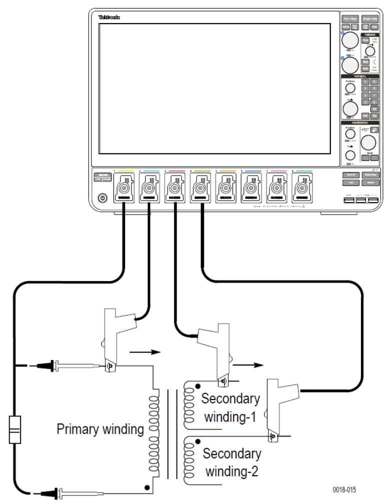
Magnetic Property measurement: Configure panel

Field or control	Description
Primary Voltage Source	Selects the source used to take the voltage measurement. Tap the field to show the list of available sources.

Table continued…

Field or control	Description
Primary Current Source	Selects the source used to take the current measurement. Tap the field to show the list of available sources.
Primary Turns	Sets the number of turns used on the primary winding. Tap the field and use the knob to set the value, or double-tap on the field and use the keyboard to enter a value.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Edge Source	Sets the source used to detect the edge. Select either **Voltage** or **Current**. Default is **Voltage**. ⚠ **Note:** For a variable switching operation, connect the gated drive signal to the edge source, as in variable mode the amplitude of the voltage acquired across the inductor varies with time. It is recommended to use gate drive signal as it is clean with no variations.
Units	Sets the measurement units. **SI** sets the dimensions of the component in meters and units of magnetics in Tesla and Amperes Turns per meter. **CGS** sets the unit of measurement to centimeters, and the result units are in Gauss and Oersted.
Cross Section	Sets the cross section dimensions of the magnetic component. The values are: • 1 nm^2 to 1 Mm^2 for **SI** • 1 ncm^2 to 1 Mcm^2 for **CGS**
Magnetic Length	Sets the cross section dimensions of the magnetic length. The values are: • 0 m to 1 Mm for **SI** • 0 cm to 1 Mcm for **CGS**
Voltage Phase	Set the secondary voltage phase.

Table continued…

Field or control	Description
Number of sec windings	Sets the number of secondary windings to measure. The range is from 1 to 6 if enable voltage sources is not checked, otherwise it is 1 to 3 secondary windings. Measure the magnetic property of secondary windings such as in a coupled inductor or a transformer that has multiple windings on the same core. Follow the connection combinations of the voltage and current probe at the primary or the main winding at the secondary or the other windings to the · dot as shown in the following figure. The voltage probe should be connected such that the voltage is read as positive when the current rises.  **Warning:** When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components used are within their ratings. For more information, refer to the topic *General safety summary*.
Winding (1-6)	Sets the signal source used to measure the selected winding. The range is from 1 to 6.
Turns (1-6)	Sets the number of turns in the respective secondary winding.
Enable Voltage Source	Configures the transformer secondary voltage source when **Enable Voltage Source** is checked.
Secondary Voltage Source	Selects the source used to take the secondary voltage. Tap the field to show the list of available sources.
Secondary Current Source	Selects the source used to take the secondary current. Tap the field to show the list of available sources.

Amplitude Analysis, Ripple Analysis, Timing Analysis power measurements: Configure panel

Field or control	Description
Source	Sets the signal source or sources used to take the measurement. If the measurement requires more than one source, multiple source fields are displayed. Tap the field to show the list of available sources.

Table continued…

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Edge	Sets the signal edge to detect (rise or fall). Used in most **Timing Analysis** measurements. Available when the power measurement = **Frequency**.
Clock Edge	Sets the clock signal edge to detect (rise, fall, or either). Used in **Timing Analysis** measurements. Available when the power measurement = **Positive Duty Cycle** or **Negative Duty Cycle**.
Line Frequency	Sets the line frequency of the power signal source. **Power Autoset** uses the selected frequency to setup the scope acquisition parameters. Available when the power measurement = **Line Ripple**.
Custom Frequency	**Custom Frequency** Sets the custom frequency of the power signal source. Available only when **Line Frequency** = **Custom**.
Switching Frequency	Specify the operating frequency of the switching device. **Power Autoset** uses the selected frequency to set the oscilloscope acquisition parameters. Frequency range is 50 Hz to 1 MHz. Only available when the power measurement = **Switching Ripple**. **Note:** Ripple RMS is measured at the configured ripple frequency.

Control Loop Response measurement: Configure panel

See *Connection setup for Frequency Response Analysis (FRA) measurements* on page 174.

Field or control	Description
Input Source	Selects the channel connected to the input side of the DUT.
Output Source	Selects the channel connected to the output side of the DUT.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Generator	Generates stimulus, as a series of sine waves with configured amplitude, which are sent to the input of the DUT. Supported signal generator is Internal and External signal source. AFG31000 series and AFG 3K series are the supported external signal source.
Connection	Opens AFG configuration menu. Enter the IP address and click **Test Connection** to connect to the signal generator. Available only when Generator = **External**.
Impedance	User configurable Impedance for the generator.

Table continued…

Field or control	Description
Points Per Decade	Sets the number of frequency points between the start and stop frequency in terms of log scale. The minimum value is 10, maximum value is 100, and the default is 10.
Start/Stop Frequency	Sets the generator start and stop frequency. The maximum stop frequency depends on the generator. The start frequency is 100 Hz, the stop frequency is 20 MHz, and the default is 100 Hz.
Amplitude Mode	Sets the built-in signal generator output amplitude mode. **Constant**: Sets the constant amplitude to all frequencies in the in-built signal generator. **Profile**: Opens a configuration menu that lets you set the amplitude for each frequency band. The generator uses the values in the table to set the output amplitude as the frequency changes.
Configure Profile	Opens **Profile** configuration menu to enter the frequency range with amplitude values. See *Configure Profile* on page 176. Available only when Amplitude mode = **Profile**.
Amplitude	Sets the generator to apply the same amplitude to all frequencies output from the generator. The minimum value is 20 mV as per internal generator, the maximum value is whatever the generator supports, and the default is 100 mV. Available when Amplitude mode = **Constant**.
Analysis Method	Allows you to toggle between FFT or Spectrum view. Default is FFT.

Note: Math sources are not supported for Control Loop Response (BODE), PSRR, and Impedance measurements, so do not configure Math for these measurements.

Power Supply Rejection Ratio (PSRR) measurement: Configure panel

See *Connection setup for Frequency Response Analysis (FRA) measurements* on page 174.

Field or control	Description
Generator	Generates stimulus, as a series of sine waves with configured amplitude, which are sent to the input of the DUT. Supported signal generator is Internal and External signal source. AFG31000 series and AFG 3K series are the supported external signal source.
Connection	Opens AFG configuration menu. Enter the IP address and click **Test Connection** to connect to the signal generator. Available only when Generator = **External**.
Impedance	Sets the output impedance of the built-in signal generator

Table continued…

Field or control	Description
Points Per Decade	Sets the number of frequency points between the start and stop frequency in terms of log scale. The minimum value is 10, maximum value is 100, and the default is 10.
Start/Stop Frequency	Sets the generator start and stop frequency. The maximum stop frequency depends on the built-in signal generator. The start frequency is 100 Hz, the stop frequency is 20 MHz, and the default is 100 Hz.
Amplitude Mode	Sets the built-in signal generator output amplitude mode. **Constant**: Sets the constant amplitude to all frequencies in the built-in signal generator. **Profile**: Opens a configuration menu that lets you set the amplitude for each frequency band. The generator uses the values in the table to set the output amplitude as the frequency changes.
Configure Profile	Opens **Profile** configuration menu to enter the frequency range with amplitude values. See *Configure Profile* on page 176. Available only when Amplitude mode = **Profile**.
Amplitude	Sets the generator to apply the same amplitude to all frequencies output from the generator. The minimum value is 20 mV as per internal generator, the maximum value is whatever the generator supports, and the default is 100 mV. Available when Amplitude mode = **Constant**.
Analysis Method	Allows you to toggle between FFT or Spectrum view. Default is FFT.

Impedance measurement: Configure panel

See *Connection setup for Frequency Response Analysis (FRA) measurements* on page 174.

Field or control	Description
Splitter	Select the splitter as **Passive** or **Active**. By default splitter is Active. *Connection setup for Frequency Response Analysis (FRA) measurements* on page 174 to view the connection diagram for active and passive splitter.
Generator	Generates stimulus, as a series of sine waves with configured amplitude, that is sent to the DUT input. Supported signal generator is Internal and External signal source. AFG31000 series and AFG 3K series are the supported external signal source.
Connection	Opens the AFG configuration menu. Enter the IP address of the external generator and click **Test Connection** to connect to the signal generator. Available only when Generator = **External**.
Impedance	Sets the output impedance of the built-in signal generator to 50 Ω.

Table continued…

Field or control	Description
Points Per Decade	Sets the number of frequency points between the start and stop frequency in terms of log scale. The minimum value is 10, maximum value is 100, and the default is 10.
Start/Stop Frequency	Sets the generator start and stop frequency. The maximum stop frequency depends on the built-in signal generator. The start frequency is 100 Hz, the stop frequency is 20 MHz.
Amplitude Mode	Sets the built-in signal generator output amplitude mode. **Constant**: Sets the constant amplitude to all frequencies in the built-in signal generator. **Profile**: Opens a configuration menu that lets you set the amplitude for each frequency band. The generator uses the values in the table to set the output amplitude as the frequency changes.
Configure Profile	Opens the **Profile** configuration menu to enter the frequency range with amplitude values. See *Configure Profile* on page 176. Available only when Amplitude mode = **Profile**.
Amplitude	Sets the generator output to apply the same amplitude to all frequencies. The minimum value is 20 mV for the internal generator, the maximum value is whatever the generator supports, and the default is 100 mV. Available when Amplitude mode = **Constant**.
Analysis Method	Allows you to toggle between FFT or Spectrum view. Default is FFT.

Note: For Impedance measurement, you are recommended to use constant amplitude for all frequencies.

Other measurement configuration panels

Reference Levels panel (Measurement configuration menu) on page 151

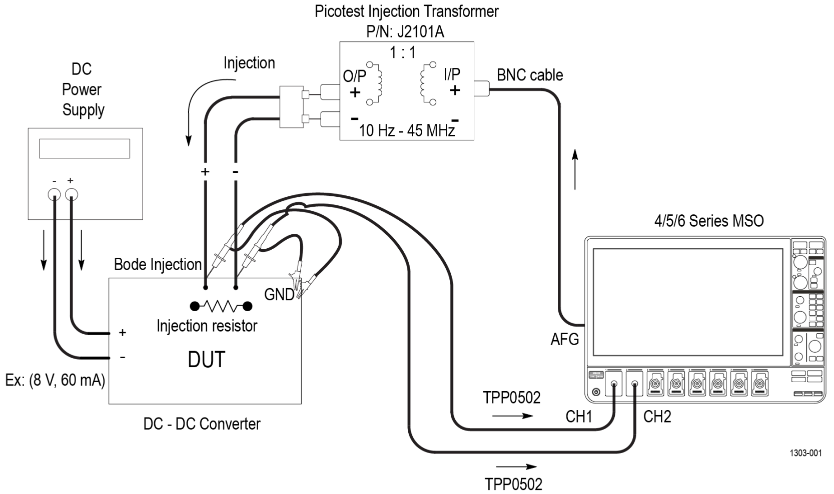
Power Measurement Name panel (Measurement configuration menu) on page 161

Configure panel (Measurement configuration menu) on page 150

Gating panel (Power measurement configuration menu) on page 184

Connection setup for Frequency Response Analysis (FRA) measurements

Frequency Response Analysis (FRA) measurements include Control Loop Response (Bode), Power Supply Rejection Ratio (PSRR), and Impedance measurements.



Note: To perform BODE measurement, inject stimulus signal from AFG over a band of frequencies to the control loop feedback path of the power convertor. To facilitate the signal injection, a small resistor needs to be inserted in the feedback loop. The injection resistor value should be of < 10 Ohms (recommended value is around 5 Ohms).

Figure 15: Connection diagram for Bode measurement

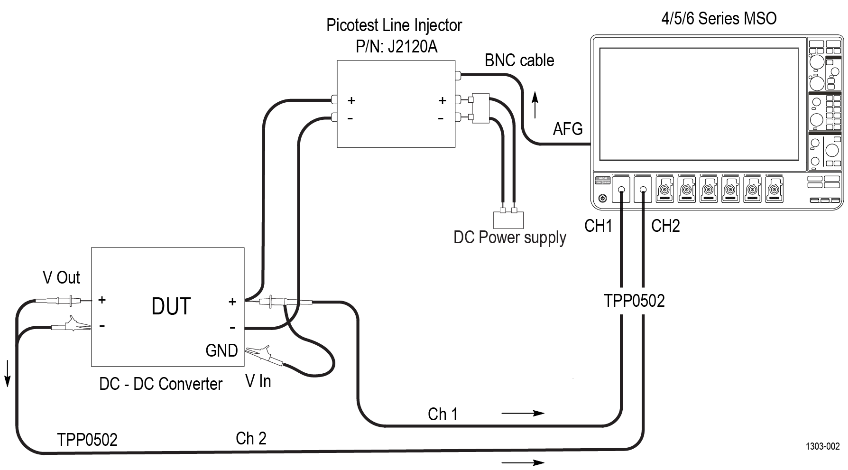
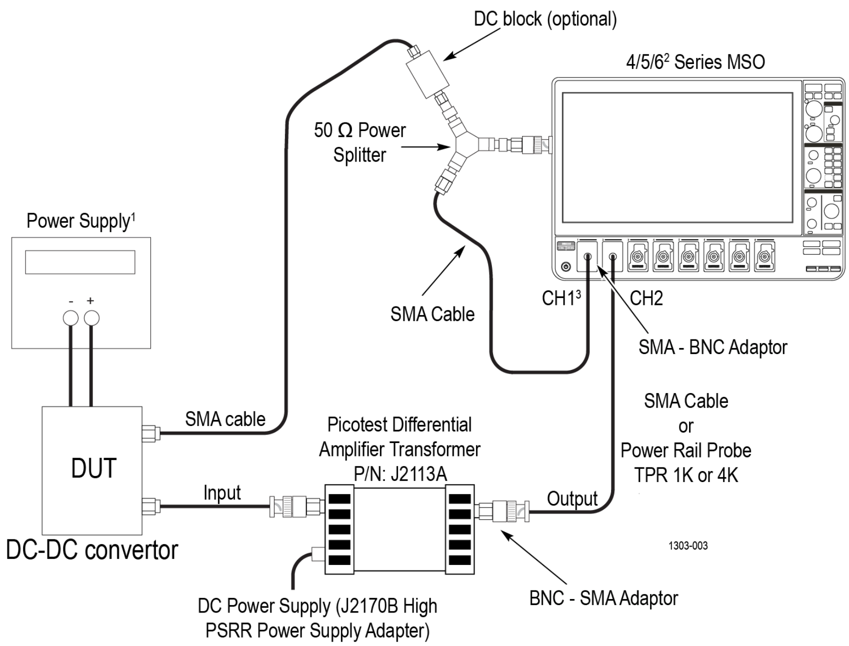
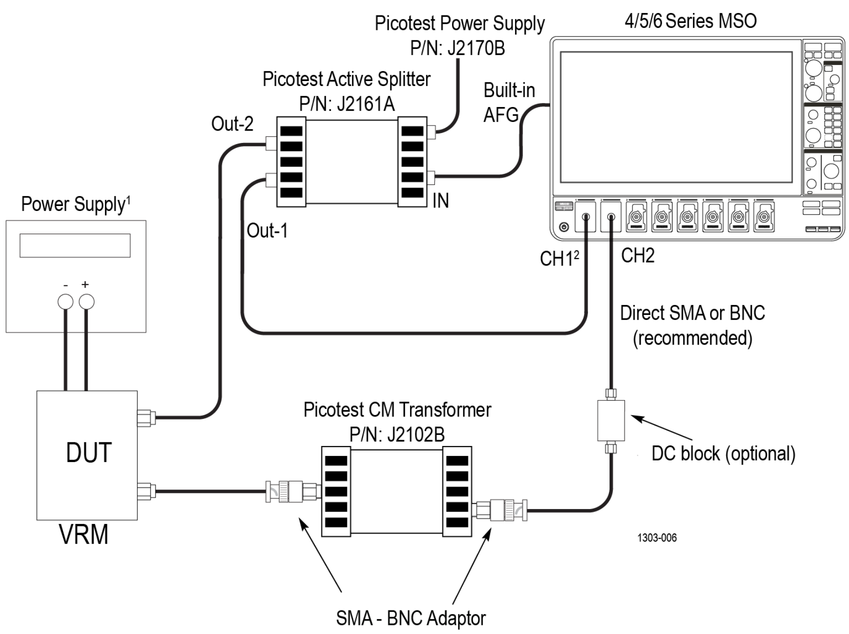


Figure 16: Connection diagram for PSRR measurement



[1]Source of power supply can be a DC power supply unit or USB connector
[2]6 Series MSO Oscilloscope contains 4 channels only
[3]It is recommended to use DC block at CH1/CH2 of the oscilloscope, if there is a DC offset in the signal.

Figure 17: Connection diagram for Impedance measurement (Passive Splitter)



[1]Source of power supply can be a DC power supply unit or USB connector
[2]It is recommended to use DC block at CH1/CH2 of the oscilloscope, if there is a DC offset in the signal.

Figure 18: Connection diagram for Impedance measurement (Active Splitter)

FRA measurements are supported for 4/5/6 oscilloscope models.

Note:

1. It is recommended to use a splitter with the built-in AFG source for Impedance measurements. If you use an external AFG, use a single channel with a splitter.

2. Negative values are not displayed in the impedance plot. They are replaced by the most previous positive value.

Use the following steps to make the FRA measurements:

1. Make the connections as shown in the diagram.
2. Power on the DUT and transformer, if required.
3. Configure the measurements and analyze the results.

Connect to the AFG

Use the Generator Connection Information menu to connect to the signal generator. This option is available only when the generator configuration is set to external.

Field or control	Description
IP address	Enter the IP address of the instrument.
Test Connection	Click to test the connection with the instrument.

Note: It is recommended to use the built-in AFG with splitter for Impedance measurement. If you use an external AFG, use single channel with splitter.

Configure Profile

Use the Configure Profile menu to configure the amplitude profile for the Power Supply Rejection Ratio (PSRR), Control Loop Response, and Impedance measurements.

It is recommended to customize the built-in signal generator amplitude values using the amplitude profile configuration for these measurements. With profiling you can set the generator at lower amplitudes, at different frequencies where the DUT is sensitive, and set higher amplitudes where the DUT is less sensitive to distortion. In general, a Switched-mode Power Supply (SMPS) is very sensitive near the zero degree cross over frequency, so it is advisable to define a profile, instead of using a constant amplitude output signal.

Use the parameters to configure the profile for PSRR and Control Loop Response measurements.

Field or control	Description
Start	Displays the start frequency.
Stop	Displays the stop frequency.
Amplitude	Shows the amplitude value that is used to set the AFG level.
Insert Step	Inserts an additional row displaying start, stop, and amplitude values.
Delete Step	Deletes the selected row.
Clear Table	Clears all the values in the table.

Use the following steps to configure the amplitude profile for a PSRR and Control Loop Response measurement:

1. Click **Configure Profile** button. Displays the configure profile dialog. By default the configure profile dialog displays two rows:

- The Start Frequency of the first row is the Start Frequency from the PSRR measurement configure panel.
- The Stop Frequency of the first row is the mean of Start and Stop Frequency values from the PSRR/Control Loop Response measurement configure panel.
- The Start Frequency of the second row is 'greater' than the Stop Frequency from the PSRR/Control Loop Response/Impedance measurement configure panel.
- The Stop Frequency of the second row is the Stop Frequency from the PSRR/Control Loop Response measurement configure panel.

2. You can change the values, to reduce the variations in the gain curve at particular frequencies where the DUT is sensitive to distortion. This can be done by making configuration changes and viewing the response in the gain/phase curve of the Control Loop Response/PSRR plot.
3. Click anywhere outside the configure profile table to account for the updates.

Below are the sample amplitude values for a DC-DC convertor DUT.

Table 4: Sample Amplitude values

Start	Stop	Amplitude
10 Hz	100 Hz	1 V
100 Hz	500 Hz	800 mV
500 Hz	1 kHz	600 mV
1 kHz	10 kHz	400 mV
10 kHz	100 kHz	350 mV
100 kHz	1 MHz	300 mV
1 MHz	10 MHz	250 mV
10 MHz	20 MHz	200 mV

Note: The stop frequency can be configured up to 50 MHz which is maximum value of build-in AFG.

Power Autoset

Sets the oscilloscope acquisition system for optimal measurement results for all active power measurements except Turn On Time, Turn Off Time, Input Capacitance, Inrush Current, Control Loop Response, Power Supply Rejection Ratio (PSRR), and Impedance measurement.

Use the following steps to run a **Power Autoset** on applicable power measurements:

1. Add the power measurements that you want to take to the Results bar.
2. Configure each power measurement individually (input voltage sources, current sources, label name, and so on).
3. Connect the input signals to the instrument and confirm the waveforms are correct and not clipping.
4. Double-tap the power measurement result badge for which you want to take a measurement.
5. Tap the **Power Autoset** button for the measurement and wait for the busy indicator to disappear. The instrument has now been optimized for that power measurement.

Note:

- In case of failure, the instrument displays a popup error message. See *Errors and Warnings*.
- When different frequency signals are connected to different channels, **Power Autoset** uses the AC signal of the lowest-numbered channel to evaluate and set up the instrument parameters. For example, If Channel 1 is connected to a DC signal, Channel 2 is connected to an AC signal of 1 MHz, and Channel 4 is connected to an AC signal of 1 kHz, then **Power Autoset** uses Channel 2 (the first channel with an AC

signal) to set up the instrument parameters. Channel 4 may not be set up to see the minimum number of cycles properly; you may need to manually set the Channel 4 parameters.

- **Power Autoset** runs all parts of instrument Autoset, even if some parts have been disabled in the **User Preferences** > **Autoset** panel. See *Autoset panel fields and controls* on page 382 for more information on the Autoset function.
- **Power Autoset** is available per measurement. You should run the Power Autoset for each measurement separately when you add measurements, to ensure optimum power measurement setup for that measurement.

6. Record the power measurement after the **Power Autoset** completes.
7. Repeat steps *4* on page 177 - *6* on page 178 for other Power measurements that you want to take.

Power Preset

Sets the oscilloscope vertical, horizontal and acquisition parameters using the user inputs like peak voltage and current for Input Capacitance, Inrush Current, Turn On Time, and Turn Off Time. For Control Loop response and PSRR measurements, it sets the built-in generator such as start frequency, impedance, and amplitude values and also sets the vertical and horizontal acquisition parameters, so that oscilloscope is ready to acquire the desired waveform once the measurement starts.

Note: The **Power Preset** button is available only for Input Capacitance, Inrush Current, Turn On Time, Turn Off Time, Control Loop Response, and Power Supply Rejection Ratio power measurements.

Use the following steps to run a **Power Preset** on applicable power measurements:

1. Add the supported power measurements to the **Results** bar.
2. Double-tap each Measurement badge and configure the measurement (input voltage sources, current sources, label name, and so on).
3. Connect the input signals.
4. Tap the **Power Preset** button in the measurement configuration to set.

   To set the vertical and horizontal acquisition parameters:

   a. Measurement configuration begins and displays a window (displayed only for FRA measurements).

      Note:

      - When the measurement runs, you cannot operate oscilloscope front panel buttons.
      - You can change the ACQ modes to Sample mode after **Power Preset** is clicked and before oscilloscope **RUN/STOP** is performed

   b. To stop the measurement configuration or to operate the front panel buttons, click the **Abort** button on the window.

      Note:

      - When the FRA measurement is configured, the window disappears.
      - When FRA measurement is run with Spectrum view and if the Acquisition subsystem error message is displayed, click OK button and proceed with the measurement. There is no need to power cycle the instrument as indicated in the message.

   c. Tap **Run/Stop** on the front panel of the oscilloscope.
5. Follow the instructions on the Measurement badge to take the power measurement.

SOA Mask definition controls and fields

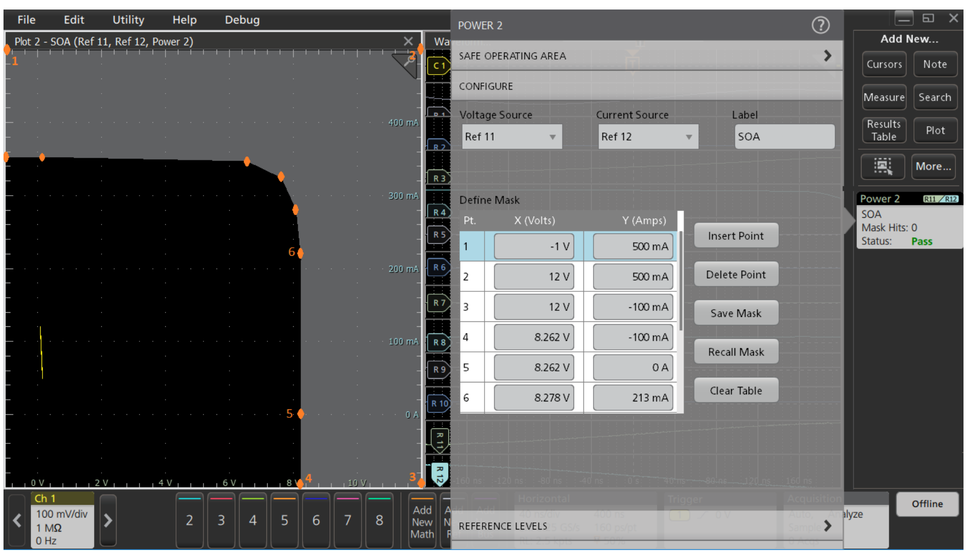
Use the SOA Mask dialog to configure the parameters to add point, delete point, save mask, and recall mask.

Use the parameters to define the linear mask for an SOA measurement.

Define Mask fields and controls

Field or control	Description
X (Volts)	Define voltage values for the mask point.
Y (Amps)	Define current values for the mask point.
Insert Point	Add voltage and current points to define mask. Points are added to the end of the existing list.
Delete Point	Deletes the selected point data row.
Save Mask	Opens the **Save As** menu to navigate to and select the location at which to save the SOA mask data as a .pwrmsk file.
Recall Mask	Opens the **Open** menu to navigate to and select the location from which to recall (load) the SOA mask data .pwrmsk file. **Note:** Mask files created with oscilloscope firmware version 1.4.x cannot be used in firmware version 1.6.x. Use version 1.6.x and later to create and recall mask files.
Clear Table	Clears the values of the mask coordinates in the table.

The following image shows the default SOA mask and its associated mask point table.



Note: You need to define both internal and external points.

Save Mask menu (SOA power measurement)

Use this menu to save a SOA power measurement mask file to a specified location.

Prerequisite: Open the **Configure** panel of the SOA measurement for which you want to save a mask file (.pwrmsk).

To open the Save Mask configuration menu:

1. Tap the **Save Mask** button to open the **Save As** menu.

2. Use the menu fields and controls to navigate to and select the location to save the SOA mask file.

Save As configuration menu (SOA mask file)

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Use the **Browse** button to quickly navigate to the location to which to save the file. Or you can tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Tap to open the Browse Save As Location configuration menu, to navigate to the location to which to save the file. See *Browse Save As Location configuration menu*.
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or the default value of Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save As Type	You can only save SOA mask files as type .pwrmsk.
Cancel	Cancels the file save action and closes the configuration menu. You can also cancel the save operation by tapping anywhere outside the menu.
Save	Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.

Recall Mask menu (SOA power measurement)

Use this menu to recall (load) a .pwrmsk mask file for an SOA power measurement plot.

Prerequisite: Open the **Configure** panel of the SOA measurement for which you want to recall a mask (pwrmsk) file.

Note: Mask files created with oscilloscope firmware version 1.4.x cannot be used in firmware version 1.6.x. Use version 1.6.x and later to create and recall mask files.

To open the Recall Mask configuration menu:

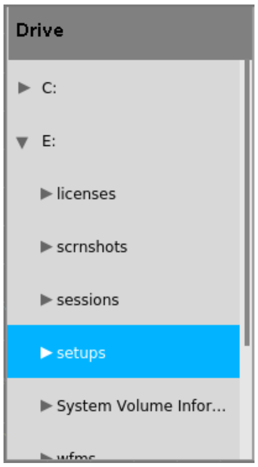
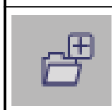
1. Tap the **Recall Mask** button to open the **Recall** menu.
2. Use the menu fields and controls to navigate to and select the mask file to recall.

File operations and Microsoft Windows 10 Operating System SSD

Instruments with Windows 10 SSD will display the standard Windows file tools to navigate to and select files and folders.

The Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. This is different from standard instruments (without Windows 10 installed), which assign a non-changing drive letter to each USB port.

Recall file configuration menu (SOA mask file)

Field or control	Description
Look in:	Shows the current directory path to the location of a file. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, up to a maximum of 20 locations.
	The Drive column lists the directory structure, opening at the root (/) level. Use to quickly navigate to a location. Tap to list the contents of the directory in the Name pane. Double-tap an item to display the directory and any subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
	Use the arrow buttons to navigate the file directory. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder. The Up arrow navigates up one level from the current folder.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File name	Lists the selected file name to recall. Tap on the file name in the Name column to add it to this field.
Files of type	The SOA mask file type (.pwrmsk) cannot be changed.
Cancel	Cancels the file recall action and closes the menu.
Recall	Recalls the selected SOA mask file and plots it on the SOA Plot view.

USB port drive names and locations

Use the following table to determine which drive to select when navigating to and/or selecting a file on system memory or a connected USB memory device.

Drive name	Drive letter	Drive or physical USB port location
MSO44, MSO46		
Root drive	**C**	User-accessible memory on the oscilloscope.

Table continued…

Drive name	Drive letter	Drive or physical USB port location
Front panel	E	USB 2.0 (top)
	F	USB 2.0 (middle)
	G	USB 2.0 (bottom)
Rear panel	H	USB 2.0 (left)
	I	USB 2.0 (right)

Drive name	Drive letter	Drive or physical USB port location
MSO54, MSO56, MSO58, MSO64, LPD64 without Windows OS		
Root drive	C	User-accessible memory on the oscilloscope.
Front panel	E	USB 3.0 (left)
	F	USB 2.0 (center)
	G	USB 2.0 (right)
Rear panel	H	USB 2.0 (top)
	I	USB 2.0 (bottom)
	J	USB 3.0 (top)
	K	USB 3.0 (bottom)
Instruments with Windows OS		
Root drive	C	User-accessible memory on the oscilloscope.
USB ports	Dynamic port letter assignment	If Windows operating system is installed, the Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. Use standard Windows procedures to mount and access network drives.

Drive name	Drive letter	Drive or physical USB port location
MSO58LP without Windows OS		
Root drive	C	User-accessible memory on the oscilloscope.
Front panel	E	USB 3.0 (left)
	F	USB 2.0 (right)
Rear panel	G	USB 2.0 (top)
	H	USB 2.0 (bottom)
	I	USB 3.0 (top)
	J	USB 3.0 (bottom)
Instruments with Windows OS		
Root drive	C	User-accessible memory on the oscilloscope.

Table continued…

Drive name	Drive letter	Drive or physical USB port location
USB ports	**Dynamic port letter assignment**	If Windows operating system is installed, the Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. Use standard Windows procedures to mount and access network drives.

Drive name	Drive letter	Drive or physical USB port location
LPD64 without Windows OS		
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**F**	USB 3.0 (left)
	E	USB 2.0 (center)
	G	USB 2.0 (right)
Rear panel	**H**	USB 2.0 (top)
	I	USB 2.0 (bottom)
	J	USB 3.0 (top)
	K	USB 3.0 (bottom)
Instruments with Windows OS		
Root drive	**C**	User-accessible memory on the oscilloscope.
USB ports	**Dynamic port letter assignment**	If Windows operating system is installed, the Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. Use standard Windows procedures to mount and access network drives.

Reference Levels panel (Power measurement configuration Menu)

Use the Reference Levels panel to set the scope of the reference level settings (global or local), the measurement reference levels (High, Mid, and Low), the units used to take measurements, and the method used to calculate the Top and Base waveform values. You can set the reference levels to be the same or different for rising and falling edges.

See *Reference Levels panel (Measurement configuration menu)* on page 151 for the Reference panel fields and controls.

Other measurement settings panels

Power Measurement Name panel (Measurement configuration menu) on page 161

Configure panel (Power measurement configuration menu) on page 162

Gating panel (Power measurement configuration menu) on page 184

Gating panel (Power measurement configuration menu)

Use Gating to confine a measurement to a certain part of a waveform.

See *Gating panel (Measurement configuration menu)* on page 157 for the Gating panel fields and controls.

See also

Power Measurement Name panel (Measurement configuration menu) on page 161

Reference Levels panel (Power measurement configuration Menu) on page 183

Configure panel (Power measurement configuration menu) on page 162

IMDA measurement configuration menu overview

Use this configuration menu to configure the measurement that is added in the IMDA tab. You can plot a measurement, and change measurement settings including source, scope (global or local), reference levels, and gating.

Prerequisites:

To get accurate results, do the following steps:

- Degauss current probe
- Deskew voltage and current channels
- Perform Oscilloscope SPC

To open the IMDA measurement configuration menu for a measurement, double-tap a IMDA measurement badge in the Results bar. The configuration menu and panels only show fields and controls relevant to the selected measurement.

The menu opens on the measurement name panel (the name of the measurement), which provides controls, display plots of the measurement, and so on. The measurement name panel only shows fields and controls relevant to the selected measurement.

1. Select the Sources of interest.
2. Select Edge configuration.
3. Provide information about the abc for electrical analysis order configured as the ABC connection on the DUT and xyz for output analysis measurements.
4. Configure the LPF and its order with the cut off frequency.
5. Configure REF levels based on noise on the acquired signals.
6. Configure gating to place cursors at the integral cycles.

IMDA Measurement configuration menu fields, controls, and panels for Motor Analysis

Field, control, or panel	Description
Plots (Measurement name panel)	Adds Phasor plot by default when Power Quality and DQ0 is added. Buttons that open Plot views of the measurement. Available plots depend on the measurement. Plot types include Phasor Diagram and Harmonics. **Note:** For PQ, Harmonics, DQO and Efficiency measurements Filtered Math waveform is added in the waveform view. To add a plot to the screen, tap the plot button. See *Add Plot configuration menu* on page 241.
Configure panel	Sets the label text, and other fields that are specific to each measurement type. See *Configure panel (Measurement configuration menu)* on page 150.
Reference Levels panel	Sets the reference levels and units used to take measurements, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values. See *Reference Levels panel (Measurement configuration menu)* on page 151.
Gating panel	Sets the measurement region (gate) used to take measurements. Select the scope of the gate setting (global or local), and the type of gating to use. See *Gating panel (Measurement configuration menu)* on page 157.

Use the following links to access information on measurement panel contents.

IMDA Measurement Name panel (Measurement configuration menu)

The IMDA Measurement Name panel (the name of the measurement) provides controls for adding display statistics to the measurement badge and opening plots of the measurement.

To open the IMDA measurement name panel, double-tap a IMDA Measurement badge. This is the default panel shown when you open a IMDA Measurement settings menu.

The contents of the Measurement Name panel depends on the measurement.

Field or control	Description
Three Phase Autoset	Sets the oscilloscope for optimal Horizontal, Vertical and Acquisition parameters for configured three phase wiring, based on the measurement. Three Phase Autoset sets up voltages and current sources based on the wiring configuration. The Autoset optimally set up the vertical, horizontal, acquisition, and trigger parameters on the oscilloscope and will turn off all the unused channel sources and turn on the configured sources.
Information	Displays the warning details.
Plots	Adds a Plot view of the measurement value to the screen. Available plots depend on the measurement. Plot types include Phasor Diagram and Harmonics.

See also

Configure panel (Measurement configuration menu) on page 150

Reference Levels panel (Measurement configuration menu) on page 151

Gating panel (Measurement configuration menu) on page 157

Configure panel (IMDA measurement configuration menu) - Motor Analysis

Use the configure panel to add a custom name (label) for the measurement and other parameters.

To open the Configure panel for a IMDA measurement:

1. Double-tap a IMDA measurement badge to open the IMDA measurement configuration menu.
2. Tap the **Configure** panel.

Direct Quadrature Zero Transformation (DQ0-IMDA DQ0 license is required)

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Source Type	Toggle between voltage source and current source.
Voltage Source	Select the input or output voltage.
Current Source	Select the input or output current.

Power Quality measurement: Configure panel

Field or control	Description
Voltage Source	Select the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Select the current source used to take the measurement. Tap the field to show the list of available sources.
Label	Set the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Calculate Over Full Cycles	Enables calculating the measurement over the entire acquisition.
Frequency Reference	Select the signal source (Voltage or Current) to use to determine the measurement frequency.

Harmonics measurement: Configure panel

Field or control	Description
Standard	Sets the standard to use for measurements. None (no Standard), IEC 61000-3-2, MIL-STD-1399, AM14, or D0-160G (Standard for Airborne equipment. Supported for harmonics measurement for single phase DUT). ⚠ **Note:** When standard is set as None with the current source, it displays an error as Low Amplitude. Change reflevels method to MinMax manually. This error is displayed when you recall the demo setup files for this measurement.

Table continued…

Field or control	Description
Harmonics	Sets the harmonics order (number of harmonics) for the selected standard. Ranges from 40 to 100.
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Line Frequency	Sets the line frequency of the power signal source. In Auto mode, the application measures the frequency of the input signal automatically. Input signal is used to compute harmonics in the current source. Default is Auto.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Harmonics Source	Selects the source of harmonics calculation source (Voltage or Current) for computation of the Harmonics.
Power Level (MIL-STD-1399)	Selects the required power level, High or Low. It is used to compute limit values for MIL-STD-1399.
Current (MIL-STD-1399)	Selects either rated or measured. Specify the value for rated or measure the Input current signal.
Start Frequency	Specify the fundamental frequency to be measured for each standard. • **None**: The values are **Auto**, 50, 60, 400, or **Custom** (specify the start Frequency) • **IEC-61000-3-2**: The values are **Auto**, 50, 60, or **Custom** • **MIL-STD-1399**: The values are **Auto**, 400, or **Custom** • **AM-14**: The values are **Auto**, 50, 60, or **Custom** • **D0-160G**: The values are **Auto**, 360, 400, 800, or **Custom** Default is **Auto**. The measured frequency value is displayed in the measurement badge.

Source Setup panel - Motor Analysis

Field or control	Description
Source Setting	Configure the Global or Local input or output, source, filter, and LL-LN conversion settings.
Global	Settings configured in the source setup panel is applicable to all active IMDA measurements.
Local	Settings configured in the source setup panel is applicable only to that measurement.
Configuration	Configure to run IMDA measurements on either input or output side of the motor drive. Available options are: • Input • Output
Input	Sets source labels to Vab, Vbc, Vca, Ia, Ib, and Ic based on the selected wiring.
Output	Sets source labels to Vxy, Vyz, Vza, Ix, Iy, and Iz based on the selected wiring.

Table continued…

Field or control	Description
Input Wiring	Select the input wiring configuration from the drop-down: • 1 Phase-2 Wire (1V1I) • 1 Phase-3 Wire (2V2I) • 3 Phase-3 Wire (2V2I) • 3 Phase-3 Wire (3V3I) • 3 Phase-4 Wire (3V3I) • 1 Phase-2 Wire DC (1V1I)
Output Wiring	Select the output wiring configuration from the drop-down: • 1 Phase-2 Wire (1V1I) • 3 Phase-3 Wire (2V2I) • 3 Phase-3 Wire (3V3I) • 3 Phase-4 Wire (3V3I)
LL to LN Conversion	Converts Line-to-Line into Line-to-Neutral using mathematical equation. All voltage computed results are divided by Phase shift of -30 degree for voltage vectors. Available when **Wiring** = **3 Phase-3 Wire (3V3I)**.
Connection	Read only control which is Line-to-Line for 3 Phase-3 Wire 2V2I and 3 Phase-3 Wire 3V3I and Line-to-Neutral for all other wirings.
Select Lines	Sets the select lines as ab-cb, ac-bc, ba-ca or xy-zy, xz-yz, yx-zx based on Input or Output configuration. Select line is visible only for 3Phase-3Wire 2V2I wiring.
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Edge Qualifier	Configure best source (with less noise) which is used to extract proper edges from the PWM source.
Low Pass Filter	Blocks the high frequency band and passes only the low frequency band of the waveform. Select filter order (No Filter, 1st (default), 2nd, or 3rd).
Cutoff Frequency (Fc)	Set the cut off frequency for LPF. Default is 500 Hz.

Input Current measurement: Configure panel

Input Current measurement: Configure panel

Field or control	Description
Wiring	Set to measure 2V2I or 3V3I for 3P3W and 3P4W based on motor configuration. They can also use Line-to-Line and Line-to-Neutral for mathematical conversion. Tap the field to show the list of available sources. 1 Phase-2 Wire (1V1I) 1 Phase-3 Wire (2V2I) 3 Phase-3 Wire (2V2I) 3 Phase-3 Wire (3V3I) 3 Phase-4 Wire (3V3I) Both ABC and XYZ combination. **Note:** Wiring toggle button with options ab-cb or ac-bc or ba-ca is available only when wiring is 2V2I.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources. The current source depends on the wiring selection.

Input Power measurement: Configure panel

Input Power measurement: Configure panel

Field or control	Description
Wiring	Set to measure 2V2I or 3V3I for 3P3W and 3P4W based on motor configuration. They can also use Line-to-Line and Line-to-Neutral for mathematical conversion. Tap the field to show the list of available sources. 1 Phase-2 Wire (1V1I) 1 Phase-3 Wire (2V2I) 3 Phase-3 Wire (2V2I) 3 Phase-3 Wire (3V3I) 3 Phase-4 Wire (3V3I) Both ABC and XYZ combination. **Note:** Wiring toggle button with options ab-cb or ac-bc or ba-ca is available only when wiring is 2V2I.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.

Table continued…

Field or control	Description
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Edge Qualifier	Configure best source (with less noise) which is used to extract proper edges from the PWM source.
Low Pass Filter	Blocks the high frequency band and passes only the low frequency band of the waveform. Select filter order (No Filter, 1st (default), 2nd, or 3rd).
Cutoff Frequency(Fc)	Default is 1 KHz. Available only when **Low Pass Filter = 1st Order, 2nd Order, or 3rd Order.**

Line Ripple measurement: Configure panel

Line Ripple measurement: Configure panel

Switching Ripple and Line Ripple are applicable to AC-AC (Industrial).

Field or control	Description
Wiring	Set to measure 2V2I or 3V3I for 3P3W and 3P4W based on motor configuration. They can also use Line-to-Line and Line-to-Neutral for mathematical conversion. Tap the field to show the list of available sources. 1 Phase-2 Wire (1V1I) 1 Phase-3 Wire (2V2I) 3 Phase-3 Wire (2V2I) 3 Phase-3 Wire (3V3I) 3 Phase-4 Wire (3V3I) Both ABC and XYZ combination. **Note:** Wiring toggle button with options ab-cb or ac-bc or ba-ca is available only when wiring is 2V2I.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Configuration	Input Output
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Line Frequency	Sets the line frequency of the power signal source. Input signal is used to compute ripple in the input source. Default is 50 Hz.
Custom Frequency	Sets the custom frequency of the power signal source. Available only when **Line Frequency = Custom**.

Switching Ripple measurement: Configure panel

Switching Ripple measurement: Configure panel

Field or control	Description
Wiring	Set to measure 2V2I or 3V3I for 3P3W and 3P4W based on motor configuration. They can also use Line-to-Line and Line-to-Neutral for mathematical conversion. Tap the field to show the list of available sources. 1 Phase-2 Wire (1V1I) 1 Phase-3 Wire (2V2I) 3 Phase-3 Wire (2V2I) 3 Phase-3 Wire (3V3I) 3 Phase-4 Wire (3V3I) Both ABC and XYZ combination. **Note:** Wiring toggle button with options ab-cb or ac-bc or ba-ca is available only when wiring is 2V2I.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Source	Select the source to take the measurement. Tap the field to show the list of available sources. xy yz
Switching Frequency	Sets the switching frequency of the power signal source. Input signal is used to compute ripple in the input source.

Phasor Diagram measurement: Configure panel

Phasor Diagram measurement: Configure panel

Field or control	Description
Wiring: 3 Phase-3 Wire (2V2I)	Set to measure 2V2I or 3V3I for 3P3W and 3P4W based on motor configuration. They can also use Line-to-Line and Line-to-Neutral for mathematical conversion. Tap the field to show the list of available sources. 1 Phase-2 Wire (1V1I) 1 Phase-3 Wire (2V2I) 3 Phase-3 Wire (2V2I) 3 Phase-3 Wire (3V3I) 3 Phase-4 Wire (3V3I) Both ABC and XYZ combination. **Note:** Wiring toggle button with options ab-cb or ac-bc or ba-ca is available only when wiring is 2V2I.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Vxy	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Vyz	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
Ix	Sets the input current value of the DUT.
Iz	Sets the input current value of the DUT.
Edge Qualifier	Configure best source (with less noise) which is used to extract proper edges from the PWM source.
Low Pass Filter	Blocks the high frequency band and passes only the low frequency band of the waveform. Select filter order (No Filter, 1st (default), 2nd, or 3rd).
Cutoff Frequency(Fc)	Set the cut off frequency for LPF. Default is 1 KHz. Available only when **Low Pass Filter = 1st Order, 2nd Order, or 3rd Order.**
Phasor Type	Allows to select amplitude computation method either RMS or Magnitude. RMS is computed on the time domain waveform. Magnitude values are computed at the fundamental or operating frequency of the signal.

Note: Filtered signal will be added as part of auto-created math when measurement is added.

Efficiency measurement: Configure panel

Field or control	Description
Voltage Source	Selects the input voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the input current source used to take the measurement. Tap the field to show the list of available sources.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
No of Outputs (1, 2, or 3)	Sets the number of outputs on the SMPS DUT, up to a maximum of three. You can configure these outputs in the application, and set up the sources (V and I), so that the efficiency per output and total efficiency for all available outputs are displayed.
Output 1 Voltage	Selects the voltage source for output one when the number of outputs is set to one.
Output 1 Current	Selects the current source for output one when the number of outputs is set to one.

Other measurement configuration panels

Reference Levels panel (Measurement configuration menu) on page 151

IMDA Measurement Name panel (Measurement configuration menu) on page 185

Configure panel (Measurement configuration menu) on page 150

Gating panel (IMDA measurement configuration menu) on page 197

Configure panel (IMDA measurement configuration menu) - DC-AC (Inverter)

Use the configure panel to set the measurements source(s), add a custom name (label) for the measurement and other parameters.

To open the Configure panel for a IMDA measurement:

1. Double-tap a IMDA measurement badge to open the IMDA measurement configuration menu.
2. Tap the **Configure** panel.

Connection setup for IMDA measurements

Following are the connection setup diagrams for 2V2I and 3V3I wiring:

1Phase2 Wire (1V1I): This is applicable to single phase AC input as in Industrial and single pair DC bus as in Inverter. This is also applicable to Half and Full Bridge configurations.

1Phase3 Wire (2V2I): This is applicable for AC input for Line-to-Neutral configuration.

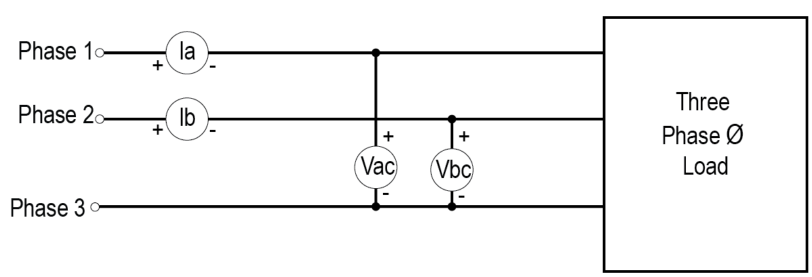


Figure 19: 2V2I 3Phase3Wire

Specify 2 voltage and 2 current sources.

Considers 3 voltage and 3 current sources.

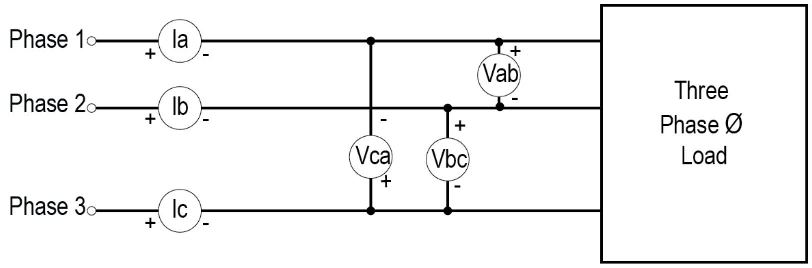


Figure 20: 3V3I 3Phase3Wire

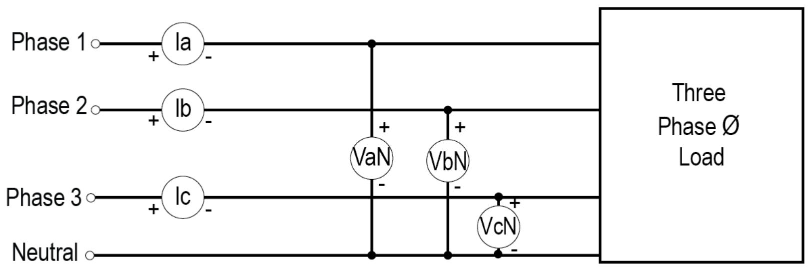


Figure 21: 3V3I 3Phase4Wire

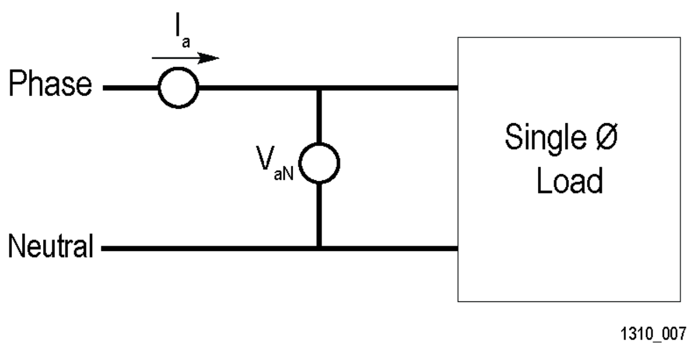


Figure 22: 1V1I 1Phase2Wire

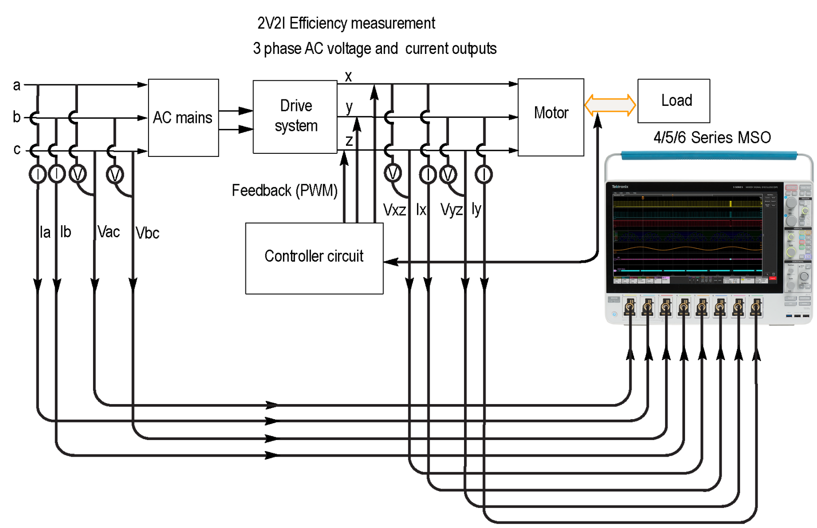


Figure 23: 3 phase motor system

The voltage probes that is used is THDP0100/0200 and current probe that is used is TCP0030A.

Where,

a,b,c is for Input Analysis

x,y,z is for Output Analysis

Drive system is for Ripple measurements.

IMDA Autoset

IMDA Autoset sets up the Voltages Current sources based on the wiring configuration. The IMDA Autoset will optimally set up the vertical, horizontal, acquisition, and trigger parameters on the oscilloscope and will be done on all active IMDA measurements. Autoset updates the RefLevels to MinMax and sets the Hysteris to 10%.

Use the following steps to run a **IMDA Autoset** on applicable IMDA measurements:

1. Add the IMDA measurements that you want to take to the Results bar.
2. Configure each IMDA measurement individually (input voltage sources, current sources, label name, and so on).
3. Connect the input signals to the instrument and confirm the waveforms are correct and not clipping.
4. Double-tap the IMDA measurement result badge for which you want to take a measurement.
5. Tap the **IMDA Autoset** button for the measurement and wait for the busy indicator to disappear. The instrument has now been optimized for that measurement.

   Note:

   - The IMDA Autoset may not work and can fail for higher DC voltages (>200 V).You have to manually adjust the Horizontal Scale and Vertical Scale/Offset to get the proper waveform.
   - In case of failure, the instrument displays a popup error message. See *Errors and Warnings*.
6. Record the IMDA measurement after the **IMDA Autoset** completes.
7. Repeat steps 4 - 6 for other IMDA measurements that you want to take.

Note:

- IMDA Autoset will turn off unused channel sources and turn on configured sources.
- When IMDA Autoset fails or there are insufficient cycles in the signal after autoset, adjust the hysteresis or low pass filter settings in the measurement configuration menu and then rerun IMDA Autoset.
- If you view **Input source mismatch** error when using a Rogowski probe (TRCP series), update the **Vertical Channel** -> **Probe setup** to **Other** and set as **A**.

Set modify custom limits

You can set and modify the custom limits for IMDA Harmonics measurement.

Set custom limits

To set the custom limits:

1. Go to **IMDA** tab, add **Harmonics** measurement from **Electrical analysis** measurement tab.
2. Select **Harmonics** measurement.
3. Select the limits as **Custom**.
4. Click **Load Limits**.
5. Browse to `C:\Users\Public\Tektronix\TekScope\Applications\Power\HarmonicsLimits` path.
6. Select IECStdClassALimits.csv. This is a sample file provided by default by the installer.

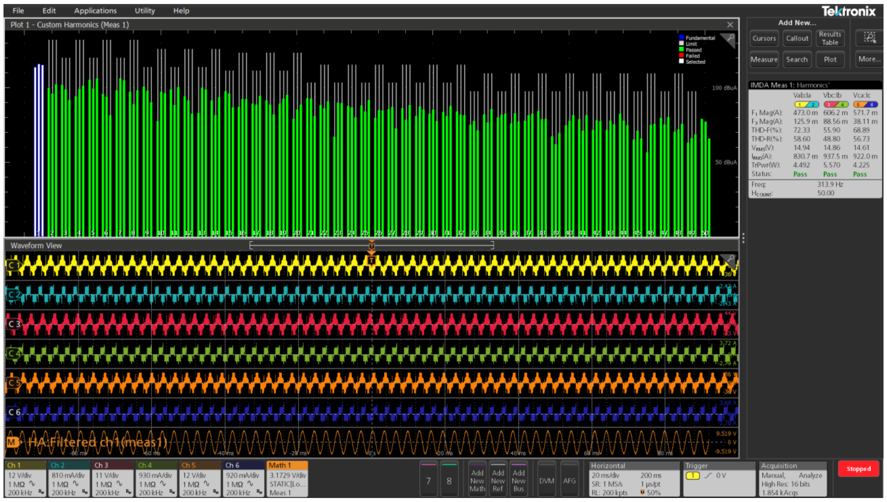
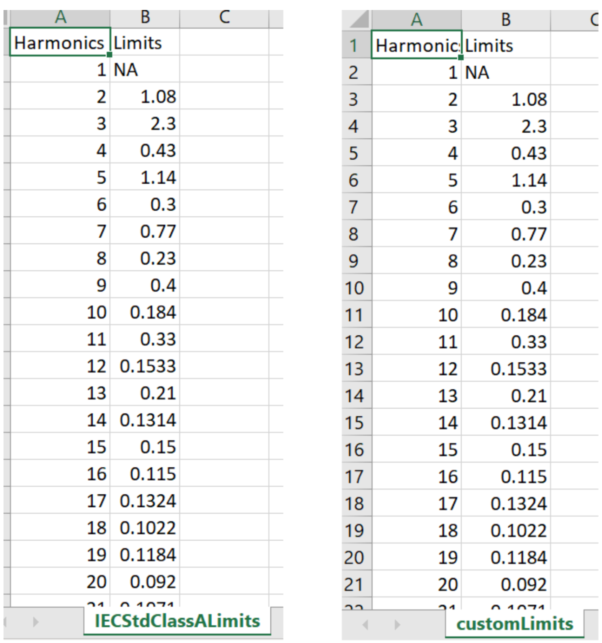


Figure 24: Harmonics with custom limits

Modify custom limits

To modify the custom limits:

1. Open the sample limits file in excel tool.



A	B
Harmonics	Limits
1	NA
2	1.08
3	2.3
4	0.43
5	1.14
6	0.3
7	0.77
8	0.23
9	0.4
10	0.184
11	0.33
12	0.1533
13	0.21
14	0.1314
15	0.15
16	0.115
17	0.1324
18	0.1022
19	0.1184
20	0.092

	A	B
1	Harmonic	Limits
2	1	NA
3	2	1.08
4	3	2.3
5	4	0.43
6	5	1.14
7	6	0.3
8	7	0.77
9	8	0.23
10	9	0.4
11	10	0.184
12	11	0.33
13	12	0.1533
14	13	0.21
15	14	0.1314
16	15	0.15
17	16	0.115
18	17	0.1324
19	18	0.1022
20	19	0.1184
21	20	0.092

Figure 25: Harmonics with custom limits

2. Edit the limits value by adding the custom limits or you can save as another file as '...sample_custom.csv' and recall this file.

Reference Levels panel (IMDA measurement configuration Menu)

Use the Reference Levels panel to set the scope of the reference level settings (global or local), the measurement reference levels (High, Mid, and Low), the units used to take measurements, and the method used to calculate the Top and Base waveform values. Hysteresis settings are critical for PWM edge extraction.

See *Reference Levels panel (Measurement configuration menu)* on page 151 for the Reference panel fields and controls.

Other measurement settings panels

IMDA Measurement Name panel (Measurement configuration menu) on page 185

Configure panel (IMDA measurement configuration menu) - Motor Analysis on page 186

Gating panel (IMDA measurement configuration menu) on page 197

Gating panel (IMDA measurement configuration menu)

Use Gating to confine a measurement to a certain part of a waveform.

See *Gating panel (Measurement configuration menu)* on page 157 for the Gating panel fields and controls.

See also

IMDA Measurement Name panel (Measurement configuration menu) on page 185

Reference Levels panel (IMDA measurement configuration Menu) on page 196

Configure panel (IMDA measurement configuration menu) - Motor Analysis on page 186

DPM measurement configuration menu overview

Use this configuration menu to add a measurement and change measurement settings including source, scope (global or local), and reference levels.

To open the DPM (Digital Power Management) measurement configuration menu for a measurement, double-tap a DPM measurement badge in the Results bar. The configuration menu and panels only show fields and controls relevant to the selected measurement.

The menu opens on the measurement name panel (the name of the measurement), which provides controls to display additional statistics to the measurement badge, display plots of the measurement, and so on. The most common DPM measurement Name fields are listed in the following table.

DPM Measurement configuration menu fields, controls, and panels

Field, control, or panel	Description
Plots (Measurement name panel)	Displays Plot views of the measurement. Available plots depend on the measurement. To add a plot to the screen, tap the plot button. See *Add Plot configuration menu* on page 241.
Configure panel	Sets the source, label text, and other fields that are specific to each measurement type. See *Configure panel (Measurement configuration menu)* on page 150.
Reference Levels panel	Sets the reference levels and units used to take measurements, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values. See *Reference Levels panel (Measurement configuration menu)* on page 151.

Use the following links to access information on measurement panel contents.

DPM Measurement Name panel (Measurement configuration menu)

The DPM Measurement Name panel (the name of the measurement) provides controls for opening plots of the measurement.

To open the DPM measurement name panel, double-tap a DPM Measurement badge. This is the default panel shown when you open a DPM Measurement settings menu.

The contents of the Measurement Name panel depends on the measurement.

Field or control	Description
Power Rail Autoset	Sets the oscilloscope acquisition system for optimal results for all active DPM measurements. See *Power Rail Autoset*.
Spectrum View Autoset	Sets the oscilloscope to acquire the frequency-domain data in parallel with the time-domain data. This frequency-domain data is processed, transformed, and displayed in a separate Spectrum View window, that is optimized to display and adjust spectrum traces independently of the time-domain waveform.
Plots	Adds a Plot view of the measurement value to the screen. Available plots depend on the measurement.

Other measurement configuration panels

Configure panel (Measurement configuration menu) on page 150

Reference Levels panel (Measurement configuration menu) on page 151

Configure panel (DPM measurement configuration menu)

Use the configure panel to set the measurements source(s), add a custom name (label) for the measurement and other parameters.

To open the Configure panel for a DPM measurement:

1. Double-tap a DPM measurement badge to open the DPM measurement configuration menu.
2. Tap the **Configure** panel.

Ripple configure panel

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Power Rail	Sets the power rail source(s) used to take the measurement. Tap the field to show the list of available sources. Select the current source.
Ripple Frequency	Sets the ripple frequency used to calculate the RMS.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Calculate One Measurement Per:	Select **Record** or **Cycle.**

Table continued…

Field or control	Description
Edge Qualifier (CLK)	Sets the edge qualifier to PWM clock source and computes ripple, overshoot or undershoot values at each cycle. Available when Calculate One Measurement Per = **Cycle**.

Overshoot and Undershoot measurements configure panel

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Power Rail	Sets the power rail source(s) used to take the measurement. Tap the field to show the list of available sources.
Reference Level	Sets the reference level needed to calculate the overshoot and undershoot of the signal.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Calculate One Measurement Per:	Select **Record** or **Cycle.**
Edge Qualifier (CLK)	Sets the edge qualifier to PWM clock source and computes ripple, overshoot or undershoot values at each cycle. Available when Calculate One Measurement Per = **Cycle**.

Turn On Time and Turn Off Time measurements configure panel

Field or control	Description
Input Source	Selects the channel connected to the input side of the DUT.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Maximum Input Voltage	Sets the maximum input voltage, in the range of -500 V to 500 V.
Input Trigger Level	Sets the input trigger level, in the range of 1 V to 500 V.
Wait Time	Sets the wait time in seconds.
Power Rail Output	Sets the sources as Analog sources or Ref or Math connected to DC rail output from the DUT.
Output Voltage	Sets the DC rail output voltages for the corresponding power rail output.

Table continued…

Field or control	Description
Power Rail Preset	Sets the oscilloscope with the configured inputs to acquire the signals of interest with a single acquisition. After performing a Power Preset, press the Run/Stop button on the front panel and then power Off/On the device under test.

DC Rail Voltage measurement configure panel

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Power Rail	Sets the power rail source(s) used to take the measurement. Tap the field to show the list of available sources. Select the current source.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Calculate One Measurement Per:	Select **Record** or **Cycle.**
Edge Qualifier (CLK)	Sets the edge qualifier to PWM clock source and computes ripple, overshoot or undershoot values at each cycle. Available when Calculate One Measurement Per = **Cycle**.

Eye Width measurement configure panel

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Power Rail	Sets the power rail source(s) used to take the measurement. Tap the field to show the list of available sources. Select the current source.
Jitter Frequency	Sets the jitter frequency required for the power rail autoset.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Power Rail Clock	Sets the power rail clock source(s) used to take the measurement. Tap the field to show the list of available sources. Select the current source. Available when Method = **Explicit Clock**, in Clock Recovery panel.

DJ, PJ, RJ, and TIE measurements configure panel

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Power Rail	Sets the power rail source(s) used to take the measurement. Tap the field to show the list of available sources. Select the current source.
Jitter Frequency	Sets the jitter frequency required for the power rail autoset.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Power Rail Clock	Sets the power rail clock source(s) used to take the measurement. Tap the field to show the list of available sources. Select the current source. Available when Method = **Explicit Clock**, in Clock Recovery panel.
Clock Edge	Sets the signal transition edge (rising, falling, or either) for evaluating the logic condition at that clock transition.

Eye High, Eye Low, and Eye Height measurements configure panel

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Power Rail	Sets the power rail source(s) used to take the measurement. Tap the field to show the list of available sources. Select the current source.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Power Rail Clock	Sets the power rail clock source(s) used to take the measurement. Tap the field to show the list of available sources. Select the current source. Available when Method = **Explicit Clock**, in Clock Recovery panel.
Bit Type	Sets the waveform bit types for eye height analysis (Transition, Non-Transition, or All). All does eye analysis using both transition and non-transition bits. Transition does eye analysis only on transition bits. A transition bit is a bit that is changing from low to high or high to low. Non-Transition does eye analysis only on nontransition bits. A nontransition bit is a bit that is not changing state.
Measure at % of the Unit Interval	Sets the horizontal position where the measurement is taken, as a percentage of the Unit Interval.

Other measurement configuration panels

Other measurement configuration panels

Reference Levels panel (Measurement configuration menu) on page 151

IMDA Measurement Name panel (Measurement configuration menu) on page 185

Configure panel (Measurement configuration menu) on page 150

Gating panel (IMDA measurement configuration menu) on page 197

Reference Levels panel (DPM measurement configuration Menu)

Use the Reference Levels panel to set the scope of the reference level settings (global or local), the measurement reference levels (High, Mid, and Low), the units used to take measurements, and the method used to calculate the Top and Base waveform values. You can set the reference levels to be the same or different for rising and falling edges.

See *Reference Levels panel (Measurement configuration menu)* on page 151 for the Reference panel fields and controls.

You can select different reference levels for all DPM measurement sources. To set the different reference levels, select **Utility** > **Preferences**. In the User Preferences window, click **Measurements** panel and select the Shared Reference Levels as **Per Source**.

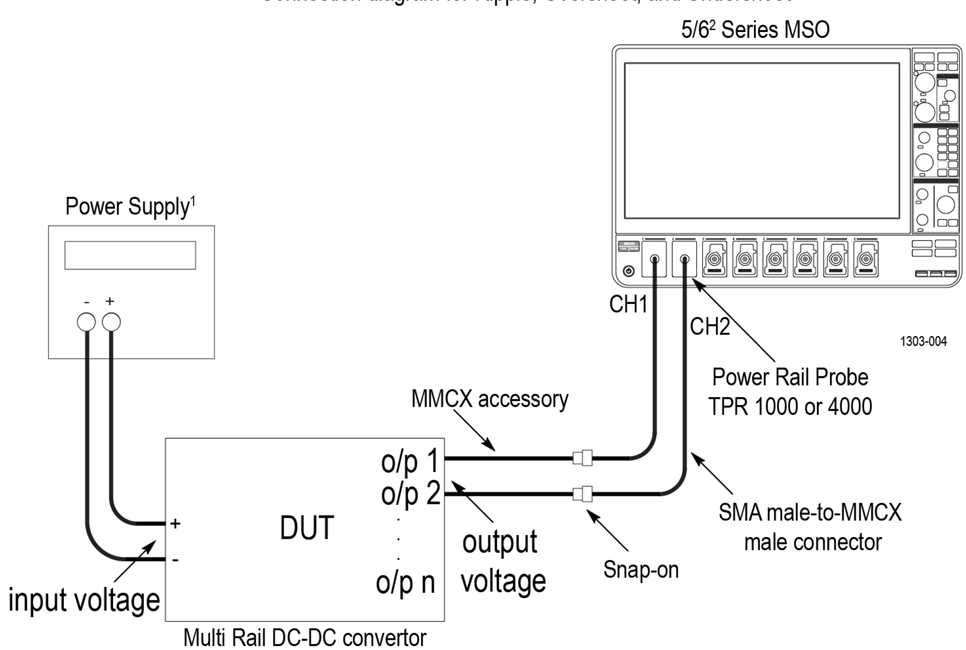
Other measurement configuration panels

DPM Measurement Name panel (Measurement configuration menu) on page 198

Configure panel (Measurement configuration menu) on page 150

Connection setup for Digital Power Management (DPM) measurements

Connection diagram for Ripple, Overshoot, and Undershoot



[1]Source of power supply can be a DC power supply unit or USB connector
[2]6 Series MSO Oscilloscope contains 4 channels only
n is the number of DC rail output of the DUT. The total number of outputs you can connect to the oscilloscope is x-1, where x is the total number of channels.

Figure 26: Connection setup for Digital Power Management (DPM) measurements

Power Rail Autoset

Sets the oscilloscope acquisition system for optimal measurement results for DPM measurements.

Use the following steps to run a **Power Rail Autoset** on applicable DPM measurements:

1. Add the DPM measurements that you want to take to the Results bar.
2. Configure each DPM measurement individually (input voltage sources, label name, and so on).
3. Connect the power rail signals to the oscilloscope channels using power rail probe.
4. Double-tap the DPM measurement result badge for which you want to take a measurement.
5. Tap the **Power Rail Autoset** button for the measurement and wait for the busy indicator to disappear. The instrument has now been optimized for that DPM measurement.

   Note:

   - In case of failure, the instrument displays a popup error message. See *Errors and Warnings*.
   - **Power Rail Autoset** runs all parts of instrument Autoset, even if some parts have been disabled in the **User Preferences** > **Autoset** panel. See *Autoset panel fields and controls* on page 382 for more information on the Autoset function.
   - **Power Rail Autoset** is available per measurement. You should run the Power Rail Autoset for each measurement separately when you add measurements, to ensure optimum DPM measurement setup for that measurement. The power rail autoset configured for a DPM jitter measurement will supersede the power rail autoset configuration of all other DPM measurements.
6. View the DPM measurement results in the results badge, after the **Power Rail Autoset** is complete.
7. Repeat steps *4* on page 177 - *6* on page 178 for other DPM measurements that you want to take.

Note: When you select multiple rails with different ripple or jitter frequencies, the Power Rail Autoset uses the best frequency value to set up the oscilloscope horizontal parameters.

Power Rail Preset

Sets the oscilloscope vertical, horizontal and acquisition parameters using the user inputs like peak voltage and current for Turn Off Time and Turn On Time measurements.

Note: The **Power Rail Preset** button is available only for **Turn On Time**, **Turn Off Time**, and **Turn on Overshoot** measurements.

Use the following steps to run a **Power Rail Preset** on applicable DPM measurements:

1. Add the supported DPM measurements to the **Results** bar.
2. Double-tap each Measurement badge and configure the measurement (input voltage sources, current sources, label name, and so on).
3. Connect the input signals.
4. Tap the **Power Rail Preset** button in the measurement configuration to set.

   To set the vertical and horizontal acquisition parameters:

   a. Measurement configuration begins and displays a window.

      Note:

      - You cannot operate the oscilloscope front panel controls while the measurement runs.
      - You can change the ACQ modes to Sample mode after **Power Rail Preset** is clicked and before the oscilloscope **Single/Seq** is performed.

   b. Press **Single Seq** on the front panel and Turn Off/On the DUT.
5. Follow the instructions on the Measurement badge to take the DPM measurement.

Spectrum View Autoset

The Spectrum View Autoset feature in 4/5/6 series instruments uses a separate hardware path in the acquisition of each channel to acquire the frequency-domain data in parallel with the time-domain data. This frequency-domain data is processed, transformed, and displayed in a separate Spectrum View window, that is optimized to display and adjust spectrum traces independently of the time-domain waveform.

Use the following steps to run a **Spectrum View Autoset** on DPM measurements:

1. Add the DPM measurements that you want to take to the Results bar.
2. Configure each DPM measurement individually (input voltage sources, label name, and so on).
3. Connect the power rail signals to the oscilloscope channels using power rail probe.
4. Double-tap the DPM measurement result badge for which you want to take a measurement.
5. Tap the **Spectrum View Autoset** button for the measurement. This performs the following settings:
   - Sets the SpectrumVu center frequency to Ripple frequency.
   - RBW and Span are set automatically to account for ripple frequencies.
   - Sets FFT Window as Kaiser-Bessel since this has less spectral leakage.
   - Turns on Spectrum View window for the configured sources.

Note:

- In case of failure, the instrument displays a popup error message. See *Errors and Warnings*.
- It is recommended to run **Power Rail Autoset** before **Spectrum View Autoset**.

Spectrum GSRB configuration

Field or Control	Description
Lock Center Frequency Across All Channels	Select to configure the **Center Frequency** value selected for all channels. Unselecting will enable the **Source** option, which will allow you to set the center frequency per source.
Source	Select the channel source from the drop-down list and configure the **Center Frequency** value. Only available when **Lock Center Frequency Across All Channels** is unselected.
Center Frequency	Set the center frequency value.
Span	Set the Span to a factor of Center Frequency. Span is common to all channels. Set a value that works for multiple rail frequencies.
Window	Select the window type from the drop-down list: • Flattop2 • Kaiser-Bessel • Rectangular • Hamming • Hanning • Blackman-Harris
RBW Mode	Select the RBW Mode as **Auto** or **Manual**.

Table continued…

Field or Control	Description
Span: RBW	Set the RBW in span. Only available when **RBW Mode = Auto**
RBW	Set the RBW in absolute values. Only available when **RBW Mode = Manual**
Lock Spectrum Time Across All Channels	Select to configure the **Spectrum Time** value selected for all channels. Unselecting will enable the **Source** option, which will allow you to set the spectrum time per source.
Source	Select the channel source from the drop-down list and configure the **Spectrum Time** value. Only available when **Lock Spectrum Time Across All Channels** is unselected.
Spectrum Time	Set the spectrum time value.

See also

Using Spectrum View on page 107

DDR measurement configuration menu overview

Use this configuration menu to add statistics to a DDR measurement badge readout, plot a measurement, and change measurement settings including source, measurement scope (global or local), reference levels, gating, and filter/limit results.

To open the DDR measurement configuration menu for a measurement, double-tap a DDR measurement badge in the Results bar. The configuration menu and panels only show fields and controls relevant to the selected measurement.

The menu opens on the measurement name panel (the name of the measurement). The measurement name panel only shows fields and controls relevant to the selected measurement. Common controls include displaying statistics in the measurement badge and creating plots of the measurement.

The most common DDR measurement name fields are listed in the following table.

DDR Measurement configuration menu fields, controls, and panels

Field, control, or panel	Description
Show Statistics in Badge (Measurement name panel)	Adds the listed statistical measurement readouts to the measurement badge readout.
Plots (Measurement name panel)	Buttons that open Plot views of the measurement. Available plots depend on the measurement. Plot types include Time Trend, Histogram, and Spectrum. To add a plot to the screen, tap the plot button.
Panels	
Configure panel	Sets the source, label text, and other fields that are specific to each measurement type.

Table continued…

Field, control, or panel	Description
Reference Levels panel	Sets the reference levels and units used to take measurements, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values.
Gating panel	Sets the measurement region (gate) used to take measurements. Select the scope of the gate setting (global or local), and the type of gating to use.
Filter/Limit Results panel	Sets the scope of the filtering setting (global or local), high and low pass filter settings, the range of measurement result limits, and the limit measurement population size.

Use the following links to access information on measurement panel contents.

DDR measurement name panel (Measurement configuration menu)

The DDR measurement name panel (the name of the measurement) provides controls for adding display statistics to the measurement badge and opening plots of the measurement.

To open and configure a DDR measurement, double-tap a DDR measurement badge.

The contents of the measurement configuration menu depend on the measurement.

Field or control	Description
Show Statistics in Badge	Adds the listed statistical measurement readouts to the measurement badge.
Plots	Adds a Plot view of the measurement to the screen. Available plots depend on the measurement. Plot types include **Time Trend**, **Histogram**, and **Spectrum**.

See also

Configure panel (DDR measurement configuration menu) on page 206

Filter/Limit results panel (DDR measurement configuration Menu) on page 207

Reference Levels panel (DDR measurement configuration Menu) on page 207

Gating panel (DDR measurement configuration menu) on page 207

Configure panel (DDR measurement configuration menu)

Use the configure panel to set the measurements source(s), add a custom name (label) for the measurement, and other parameters.

To open the Configure panel for a DDR measurement:

1. Double-tap a DDR measurement badge to open the DDR measurement configuration menu.
2. Tap the **Configure** panel.

Configure panel

Field or control	Description
Source	Sets the signal source used to take the measurement. Tap the field to show the list of available sources. Select the current source.

Table continued…

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

Other measurement configuration panels

Filter/Limit results panel (DDR measurement configuration Menu) on page 207

Reference Levels panel (DDR measurement configuration Menu) on page 207

Gating panel (DDR measurement configuration menu) on page 207

Reference Levels panel (DDR measurement configuration Menu)

Use the Reference levels panel to set the scope of the reference level settings (global or local), the measurement reference levels (High, Mid, and Low), the units used to take measurements, and the method used to calculate the Top and Base waveform values. You can set the reference levels to be the same or different for rising and falling edges.

See *Reference Levels panel (Measurement configuration menu)* on page 151 for the Reference panel fields and controls.

Other measurement settings panels

Configure panel (DDR measurement configuration menu) on page 206

Filter/Limit results panel (DDR measurement configuration Menu) on page 207

Gating panel (DDR measurement configuration menu) on page 207

Gating panel (DDR measurement configuration menu)

Use Gating to confine a measurement to a certain part of a waveform.

See *Gating panel (Measurement configuration menu)* on page 157 for the Gating panel fields and controls.

See also

Configure panel (DDR measurement configuration menu) on page 206

Filter/Limit results panel (DDR measurement configuration Menu) on page 207

Reference Levels panel (DDR measurement configuration Menu) on page 207

Filter/Limit results panel (DDR measurement configuration Menu)

Use these settings to apply a High Pass and/or Low Pass filter to block specified frequency band components when taking measurements. Use the limit controls to set range of measurement values to measure, and the number of measurements to take (population).

See *Filter/Limit Results panel (Measurement Settings menu)* on page 158 for the Filter/Limit Results panel fields and controls.

Other measurement settings panels

Configure panel (DDR measurement configuration menu) on page 206

Reference Levels panel (DDR measurement configuration Menu) on page 207

Gating panel (DDR measurement configuration menu) on page 207

Bus configuration menu

Use the Bus menu to select the bus type to display, configure the input sources, and set how to display the bus on the screen.

To open the Bus configuration menu:

- For an existing bus, double-tap the **Bus** badge in the Settings bar.
- To add a new Bus badge on the Settings bar, tap the **Add New Bus** button. This adds the Bus badge to the Settings bar and opens the Bus configuration menu.

Bus configuration menu - fields and controls

Field or control	Description
Display	Toggles bus display on or off.
Label	Enter label text in this field. The default label is the name of the bus type.
Position	Sets the vertical position of the bus waveform. Default is 0 (center of graticule).
Set to 0	Sets the vertical position of the bus waveform to 0 (center graticule).
Bus Type	Select a bus from the drop down list. The Parallel bus type comes standard on the instrument. Serial buses require purchase and installation of serial bus triggering and analysis options.
Source configuration	A set of fields and controls that set the bus signal input parameters. Shown fields depend on the selected bus type. See the individual bus configuration help topics for information on their settings.
Display format	Enables showing just the decoded bus or both the bus and its digital waveforms. You can also tap on the + symbol on the bus waveform to toggle between showing the bus only or showing bus and source waveforms.
Decode format	Sets how decoded data information is shown in the bus. Select from listed formats. Available formats depend on the bus type.

Use the following links to access information on specific Bus configuration menus.

8b10b serial bus search configuration menu

Use the 8b10b Search configuration menu to define conditions to search for and mark on a 8b10b bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the 8b10b bus that you want to search.
Mark On	Select the type of information for which to search.

Table continued…

Field or control	Description
Format	Sets the number of data bytes for which to search (one to five bytes). Use the **A** knob to change the value. Only available when **Mark On** = **Symbols**.
Type	Sets the type to search (**Data** or **Control**). Only available when **Mark On** = **Symbols** and **Format** = **8-Bit**.
Disparity	Sets the disparity type as **Positive** or **Negative** or **Either**. Only available when **Mark On** = **Symbols** and **Format** = **8-Bit**, **10-Bit**, and **Symbol**. ⚠ **Note:** Set the disparity value and then query the 10 bit data value.
Value	Sets the value for the format. • When **Format** = **8-Bit**, you can set 8-Bit value in Binary or Hex format. • When **Format** = **10-Bit**, you can set 10-Bit value in Binary or Hex format. • When **Format** = **Symbol**, you can set values in decimal format. Only available when **Mark On** = **Symbols**.
10Bit RD+	Sets the RDpositive 10 bit value. It displays the corresponding 10 bit value for 8 bit or symbol. Set the Binary and Hex values Available when Format = 10 Bit
10Bit RD+	Sets the RDnegative 10 bit value. It displays the corresponding 10 bit value for 8 bit or symbol. Set the Binary and Hex values Available when Format = 10 Bit
Error Type	Sets the error type (**Symbol or Disparity**) for which to search and mark on the waveform. Only available when **Mark On** = **Errors**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. 8b10b is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. 8b10b is currently not supported as a bus trigger source.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

8b10b serial bus configuration menu

ARINC 429 serial bus menu

Use the ARINC 429 bus menu to set up and decode a ARINC 429 avionics network serial data bus waveform.

To set up the ARINC 429 avionics serial data bus:

- Tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the new Bus badge. Set the **Bus Type** to **ARINC429**.
- To change the settings on an existing ARINC 429 serial bus waveform, double-tap the **Bus** waveform badge and make necessary changes.

ARINC 429 serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text using an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **ARINC429**.
Polarity	Select the polarity to match the ARINC 429 bus being acquired.
Source	Select the ARINC 429 signal source.
High Threshold, Low Threshold	Sets the valid high and low threshold values for the signal source.
Bit Rate	Sets the bit rate to 12,500, 100,000, or Custom.
Custom Rate	Sets a custom data bit rate. To set the value, tap the field and use the Multipurpose knob, double-tap the field and use the Custom Rate virtual keypad, or double-tap the field and use an attached keyboard. Only available when Bit Rate = Custom.
Data Format	Sets the data format to Data (19 bits), SDI (Source/Destination Identifiers) plus data (21 bits), or SDI plus Data plus Sign/Status Matrix (SSM) (23 bits.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and Mixed Hex.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

ARINC 429 serial bus search configuration menu on page 252

Audio serial bus configuration menu

Use the Audio bus menu to set up and display Audio Type I2S, Left Justified (LJ), Right Justified (RJ), or TDM Audio serial bus waveforms.

To set up the Audio serial bus:

- To create a new Audio bus waveform, tap the **Add New Bus** button on the Settings bar and open the Bus configuration menu. Set the **Bus Type** to **Audio**.
- To change the settings on an existing Audio serial bus waveform, double-tap the **Bus** waveform badge and make the necessary changes.

Audio serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **Audio**.
Audio Type	Sets the digital audio signal type. Select from the drop-down list.
Bit Order	Set the waveform to decode with most-significant (MS) bit first or least-significant (LS) bit first.
Bit Clock	Set the signal source, logic level threshold, and polarity for the Bit Clock signal.
Word Select	Set the signal source, logic level threshold, and normal or invert signal setting for the Word signal.
Data	Set the signal source, logic level threshold, and logic definition (active high or low) for the Data signal.
Word Size (Audio Type = I2S, LJ, or RJ)	Set the number of bits used in a Word for the selected audio type (8, 12, 16, 18, 20, 24, 28, or 32-bits).
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information.
TDM-specific settings	
Frame Sync	Set the signal source, logic level threshold, and polarity for the frame sync signal.
Data Bits per Channel	Set the number of data bits per audio channel.
Clock Bits per Channel	Set the number of clock bits per audio channel.
Channels per Frame	Set the number of audio channels per data frame.
Bit Delay	Sets the bit delay (number of bits).

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

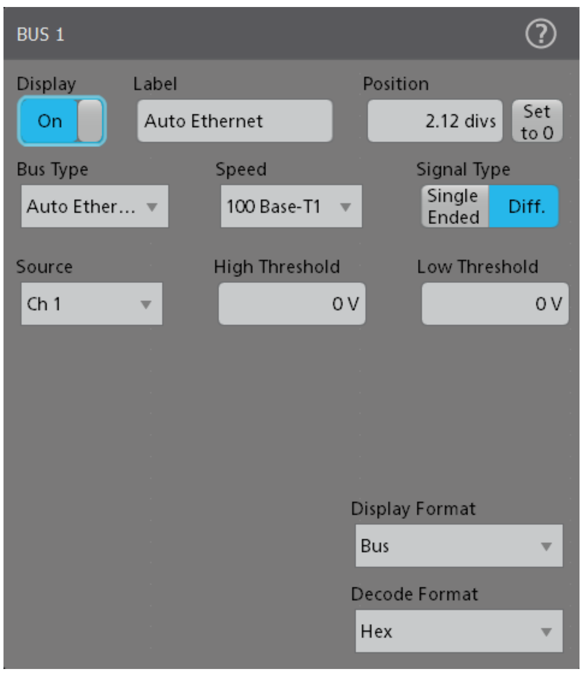
Audio serial bus search configuration menu on page 254

Auto Ethernet serial bus configuration menu

Use the Auto Ethernet bus menu to set up and display an Auto Ethernet serial bus waveform.

To set up a Auto Ethernet serial bus:

- To add a new Auto Ethernet serial bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **Auto Ethernet**.
- To change the settings on an existing Auto Ethernet serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.



You can *configure search for Auto Ethernet* and define conditions to search for and mark on an Auto Ethernet bus waveform.

Auto Ethernet serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.

Table continued…

Field or control	Description
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **Auto Ethernet**.
Speed	Sets the network signal speed as **100Base-T1**.
Signal type	Sets the signal type to Single-Ended or Differential.
Source	Select the channel source for the signal from a differential probe. Only available when Signal Type = Diff.
High Threshold	Sets the threshold value for a logic 1 value. Only available when Signal Type = Diff.
Low Threshold	Sets the threshold value for a logic 0 value. Only available when Signal Type = Diff.
D+ Input, D- Input	Sets the signal sources and threshold values for the Single Ended data+ and - signals. Only available when Signal Type = Single-Ended.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Auto Ethernet serial bus search configuration menu on page 255

CAN serial bus configuration menu

Use the CAN bus menu to set up and display a CAN (Controller Area Network) or CAN FD (CAN Flexible Datarate) serial bus waveform.

To create a new CAN bus waveform:

1. Tap **Add New Bus** on the Settings bar.
2. Double-tap the **CAN** badge to open the bus configuration menu.
3. Set the **Bus Type** to **CAN**.

To change the settings on an existing CAN serial bus waveform, double-tap the **CAN** waveform badge on the Settings bar to open the configuration menu.

CAN serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **CAN** to set up and display a CAN bus waveform.
Signal Type	Sets the CAN signal type to decode. The default is **CAN_H**.
CAN Standard	Sets the CAN signal standard to decode. The default is **CAN 2.0**.
Source	Select the signal source from listed analog and digital channels.
Threshold	Sets the high/low logic transition level.
Sample Point	Sets the sample point from 5% to 95% of the position within the bit period or the unit interval.
Bit Rate	Sets the bit rate of your CAN bus serial data. To enter a custom bit rate, select **Custom** and enter the custom bit rate in the **Custom Rate** input box. Only available when **CAN Standard** = **CAN 2.0**.
SD Bit Rate	Sets the SD bit rate of your CAN FD serial bus data. To enter a custom bit rate, select **Custom** and enter the custom bit rate in the **Custom Rate** input box. Only available when **CAN Standard** = **CAN FD (ISO)** or **CAN FD (non-ISO)**.
FD Bit Rate	Sets the FD bit rate of your CAN FD serial bus data. To enter a custom bit rate, select Custom and enter the custom bit rate in the Custom Rate input box. Only available when **CAN Standard** = **CAN FD (ISO)** or **CAN FD (non-ISO)**.

Table continued…

Field or control	Description
Custom rate	Sets the custom bit rate to use to decode the signal. Tap the field and use the Multipurpose knob to change the value, or double-tap on the field and use the virtual keypad to enter a custom bit rate. Only available when **Bit Rate** or **SDI Bit Rate** = **Custom.**
Display Format	**Bus** sets the Waveform view to show just the decoded bus information. **Bus and Waveform** sets the Waveform view to show both the decoded bus and the source signal waveforms. You can also tap on the **+** symbol on the bus waveform to toggle between showing the bus only or showing bus and source waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, **Mixed Hex**, and **Symbolic**. Selecting **Symbolic** lets you navigate to and load a dbc decode label file for the current CAN waveform. The .dbc file is a proprietary description of a CAN bus system and its message and signal labels. Loading the .dbc file displays the message and signal information on the decoded CAN waveform. Selecting a .dbc file immediately loads the .dbc contents into the instrument. **Note:** .dbc files are not saved as part of a save session operation. To remove the .dbc symbolic information from the waveform, select a different **Decode Format**.
Browse	Opens the standard **File Open** dialog to navigate to and load a .dbc file to apply to the bus being created. The **File Open** dialog automatically filters for the .dbc file extension. Only available when **Decode Format** = **Symbolic**.
Path field	Use to enter the path to a .dbc file, or select from a drop-down list of recently accessed .dbc files. Only available when **Decode Format** = **Symbolic**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

CAN serial bus search configuration menu (when not using a .dbc symbol definition file) on page 257

DPHY serial bus configuration menu

Use the DPHY bus menu to set up and display an DPHY (clock, data + and data -) serial bus waveform.

To set up the DPHY serial bus:

- To add a new DPHY bus, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **DPHY**.
- To change the settings on an existing DPHY serial bus, double-tap the DPHY **Bus** badge and make necessary changes.

DPHY serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **DPHY**.
Protocols	Select the protocol to decode. Available options are **CSI** and **DSI**. Default is CSI.
LP Direction	Set the LP (Low Power) direction, host to peripheral or peripheral to host. Available options are **Forward** and **Reverse**. Available only when **Protocols** = **DSI**.
Source	Select the channel source for the signal.
Data Threshold	Select the data threshold for Dp and Dn signals.
LP Threshold	Select the LP threshold for Dp and Dn signals.
Dp Input, Dn Input	Sets the signal sources.
8b9b Encoding	Turns on or off 8b9b line encoding to support additional control features. Default is OFF.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Mixed Hex**, and **Binary**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

DPHY serial bus search configuration menu on page 261

Ethernet serial bus menu

Use the Ethernet bus menu to set up and display an Ethernet 10BaseT or 100BaseT serial bus waveform.

To use the Ethernet serial bus menu:

- To create a new Ethernet bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **Ethernet**.
- To change the settings on an existing Ethernet serial bus waveform, double-tap the Ethernet **Bus** waveform badge to open the configuration menu.

Ethernet serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **Ethernet**.
Speed	Sets the network signal speed.
Signal Type	Sets the signal type to Single Ended or Differential.
Source	Sets the signal source for Differential. Only available when Signal Type = Diff.
D+ Input, **D- Input**	Defines the signal sources and threshold values for the Single Ended data+ and - signals. Only available when Signal Type = Single Ended.
High Threshold	Sets the threshold value for a logic 1 value. Only available when Signal Type = Diff.
Low Threshold	Sets the threshold value for a logic 0 value. Only available when Signal Type = Diff.
IPv4	Set to Yes if the Ethernet signal being measured uses Internet Protocol version 4 (IPv4).
Q-(VLAN)	Set to Yes if the Ethernet signal being measured uses IEEE 802.1Q virtual LANs.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, Mixed ASCII, and Mixed Hex.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Ethernet serial bus search configuration menu on page 272

FlexRay serial bus configuration menu

Use the Flexray bus menu to set up and display a Flexray automotive network serial bus waveform.

To set up the FlexRay serial bus:

- To create a new FlexRay bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **FlexRay**.
- To change the settings on an existing FlexRay serial bus waveform, double-tap the **Bus** waveform badge and make necessary changes.

FlexRay serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text using an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **FlexRay**.
Signal Type	Select the FlexRay signal type being measured.
Channel Type	Set to A or B channel.
Source	Select the FlexRay signal source.
Threshold	Sets the threshold value for the TX or RX signal type.
High Threshold, Low Threshold	Sets the high and low threshold values for the BM Inverted and Bdiff/BP signal types.
Bit Rate	Select a bit rate. To set a custom bit rate, select Custom and enter a value in the Custom Rate field.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

FlexRay serial bus search configuration menu on page 277

I2C serial bus configuration menu

Use the I2C bus menu to set up and display an I^2C (Inter-Integrated Circuit) serial bus waveform.

To set up the I^2C serial bus menu:

- To create a new I^2C bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **I2C**.
- To change the settings on an existing I^2C serial bus waveform, double-tap the I^2C **Bus** waveform badge and make necessary changes in the configuration menu.

I2C serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **I2C**.
SCLK Input	Sets the source and threshold level for the Serial Clock Line signal.
SDA Input	Sets the source and threshold level for the Serial Data signal.
Include R/W bit in Address	Select **Yes** to display 7-bit addresses as eight bits, where the eighth bit (LSB) is the R/W bit, or display 10-bit addresses as 11 bits, where the third bit is the R/W bit. Select **No** to display 7-bit addresses as seven bits, and 10-bit addresses as ten bits.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

I2C serial bus search configuration menu on page 279

I3C serial bus configuration menu

Use the I3C bus menu to set up and display a decoded I^3C serial bus waveform.

To set up the I^3C serial bus menu:

- To create a new I^3C bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **I3C**.
- To change the settings on an existing I^3C serial bus waveform, double-tap the I^3C **Bus** waveform badge and make necessary changes in the configuration menu.

I3C serial bus menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	Creates a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in **Stacked** mode), or center screen in **Overlay** mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **I3C**.
SCLK Input	Sets the source and threshold level for the Serial Clock Line signal.
SDA Input	Sets the source and threshold level for the Serial Data signal.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

I3C serial bus Search configuration menu on page 282

LIN serial bus configuration menu

Use this menu to set up and display a LIN (Local Interconnect Network) serial bus waveform.

To set up the LIN serial bus:

- To create a new LIN bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **LIN**.
- To change the settings on an existing LIN serial bus waveform, double-tap the LIN **Bus** waveform badge and make necessary changes.

LIN serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **LIN**.
Source	Set the signal source from available analog or digital channels.
Threshold	Set the threshold level to define a logic high level.
Polarity	Select the polarity to match the LIN bus being acquired.
LIN Standard	Select the standard to match the LIN bus being acquired.
Bit rate	Sets the bit rate. To enter a custom bit rate, tap Custom and enter the custom bit rate in the Custom Rate input box.
Include Parity Bits with ID	Set to Yes to include parity bits with the ID.
Sample Point	Sets the sample point from 5% to 95% of the position within the bit period or the unit interval.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and Mixed.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

LIN serial bus search configuration menu on page 290

MIL-STD-1553 serial bus menu

Use the MIL-STD-1553 bus menu to set up and decode a MIL-STD-1553 aeronautic network serial data bus waveform.

To set up the MIL-STD-1553 serial data bus:

- To create a new MIL-STD-1553 bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **MIL-STD-1553**.
- To change the settings on an existing FlexRay serial bus waveform, double-tap the **Bus** waveform badge and make necessary changes.

MIL-STD-1553 serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text using an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **MIL-STD-1553**.
Polarity	Select the polarity to match the MIL-STD-1553 bus being acquired.
Source	Select the MIL-STD-1553 signal source.
High Threshold, Low Threshold	Sets the valid high and low threshold values for the signal source.
RT Maximum	Sets the maximum valid response time (RT) for a command.
RT Minimum	Sets the minimum valid response time (RT) for a command.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, Mixed ASCII, and Mixed Hex.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

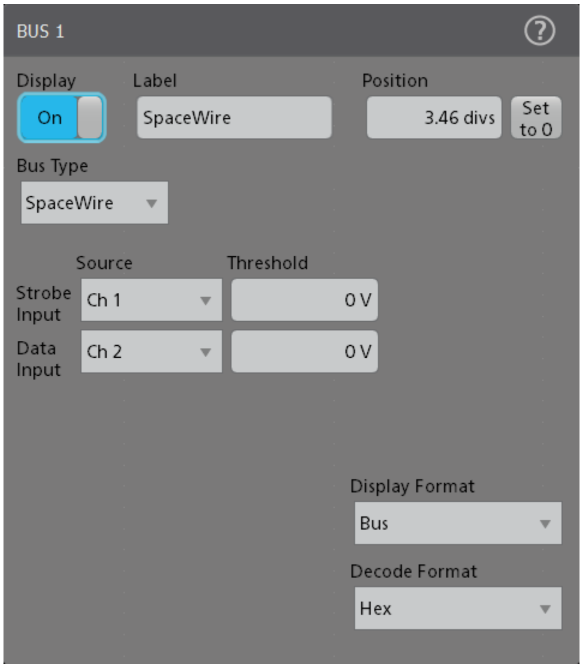
MIL-STD-1553 serial bus search configuration menu on page 291

NRZ serial bus configuration menu

Use the NRZ bus menu to set up and display an NRZ serial bus waveform.

To set up a NRZ serial bus:

- To add a new NRZ bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double-tapping the badge. Set the **Bus Type** to **NRZ**.
- To change the settings on an existing NRZ serial bus waveform, double-tap the **NRZ** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.



You can *configure search for NRZ* and define conditions to search for and mark on an NRZ bus waveform.

NRZ serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **NRZ**.
Bit Order	Set the waveform to decode with most-significant (MS) bit first or least-significant (LS) bit first.
Source	Select the signal source from listed analog and digital channels.

Table continued…

Field or control	Description
Threshold	Sets the threshold value for the signal source.
Polarity	Select the polarity to match the NRZ bus being acquired.
Bit Rate	Set the bit rate. You can configure values from 1 bps to 1 Gbps.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

NRZ serial bus search configuration menu on page 293

Parallel Bus configuration menu

Use this menu to set up and display a parallel bus waveform. Parallel bus decoding and triggering is included with the oscilloscope.

To set up the parallel bus:

- To create a new parallel bus, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **Parallel**.
- To change the settings on an existing parallel bus waveform, double-tap the **Bus** waveform badge to open the configuration menu, and make necessary changes.

Parallel bus configuration menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to Parallel to define a parallel bus.
Clocked Data	Toggles Yes or No to use a clock signal to recover the data bits from the bus inputs.
Clock Source	Sets the source for the bus clock signal. The source can be an analog or digital channel. Only available when Clocked Data is set to Yes.

Table continued…

Field or control	Description
Polarity	Sets the clock signal edge to use for timing reference. Only available when Clocked Data is set to Yes.
Threshold	Sets the threshold value to determine high logic value. Only available when Clocked Data is set to Yes.
Define Inputs	Opens a Parallel Bus - Define Inputs configuration menu to set the signal sources and the bit order (MSB to LSB) for the bus. See *Parallel Bus - Define Inputs menu* on page 225.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Parallel bus search configuration menu on page 294

Parallel Bus - Define Inputs menu

Use this menu to select the signal sources and order for the parallel bus waveform

To access the Parallel Bus - Define Inputs menu, double-tap a Parallel Bus badge to open the configuration menu, and tap the **Define Inputs** button.

Parallel Bus - Define Inputs menu fields and controls

Field or control	Description
Parallel bus definition list	Lists the signal source and thresholds of selected channels or waveforms. The MSB is at the top of the list. To add a signal to the Parallel bus definition list, tap a source button in the Sources list. The button moves from the Sources list to the bottom of the bus list. Use the arrow buttons to the right of the field to move a selected signal up or down in the list. To remove a signal from the Parallel bus (and return it to the Sources list), tap on the signal source button. To change the threshold value for individual channels, tap in a selected Threshold field and use the assigned multipurpose knob, or double-tap the field to open the keypad and enter values.
Sources	Lists all available sources to use for a parallel bus. To add a source to the Parallel bus definition list, tap a source button. The button moves from the Sources list to the bottom of the bus list.

Table continued…

Field or control	Description
Set All Thresholds	Sets all thresholds in the Parallel bus definition list to the specified value. Enter a value and tap Apply to set the values.

Other bus types

Serial bus decode and analysis features are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Parallel bus search configuration menu on page 294

PSI5 serial bus configuration menu

Use the PSI5 bus menu to set up and display an PSI5 (Peripheral Sensor Interface) serial bus waveform.

To set up the PSI5 serial bus:

- To create a new PSI5 bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **PSI5**.
- To change the settings on an existing PSI5 serial bus waveform, double-tap the PSI5 **Bus** waveform badge and make necessary changes.

PSI5 serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **PSI5** to set up and display a PSI5 bus waveform.
Direction	Select the direction.
Sensor To ECU	Sets the BUS to Sensor to ECU mode.
ECU to Sensor	Sets the BUS to ECU to Sensor mode.
Source	Set the signal source from available analog or digital channels.
Threshold	Set the threshold level to define a logic high level.

Table continued…

Field or control	Description
Mode	Configure the speed. Standard (125 kbps) Fast (189 kbps) Slow (83.3 kbps)
Data A	Set the number of bits for data region A.
Data B	Set the number of bits for data region B.
Serial Messaging	Turns On/Off the Serial channel messaging
Frame Control	Indicates the type of frame, data content or identifies the sensor. The values are 0,1,2,3, and 4.
Status	Set the sensor status from 0 to 2 bits.
Sync Mode	Tooth Gap Pulse Width Only available when **Direction = ECU to Sensor** .
Bit Period	Only available when **Direction = ECU to Sensor** and **Sync Mode = Tooth Gap**.
Data Format	Nibble Byte Only available when **Direction = ECU to Sensor** .
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

PSI5 serial bus search configuration menu on page 263

SVID serial bus configuration menu

Use the SVID bus menu to set up and display an SVID (Serial VID (clock, data, alert)) serial bus waveform.

To set up the SVID serial bus:

- To create a new SVID bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **SVID**.

- To change the settings on an existing SVID serial bus waveform, double-tap the SVID **Bus** waveform badge and make necessary changes.

SVID serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set the **SVID** to set up and display a SVID bus waveform.
Clock Input	Set the source and threshold level for the clock input signal.
Data Input	Sets the source and threshold level for the data input signal.
Alert Input	Sets the source and threshold level for the alert input signal.
Threshold	Set the threshold level to define a logic high level. Sets the threshold value for the signal source.
Display Format	Sets the waveform view to show the decoded bus information, or the decoded bus and the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SVID serial bus search configuration menu on page 267

MDIO serial bus configuration menu

Use the MDIO bus menu to set up and display an MDIO (Management Data Input Output) serial bus waveform.

To set up the MDIO serial bus:

- To create a new MDIO bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **MDIO**.
- To change the settings on an existing MDIO serial bus waveform, double-tap the MDIO **Bus** waveform badge and make necessary changes.

MDIO serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set the **MDIO** to set up and display a MDIO bus waveform.
Clock Input	Set the source and threshold level for the clock input signal.
Data Input	Set the source and threshold level for the data input signal.
Threshold	Set the threshold level to define a logic high level. Sets the threshold value for the signal source.
Display Format	Sets the waveform view to show the decoded bus information, or the decoded bus and the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

MDIO serial bus search configuration menu on page 269

eUSB serial bus configuration menu

Use the Embedded USB bus menu to set up and display an eUSB 2.0 (Embedded Universal Serial Bus) waveform.

To set up a eUSB serial bus:

- To create a new eUSB bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **eUSB**.
- To change the settings on an existing eUSB serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

eUSB serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **eUSB**.
Speed	Set the speed to match the acquiring speed of the eUSB bus. Available data rate options are **Full (12 Mbps)**, **High Speed (480 Mbps)**, and **Low (1.5 Mbps)**.
Mode	Set the mode of operation to Native or Repeater. Default is Native. • **Native** mode is used for inter chip disconnect. • **Repeater** is used to communicate eUSB2 port with USB2.0 through repeater that translates between 2USB2 signaling and USB2.0 signaling. • **Repeater Peripheral**
Signal Type	Set to match the eUSB signal you are acquiring (Single Ended or Differential). Use a differential probe to acquire the differential USB signal. Both Single Ended and Diff can be used to measure High speed USB signals. Single Ended is used to measure Full (12 Mbps) and Low (1.2 Mbps) speed USB signals. **Note:** Signal Type selection is applicable only in High Speed.
Source	Select the channel source for the signal from a differential probe. Only available when **Signal Type** = **Diff**.
Threshold	Set the threshold level.
Data Threshold	Set the data threshold for D+ signal and D- signal. Only available when **Signal Type** = **Single Ended**.
High Threshold	Set the high threshold for D+ signal and D- signal.
Low Threshold	Set the low threshold for D+ signal and D- signal.
D+ Input	Select the channel source and set the threshold level for the Data+ signal.
D- Input	Select the channel source and set the threshold level for the Data- signal.
Display Format	Sets the waveform view to show the decoded bus information or the decoded bus plus the source signal waveforms.

Table continued…

Field or control	Description
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, **Mixed ASCII**, and **Mixed Hex**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

eUSB serial bus search configuration menu on page 274

Manchester serial bus configuration menu

Use the Manchester bus menu to set up and display a manchester waveform.

To set up a Manchester serial bus:

- To create a new Manchester bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **Manchester**.
- To change the settings on an existing Manchester serial bus, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

Manchester serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus decode. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **Manchester**.
Set the edge for bit '0'	Set the edge for bit 0.
Data Rate	Set the data rate for the manchester bus. Default data rate is 125 kb/s.
Source	Select the signal source.
Threshold	Set the threshold level.
Start Index	Select the start index. Default is one edge. Unit is edge.

Table continued…

Field or control	Description
Packet View	Set to **On** to show packet field level decode and **Off** show bit level decode. Default is ON.
Idle Bits	Set the idle bits depending on the idle period of time. Default is 1.2.
Tolerance	Set the tolerance for giving bandwidth to edge in the bit for bit error Default is 10 %.
Sync Bits	Set the number of bits that define sync pattern for the manchester bus. Default is 1 bit Only available when **Packet View** = **On**.
Parity	Set the manchester decode in the packet from the waveform being acquired. Default is None. Only available when **Parity View** = **On**.
Bit Order	Set the order of bit to Most Significant Bit first (MSB) or Least Significant Bit first (LSB). Only available when **Packet View** = **On**. Default is MSB.
Word Count	Set the number of words to be decoded in a packet. Only available when **Packet View** = **On**. Default is 1.
Word Size	Set the number of bits in each payload. Only available when **Packet View** = **On**. Default is 8 bits.
Display Format	Sets the waveform view to show the decoded bus information. Format is **Bus**.
Header	Set the number of bits in the header of a packet. Only available when **Packet View** = **On**. Default is 0 bits.
Trailer	Set the number of bits in the trailer of a packet. Only available when **Packet View** = **On**. Default is 0 bits.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex** and **Binary**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Manchester serial bus search configuration menu on page 280

RS232 serial bus menu

Use this menu to set up and display an RS232 serial bus waveform.

To set up a RS232 serial bus:

- To create a new RS232 bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **RS232**.
- To change the settings on an existing RS232 serial bus waveform, double-tap the RS232 **Bus** waveform badge to open the configuration menu and make necessary changes.

RS232 serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **RS232**.
Bit Rate	Sets the data bit rate. To enter a custom bit rate, tap Custom and enter the custom bit rate in the Custom Rate input box.
Source	Set the signal source from available analog or digital channels.
Threshold	Set the threshold level to define a logic high level.
Polarity	Select the polarity to match the RS232 bus being acquired. Use Normal polarity for RS-232 signals, and Inverted polarity for RS-422, RS-485, and UART buses.
Data Bits	Set the number of bits that define a data packet for your RS232 bus.
Parity	Set the parity to match the RS232 bus being acquired.
Packet View	Set to On to show decoded packet level information on the bus waveform.

Table continued…

Field or control	Description
End of packet	select the appropriate end of packet value to match the RS232 bus being acquired. Available when Packet View = On.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and ASCII.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

RS-232 serial bus search configuration menu on page 295

SENT serial bus configuration menu

Use the SENT bus menu to set up and display a SENT (Single Edge Nibble Transmission) serial bus waveform.

To add a SENT serial bus waveform to the display:

1. Tap **Add New Bus** on the Settings bar.
2. Double-tap the new bus badge to open the bus configuration menu.
3. Set the **Bus Type** to **SENT**.
4. Use the menu fields and controls to configure the **SENT** bus parameters.

To change the settings on an existing CAN serial bus waveform, double-tap the **SENT** bus badge to open the configuration menu.

SENT serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and use the virtual keyboard to enter label text, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **SENT** to set up and display a SENT serial bus waveform.
Source	Select the signal source from the listed analog and digital channels.

Table continued…

Field or control	Description
Threshold	Sets the high/low logic transition level.
Polarity	Sets the signal edge to use to determine the width of the clock tick pulse. **Normal** (default) corresponds to a falling edge polarity that determines how many ticks wide a pulse is. **Inverted** corresponds to a rising edge polarity.
Clock Tick	Sets the time period of the clock tick. The valid range is from 1 μs to 300 μs. The default is 3 μs.
Tick Tolerance	Sets the tolerance, as a percent, that is acceptable for the clock tick signal to be recognized. The valid tolerance range is from 1% to 30%. The default tolerance is 20%.
Fast Data Channels	Sets the number of fast data channels. The default is 2.
Data Nibbles	Sets the number of data nibbles to detect in the serial signal (**3**, **4**, or **6**). Available when **Fast Data Channels** = **1**.
Channel Widths (C1/C2)	Sets the number of bits per channel when using two fast data channels (**12/12**, **14/10**, or **16/8**). Available when **Fast Data Channels** = **2**.
Pause Pulse	Sets the instrument to detect a Pause pulse in the serial data. The default value is Yes.
Slow Channel	Sets the slow channel characteristics. Tap and select from the available list of slow channel types. The default value is None.
Display	**Bus** sets the waveform view to show just the decoded bus information. **Bus and Waveform** sets the waveform view to show both the decoded bus and the source signal waveforms. You can also tap on the + symbol on the bus waveform to toggle between showing the bus only or showing bus and source waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, **Mixed Decimal**, and **Mixed Hex**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SENT serial bus search configuration menu on page 296

SpaceWire serial bus configuration menu

Use the SpaceWire bus menu to set up and display an SpaceWire serial bus waveform.

To set up a SpaceWire serial bus:

- To add a new SpaceWire bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double-tapping the badge. Set the **Bus Type** to **SpaceWire**.
- To change the settings on an existing SpaceWire serial bus waveform, double-tap the **SpaceWire** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

You can *configure search for SpaceWire* and define conditions to search for and mark on an SpaceWire bus waveform.

SpaceWire serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to Bus type to **SpaceWire**.
Strobe Input	Sets the source and threshold level for the signal.
Data Input	Sets the source and threshold level for the signal.
Sync	Sets the synchronization. Available options are Data, Auto, NULL, and Custom.
Data	Set the data. Available when **Sync** = **Data**. Use *Configure pattern editor* on page 301 to customize the data.
Count	Displays the count. Available when **Sync** = **NULL**.
Custom	Set the custom value. Available when **Sync** = **Custom**. Use *Configure pattern editor* on page 301 to customize the custom value.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SpaceWire serial bus search configuration menu on page 299

SPI serial bus configuration menu

Use the SPI bus menu to set up and display an SPI (Serial Peripheral Interface) synchronous serial bus waveform.

To set up the SPI serial bus:

- To create a new SPI bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **SPI**.
- To change the settings on an existing SPI serial bus waveform, double-tap the **Bus** waveform badge to open the configuration menu, and make necessary changes.

SPI serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **SPI**.
Data Inputs	Select the number of inputs (One or Two). One will enable to configure one data input. Two enables to configure MOSI and MISO.
Framing	Set to Slave Select (SS) or Idle framing modes.
SCLK Input	Select the channel source and threshold level for the Serial Clock signal (output from master). Set the Polarity to rising or falling edge of the clock signal used by the master device to start transferring bits. Available when **Framing = SS** and **Data Inputs** = **One**, and **Framing** = **Idle**.
SS Input	Select the channel source and threshold level for the Slave Select signal to start communications with the slave device. Set the Polarity to use Active High or Active Low logic for the SS signal. Available when **Framing** = **SS**.
Data Input	Select the channel source and threshold level for the data input. Set the Polarity to use Active High or Active Low for the SS signal. Available when **Data Inputs** = **One**.

Table continued…

Field or control	Description
MOSI Input	Select the channel source and threshold level for the MOSI (Master In Slave Input) input. Set the Polarity to use Active High or Active Low logic for the MOSI signal. Available when **Data Inputs** = **Two**, and **Framing** = **SS/Idle**.
MISO Input	Select the channel source and threshold level for the MISO (Master In Slave Output) input. Set the Polarity to use Active High or Active Low logic for the MISO signal. Available when **Framing** = **SS/Idle** and **Data Inputs** = **Two**.
Word Size	Enter the word size, in bits. Minimum is 4, maximum is 32, and default is 8.
Bit Order	Set to most significant bit MS First or least significant bit LS First.
Idle Time (Framing =)	Set the idle frame time. Available when **Framing** = **Idle**.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms. This depends on data input. Available only when **Data Input** = **One**.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex** and **Binary**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SPI serial bus search configuration menu on page 298

SPMI serial bus configuration menu

Use the SPMI bus menu to set up and display an SPMI (System Power Management Interface) serial bus waveform.

To add an SPMI serial bus waveform to the display:

1. Tap **Add New Bus** on the Settings bar.
2. Set the **Bus Type** to **SPMI**.
3. Use the menu fields and controls to configure the **SPMI** bus parameters.

To change the settings on an existing SPMI serial bus waveform, double-tap the **SPMI** bus badge to open the configuration menu.

SPMI serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.

Table continued…

Field or control	Description
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **SPMI** to set up and display a SPMI bus waveform.
SCLK Input	Sets the signal **Source** and the high/low logic transition **Threshold** level for the **SCLK** input signal.
SDATA Input	Sets the signal **Source** and the high/low logic transition **Threshold** level for the **SDATA** input signal.
Display	**Bus** sets the waveform view to show just the decoded bus information. **Bus and Waveforms** sets the waveform view to show both the decoded bus and the source signal waveforms. You can also tap on the + symbol on the bus waveform to toggle between showing the bus only or showing bus and source waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SPMI serial bus search configuration menu on page 300

USB serial bus configuration menu

Use the USB bus menu to set up and display an USB 2.0 (Universal Serial Bus) waveform.

To set up a USB serial bus:

- To create a new USB bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **USB**.
- To change the settings on an existing USB serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

USB serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.

Table continued…

Field or control	Description
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **USB**.
Speed	Set the speed to match the USB bus you are acquiring. Selecting High Speed (480 Mbps) sets the Signal Type to Differential.
Signal type	Set to match the USB signal you are acquiring (Single Ended or Differential). Use a differential probe to acquire the differential USB signal. Selecting High Speed (480 Mbps) sets the Signal Type to Differential. Both Single Ended and Diff can be used to measure Full (12 Mbps) and Low (1.2 Mbps) speed USB signals.
Source	Select the channel source for the signal from a differential probe. Only available when Signal Type = Diff.
High Threshold	Set the high threshold level for the differential signal.
Low Threshold	Set the low threshold level for the differential signal.
D+ Input	Select the channel source and set the threshold level for the Data+ signal. Only available when Signal Type = Single Ended.
D- Input	Select the channel source and set the threshold level for the Data- signal. Only available when Signal Type = Single Ended.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

USB serial bus search configuration menu on page 303

Add Plot configuration menu

Use this configuration menu to select and plot the amplitude and time relationship of two (XY) or three (XYZ) signals, which can be sourced from channel, math, or reference waveforms, or plot a TIE measurement eye diagram.

These plots differ from the Measurement plots (added from a Measurement configuration menu), which plot that measurement's value, not the signal source value(s).

To open the **Add Plot** configuration menu, tap the **Add New... Plot** button.

To add a plot:

1. Select a plot type.
2. Select the signal source or sources.
3. Tap **Add**. You can also double tap a plot to add it to the screen. The plot is added to the screen.

Add Plot menu fields and controls

Field or control	Description
Sources	Sets the input sources for the plot. The number of source fields listed depends on the plot.
XY, **XYZ**, **Eye Diagram**	Select the plot type to add to the screen. **Eye Diagram** adds a TIE measurement badge to the Results bar and adds the eye diagram to the screen.
Add	Adds the selected plot to the screen. You can also double tap on a plot type to add it to the screen.

Configuring plots

Each plot has a menu that lets you configure that plot's settings. See *Plot configuration menus* on page 333.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

Add Results Table configuration menu

Use this configuration menu to add a table of all active measurements, searches, or bus decode values. Results tables show values in a spreadsheet-like format.

To open the **Add Results Table** configuration menu, tap the **Add New... Results Table** button.

To add a result table to the screen:

1. Tap **Add New... Results Table**.
2. Select a table type.
3. Tap **Add**. You can also double-tap a table type to add it to the screen.

Add Results Table menu fields and controls

Field or control	Description
Measurements	Display a table of all the measurements in the Results bar.

Table continued…

Field or control	Description
Search	Display a table of all defined searches with each search shown on its own tab.
Bus Decode	Display a table of the bus decode results.
Harmonics/Custom	Displays a table of the Harmonics measurement results. Custom displays a table of the Frequency Response Analysis measurement results. Custom is available only for Inrush Current, Input Capacitance, Turn-on Time, Turn-off time, Control loop response, PSRR, and Impedance measurements.
Add	Displays the selected table type. You can add as many tables as you want to the screen.

Results Tables operations overview

Results tables list summaries of all active measurements, bus decode activity, and search results in a spreadsheet format. Use Results tables to quickly compare values or save the results to a report.

Results Table - general operations

- Double-tap anywhere on a results table to open its configuration menu.
- To scroll up and down in a results table, tap the table and use Multipurpose knob **A** to scroll and select table rows.
- To move a column, touch and drag the column title to a new position in the table.
- To add or remove a column from the table, double-tap the results table to open the table configuration menu and select from available columns to add or remove.
- To resize a column width, use the mouse to position the cursor on the column name border to change, then click and drag the column border to resize that column.
- To configure or delete a single measurement in a Measurement Results table, touch and hold on a table row to open a right-click menu that lets you configure or delete that measurement.
- You cannot sort the contents of a column.

See also

Measurement Table configuration menu on page 242

Bus Decode Results table configuration menu on page 245

Search Results table menu on page 244

Custom Results table configuration menu on page 247

Measurement Table configuration menu

Use this menu to select which statistics to show in the **Measurement Results** table, select the number of digits to show in each column, save a table, and so on.

To open the Measurement Table configuration menu, double-tap anywhere in a Measurement Results table.

Measurement Table menu

Field or control	Description
Table Settings panel	

Table continued…

Field or control	Description
Statistics	Sets the amount of statistical information you want to display for each measurement. **Both** shows both the accumulated results and the current acquisition results. **All Acquisitions** shows statistics for all acquisitions from the last action that cleared acquisition memory. **Current Acquisition** shows the statistics for just the current acquisition.
Show Cycle-to-Cycle Variation	For measurements where this applies, this function allows you to see the mean or average variation between measurements performed on each available cycle in the waveform.
Show tabs with all measurement occurrences	The measurement results table becomes tabbed, with a tab for each measurement. The contents of the current measurement results table are shown on the first tab "Statistics". This tab is always shown first. A tab for each defined measurement is shown to the right of the statistics tab. The left-to-right order of the measurement tabs is the same as the top-to-bottom order of the measurement badges in the Results Bar. Power measurements, Jitter Summary and any other measurement that display more than one value in their Results badge are not shown on their own tabs in the Results Bar. The contents of a measurement tab are calculated when the tab is selected (so the user can see it) and when the contents of the table are saved. If the user rearranges the order of the measurement badges, the order of the tabs in the Results table also update. When a row is selected, the zoom window moves to that occurrence in the record. When the zoom window is moved, the nearest result is highlighted. When the table contents are saved, the content of all tabs come from the same acquisition. Each results table measurement tab content is saved to a separate file, with the measurement badge name used in the file name. If a user-provided file name is specified, that name is appended to the front of the measurement file name (Example: Data_Meas1.csv).
Save Table	Opens a menu with which to save the results table data to a file. See *Save As configuration menu (Measurement Results Table)* on page 244.
Column Visibility panel	
Label	Adds a Label column to show the user-defined label for all measurements. If no user-defined label exists, the column shows the default measurement name.
Peak-to-Peak	Adds a Peak-to-Peak column and shows a Pk-Pk readout for all relevant measurements.
Check boxes to select columns to display	Select or clear individual statistic check boxes to add or remove that column from the table.
Column Resolution panel	
Measurement column resolution	Sets the resolution (number of digits to display) in each measurement column.
Set to Defaults	Sets all columns to show 5 digit readout resolution.

Measurements Results Table operations

- Double-tap anywhere on a results table to open its configuration menu.

- To configure or delete a single measurement in a Measurement Results table, touch and hold on a table row to open a right-click menu that lets you configure or delete that measurement.
- If you add or delete a measurement to the Results bar, that measurement is automatically added to or deleted from an existing Measurement Results table.
- To scroll up and down in a results table, tap the table and use Multipurpose knob **A** to scroll and select table rows.
- To move a column, touch and drag the column title to a new position in the table.
- To add or remove a column from the table, double-tap the results table to open the table configuration menu and select from available columns to add or remove.
- To resize a column width, use the mouse to position the cursor on the column name border to change, then click and drag the column border to resize that column.
- You cannot sort the contents of a column.

Save As configuration menu (Measurement Results Table)

Use the Save As configuration menu to set the location and file name to which to save the contents of a results table, as a comma-separated value (csv) format file.

To open the Save As configuration menu to save a results table to a file:

1. Double-tap anywhere in the Measurement Results table to open the Measurement Table configuration menu.
2. Tap the **Other** panel.
3. Tap **Save Table**.

Save As menu, Measurement Results table fields and controls

Field or control	Description
Save Location	Sets the location to which to save the file. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, for the current save type, up to a maximum of 20 locations.
Browse	Tap to open the Browse Save As Location dialog, to navigate to and select the location to which to save the file. See *Browse Save As Location configuration menu*.
File Name	The file name assigned to the file. The default file name is Tek000. The file name increments on the next save (Tek001, Tek002, and so on). Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the on-screen keyboard and enter a file name.
Save as Type	You can only save table results as comma separated values (.csv) files.

Search Results table menu

Use this menu to configure the content of the Search results table.

To open the Search Results Table configuration menu, double-tap anywhere in the Search Results table. If there are multiple search results tables, tap the tab of the search table to configure and then double-tap anywhere in that table.

Search Results Table menu

Field or control	Description
Table Settings	
Location Timestamp Resolution	Sets the search mark timestamp resolution to display. The timestamp shows the time of each mark relative to the first acquired search mark. **Short** shows a rounded version of the timestamp data. ⚠ **Note:** The actual timestamp data is not truncated by the Short setting, and is retained in the acquisition data. **Precise** adds additional columns to the table to display the full timestamp value by individual time units (seconds, milliseconds, microseconds, and so on).
Delta resolution	Sets the timestamp resolution to display for mark deltas. Delta shows the time difference between each search mark. **Short** shows the delta time between marks as a rounded version of the timestamp data. ⚠ **Note:** The actual timestamp data is not truncated by the Short setting, and is retained in the acquisition data. **Precise** adds additional columns to the table to display the full delta timestamp value by individual time units (seconds, milliseconds, microseconds, and so on).
Save Table	Opens a menu with which to save the results table data to a file. See *Save As configuration menu (Measurement Results Table)* on page 244.
Column Visibility panel	
Check boxes to select columns to display	Select or clear individual check boxes to add or remove that column from the table.

Search Results Table operations

- If there are multiple searches, each search has a tab in the table. Tap the tab of the search to display and/or configure, and then double-tap anywhere in that table to open the configuration menu for that table.
- Double-tap anywhere on a results table to open its configuration menu.
- If you add or delete a search to the Results bar, that search is automatically added to or deleted from an existing Search Results table.
- To scroll up and down in a results table, tap the table and use Multipurpose knob **A** to scroll and select table rows.
- To move a column, touch and drag the column title to a new position in the table.
- To add or remove a column from the table, double-tap the results table to open the table configuration menu and select from available columns to add or remove.
- To resize a column width, use the mouse to position the cursor on the column name border to change, then click and drag the column border to resize that column. If the column width change forces the truncation of data, hovering over a cell displays the full information content of that cell (subject to significant digits limitations on numeric entries).
- You cannot sort the contents of a column.
- You cannot change the order of rows.
- After the search events are displayed in the table, select any row and the waveform zoom points to the respective event in the display. Vice versa, when you move the zoom window, the corresponding row will be selected in the table.

Bus Decode Results table configuration menu

Use this configuration menu to select which information to show in each Bus Decode Results table, or save a Bus Decode Results table to a file.

To open the Bus Decode Table configuration menu, double-tap anywhere in the Bus Decode Results table.

If there are multiple bus decode results tables, each bus has a tab in the table. Tap the tab of the bus to display and/or configure, and then double-tap anywhere in that table to open the configuration menu for that table.

Bus Decode Results table menu

Field or control	Description
Column Visibility panel	
Check boxes for columns in table to display	Select or clear individual check boxes to add or remove that column from the table. The listed columns depend on the selected bus table in the Bus Decode Results table. Select the tab of the bus, at the top of the table, to which you want to make changes, then double-tap the table to open the configuration menu for that table.
Other panel	
Save Table	Opens the Save As dialog to let you save the table data to a comma-separated values (.csv) file. See *Save As configuration menu (Bus Decode Results table)* on page 246.

Bus Decode Results Table operations

- Each bus in a Bus Decode Results table has its own tab. Tap a tab to show the results for that bus.
- Selecting a Bus in the Bus Decode Table configuration menu does not select and display the tab for that bus. Select a tab before configuring the table for that tab.
- Double-tap anywhere on a results table to open the configuration menu for that tab.
- If you add or delete a bus to the Settings bar, that bus is automatically added to or deleted from an existing Bus Decode Results table.
- To scroll up and down in a results table, tap the table and use Multipurpose knob **A** to scroll and select table rows.
- To move a column, touch and drag the column title to a new position in the table.
- To add or remove a column from the table, double-tap the results table to open the table configuration menu and select from available columns to add or remove.
- To resize a column width, use the mouse to position the cursor on the column name border to change, then click and drag the column border to resize that column.
- You cannot sort the contents of a column.
- When both Fast Frame and one or more buses are in use, the Bus Decode Results table will be modified as follows:
  - A new column called Frame is added between Index and Start Time. This column shows the frame number the serial packet was captured in. Note that more than one packet could be captured in a single frame, so this number may be the same for multiple rows.
  - The bus decode table will show all packets from all frames.
- After the decoded packets are displyed in the table, select any row and the waveform zoom points to the respective packet in the bus display. Vice versa, when you move the zoom window, the coresponding packet in the table will be selected.

Save As configuration menu (Bus Decode Results table)

Use this configuration menu to set the location and file name to which to save the contents of a results table, as a comma-separated value (csv) format file.

To open the Save As configuration menu:

1. Double-tap anywhere in the Bus Decode Results table to open the Bus Decode Results configuration menu.
2. Tap the **Other** panel.
3. Tap **Save Table**.

Save As menu (Bus Decode Results table) fields and controls

Field or control	Description
Save Location	Sets the location to which to save the file. The default value is the last location to which a file of this type was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, for the current save type, up to a maximum of 20 locations.
Browse	Tap to open the Browse Save As Location configuration menu, to navigate to and select the location to which to save the file. See *Browse Save As Location configuration menu*.
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a default name created by the instrument if this file type has not previously been saved with a custom file name. The default file name uses the format Tek000. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save as Type	You can only save table results as comma separated values (.csv) files.

Custom Results table configuration menu

Use this configuration menu to select the information to be displayed in the Custom Results Table, or save a Custom table to a file.

To open a Custom Results Table configuration menu, double-tap anywhere in the Custom Results table.

Note:

- If multiple harmonics results tables are displayed, each result is in a separate tab. Tap the harmonics tab of interest to open that table, and then double-tap in the Harmonics results table area to open that table's configuration menu.
- Custom is displayed only for Control Loop Response and Power Supply Rejection Ratio measurements.

Custom Table configuration menu

Field or control	Description
Table Settings panel	
Column Visibility	Select or clear individual check boxes to add or remove that column from the table.
Units panel: Available only for Harmonics measurement.	
Units	Sets the vertical scale used to display measurement data. Select Log or Linear.
Harmonics	Select **All** to show all harmonics, **Odd** to show just odd harmonics, or **Even** to show just even harmonics in the table. **Note:** Changing this field also changes the harmonics bars shown in the Harmonics bar plot.

Table continued…

Field or control	Description
Save panel	
Save Table	Opens a menu with which to save the Custom Results table data to a comma-separated value format file. See *Save As configuration menu (Custom Results Table)* on page 248.

Save As configuration menu (Custom Results Table)

Use this configuration menu to set the location and file name to save the Custom Results table content as a comma-separated value (csv) format file.

To open the Save As configuration menu to save a Custom Results table to a file:

1. Double-tap anywhere in the Custom Results table to open the Custom Table configuration menu.
2. Tap the **Save** panel.
3. Tap the **Save Table** button.

Save As configuration menu (Custom Results Table)

Field or control	Description
Save Location	Sets the location to save the file. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter the location Use and external keyboard or double-tap on the file name to open the on-screen keyboard and enter the path details. You can also use the Browse button to open a menu to navigate to and select the location to which to save the file. Tap the down arrow icon on the right end of the field to open the list of recently saved file locations, for the current save type, up to a maximum of 20 locations.
Browse	Tap to open the Browse Save As Location dialog, to navigate to and select the location to which to save the file. See *Browse Save As Location configuration menu*.
File Name	The file name assigned to the file. The default file name is Tek000. The file name increments on the next save (Tek001, Tek002, and so on). Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the on-screen keyboard and enter a file name.
Save as Type	You can only save a Custom Results table to a comma-separated value (csv) format file.

Annotation and navigation on waveform plots/data and results table

Annotation and Navigation are applicable for all the cycle based measurements such as Switching Loss, di/dt, dv/dt, Timing Analysis, and Amplitude Analysis.

To navigate on the results table follow the steps:

1. Tap **Add New... Measurement**.
2. Select the appropriate signals source or sources.
3. Select a supported power measurement (Switching Loss, di/dt, dv/dt, $R_{DS(on)}$, Timing Analysis, or Amplitude Analysis) and tap **Add**.
4. Double-tap the added power measurement badge to open that measurement's configuration menu.
5. Tap **Power Autoset** button for live signals.

6. Select the **Time Trend** Plot if the measurement includes it.
7. Push the **Single/Seq** button on the front panel.
8. Single-tap the measurement badge to display the navigation buttons on the badge.
9. Tap the **Next** or**Previous** buttons to navigate to the next or previous measurement region of the waveform view. The values for each measurement region are displayed above the navigation buttons on the badge. The selected measurement regions are also highlighted in the Zoom view on the Waveform view.
10. Tap the **Min** or **Max** buttons to navigate to the measurement region containing the minimum and maximum measurement values.

Note:

- $R_{DS(on)}$ supports an annotation feature that marks the minimum value of $R_{DS(on)}$ with a line in each switching cycle.
- When $R_{DS(on)}$ is added, a Math waveform is created. Math (resistance curve) equation can be a V/I or ΔV/ ΔI based on the measurement configuration, where V/I is static and ΔV/ ΔI is dynamic in nature.
- You can use the Measurement badge **Previous** and **Next** buttons to navigate from switching cycle to switching cycle. You can also view the minimum $R_{DS(on)}$ value marked in the respective switching cycle.
- By default, $R_{DS(on)}$ places the annotation lines on the switching cycle where the minimum $R_{DS(on)}$ value occurs. When you select the measurement results badge, the navigation function enables zoom mode and centers the minimum $R_{DS(on)}$ value for the switching cycle in the display.

Navigation on Bar Graph and Harmonics Results Table

1. Add **Harmonics** measurement and configure the parameters.
2. If the source is a live channel, tap **Power Autoset** button.
3. Add Harmonic **Bar Graph**.
4. Tap the Results table and add Harmonics table.
5. Press the **Single/Seq** button, on the front panel.
6. Tap **Harmonics** measurement badge, which expands to display the previous and next button.
7. Tap the navigation button to navigate on the next/previous bar on the Bar Graph and to the respective row in the Harmonic table. This navigation is also vice versa and can be navigated by a single tap on a column or row or using the navigation button.

Note:

- When all the displays are synchronized, Results table row will be highlighted and the selected Bar will be displayed in White color.
- If more than one Bar Graph is added, then all the Bar Graphs will be associated with the Results Table at the same time.

Search configuration menu overview

Use the Search configuration menu to define conditions that you want to mark on a channel or waveform signal.

To open the Search configuration menu, double-tap a Search badge in the results bar.

If there are no Search badges on the Results bar, tap the **Add New... Search** button. A Search badge is added to the Settings bar, and the Search configuration menu opens to search type of Edge (default).

The search types and settings are similar to their corresponding trigger types (Edge, Pulse Width, Runt, and so on).

You can create multiple searches for the same waveform. Each occurrence of the search condition is marked with a triangle along the top of the waveform. Marks for each search are shown in a different color.

Use the following links to access information on specific Search settings.

Bus Search configuration menus

Use a Bus search to search for and mark bus-related events (Start, Stop, Missing Acq, Address, Data, and so on) on a bus waveform.

To create a new Bus search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Bus**.
3. Select the bus **Source.**
4. Use the search menu fields to set the search parameters.

To change the settings of an existing search, double-tap the search badge to open its configuration menu and make necessary changes.

Select a link to view the configuration menu settings for a specific bus.

8b10b serial bus search configuration menu on page 208

ARINC 429 serial bus search configuration menu on page 252

Audio serial bus search configuration menu on page 254

Auto Ethernet serial bus search configuration menu on page 255

CAN serial bus search configuration menu (when not using a .dbc symbol definition file) on page 257

Ethernet serial bus search configuration menu on page 272

eUSB serial bus search configuration menu on page 274

FlexRay serial bus search configuration menu on page 277

I2C serial bus search configuration menu on page 279

I3C serial bus Search configuration menu on page 282

LIN serial bus search configuration menu on page 290

MDIO serial bus search configuration menu on page 269

MIL-STD-1553 serial bus search configuration menu on page 291

NRZ serial bus search configuration menu on page 293

Parallel bus search configuration menu on page 294

RS-232 serial bus search configuration menu on page 295

SENT serial bus search configuration menu on page 296

SpaceWire serial bus search configuration menu on page 299

SPI serial bus search configuration menu on page 298

SPMI serial bus search configuration menu on page 300

SVID serial bus search configuration menu on page 267

USB serial bus search configuration menu on page 303

PSI5 serial bus configuration menu on page 226

Other search types

Edge Search configuration menu on page 305

Logic search configuration menu on page 306

Pulse Width Search configuration menu on page 308

Rise/Fall Time Search configuration menu on page 310

Runt Search configuration menu on page 311

Setup and Hold Search configuration menu on page 312

Timeout Search configuration menu on page 314

Window Search configuration menu

DDR Read Search configuration menu on page 316

DDR Write Search configuration menu on page 318

DDR Read & Write Search configuration menu on page 320

8b10b serial bus search configuration menu

Use the 8b10b Search configuration menu to define conditions to search for and mark on a 8b10b bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the 8b10b bus that you want to search.
Mark On	Select the type of information for which to search.
Format	Sets the number of data bytes for which to search (one to five bytes). Use the **A** knob to change the value. Only available when **Mark On** = **Symbols**.
Type	Sets the type to search (**Data** or **Control**). Only available when **Mark On** = **Symbols** and **Format** = **8-Bit**.
Disparity	Sets the disparity type as **Positive** or **Negative** or **Either**. Only available when **Mark On** = **Symbols** and **Format** = **8-Bit**, **10-Bit**, and **Symbol**. **Note:** Set the disparity value and then query the 10 bit data value.

Table continued…

Field or control	Description
Value	Sets the value for the format. • When **Format** = **8-Bit**, you can set 8-Bit value in Binary or Hex format. • When **Format** = **10-Bit**, you can set 10-Bit value in Binary or Hex format. • When **Format** = **Symbol**, you can set values in decimal format. Only available when **Mark On** = **Symbols**.
10Bit RD+	Sets the RDpositive 10 bit value. It displays the corresponding 10 bit value for 8 bit or symbol. Set the Binary and Hex values Available when Format = 10 Bit
10Bit RD+	Sets the RDnegative 10 bit value. It displays the corresponding 10 bit value for 8 bit or symbol. Set the Binary and Hex values Available when Format = 10 Bit
Error Type	Sets the error type (**Symbol or Disparity**) for which to search and mark on the waveform. Only available when **Mark On** = **Errors**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. 8b10b is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. 8b10b is currently not supported as a bus trigger source.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

8b10b serial bus configuration menu

ARINC 429 serial bus search configuration menu

Use the ARINC 429 Search configuration menu to define conditions to search for and mark on an ARINC 429 bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the **ARINC 429** bus to search.

Table continued…

Field or control	Description
Mark When	Sets the type of information for which to search.
Trigger When Label	Sets the label condition for which to search.
Label	Sets the label pattern for which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** = **Label** or **Label & Data**.
Label Low	Sets the low value of the label pattern range for which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** = **Label** and **Mark When Label** = **Inside Range** or **Outside Range**.
Label High	Sets the high value of the label pattern range for which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** = **Label** and **Mark When Label** = **Inside Range** or **Outside Range**.
Trigger When Data	Sets the data condition for which to search.
Data	Sets the data pattern for which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** = **Data** or **Label & Data**.
Data Low	Sets the low value of the data pattern range for which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** = **Data** and **Mark When Data** = **Inside Range** or **Outside Range**.
Data High	Sets the high value of the data pattern range for which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** = **Data** and **Mark When Data** = **Inside Range** or **Outside Range**.
SSM	Sets the Sign/Status Matrix (SSM) bit condition for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** = **Data** or **Label & Data**.

Table continued…

Field or control	Description
SDI	Sets the Source/Destination Identifier (SDI) bit condition for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** = **Data** or **Label & Data**.
Error Type	Sets the error condition for which to search. Available when **Mark When** = **Error**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

ARINC 429 serial bus menu on page 210

Audio serial bus search configuration menu

Use the Audio Search configuration menu to define conditions to search for and mark on an Audio bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the **Audio** bus to search.
Mark On	Select the type of information for which to search.
Data	Sets the data pattern for which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data**.
Word	Sets the audio word channel for which to search. Only available when **Mark On** = **Data**.

Table continued…

Field or control	Description
Mark When	Sets the mark when condition for the specified data pattern. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Audio serial bus configuration menu on page 211

Auto Ethernet serial bus search configuration menu

Use the Auto Ethernet Search configuration menu to define conditions to search for and mark on an Auto Ethernet bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the Auto Ethernet bus that you want to search.
Mark On	Select the type of information for which to search.
MAC Address Destination, MAC Address Source	Sets the MAC destination and/or source address pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Mac Address**.

Table continued…

Field or control	Description
Q-Tag	Sets the Q-tag pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Q-Tag Control Information**.
MAC Length/Type	Enter the MAC length or type pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **MAC Length/Type**.
IP Protocol	Sets the IP protocol pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **IP Header**.
Source, Destination Address	Sets the source and/or destination IP address pattern for which to search. Tap a **Hex** or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **IP Header**.
Source Port, Destination Port	Sets the source and/or destination TCP header port pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **TCP Header**.
Sequence Number	Sets the TCP header sequence number pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **TCP Header**.
Ack Number	Sets the TCP header ack number pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **TCP Header**.
Data Bytes	Sets the number of client data bytes for which to search (one to sixteen bytes). Use the **A** knob to change the value. Only available when **Mark On** = **Client Data**.
Byte Offset	Sets the client data byte offset (**Don't Care** or the number of bytes). Tap the field and use the **A** knob to change the value. Only available when **Mark On** = **Client Data**.

Table continued…

Field or control	Description
Client Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex** or **ASCII** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Client Data.**
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **MAC Length/Type** or **Client Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. Auto Ethernet is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. Auto Ethernet is currently not supported as a bus trigger source.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Auto Ethernet serial bus configuration menu on page 212

CAN serial bus search configuration menu (when not using a .dbc symbol definition file)

Use the CAN Search configuration menu to define conditions to search for and mark on an CAN bus waveform. This menu supports searches for CAN bus events when a .dbc symbol definition file is not loaded.

Note: See *CAN serial bus search configuration menu (when using a .dbc symbol definition file)* on page 259 for CAN serial bus searches that use a loaded .dbc symbol definition file.

Field or control	Description
Display	Enables or disables displaying waveform search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.

Table continued…

Field or control	Description
Source	Select the **CAN** bus that you want to search.
Mark On	Sets the type of information for which to search and mark on the bus waveform acquisition. **Start of Frame** puts a search mark at each occurrence of a CAN frame start. **Type of Frame** puts a search mark at each occurrence of the frame type as set in the Frame Type field. **Identifier** puts a search mark at each occurrence of a CAN identifier frame as set in the **Direction**, **Identifier Format**, and **Identifier** fields. Not available when the bus badge **Decode Format** is set to **Symbolic**. **Data** puts a search mark at the occurrence of the specified data fields (**Data Bytes**, **Data Offset**, **Mark When**, and **Data**). Not available when the bus badge **Decode Format** is set to **Symbolic**. **Id & Data** puts a search mark at each occurrence of the specified Id and data fields. Not available when the bus badge **Decode Format** is set to **Symbolic.** **End of Frame** puts a search mark at each occurrence of a CAN frame end. **Error** puts a search mark at each occurrence of the specified error condition (**Missing Ack**, **Bit Stuffing**, **FD Form Error**, or **Any Error**).
Frame Type	Sets the frame type for which to search and mark on the bus waveform (**Data Frame**, **Remote Frame**, **Error Frame**, **Overload Frame**). Only available when **Mark On** = **Type of Frame**.
Error Type	Sets the error type for which to search and mark on the bus waveform. (**Missing Ack**, **Bit Stuffing**, or **Any Error**. Only available when **Mark On** = **Error**.
Direction	Sets the Identifier's transfer direction (**Read**, **Write**, or **Either**) for which to search. Only available when **Mark On** = **Identifier**.
Identifier Format	Sets the CAN identifier format bit length for which to search (**Standard** 11-bit, or **Extended** 29-bit for CAN 2.0B). Only available when **Mark On** = **Identifier** or **ID & Data**.
Identifier	Sets the identifier pattern for which to search. The number of bits shown depends on the **Identifier Format** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Identifier** or **ID & Data**.
Data Bytes	Sets the number of data bytes for which to search (one to eight bytes). Use the **A** knob to change the value. Only available when **Mark On** = **Data** or **ID & Data**.

Table continued…

Field or control	Description
Data Offset	Sets the data offset value. Use the **A** knob to change the value. Only available when **Mark On** = **Data** or **ID & Data**.
Mark When	Sets the mark when condition (mark the waveform when the waveform is =, ≠, <, >, ≤, or ≥ the data conditions). Only available when **Mark On** = **Data** or **ID & Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data** or **ID & Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit to change. Use the **B** knob to change the value of the digit.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

CAN serial bus configuration menu on page 213

CAN serial bus search configuration menu (when using a .dbc symbol definition file)

Use the CAN Search configuration menu to define conditions to search for and mark on an CAN bus waveform. This menu supports searches for CAN bus events based on labels derived from a loaded .dbc symbol definition file.

Field or control	Description
Display	Enables or disables displaying waveform search marks for this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the **CAN** bus that you want to search and mark.

Table continued…

Field or control	Description
Mark On	Sets the type of information for which to search and mark on the bus waveform acquisition. **Start of Frame** puts a search mark at each occurrence of a CAN frame start. **FD Bits** puts a search mark at each occurrence of the specified **BRS Bit** (Bit Rate Switch) and **ESI Bit** (Error State Indicator) states. Not available when the bus badge **Decode Format** is set to **Symbolic**. **End of Frame** puts a search mark at each occurrence of a CAN frame end. **Error** puts a search mark at each occurrence of the specified error condition (**Missing Ack**, **Bit Stuffing**, **FD Form Error**, or **Any Error**). **Symbolic** puts a search mark at each occurrence of a specified message or signal condition as defined by an installed .dbc symbol definition file. Only available when the bus badge **Decode Format** is set to **Symbolic**.
Configure	Opens the **CAN DBC Symbolic Configuration** menu, which sets the CAN bus search parameters based on the message and signal labeling derived from an installed .dbc symbol definition file. See *CAN serial bus search configuration menu (when using a .dbc symbol definition file)* on page 259.
Message, **Signal**, and **Value** fields	These fields display the actual search data values for the **Message**, **Signal**, and **Value** fields as set in the **CAN DBC Symbolic Configuration** menu. These fields are shown for reference and are not editable.
BRS Bits	Sets the **BRS** (Bit Rate Switch) bit state for which to search and mark at each occurrence on the bus waveform. Only available when **Mark On** = **FD Bits**.
ESI Bits	Sets the **ESI** (Error State Indicator) bit state for which to search and mark at each occurrence on the bus waveform. Only available when **Mark On** = **FD Bits**.
Error Type	Sets the error type for which to search and mark on the bus waveform. (**Missing Ack**, **Bit Stuffing**, or **Any Error**. Only available when **Mark On** = **Error**.
A, B knob controls	Use the **A** knob to select (highlight) the digit to change. Use the **B** knob to change the value of the digit.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

CAN serial bus configuration menu on page 213

CAN DBC Symbolic Configuration menu

Use this menu to set the CAN .dbc symbol definitions for which to search. This menu is accessed from a CAN Bus Search menu when a .dbc symbol definition file is loaded on the instrument. Load the .dbc file when you first add and configure the CAN bus.

Field or control	Description
Symbolic Type	Sets whether to search for **Message** CAN or search on both **Message and Signal** code.
Message	Sets the CAN message label for which to search. Use the drop-down list to select from message labels derived from the installed .dbc file.
Signal	Sets the CAN signal label for which to search. Use the drop-down list to select from signal labels derived from the installed .dbc file.
Signal Encoded	Enables searching for encoded signal values (when selected) or specific values (when not selected). Use the drop-down list to select from encoded signal types derived from the installed .dbc file. Value searching is always available for a signal. Signals, however, are not necessarily encoded, so searches on signal encoding are not always available.
Value	Sets a specific **Signal** value for which to search. Use knob A to set a value, or double-tap the field and enter a value using the virtual keyboard.

See also

CAN serial bus search configuration menu (when using a .dbc symbol definition file) on page 259

CAN serial bus configuration menu on page 213

CAN serial bus configuration menu on page 213

DPHY serial bus search configuration menu

Use the DPHY Search configuration menu to define the conditions to search for and mark on an DPHY bus.

Table 5: DPHY search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the DPHY bus that you want to search.

Table continued…

Field or control	Description
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates will show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
SoT	Select the Start of Transmission (SoT) for each transmission.
EoT	Select the End of Transmission (EoT) for each transmission.
Mode	Set the mode of operation to HS or LP. Available only when **Mark On** = **Data**, **Scrambling**, **Compression**, **Packets**, and **Errors**.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Tap the field and use the A knob to change the value. Available only when **Mark On** = **Data**.
Data	Sets the data packet pattern for which to search. The number of bits shown depends on the Data Bytes setting. Use in conjunction with the Mark When field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** = **Data**.
Scrambling	Search the scrambling mode command.
Compression	Search for the compression mode command.
Packet Type	Sets the packet type for which to search. Available options are **Short** or **Long**. Default is Short. Available only when **Mark On** = **Packets**.
Packet Name	Sets the packet name for which to search. Available only when **Mark On** = **Packets**.
BusTurnAround	Searches the bus turn around.
Escape	Search for the escape entry mode.
Stop	Search for escape mode exit.
Error Type	Sets the error type for which to search. Available when **Mark On** = **Error**. Available options are **Any**, **ECC**, or **CRC**. Default is Any.

Table continued…

Field or control	Description
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. DPHY is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. DPHY is currently not supported as a bus trigger source.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

DPHY serial bus configuration menu on page 216

PSI5 serial bus search configuration menu

Use the PSI5 Search configuration menu to define conditions to search for and mark on an PSI5 bus waveform. You can have multiple searches on the same bus.

Table 6: PSI5 search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the PSI5 bus that you want to search.

Table continued…

Field or control	Description
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates to show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings. When **Direction = ECU to Sensor**, the available settings are: • Start • Data • Function Code • Sensor Address • Register Address • CRC Error When **Direction = Sensor to ECU**, the available settings are: • Start • Status Available when the status bit is configured in bus configuration. • Data • Block ID • Sensor Status • Error
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. PSI5 is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. PSI5 is currently not supported as a bus trigger source.

Mark On = Start

This table lists the menu fields displayed when **Mark On** is set to **Start**.

Field or control	Description
Start	Sets the search to mark **Start** events. A **Start** event occurs with two zeros when direction is from sensor to ECU. When direction is from ECU to sensor, Start event occurs with 011111110 when sync mode is pulse width, and 010 when sync mode is tooth gap. When direction is from Sensor to ECU, Start event occurs with 00. There are no menu fields or controls displayed when this item is selected. When **Direction = ECU to Sensor** and **Direction = ECU to Sensor**.

Mark On = Data

This table lists the menu fields displayed when **Mark On** is set to **Data**.

Field or control	Description
Data Bits	When sync mode is tooth gap, data bits are 4 and 8 bits. When sync mode is pulse width data bits are 4 , 8 and 20
Data	Sets the search to mark events that meet the specified data value. For Sensor to ECU, it displays Region A. If the Data B field is configured Bus configuration, displays Region B as data search option. For ECU to Sensor, data search displays Data Bits and Value. In configuration Sync mode is Tooth Gap, the number of data bits is 4 and 8. In configuration Sync Mode is Pulse Width, the data bits are 4, 8, and 20. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Data** and **Direction = ECU to Sensor**.

On = Function Code

This table lists the menu fields displayed when **Mark On** is set to **Function Code**.

Field or control	Description
Function Code	Sets the search to mark events that meet the specified data value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Function Code** and **Direction = ECU to Sensor**.

Mark On = Sensor Address

This table lists the menu fields displayed when **Mark On** is set to **Sensor Address**.

Field or control	Description
Address	Sets the sensor address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Sensor Address** and **Direction = ECU to Sensor**.

Mark On = Register Address

This table lists the menu fields displayed when **Mark On** is set to **Register Address**.

Field or control	Description
No. of Bits	2-Bit 6-Bit 8-Bit Only available when **Mark On = Register Address** and **Direction = ECU to Sensor**.

Table continued…

Field or control	Description
Register Address	Sets the register address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Register Address** and **Direction = ECU to Sensor**.

Mark On = CRC Error

This table lists the menu fields displayed when **Mark On** is set to **CRC Error**.

Field or control	Description
CRC Error	Sets the search to mark **CRC Error** events. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = CRC Error** and **Direction = ECU to Sensor**.

Mark On = Sensor Status

This table lists the menu fields displayed when **Mark On** is set to **Sensor Status**.

Field or control	Description
Type	Sets the search to mark events that meet the specified type. Available options are **Sensor Ready**, **Sensor Busy**, **Sensor Defect**, **Sensor Unlocked**, and **Service Mode**. Only available when **Mark On = Sensor Status** and **Direction = Sensor to ECU**.

Mark On = Block ID

This table lists the menu fields displayed when **Mark On** is set to **Block ID**.

Field or control	Description
CRC Error	Sets the search to mark **Block ID** events. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Block ID** and **Direction = Sensor to ECU**.

Mark On = Status

This table lists the menu fields displayed when **Mark On** is set to **Status**.

Field or control	Description
CRC Error	Sets the search to mark **Status** events. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Status** and **Direction = Sensor to ECU**.

Mark On = Error

This table lists the menu fields displayed when **Mark On** is set to **Error**.

Field or control	Description
Error	Sets the search to mark **Error** type. Available error types are **Any**, **Parity**, **CRC**, and **Response Code**. Only available when **Mark On = Error** and **Direction = Sensor to ECU**.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

PSI5 serial bus configuration menu on page 226

SVID serial bus search configuration menu

Use the SVID Search configuration menu to define the conditions to search for and mark on an SVID bus waveform. You can have multiple searches on the same bus.

Table 7: SVID search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the SVID bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates will show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. SVID is currently not supported as a bus trigger source.

Table continued…

Field or control	Description
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. SVID is currently not supported as a bus trigger source.

Mark On = Start

This table lists the menu fields displayed when **Mark On** is set to **Start**.

Field or control	Description
Start	A **Start** event occurs when the SDA line transitions from a High to Low state while the SCL line remains High. There are no menu fields or controls displayed when this item is selected.

Mark On = Slave Address

This table lists the menu fields displayed when **Mark On** is set to **Start**.

Field or control	Description
Slave Address	Sets the slave address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Slave Address**.

Mark On = Command

This table lists the menu fields displayed when **Mark On** is set to **Command**.

Field or control	Description
Command Type	Sets the search to mark **Command** events. Available Command Types: • **Get** • **Set** • **Optional** Available when **Mark On** = **Command**.
Command Response	Set the command response. • Available options for Get are GetReg, GetRegVendor, and GetRegTestCfg. • Available options for Set are SetVID Fast, SetVID Slow, SetVID Decay, SetPS, SetRegAddr, SetRegData, SetRegAddrVendor, SetRegDataVendor, SetRegAddrTestCfg, and SetRegDataTestCfg. • Available options for Optional are Test Mode, SwtWP, GetRegPktRecent, GetRegPktAlert, GetRegPktBad, and GetRegVrEvent. Available when **Mark On** = **Command**.

Mark On = Payload

This table lists the menu fields displayed when **Mark On** is set to **Payload**.

Field or control	Description
Payload Type	Sets the search to mark events that meet the specified address for a Device or register location. The **Payload** sets the number of bits in the field. Available payload types are Master, Slave, or Either. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Payload**.

Mark On = Errors

This table lists the menu fields displayed when **Mark On** is set to **Errors**.

Field or control	Description
Errors	Sets the search to mark events that meet the specified address for a Device or register location. The **Errors** sets the number of bits in the field. Tap the **Binary** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Errors**.
Error Type	Set the error type from the drop down. Available error types are Any, NACK, Reject, Mixed, and Parity.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SVID serial bus configuration menu on page 227

MDIO serial bus search configuration menu

Use the MDIO Search configuration menu to define the conditions to search for and mark on an MDIO bus waveform. You can have multiple searches on the same bus.

Table 8: MDIO search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.

Table continued…

Field or control	Description
Source	Select the MDIO bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates to show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. MDIO is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. MDIO is currently not supported as a bus trigger source.

Mark On = Start Packet

This table lists the menu fields displayed when **Mark On** is set to **Start Packet**.

Field or control	Description
Start Packet	Sets the search to mark **Start** events. A **Start Packet** event occurs when the SDA line transitions from a High to Low state while the SCL line remains High. There are no menu fields or controls displayed when this item is selected.

Mark On = OpCode

This table lists the menu fields displayed when **Mark On** is set to **OpCode**.

Field or control	Description
OpCode	Sets to search two bit operation codes. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark When** = **OpCode**.

Mark On = Physical Address

This table lists the menu fields displayed when **Mark On** is set to **Physical Address**.

Field or control	Description
Physical Address	Set the 5 bit long field to search for the physical PHYs which are used to interface to the network. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.

Mark On = Register Address

This table lists the menu fields displayed when **Mark On** is set to **Register Address**.

Field or control	Description
Register Address	Sets the 5 bit long field to search the register to be written to or read from. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Register Address**.

Mark On = Device Type

This table lists the menu fields displayed when **Mark On** is set to **Device Type**.

Field or control	Description
Device Type	Sets the 5 bit long field to search for the device type of clause 45. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Device Type**.

Mark On = Data

This table lists the menu fields displayed when **Mark On** is set to **Data**.

Field or control	Description
Data	Sets the search to mark events that meet the specified data value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Data**.

Mark On = Address

This table lists the menu fields displayed when **Mark On** is set to **Address**.

Field or control	Description
Address	Sets the 16 bit long field search for the specified address. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Address**.

Mark On = Error

This table lists the menu fields displayed when **Mark On** is set to **Error**.

Field or control	Description
Error	Sets the error on which to search. Available when **Mark On** = **Error**.
Error Type	Sets the error type on which to search. Select the error type from the drop-down. Available error types are: • OpCode • Device Type • Any Available when **Mark On** = **Error**.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

MDIO serial bus configuration menu on page 228

Ethernet serial bus search configuration menu

Use the Ethernet Search configuration menu to define conditions to search for and mark on an Ethernet bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the Ethernet bus that you want to search.
Mark On	Select the type of information for which to search.
MAC Address Destination, MAC Address Source	Sets the MAC destination and/or source address pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Mac Addresses**.
Q-Tag	Sets the Q-tag pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Q-Tag Control Information**.

Table continued…

Field or control	Description
MAC Length/Type	Enter the MAC length or type pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when M**ark On** = **MAC Length/Type**.
IP Protocol	Sets the IP protocol pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **IP Header**.
Source, Destination Address	Sets the source and/or destination IP address pattern for which to search. Tap a **Hex** or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **IP Header**.
Source Port, Destination Port	Sets the source and/or destination TCP header port pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **TCP Header**.
Sequence Number	Sets the TCP header sequence number pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **TCP Header**.
Ack Number	Sets the TCP header ack number pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **TCP Header**.
Data Bytes	Sets the number of client data bytes for which to search (one to sixteen bytes). Use the **A** knob to change the value. Only available when **Mark On** = **Client Data**.
Byte Offset	Sets the client data byte offset (**Don't Care** or the number of bytes). Tap the field and use the **A** knob to change the value. Only available when **Mark On** = **Client Data**.
Client Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex** or **ASCII** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Client Data.**

Table continued…

Field or control	Description
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **MAC Length/Type** or **Client Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Ethernet serial bus menu on page 217

eUSB serial bus search configuration menu

Use the eUSB Search configuration menu to define conditions to search for and mark on an eUSB bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the eUSB bus that you want to search.
Mark On	Select the type of information for which to search.
Sync Bits Search	Select to search for the number of sync bits. Available when **Mark On = Sync**.
Sync Bits	Set the number of sync bits. By default Sync Bits are OFF. ⚠ **Note:** Sync Bits will be On, only when Speed is High in the Bus configuration. Available when **Mark On = Sync**, **Sync Bits Search = ON** and **Mark When Sync Bits =** all except, **Inside Range** and **Outside Range**.

Table continued…

Field or control	Description
Handshake Type	Sets the handshake packet type for which to search. Available options are **Any (XX10)**, **ACK (0010)**, **NAK (1010)**, **STALL (1110)**, or **NYET (0110)**. **Note:** NYET is applicable only in High Speed. Only available when **Mark On** = **Handshake Packet**.
Packet Type	Sets the special packet type for which to search. Available options are **Any (XX00)**, **PRE (1100)**, and **RESERVED (0000)**. **Note: ERR (1100)**, **SPLIT (1000)**, **PING (0100)** are available only for High Speed. Only available when **Mark On** = **Special Packet**.
Hub Address	Sets the special packet address pattern for which to search. Tap the **Binary** or **Hex** and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Special Packet** and **Packet Type** = **Any (XX00)**, **PRE (1100)**, or **RESERVED (0000)**. Available when high speed is selected and **Mark On** = **Special Packet**, **ERR (1100)**, **SPLIT (1000)**, **PING (0100)** are displayed.
Port Address	Sets the special packet address pattern for which to search. Tap the **Binary** or **Hex** and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Special Packet** and **Packet Type** = **Any (XX00)**, **PRE (1100)**, or **RESERVED (0000).**. Available when high speed is selected and **Mark On** = **Special Packet**, **ERR (1100)**, **SPLIT (1000)**, **PING (0100)** are displayed.
Error Type	Sets the error type for which to search. Available options are **PID Check Bits**, **Token CRC5**, **Data CRC16**, and **Bit Stuffing**. Bit Stuffing is not available when High Speed is selected inthe configuration. Only available when **Mark On** = **Error**.
Token Type	Sets the token packet type for which to search. Only available when **Mark On** = **Token Packet**.
Mark When Address	Sets the token type pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Token Packet**, **Token Type** = all except **SOF (0101)**, **Mark When Address** = **Inside Range**, and **Outside Range**.

Table continued…

Field or control	Description
Address	Sets the token packet address pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Token Packet** and **Token Type** = all except **SOF (0101)**. When **Mark On** = **Register Access Protocol** and **Fields** = **Address**.
Endpoint	Sets the token packet endpoint pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Token Packet** and **Token Type** = all except **SOF (0101)**.
Frame Number	Sets the frame number pattern for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Token Packet** and **Token Type** = **SOF (0101)**.
Data Packet Type	Sets the data packet type for which to search. Only available when **Mark On** = **Data Packet**. Available options are **ANY (1011)**, **DATA0 (1011)**, and **DATA1 (1011)**. Available options **DATA2 (0111)** and **MDATA (1111)** when High Speed is selected in the configuration.
Data Bytes	Sets the number of data bytes for which to search (one to sixteen bytes). Tap the field and use the **A** knob to change the value. Only available when **Mark On** = **Data Packet**.
Byte Offset	Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the **A** knob to change the value. Only available when **Mark On** = **Data Packet**.
Data	Sets the data packet pattern for which to search. The number of bits shown depends on the Data Bytes setting. Use in conjunction with the Mark When field to specify the exact search condition. Tap the **Binary** or **Hex** or **ASCII** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data Packet** and **Mark On**=**Data Packet**=**Register Access Protocol**, **Fields**=**Data**.
Reset	Select to search all the reset events. It indicates bus reset.

Table continued…

Field or control	Description
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** or **ASCII** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Token Packet** or **Data Packet**.
Resume/Wake	Select to search for all the Resume/Wakeup events.
Connect	Select to search for all the connected events. This event is triggered when the device is connected after POR.
Control Message	Select to search for all the control messages.
Suspend	Select to search all the suspended events.
Device Chirp	Select to search for all the device chirp events.
Host chirp	Select to search for all the host chirp events.
Port Reset	Select to search for all the port reset events.
Port Configuration	Select to search for port configuration events.
Fields	Select the fields. Available options are Command, Address, or Data. Only available when **Mark On** = **Register Access Protocol**.
RAP Command	Only available when **Mark On** = **Register Access Protocol** and **Fields**= **Command**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. eUSB is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. eUSB is currently not supported as a bus trigger source.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

eUSB serial bus configuration menu on page 229

FlexRay serial bus search configuration menu

Use the FlexRay Search configuration menu to define conditions to search for and mark on an FlexRay bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.

Table continued…

Field or control	Description
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the FlexRay bus for which to search.
Mark On	Select the type of information for which to search.
Indicator Bits	Select the defined indicator bits type for which to search from the drop-down list. Only available when **Mark On** = **Indicator Bits**.
Indicator Bits	Enter the indicator bits for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Header Fields**.
Identifier	Enter the frame identifier pattern for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Frame ID** or **Header Fields**.
Cycle Count	Enter the cycle count pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Cycle Count** or **Header Fields**.
Payload Length	Enter the payload length pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Header Fields**.
Header CRC	Enter the header CRC pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Header Fields**.
Data Bytes	Enter the number of data bytes for which to search (one to sixteen bytes). Use the **A** knob to change the value. Only available when **Mark On** = **Data** or **Identifier & Data**.

Table continued…

Field or control	Description
Data	Enter the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data** or **Identifier & Data**.
Byte Offset	Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the **A** knob to change the value. Only available when **Mark On** = **Data** or **Identifier & Data**.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Frame ID**, **Cycle Count**, **Data**, or **Identifier & Data**.
Frame Type	Sets the end of frame type for which to search. Only available when **Mark On** = **End of Frame**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

FlexRay serial bus configuration menu on page 218

I2C serial bus search configuration menu

Use the I2C Search configuration menu to define conditions to search for and mark on an I2C bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.

Table continued…

Field or control	Description
Source	Select the I^2C bus that you want to search.
Mark When	Select the type of information for which to search.
Direction	Sets the transfer direction for which to search. Only available when **Mark When** = **Address** or **Address & Data**.
Addressing Mode	Sets the slave device address length (7 bits or 10 bits long). Only available when **Mark When** = **Address** or **Address & Data**.
Address	Sets the address pattern for which to search. The number of bits shown depends on the **Address Mode** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark When** = **Address** or **Address & Data**.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Use the **A** knob to change the value. Only available when **Mark When** = **Data** or **Address & Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark When** = **Data** or **Address & Data**.
A, B knob controls	Use the **A** knob to select (highlight) the character to change. Use the **B** knob to change the value of the character.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

I2C serial bus configuration menu on page 219

Manchester serial bus search configuration menu

Use the Manchester Search configuration menu to define conditions to search for and mark on an manchester bus decode. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.

Table continued…

Field or control	Description
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the Manchester bus that you want to search.
Mark On	Select the type of information for which to search.
Sync	Sets the synchronization type for which to search. Tap the **Binary** or **Hex** and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Sync Bits**.
Header	Sets the header for which to search. Tap the **Binary** or **Hex** and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Header**.
Data Words	Set the length of the data words for which to search, in the range of integer value Default value is 1, maximum value is 5. Only available when **Mark On** = **Data**.
Data	Sets the data type for which to search. Tap the **Binary** or **Hex** and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data**.
Trailer	Sets the trailer for which to search. Tap the **Binary** or **Hex** and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Trailer**.
Error Type	Sets the error type for which to search. Available types are **Manchester** and **Parity**. Only available when **Mark On** = **Errors**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. Manchester is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. Manchester is currently not supported as a bus trigger source.

Note: In the bus configuration, when the packet view is

- **On**, the displayed Search options are **Sync Bits**, **Header**, **Data**, **Trailer**, and **Errors**.
- **Off**, the displayed Search options are **Data** and **Errors**.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Manchester serial bus configuration menu on page 231

I3C serial bus Search configuration menu

Use the I3C Search configuration menu to define conditions to search for and mark on an I3C bus waveform. You can have multiple searches on the same bus.

To access an I3C serial bus Search menu, double-tap on a I3C Search badge in the Results bar. If there are no I3C Search badges, tap the **Search** button and add an I3C search.

Note: You must have an I3C serial bus waveform in the **Waveform View** before you can create a Search for the bus.

Table 9: I3C search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Source	Select the I3C bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates to show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. I3C is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. I3C is currently not supported as a bus trigger source.

Mark On = Start

This table lists the menu fields displayed when **Mark On** is set to **Start**.

Field or control	Description
Start	Sets the search to mark **Start** events. A **Start** event occurs when the SDA line transitions from a High to Low state while the SCL line remains High. There are no menu fields or controls displayed when this item is selected.

Mark On = Repeated Start

This table lists the menu fields displayed when **Mark On** is set to **Repeated Start**.

Field or control	Description
Repeated Start	Sets the search to mark **Repeated Start** events. A **Repeated Start** is when two or more instances of a **Start** in a row occur without an intervening Stop. There are no menu fields or controls displayed when this item is selected.

Mark On = Address

This table lists the menu fields displayed when **Mark On** is set to **Address**.

Field or control	Description
Direction	Sets the search to mark address events of the specified direction (**Read**, **Write**, or **Either**).
Addressing Mode	Sets the search to mark **7-Bit** or **10-Bit** address events.
Address	Sets the search to mark events that meet the specified address for a Device or register location. The **Addressing Mode** sets the number of bits in the field. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.

Mark On = Data

This table lists the menu fields displayed when **Mark On** is set to **Data**.

Field or control	Description
Data Bytes	Sets the number of data bytes to show in the **Data** field. Valid range is 1-5.
Data	Sets the search to mark events that meet the specified data value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.

Mark On = I3C SDR Direct fields

This table lists the menu fields available in the **I3C SDR Direct** field when **Mark On** is set to **I3C SDR Direct**.

I3C SDR Direct field	Description and menu fields
Enable Slave	Sets the search to mark Enable events (ENEC). Events occur when the Master enables Slave-initiated interrupts on the I3C Bus, such as to indicate a Hot-Join event or request an interrupt. There are no menu fields or controls displayed when this item is selected.
Disable Slave	Sets the search to mark Disable events (DISEC). Events occur when the Master disables Slave-initiated interrupts on the I3C Bus. There are no menu fields or controls displayed when this item is selected.

Table continued…

I3C SDR Direct field	Description and menu fields
Enter Activity State	Sets the search to mark **Enter Activity State** events (ENTAS0, ENTAS1, ENTAS2, ENTAS3). Events occur when the Master sets an activity mode state (0, 1, 2, or 3). There are no menu fields or controls displayed when this item is selected.
Reset Dynamic Address	Sets the search to mark **Reset Dynamic Address** events (RSTDAA). Events occur when a Master device tells a Slave device to forget its current Dynamic address and wait for a new address assignment. There are no menu fields or controls displayed when this item is selected.
Set Max Write Length	Sets the search to mark **Set Max Write Length** events (SETMWL). Events occur when the Master sets a maximum data write length for a Slave device. There are no menu fields or controls displayed when this item is selected.
Set Max Read Length	Sets the search to mark **Set Max Read Length** events (SETMRL). Events occur when the Master sets a maximum data read length for a Slave device. There are no menu fields or controls displayed when this item is selected.
Set Exchange Time	Sets the search to mark **Set Exchange Time** events (SETXTIME). Events occur when Master and Slave devices exchange event timing information to collect or reconstruct timestamps, synchronize controls, and do other related tasks. There are no menu fields or controls displayed when this item is selected.
Set Dynamic Address	Sets the search to mark **Set Dynamic Address** events (SETDASA). Events occur when the Master assigns a dynamic address to one Slave device using the Slave's static address. There are no menu fields or controls displayed when this item is selected.
Set New Dynamic Address	Sets the search to mark **Set New Dynamic Address** events (SETNEWDA). Events occur when the Master assigns a new dynamic address to an I3C Slave device. There are no menu fields or controls displayed when this item is selected.
Get Max Write Length	Sets the search to mark **Get Max Write Length** events (GETMWL). Events occur when the Master gets a maximum data write length in bytes for a slave device. There are no menu fields or controls displayed when this item is selected.
Get Max Read Length	Sets the search to mark **Get Max Read Length** events (GETMRL). Events occur when the Master gets a maximum data read length in bytes for a slave device. There are no menu fields or controls displayed when this item is selected.
Get Provisional ID	Sets the search to mark **Get Provisional ID** events (GETPID). Events occur when the Master sends a Get request to one I3C Slave device to return its 48-bit provisional ID to the Master device. There are no menu fields or controls displayed when this item is selected.

Table continued…

I3C SDR Direct field	Description and menu fields
Get Bus Characteristics	Sets the search to mark **Get Bus Characteristics** events (GETBCR). Events occur when the Master sends a Get request to one I3C Slave device to return its Bus Characteristics Register (BCR) contents. There are no menu fields or controls displayed when this item is selected.
Get Dev Characteristics	Sets the search to mark **Get Dev Characteristics** events (GETDCR). Events occur when the Master sends a Get request to one I3C Slave device to return its Device Characteristics Register (DCR) contents. There are no menu fields or controls displayed when this item is selected.
Get Slave Current Status	Sets the search to mark get device status events (GETSTATUS). When the **Packet** button is set to **Direct**, all events are marked. When the **Packet** button is set to **Response**, the menu displays the following fields to refine the search: **Slave Address** sets the Slave's original 7-bit static I^2C address (or 00 if the slave does not have a static I^2C address) for which to search. **Get Status MSB** sets the most significant bits of the Slave's status register for which to search. These 8 bits contain vendor-specific information. **Get Status LSB** sets the least significant bits of the Slave's status register for which to search. These 8 bits contain pending interrupt, protocol error, and activity mode bits.
Get Accept Mastership	Sets the search to mark **Get Accept Mastership** events (GETACCMST). Events occur when a Master request occurs, and when the Current Master offers Mastership to an I3C Secondary Master. There are no menu fields or controls displayed when this item is selected.
Set Bridge Direct Target	Sets the search to mark **Set Bridge Direct Target** events (SETBRGTGT). A Bridge device is a device on the I3C Bus that allows conversion from the native I3C Bus protocol to another protocol such as UART, SPI, and others). When the **Packet** button is set to **Direct**, all **Set Bridge Direct Target** events are marked. When the **Packet** button is set to **Response**, the menu displays the following fields to refine the search: **Slave Address** sets the Slave's original 7-bit static I2C address, or 00 if the slave does not have a static I2C address. **Dynamic Address** sets the 7-bit slave device address for which to search. **ID** sets the 16-bit unambiguous identifier for the Bridged device, for which to search.

Table continued…

I3C SDR Direct field	Description and menu fields
Get Max Data Speed	Sets the search to mark **Get Max Data Speed** events (GETMXDS) for a specific Slave device. Events occur when the Master asks a Slave for its SDR Mode maximum Read and Write data speeds (& optionally max. Read Turnaround time) When the **Packet** button is set to **Direct**, all **Get Max Data Speed** events are marked. When the **Packet** button is set to **Response**, the menu displays the following fields to refine the search: **Slave Address** sets the 7-bit address of a specific Slave device for which to search. **Max Write** is an 8-bit field that sets the maximum sustained data write speed (Master to Slave) for which to search. **Max Read** is an 8-bit field specifying the maximum sustained data read speed (Slave to Master) and Clock to Data Turnaround Time for which to search. **Max Read Turn** is a 24-bit field specifying the Maximum Read Turnaround Time for which to search.
Get HDR Capability	Sets the search to mark **Get HDR Capability** (GETHDRCAP) events. Events occur when the Master queries a Slave device to determine what HDR Mode(s) that device supports. There are no menu fields or controls displayed when this item is selected.
Get Exchange Time	Sets the search to mark **Get Exchange Time** events (GETXTIME). Use the following fields to refine the search: When the **Packet** button is set to **Direct**, all **Get Exchange Time** events are marked. When the **Packet** button is set to **Response**, the menu displays the following fields to refine the search: **Slave Address** sets the 7-bit address of a specific Slave device for which to search. **Supported Modes Byte** is an 8-bit field that sets which Timing Control Mode(s) to search for on the specified Slave device. **State Byte** is an 8-bit field that sets the current Timing Control Mode (if any) of the Slave device for which to search, and whether any counter overflows have occurred since the most recent previous check. **Frequency Byte** is an 8-bit field that sets the Slave’s internal oscillator frequency for which to search. **Inaccuracy Byte** is an 8-bit field that sets the maximum variation of the Slave’s internal oscillator for which to search.

Mark On = I3C SDR Broadcast fields

The following table lists the fields available when **Mark On** is set to **I3C SDR Broadcast**.

I3C SDR Broadcast field	Description and menu fields
Enable Slave	Sets the search to mark **Enable Slave** events (ENEC) by the Master device. Enable Slave controls when Slave-initiated traffic is enabled (allowed) by the Master on the I3C Bus. This function governs a Slave's attempts to request an Interrupt, to request Mastership, or to signify a Hot-Join event. Use the following field to refine the search: **Event Byte** sets the Slave enable value for which to search.
Disable Slave	Sets the search to mark **Disable Slave** events (DISEC) by the Master device. Disable Slave controls when Slave-initiated traffic is disabled (not allowed) by the Master on the I3C Bus. This function governs a Slave's attempts to request an Interrupt, to request Mastership, or to signify a Hot-Join event. Use the following field to refine the search: **Event Byte** sets the Slave disable value for which to search.
Enter Activity State	Sets the search to mark **Enter Activity State** events (ENTAS0, ENTAS1, ENTAS2, ENTAS3). Events occur when the Master sets an activity mode state (0, 1, 2, or 3). There are no menu fields or controls displayed when this item is selected.
Reset Dynamic Address	Sets the search to mark **Reset Dynamic Address** events (RSTDAA). Events occur when the Master clears/resets one or all Master-assigned Dynamic addresses from Slave devices. There are no menu fields or controls displayed when this item is selected.
Enter Dynamic Address	Sets the search to mark **Enter Dynamic Address Assignment** events (ENTDAA). Events occur when the Master requests Slave devices to enter the Dynamic Address Assignment process. Use the following fields to refine the search: **Slave Address** sets the 7-bit address of a Slave device for which to search. **ID** sets the Slave ID for which to search. **BCR** sets the Bus Control Register values for which to search. **DCR** sets the Device Control Register values for which to search.
Set Max Write Length	Sets the search to mark **Set Max Write Length** events (SETMWL). Events occur when the Master sets a maximum data write length in bytes for a slave device. There are no menu fields or controls displayed when this item is selected.
Set Max Read Length	Sets the search to mark **Set Max Read Length** events (SETMRL). Events occur when the Master sets a maximum data read length in bytes for a slave device. There are no menu fields or controls displayed when this item is selected.

Table continued…

I3C SDR Broadcast field	Description and menu fields
Define List of Slaves	Sets the search to mark **Define List of Slaves** events (DEFSLVS). Events occur when Secondary Masters are told the address of the current Master and any Slaves that are present on the I3C Bus. Use the following fields to refine the search: **Dynamic Address** sets the devices' dynamic 7-bit address for which to search. **DCR Type** sets the Device Control Register values for which to search. **BCR Type** sets the Bus Control Register values for which to search. **Static Address** sets the devices' 7-bit static address for which to search.
Enter Test Mode	Sets the search to mark **Enter Test Mode** events (ENTTM). Events occur when all I3C devices are told that the Master is entering a specified Test Mode during manufacturing or Device test. Use the following field to refine the search: **Test Mode** sets the text mode value for which to search.
Set Exchange Time	Sets the search to mark **Set Exchange Timing Information** events (SETXTIME). Events occur when Master(s) and Slave(s) are directed to exchange event timing information for purposes including synchronizing controls, collecting or reconstructing timestamps, and specifying the timing data procedure. There are no menu fields or controls displayed when this item is selected.
Set Static as Dynamic Address	Sets the search to mark events where the Master assigns a Dynamic address to a Slave with a known Static address. There are no menu fields or controls displayed when this item is selected.

Mark On = I3C DDR

The following table lists the fields available when **Mark On** is set to **I3C DDR**.

I3C DDR field	Description and menu fields
I3C DDR	Sets the search to mark Double Data Rate (DDR) events. Use the following fields to refine the search: **Slave Address** sets the 7-bit address of a Slave device for which to search. **Command Code** sets the read/write values of a command word for which to search. **Data Words** sets the length of the data words for which to search, in the range of integer value 1 (16 bits) to 5 (80 bits). **Data** sets the data value for which to search.

Mark On = Errors fields

The following table lists the fields available when **Mark On** is set to **Errors**.

Errors field	Description
Missing Ack	Sets the search to mark missing ACK error events. There are no menu fields or controls displayed when this item is selected.
T-Bit	Sets the search to mark **T-Bit** (transition bit) error events. T-bits are an alternative to the ACK/NACK mechanism. There are no menu fields or controls displayed when this item is selected.
Parity	Sets the search to mark **Parity** error events. There are no menu fields or controls displayed when this item is selected.
Broadcast Address	Sets the search to mark **Broadcast Address** error events (an error with Broadcast Address commands intended for multiple Slave devices). There are no menu fields or controls displayed when this item is selected.
Preamble	Sets the search to mark **Preamble** error events. The Preamble is the set of bits preceding the data words in HDR-DDR. There are no menu fields or controls displayed when this item is selected.
CRC5	Sets the search to mark **CRC5** (Cyclic Redundancy Check, with fifth-order polynomial length) error events. There are no menu fields or controls displayed when this item is selected.

Mark On = Hot-Join

The following table lists the fields available when **Mark On** is set to **Hot-Join**.

Hot-Join field	Description and menu fields
Hot-Join	Sets the search to mark **Hot-Join** events. Hot-Join events allow Slaves to join the I3C Bus after it is already configured. There are no menu fields or controls displayed when this item is selected.

Mark On = Direct Message End

The following table lists the fields available when **Mark On** is set to **Direct Message End**.

Direct Message End field	Description and menu fields
Direct Message End	Sets the search to mark **Direct Message End** events. These events are end of message notifiers for messages sent to specific Slave devices. There are no menu fields or controls displayed when this item is selected.

Mark On = Stop

The following table lists the fields available when **Mark On** is set to **Stop**.

Stop field	Description and menu fields
Stop	Sets the search to mark **Stop** events. A Stop event is the I3C Bus condition of a Low to High transition on the SDA line while the SCL line remains High. There are no menu fields or controls displayed when this item is selected.

Mark On = HDR Restart

The following table lists the fields available when **Mark On** is set to **HDR Restart**.

HDR Restart field	Description and menu fields
HDR Restart	Sets the search to mark **HDR Restart** events. An event occurs when an **HDR Restart** pattern is sent on the bus. There are no menu fields or controls displayed when this item is selected.

Mark On = HDR Exit

The following table lists the fields available when **Mark On** is set to **HDR Exit**.

HDR Exit field	Description and menu fields
HDR Exit	Sets the search to mark **HDR Exit** events. An event occurs when an **HDR Exit** pattern is sent on the bus. There are no menu fields or controls displayed when this item is selected.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

I3C serial bus configuration menu on page 220

LIN serial bus search configuration menu

Use the LIN Search configuration menu to define conditions to search for and mark on an LIN bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the LIN bus that you want to search.
Mark On	Select the type of information for which to search.

Table continued…

Field or control	Description
Identifier	Sets the identifier pattern for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Identifier** or **Identifier & Data**.
Data Bytes	Sets the number of data bytes for which to search (one to four bytes). Use the **A** knob to change the value. Only available when **Mark On** = **Data** or **Identifier & Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data** or **Identifier & Data**.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a **Data Low** and **Data High** boundary pattern for the specified search range. Only available when **Mark On** = **Data** or **Identifier & Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

LIN serial bus configuration menu on page 221

MIL-STD-1553 serial bus search configuration menu

Use the MIL-STD-1553 Search configuration menu to define conditions to search for and mark on an MIL-STD-1553 bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the MIL-STD-1553 bus that you want to search.

Table continued…

Field or control	Description
Mark On	Select the type of information for which to search.
Transmit/Receive Bit	Sets the transmit or receive bit state for which to search. Only available when **Mark On** = **Command**.
Mark when RT Address	Sets the RT address condition for which to search. When set to **Inside Range** or **Outside Range**, fields are displayed to set a low and high address for the specified search range. Only available when **Mark On** = **Command**.
Parity	Sets the parity state for which to search. Only available when **Mark On** = **Command**.
Address	Sets the address value for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Command**.
Low Address	Sets the low address value for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Command** and **Mark When RT Address** = **Inside Range** or **Outside Range**
Subaddress/Mode	Sets the subaddress or mode value for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Command** and **Mark When RT Address** = **Inside Range** or **Outside Range**
High Address	Sets the high address value for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Command** and **Mark When RT Address** = **Inside Range** or **Outside Range**
Word Count/Mode Count	Sets the word count or mode count value for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Command** and **Mark When RT Address** = **Inside Range** or **Outside Range**

Table continued…

Field or control	Description
Status Word Bits	Sets the status word pattern for which to search. Tap the field and use the **A** and **B** knobs to select and change the values. Selecting a bit shows a short description of that bit's function. Only available when **Mark On** = **Status** and **Mark When RT Address** = **Inside Range** or **Outside Range**
Data	Sets the data pattern for which to search. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Data**.
Error Type	Sets the error condition for which to search. Available when **Mark On** = **Error**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

MIL-STD-1553 serial bus menu on page 222

NRZ serial bus search configuration menu

Use the NRZ Search configuration menu to define conditions to search for and mark on a NRZ bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Source	Select the NRZ bus that you want to search.
Mark On	Select the type of information for which to search.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Use the **A** knob to change the value. Only available when **Mark On** = **Data**.

Table continued…

Field or control	Description
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. NRZ is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. NRZ is currently not supported as a bus trigger source.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

NRZ serial bus configuration menu on page 223

Parallel bus search configuration menu

Use the Parallel Search configuration menu to define conditions to search for and mark on an Parallel bus waveform. You can have multiple searches on the same bus.

Note: Parallel bus search is standard on all instruments.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the parallel bus that you want to search.
Data	Sets the data pattern for which to search. The number of bits shown depends on how the parallel bus is defined. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

Parallel Bus configuration menu on page 224

RS-232 serial bus search configuration menu

Use the RS-232 Search configuration menu to define conditions to search for and mark on an RS-232 bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the RS232 bus that you want to search.
Mark When	Select the type of information for which to search.
Data Bytes	Sets the number of data bytes (1 byte = 8 bits) for which to search (one to ten bytes). Use the **A** knob to change the value. Only available when **Mark When** = **Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Words** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark When** = **Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

RS232 serial bus menu on page 233

SENT serial bus search configuration menu

Use the SENT Search configuration menu to define conditions to search for and mark on an SENT bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the **SENT** bus that you want to search.
Mark On	Select the type of information for which to search.
Channel	Sets the SENT channel type for which to search. Available when **Mark On** = **Start of Packet**.
Status / Communications	Sets the value of the status/communications nibble for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Fast Channel**.
Fast Channel 1	Sets the condition and value of the fast channel 1 data for which to search. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤). The default is =. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Fast Channel**.
Fast Channel 2	Sets the condition and value of the fast channel 2 data for which to search. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤). The default is =. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Fast Channel**.
Counter	Sets the condition and value of the counter data for which to search. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤). The default is =. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Fast Channel**.

Table continued…

Field or control	Description
Inverted Nibble	Sets the value of the inverted nibble data for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Fast Channel**.
Message ID	Sets the value of the message ID data for which to search. Available when **Mark On** = **Slow Channel**.
Data	Sets the condition and value of the slow channel data for which to search. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤). The default is =. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Slow Channel**.
Mark When	Sets the mark when condition. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤, **Inside Range**, **Outside Range**). The default is =. When set to **Inside Range** or **Outside Range**, fields are displayed to set high and low boundary values for the number of clock ticks for which to mark. Available when **Mark On** = **Pause Pulse**.
Number of Ticks	Sets the number of Pause Pulse ticks for which to search. Tap the **Ticks High** or **Tick Low** field and use the **A** and **B** knobs to set the values. Available when **Mark On** = **Pause Pulse** and **Mark When** is set to =, ≠, >, <, ≥, or ≤.
Ticks High, **Ticks Low)**	Sets the highest and lowest values for the range of Pause Pulse ticks for which to search. Tap the **Ticks High** or **Tick Low** field and use the **A** and **B** knobs to set the values. Available when **Mark On** = **Pause Pulse** and **Mark When** = **Inside Range** or **Outside Range**.
Error Type	Sets the error type for which to search. Tap the arrow and select the error condition. Available when **Mark On** = **Error**.
CRC Type	Sets the CRC error type for which to search. Available when **Mark On** = **Error** and **Error Type** = **CRC**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SENT serial bus configuration menu on page 234

SPI serial bus search configuration menu

Use the SPI Search configuration menu to define conditions to search for and mark on an SPI bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the SPI bus that you want to search.
Mark On	Select the type of information for which to search.
Search Source	Set the search source. Set either MOSI or MISO. Only available when **Mark On** = **SS Active**.
Data Words	Sets the number of data words defined based on the word size in bus configuration. If the word size in bus configuration is 4, then data words will go to 32. Maximum data words is word size <=128. Only available when **Mark On** = **Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Words** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SPI serial bus configuration menu on page 237

SpaceWire serial bus search configuration menu

Use the SpaceWire Search configuration menu to define conditions to search for and mark on a SpaceWire bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the SpaceWire bus that you want to search.
Mark On	Select the type of information for which to search.
Control Code Type	Sets the control code type (**Null** or **Time Code**) for which to search and mark on the waveform. Only available when **Mark On** = **Control Code**.
Control Character Type	Sets the control character type (**Flow Control Token, Escape Code, or End Of Packet)**. Only available when **Mark On** = **Control Character**.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Use the **A** knob to change the value. Available when **Mark On** = **Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Data**.
Errors	Sets the error type (**Error End of Packet, Escape, or Parity**) for which to search and mark on the waveform. Available when **Mark On** = **Errors**.
Time Code	Sets the time code pattern for which to search. The number of bits searched is 6. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Control Code** and **Control Code Type** = **Time Code**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. SpaceWire is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. SpaceWire is currently not supported as a bus trigger source.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SpaceWire serial bus configuration menu on page 235

SPMI serial bus search configuration menu

Use the SPMI Search configuration menu to define conditions for which you want to search and mark on an SPMI bus signal. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying marks from this search on the bus waveform.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the **SPMI** bus for which to search.
Mark On	Select the type of information for which to search.
Slave Address	Sets the slave address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Reset**, **Authenticate**, **Register Read**, **Register Write**, **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, **Ext. Register Write Long**, **DD Block Slave Read**, or **Register 0 Write**.
Master Address	Sets the master address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Master Read**, **Master Write**, or **DD Block Master Read**.
Register Address	Sets the register address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Master Read**, **Master Write**, **Register Read**, **Register Write**, **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, or **Ext. Register Write Long**.
Data	Sets the data value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Master Read**, **Master Write**, **Register Read**, **Register Write**, **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, **Ext. Register Write Long**, **DD Block Master Read**, **DD Block Slave Read**, or **Register 0 Write**.

Table continued…

Field or control	Description
Data Bytes	Sets the number of data bytes for which to search. Tap the field and use the **A** knob to change the value. Or double-tap on the field and use the virtual keypad to enter a value. Available when **Mark On** = **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, or **Ext. Register Write Long**.
No Response	Sets to search on data that is all zeros (no response). All values in the Data field are set to zero and cannot be edited. Available when **Mark On** = **Master Read**, **Register Read**, **Extended Register Read**, **Ext. Register Read Long**, **DD Block Master Read**, **DD Block Slave Read**, or **Transfer Bus Ownership**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SPMI serial bus configuration menu on page 238

Configure pattern editor

Use the pattern editor keypad to configure the pattern for the data or custom. To open the pattern editor, double-tap inside a Data/Custom field that requires alphanumeric values.

Data Pattern Editor

Table 10: Data Pattern Editor fields and controls

Keys	Description
Entry field	Sets the Data and hex values (D-XX). D is the data and XX is the hex value which can be 0-9 and A-F.
0-9 and A-F	Sets the Hex value.
CLEAR	Clears all values from the input entry field.
Bksp	Deletes characters to the left of the insert text marker position.
Cancel	Cancels all the values from the input entry field.
Enter	Cancels all the values from the input entry field.

Custom Pattern Editor

Table 11: Custom Pattern Editor fields and controls

Keys	Description
NULL	Enters N in the input box.
FCT	Enters F in the input box.
EOP	Enters EOP in the input box.

Table continued…

Keys	Description
EEP	Enters EE in the input box.
T-XX	Sets the timecode and the hex values (T-XX). T is the Timecode and XX is the hex value which can be 0-9 and A-F.
D-XX	Sets the data and the hex values (T-XX). T is the data and XX is the hex value which can be 0-9 and A-F.
0-9 and A-F	Sets the Hex value.
CLEAR	Clears all values from the input entry field.
Bksp	Cancels all the values from the input entry field.
Cancel	Cancels all the values from the input entry field.
Enter	Cancels all the values from the input entry field.

⚠ **Note:** Max of eight and min of two patters can be added. Each pattern should be separated by a space.

SPMI serial bus search configuration menu

Use the SPMI Search configuration menu to define conditions for which you want to search and mark on an SPMI bus signal. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying marks from this search on the bus waveform.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the **SPMI** bus for which to search.
Mark On	Select the type of information for which to search.
Slave Address	Sets the slave address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Reset**, **Authenticate**, **Register Read**, **Register Write**, **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, **Ext. Register Write Long**, **DD Block Slave Read**, or **Register 0 Write**.
Master Address	Sets the master address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Master Read**, **Master Write**, or **DD Block Master Read**.

Table continued…

Field or control	Description
Register Address	Sets the register address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Master Read**, **Master Write**, **Register Read**, **Register Write**, **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, or **Ext. Register Write Long**.
Data	Sets the data value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Master Read**, **Master Write**, **Register Read**, **Register Write**, **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, **Ext. Register Write Long**, **DD Block Master Read**, **DD Block Slave Read**, or **Register 0 Write**.
Data Bytes	Sets the number of data bytes for which to search. Tap the field and use the **A** knob to change the value. Or double-tap on the field and use the virtual keypad to enter a value. Available when **Mark On** = **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, or **Ext. Register Write Long**.
No Response	Sets to search on data that is all zeros (no response). All values in the Data field are set to zero and cannot be edited. Available when **Mark On** = **Master Read**, **Register Read**, **Extended Register Read**, **Ext. Register Read Long**, **DD Block Master Read**, **DD Block Slave Read**, or **Transfer Bus Ownership**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

SPMI serial bus configuration menu on page 238

USB serial bus search configuration menu

Use the USB Search configuration menu to define conditions to search for and mark on an USB bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See *Act On Event configuration menu* on page 128 for the available fields and controls.

Table continued…

Field or control	Description
Search Type	Set to **Bus**.
Source	Select the USB bus that you want to search.
Mark On	Select the type of information for which to search.
Handshake Type	Sets the handshake packet type for which to search. Only available when **Mark On** = **Handshake Packet**.
Packet Type	Sets the special packet type for which to search. Only available when **Mark On** = **Special Packet**.
Error Type	Sets the error type for which to search. Only available when **Mark On** = **Error**.
Address	Sets the token packet address pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Token Packet** and **Token Type** = all except **SOF (0101)**.
Token Type	Sets the token packet type for which to search. Only available when **Mark On** = **Token Packet**.
Endpoint	Sets the token packet endpoint pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Token Packet** and **Token Type** = all except **SOF (0101)**.
Frame Number	Sets the frame number pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Token Packet** and **Token Type** = **SOF (0101)**.
Data Packet Type	Sets the data packet type for which to search. Only available when **Mark On** = **Data Packet**.
Data Bytes	Sets the number of data bytes for which to search (one to sixteen bytes). Tap the field and use the **A** knob to change the value. Only available when **Mark On** = **Data Packet**.
Byte Offset	Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the **A** knob to change the value. Only available when **Mark On** = **Data Packet**.

Table continued…

Field or control	Description
Data	Sets the data packet pattern for which to search. The number of bits shown depends on the Data Bytes setting. Use in conjunction with the Mark When field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Data Packet**.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** = **Token Packet** or **Data Packet**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

USB serial bus configuration menu on page 239

Edge Search configuration menu

Use the Edge search to mark when the specified edge condition occurs on an analog, digital, math, or reference waveform.

To create a new edge search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Edge**.
3. Select the search **Source.**
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

Edge Search configuration menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.

Table continued…

Field or control	Description
Search Type	Set to **Edge**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Slope	Sets the signal transition direction to detect. (rising, falling, or either direction).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus on page 250

Logic search configuration menu on page 306

Pulse Width Search configuration menu on page 308

Rise/Fall Time Search configuration menu on page 310

Runt Search configuration menu on page 311

Setup and Hold Search configuration menu on page 312

Timeout Search configuration menu on page 314

Window Search configuration menu

DDR Read Search configuration menu on page 316

DDR Write Search configuration menu on page 318

DDR Read & Write Search configuration menu on page 320

Logic search configuration menu

Use the Logic search to mark when specified logic conditions occur on an analog, digital, math, or reference waveform.

To create a new logic search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Logic**.
3. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

Logic Search configuration menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.

Table continued…

Field or control	Description
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Logic**.
Use Clock Edge?	Enables or disables finding logic patterns that occur on the specified clock edge. Yes places marks on the clock waveform wherever the logic pattern occurs. No places marks on the input signal waveform(s) wherever the logic pattern occurs.
Logic Pattern: Define Inputs	Opens the Logic Search-Define Inputs configuration menu where you define the logic state (High, Low, or Don't Care), and the signal threshold level that defines the logic state for each analog or digital signal. See *Define Inputs*.
Mark When	Defines the waveform logic event to mark, when Use Clock Edge is set to No. • **Goes True**: All conditions change to a true state. • **Goes False**: All conditions change to a false state. • **Is True > Limit**: Condition remains true longer than a specified time. • **Is True < Limit**: Condition remains true for less than a specified time. • **Is True = Limit**: Condition remains true for a specified time (within ± 5%). • **Is True ≠ Limit**: Condition does not remain true for a specified time (within ± 5%).
Clock Source	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform.
Clock Edge	Sets the polarity of the clock edge (rising or falling) for evaluating the other menu conditions. The Logic menu also lets you set the clock edge to either edge.
Clock Threshold	Sets the threshold level that the clock signal must pass through to be considered a valid transition. The clock threshold value is independent of the input signal threshold(s).
Define Logic	Sets the logic condition that must occur with all inputs. • **AND**: All conditions are true. • **OR**: Any condition is true. • **NAND**: One or more conditions are true. • **NOR**: No conditions are true.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus on page 250

Edge Search configuration menu on page 305

Pulse Width Search configuration menu on page 308

Rise/Fall Time Search configuration menu on page 310

Runt Search configuration menu on page 311

Setup and Hold Search configuration menu on page 312

Timeout Search configuration menu on page 314

Window Search configuration menu

DDR Read Search configuration menu on page 316

DDR Write Search configuration menu on page 318

DDR Read & Write Search configuration menu on page 320

Logic Search - Define Inputs configuration menu

Use the Define Inputs menu to select the logic condition to search on, and the logic threshold value, for each channel.

To open the Logic Search-Define Inputs configuration menu:

1. Double-tap a **Logic** Search badge on the Settings bar.
2. Tap the **Logic Pattern > Define Inputs** invoker button.

Logic Search - Define Inputs configuration menu fields and controls

Field or control	Description
Ch(x) (analog channels) or D(x) (digital Channels	Use to select the signal sources logic condition on which to perform the logic search (**High**, **Low**, **Don't Care**). If a channel is a digital channel, tap the **+** symbol to open the list of digital inputs (D0-D7) from which to select individual logic conditions for the digital signals. To set the threshold levels for digital channels, double-tap the digital Channel badge to open its configuration menu. Use the **Threshold** field to set the signal level that must be exceeded for that signal to be true (logical 1).
Set All	Sets all signal sources to detect a logic **High**, **Low**, or **Don't Care** condition.

Pulse Width Search configuration menu

Use the Pulse Width search to mark a waveform whenever the specified pulse width condition occurs.

To create a new pulse width search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Pulse Width**.
3. Select the search **Source.**
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

Pulse Width Search menu fields and controls

>

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.

Table continued…

Field or control	Description
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Pulse Width**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	• **< Limit**: A pulse width is less than the specified time limit. • **> Limit**: A pulse width is greater than the specified time limit. • **= Limit**: A pulse width is equal to the specified time limit. • **≠ Limit**: A pulse width does not equal (is greater than or less than) the specified time limit. • **Inside Range**: A pulse width is in the specified time range. • **Outside Range**: A pulse width is outside of the specified time range.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Time Limit	Sets the time period condition to be met.
High Time Limit	Sets the longest acceptable pulse width time period for the range condition. Only available when Mark When = Inside Range or Outside Range.
Low Time Limit	Sets the shortest acceptable pulse width time period for the range condition. Only available when Mark When = Inside Range or Outside Range.
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus on page 250

Edge Search configuration menu on page 305

Logic search configuration menu on page 306

Rise/Fall Time Search configuration menu on page 310

Runt Search configuration menu on page 311

Setup and Hold Search configuration menu on page 312

Timeout Search configuration menu on page 314

Window Search configuration menu

DDR Read Search configuration menu on page 316

DDR Write Search configuration menu on page 318

DDR Read & Write Search configuration menu on page 320

Rise/Fall Time Search configuration menu

Use the Rise/Fall Time search to mark occurrences where the rise or fall time is less than, greater than, equal to, or not equal to a specified time limit.

To create a new rise/fall time search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Rise/Fall Time**.
3. Select the search **Source.**
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

Rise/Fall Time Search configuration menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Rise/Fall Time**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	• **< Limit**: A signal has a rise/fall time less than the specified time limit. • **> Limit**: A signal has a rise/fall time greater than the specified time limit. • **= Limit**: A signal has a rise/fall time that is equal to the specified time limit (±5%). • **≠ Limit**: A signal has a rise/fall time that does not equal (is greater than or less than) the specified time limit (±5%).
Time Limit	Sets the time period condition to be met.
Slope	Sets the signal transition direction to detect. (rising, falling, or either direction).
Upper Threshold	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus on page 250

Edge Search configuration menu on page 305

Logic search configuration menu on page 306

Runt Search configuration menu

Use the Runt search to mark a waveform where a pulse crosses one threshold but fails to cross a second threshold before recrossing the first threshold.

To create a new runt search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Runt**.
3. Select the search **Source.**
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

Runt Search configuration menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Runt**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	• **Occurs**: A runt signal event occurs. • **< Limit**: A runt signal event occurs that has a pulse width less than the specified time limit. • **> Limit**: A runt signal event occurs that has a pulse width greater than the specified time limit. • **= Limit**: A runt signal event occurs that has a pulse width that is equal to the specified time limit (±5%). • **≠ Limit**: A runt signal event occurs that has a pulse width that does not equal (is greater than or less than) the specified time limit (±5%).
Time Limit	Sets the time period condition to be met. Only available when Mark When = < Limit, > Limit, = Limit, or != Limit.

Table continued…

Field or control	Description
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Upper Threshold	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus on page 250

Edge Search configuration menu on page 305

Logic search configuration menu on page 306

Pulse Width Search configuration menu on page 308

Rise/Fall Time Search configuration menu on page 310

Setup and Hold Search configuration menu on page 312

Timeout Search configuration menu on page 314

Window Search configuration menu

DDR Read Search configuration menu on page 316

DDR Write Search configuration menu on page 318

DDR Read & Write Search configuration menu on page 320

Setup and Hold Search configuration menu

Use the Setup and Hold search type to mark a waveform when a data signal changes state inside of a specified setup and hold time, relative to a specified clock signal.

To create a new setup and hold search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Setup & Hold**.
3. Select the search **Clock Source.**
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

Setup & Hold Search configuration menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Setup & Hold**.
Clock Source	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform.
Clock Level	Sets the threshold level that the clock signal must pass through to be considered a valid transition. The clock threshold value is independent of the input signal threshold(s).
Clock Edge	Sets the polarity of the clock edge (rising or falling) for evaluating the other menu conditions. The Logic menu also lets you set the clock edge to either edge.
Data Sources	Sets the data signal source(s). All selected sources must meet the specified setup and hold times. See *Setup and Hold Search - Define Inputs configuration menu* on page 313.
Setup Time	Sets the length of time that data signal should be stable and not change before a clock edge occurs.
Hold Time	Sets the length of time that data signal should be stable and not change after a clock edge occurs.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus on page 250

Edge Search configuration menu on page 305

Logic search configuration menu on page 306

Pulse Width Search configuration menu on page 308

Rise/Fall Time Search configuration menu on page 310

Runt Search configuration menu on page 311

Timeout Search configuration menu on page 314

Window Search configuration menu

DDR Read Search configuration menu on page 316

DDR Write Search configuration menu on page 318

DDR Read & Write Search configuration menu on page 320

Setup and Hold Search - Define Inputs configuration menu

Use the Define Inputs menu to select the data signal source(s) and set their threshold level(s).

To open the Setup & Hold Search - Define Inputs menu:

1. Double-tap a **Setup & Hold** Search badge on the Results bar.
2. Tap the **Data Sources > Define Inputs** button.

Setup and Hold Search - Define Inputs configuration menu fields and controls

Field or control	Description
Ch(x) (analog channels) or D(x) (digital Channels	Use to add (**Include**) or exclude (**Don't Include**) the data signal(s) from available input channels and waveforms. If a channel is a digital channel, tap the **+** symbol to open the list of digital inputs (D0-D7) from which to select for that channel. To set the threshold levels for digital channels, double-tap the digital Channel badge to open its configuration menu. For analog channels, use the threshold field to set the data signal level that must be exceeded for the signal transition to be true.
Set All	Use to **Include** or **Don't Include** all available channels and waveforms as data signals.

Timeout Search configuration menu

Use the Timeout search to mark a waveform when it does not detect an expected pulse transition within a specified period of time, such as when a signal gets stuck either high or low.

To create a new timeout search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Timeout**.
3. Select the search **Source.**
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

Timeout Search menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Timeout**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	• **Stays High**: The signal stays above the specified threshold level longer than the specified time. • **Stays Low**: The signal stays below the specified threshold level longer than the specified time. • **Either**: The signal stays above or below the specified threshold level longer than the specified time.
Threshold	Sets the amplitude level that the signal must pass through to be considered a valid transition.

Table continued…

Field or control	Description
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Time Limit	Sets the time period condition to be met.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus on page 250

Edge Search configuration menu on page 305

Logic search configuration menu on page 306

Pulse Width Search configuration menu on page 308

Rise/Fall Time Search configuration menu on page 310

Runt Search configuration menu on page 311

Setup and Hold Search configuration menu on page 312

Window Search configuration menu

DDR Read Search configuration menu on page 316

DDR Write Search configuration menu on page 318

DDR Read & Write Search configuration menu on page 320

USB serial bus configuration menu

Use the USB bus menu to set up and display an USB 2.0 (Universal Serial Bus) waveform.

To set up a USB serial bus:

- To create a new USB bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **USB**.
- To change the settings on an existing USB serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

USB serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.

Table continued…

Field or control	Description
Bus Type	Set to **USB**.
Speed	Set the speed to match the USB bus you are acquiring. Selecting High Speed (480 Mbps) sets the Signal Type to Differential.
Signal type	Set to match the USB signal you are acquiring (Single Ended or Differential). Use a differential probe to acquire the differential USB signal. Selecting High Speed (480 Mbps) sets the Signal Type to Differential. Both Single Ended and Diff can be used to measure Full (12 Mbps) and Low (1.2 Mbps) speed USB signals.
Source	Select the channel source for the signal from a differential probe. Only available when Signal Type = Diff.
High Threshold	Set the high threshold level for the differential signal.
Low Threshold	Set the low threshold level for the differential signal.
D+ Input	Select the channel source and set the threshold level for the Data+ signal. Only available when Signal Type = Single Ended.
D- Input	Select the channel source and set the threshold level for the Data- signal. Only available when Signal Type = Single Ended.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities (when supported) to the **Trigger** menu. See the *Bus configuration menu* on page 208 for links to all serial bus configuration menus.

See also

Bus Trigger configuration

Bus Search configuration menus on page 250

USB serial bus search configuration menu on page 303

DDR Read Search configuration menu

Use the DDR Read search to search read events on source waveforms.

To create a new DDR Read search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **DDR Read**.
3. Select the search **Sources**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

DDR Read Search configuration menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **DDR Read**.
Source	Lists the source channel or waveform to use to search.
Standard	Sets the DDR standard (**DDR3** or **LPDDR3**).
Data Rate	Sets the data rate options for the selected standard. Data rate values for the **DDR3** standard are 800, 1066, 1333, 1600, 1866, 2133, Auto, and Custom. Default is 800. Data rate values for the **LPDDR** standard are 333, 800, 1066, 1200, 1333, 1466, 1600, 1866, 2133, Auto, and Custom. Default is 333.
Custom Data Rate	Sets the custom date rate. Default is 800 MT/s. Available only when **Data Rate = Custom**.
Burst Detection Method	Sets the burst detection method to use for the measurement. • DQ/DQS Phase Alignment • Chip Select, Latency + DQ/DQS Phase Alignment • Logic State + Burst latency
Postamble Length (tCK)	Sets the postamble length value. Default is 500 mtCK.
Preamble Type	Sets the preamble type (**Static** or **Dynamic**). Default is **Static**.
DQ/DQS Levels	Sets to **Auto** or **Manual**. Default is **Auto**.
Configure	Sets the DQ/DQS level values. See *DQ/DQS Levels Configure menu* on page 322. Only available when **DQ/DQS Levels = Manual**.
Source 3 (CS)	Sets the CS (Chip Select) signal source. Default is Ch 4. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.
CAS Min	Sets the CAS (Column Access Strobe) minimum cycles value. Default value is 2.0 cycles. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.

Table continued…

Field or control	Description
CAS Max	Sets the CAS maximum cycles value. Default value is 3.0 cycles. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**. **Note:** The CAS maximum value should be greater than or equal to the CAS minimum value. If the min value is equal to the max value, the max value automatically increases.
CS Mode	Sets the CS (Chip Select) to Auto or Manual mode. Default is Auto. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.
CS Level	Sets the CS (Chip Select) level value. Default is 500 mV. Available only when **CS Mode = Manual**.
CS Active	Sets the CS signal active true state to Low or High. Default is Low. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.
Logic Pattern	Sets the Logic pattern values. See *DDR Input Configure menu* on page 323. Available only when **Burst Detection Method = Logic State + Burst latency** .
Burst Latency	Sets the burst latency value. Default is 2.5 Cycles. Available only when **Burst Detection Method = Logic State + Burst latency** .
Tolerance	Sets the tolerance value. Default is 1 Cycle.
Burst Length	Sets the burst length. Default is 8 UI.

DDR Write Search configuration menu

Use the DDR Write search to search the write events on source waveforms.

To create a new DDR Write search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **DDR Write**.
3. Select the search **Sources**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

DDR Write Search configuration menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **DDR Read**.
Source	Lists the source channel or waveform to use to search.
Standard	Sets the DDR standard (**DDR3** or **LPDDR3**).
Data Rate	Sets the data rate options for the selected standard. Data rate values for the **DDR3** standard are 800, 1066, 1333, 1600, 1866, 2133, Auto, and Custom. Default is 800. Data rate values for the **LPDDR** standard are 333, 800, 1066, 1200, 1333, 1466, 1600, 1866, 2133, Auto, and Custom. Default is 333.
Custom Data Rate	Sets the custom date rate. Default is 800 MT/s. Available only when **Data Rate = Custom**.
Burst Detection Method	Sets the burst detection method to use for the measurement. • DQ/DQS Phase Alignment • Chip Select, Latency + DQ/DQS Phase Alignment • Logic State + Burst latency
Postamble Length (tCK)	Sets the postamble length value. Default is 500 mtCK.
Preamble Type	Sets the preamble type (**Static** or **Dynamic**). Default is **Static**.
DQ/DQS Levels	Sets to **Auto** or **Manual**. Default is **Auto**.
Configure	Sets the DQ/DQS level values. See *DQ/DQS Levels Configure menu* on page 322. Only available when **DQ/DQS Levels = Manual**.
Source 3 (CS)	Sets the CS (Chip Select) signal source. Default is Ch 4. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.
CAS Min	Sets the CAS (Column Access Strobe) minimum cycles value. Default value is 2.0 cycles. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.

Table continued…

Field or control	Description
CAS Max	Sets the CAS maximum cycles value. Default value is 3.0 cycles. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**. ⚠ **Note:** The CAS maximum value should be greater than or equal to the CAS minimum value. If the min value is equal to the max value, the max value automatically increases.
CS Mode	Sets the CS (Chip Select) to Auto or Manual mode. Default is Auto. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.
CS Level	Sets the CS (Chip Select) level value. Default is 500 mV. Available only when **CS Mode = Manual**.
CS Active	Sets the CS signal active true state to Low or High. Default is Low. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.
Logic Pattern	Sets the Logic pattern values. See *DDR Input Configure menu* on page 323. Available only when **Burst Detection Method = Logic State + Burst latency** .
Burst Latency	Sets the burst latency value. Default is 2.5 Cycles. Available only when **Burst Detection Method = Logic State + Burst latency** .
Tolerance	Sets the tolerance value. Default is 1 Cycle.
Burst Length	Sets the burst length. Default is 8 UI.

DDR Read & Write Search configuration menu

Use the DDR Read & Write search to search both read and write events on source waveforms.

To create a new DDR Read & Write search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **DDR Read & Write**.
3. Select the search **Sources**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

DDR Read & Write Search configuration menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **DDR Read**.
Source	Lists the source channel or waveform to use to search.
Standard	Sets the DDR standard (**DDR3** or **LPDDR3**).
Data Rate	Sets the data rate options for the selected standard. Data rate values for the **DDR3** standard are 800, 1066, 1333, 1600, 1866, 2133, Auto, and Custom. Default is 800. Data rate values for the **LPDDR** standard are 333, 800, 1066, 1200, 1333, 1466, 1600, 1866, 2133, Auto, and Custom. Default is 333.
Custom Data Rate	Sets the custom date rate. Default is 800 MT/s. Available only when **Data Rate = Custom**.
Burst Detection Method	Sets the burst detection method to use for the measurement. • DQ/DQS Phase Alignment • Chip Select, Latency + DQ/DQS Phase Alignment • Logic State + Burst latency
Postamble Length (tCK)	Sets the postamble length value. Default is 500 mtCK.
Preamble Type	Sets the preamble type (**Static** or **Dynamic**). Default is **Static**.
DQ/DQS Levels	Sets to **Auto** or **Manual**. Default is **Auto**.
Configure	Sets the DQ/DQS level values. See *DQ/DQS Levels Configure menu* on page 322. Only available when **DQ/DQS Levels = Manual**.
Source 3 (CS)	Sets the CS (Chip Select) signal source. Default is Ch 4. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.
CAS Min	Sets the CAS (Column Access Strobe) minimum cycles value. Default value is 2.0 cycles. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.

Table continued…

Field or control	Description
CAS Max	Sets the CAS maximum cycles value. Default value is 3.0 cycles. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**. **Note:** The CAS maximum value should be greater than or equal to the CAS minimum value. If the min value is equal to the max value, the max value automatically increases.
CS Mode	Sets the CS (Chip Select) to Auto or Manual mode. Default is Auto. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.
CS Level	Sets the CS (Chip Select) level value. Default is 500 mV. Available only when **CS Mode = Manual**.
CS Active	Sets the CS signal active true state to Low or High. Default is Low. Available only when **Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment**.
Logic Pattern	Sets the Logic pattern values. See *DDR Input Configure menu* on page 323. Available only when **Burst Detection Method = Logic State + Burst latency** .
Burst Latency	Sets the burst latency value. Default is 2.5 Cycles. Available only when **Burst Detection Method = Logic State + Burst latency** .
Tolerance	Sets the tolerance value. Default is 1 Cycle.
Burst Length	Sets the burst length. Default is 8 UI.

DQ/DQS Levels Configure menu

Use the DQ/DQS Levels Configure menu to set strobe, data, hysteresis, and margin values.

DQ/DQS Levels Configure menu

This menu is accessed from the **Configure** button in a **DDR Read**, **DDR Write**, or **DDR Read & Write** search menu, when **DQ/DQS Levels** is set to **Manual**.

Field or control	Description
High	Sets the high value for Strobe and Data. Default value is 0.
Mid	Sets the mid value for Strobe and Data. Default value is 0.
Low	Sets the low value for Strobe and Data. Default value is 0.
Hysteresis	Sets the hysteresis. Default value is 10%.

Table continued…

Field or control	Description
Margin	Sets the margin. Default value is 20%.

DDR Input Configure menu

Use the Input configure menu to set the logic pattern input values for CS, RAS, CAS, and WE values.

Logic Pattern Levels Input Configure menu

This menu is accessed from the **Define Inputs** button in **DDR Read**, **DDR Write**, or **DDR Read & Write** search menu, when **Logic State + Burst Latency** is selected as the **Burst Detection method**.

Field or control	Description
CS	Sets the chip select source, level, and the threshold value.
RAS	Sets the row access strobe source, level, and the threshold value.
CAS	Sets the column access strobe source, level, and the threshold value.
WE	Sets the write enable source, level, and the threshold value.

Cursor configuration menu

Cursors are on-screen lines (bars) that you position in a Waveform or Plot views to take manual measurements on signals. They appear as horizontal and/or as vertical lines.

To open the Cursors configuration menu:

1. Double-tap a cursor readout or cursor line, or
2. Touch and hold a cursor readout or cursor line and select **Configure Cursors** from the right-click menu.

To display cursors on the screen:

1. Tap the **Add New... Cursors** button of the display, or
2. Push the **Cursors** front-panel button to toggle cursors on or off.

Cursor configuration menu fields and controls

Some fields or controls are only available when certain other controls are selected.

Field or control	Description
Display	Tap to toggle the cursor display **On** or **Off**.
Readout Location	Tap to set where to display cursor readouts. **Graticule** displays the cursor readouts on the screen cursor bars (default method). You can move the readouts on cursors by touching and dragging them along the cursor bar. **Badge** removes the screen cursor readouts and displays the cursor information in a **Cursors** badge in the Results bar. The **Cursors** badge content changes as you change the cursor type.

Table continued…

Field or control	Description
Cursor Type	Tap to select the cursor type from the drop-down list. **Waveform** cursors measure vertical amplitude and horizontal time parameters simultaneously at the point the cursor intersects a waveform. The cursor intersect point tracks waveform amplitude changes. **V Bars** are vertical cursors that measure horizontal parameters (typically time). They are not associated with the waveform, but show the time position of the cursor in the waveform record. **H Bars** are horizontal cursors that measure amplitude (typically in volts or amperes). They are not associated with the waveform, but show the amplitude position of the cursor. **V&H Bars** cursors measure vertical and horizontal parameters simultaneously. They are not associated with the waveform, but show the time and amplitude position of the cursors.
Source Waveform	Tap to select the source waveform (the waveform on which to display the cursors). **Same** places both cursors on the same waveform. **Split** allows each cursor to be on a different waveform. Only available when **Cursor Type** = **Waveform**.
Readout Type	Tap to select the type of readout to display for a XY or XYZ plot. This control is only available when the Cursor Type is set to Waveform or V&H Bars. **Rectangular** displays the position values and the difference (delta) between cursors. **Polar** displays the polar information for each cursor. The top value is the magnitude of the vector drawn from the origin of the plot to the location of the cursor. The bottom value is the angle of the vector drawn from the origin of the plot to the location of the cursor.
Source	Tap to select the source waveform from the drop-down list. **Selected Waveform** automatically moves the waveform cursors to the selected source. Or select a specific source from the drop-down list. Only available when **Source Waveform** = **Same**.
Cursor A Source, Cursor B Source	Tap to select the waveform sources for Cursor A and B. Only available when **Source Waveform** = **Split**.
Cursor A X-Position	Tap to set a specific x-axis position for Cursor A using the multipurpose knob, or double-tap to set the position using the keypad.
Cursor B X-Position	Tap to set a specific x-axis position for Cursor B using the multipurpose knob, or double-tap to set the X-Position using the keypad.
Cursor A Y-Position	Tap to set a specific Y-axis position for Cursor A using the multipurpose knob, or double-tap to set the position using the keypad. Only available when **Cursor Type** = **H Bars** or **V&H Bars**.
Cursor B Y-Position	Tap to set a specific Y-axis position for Cursor B using the multipurpose knob, or double-tap to set the X-Position using the keypad. Only available when **Cursor Type** = **H Bars** or **V&H Bars**.

Table continued…

Field or control	Description
Cursor Mode	Tap to select the cursor mode. **Independent** mode sets multipurpose knobs A and B to move each cursor separately. **Linked** mode sets multipurpose knob A to move both cursors at the same time. Knob B will still move cursor B independently of knob A.

Date and Time configuration menu

Use this menu to set the date, time format, and time zone.

To open the Date and Time configuration menu, double-tap on the Date/Time badge in the lower-right corner of the oscilloscope display.

Date and Time configuration menu fields and controls

Field or control	Description
Display	Tap to toggle display of the date and time On or Off. When turned off, double-tap on the blank area below the Run/Stop button, in the lower-right corner of the oscilloscope display, to open the configuration menu and set display to On.
Time Format	Tap to select either 12 Hour or 24 Hour time format.
Time Zone	Tap and select the desired time zone from the drop-down list.
Automatically adjust clock for Daylight Saving Time	Sets the oscilloscope to automatically change the time to account for Daylight Saving Time changes.

Note: You cannot set a specific time; the time is preset at the factory.

More (Draw A Box Menu)

Use this menu to toggle the Waveform view box draw mode between Zoom area mode, Visual Trigger area mode, and Mask segment mode.

To use the **More (Draw a Box)** menu:

1. Double-tap the **More** button (located at the up right of the Results Bar) and it opens the **Draw a Box** window.
2. Tap **Zoom** on the draw a box window to enable drawing a zoom box area on the screen. Draw a Box mode stays in Zoom mode until changed.
3. Tap **Visual Trigger** on the draw a box window to enable drawing Visual Trigger areas on the screen. Draw a Box mode stays in **Visual Trigger** mode until changed.
4. Tap **Mask** on the draw a box window to enable drawing Mask Testing segments on the screen. Draw a Box mode stays in **Mask** mode until changed.
5. Tap outside the menu. The **Draw a Box icon** changes to reflect the function available when you next select the **Draw a Box** icon.

See also

Turn on Zoom mode

Visual Trigger concepts

Create Visual Trigger areas

Edit visual trigger areas on the screen

Edit visual trigger areas using the Area menu

Mask testing waveforms

Horizontal setting panel

Use this menu to select the horizontal mode, set horizontal parameters, and enable trigger delay.

To open the Horizontal configuration menu, double-tap the Horizontal badge in the Settings bar.

⚠ **Note:** TekScope Remote Control is off, there is no Horizontal configuration menu.

Field or control	Description
Horizontal Mode	Tap to select either Automatic or Manual horizontal mode. In Automatic mode you can set the Minimum Sample Rate and Horizontal Scale. In Manual mode you can set the Sample Rate and Record Length. Horizontal Scale is a dependent variable calculated from the sample rate and record length.
Minimum Sample Rate	Sets the minimum sample rate for acquisitions. Changing horizontal scale will not reduce the sample rate below this value. This can result in partial waveform records at lower frequencies, but at the same time would provide more sample points for a few cycles of the signal. Available when Horizontal Mode = Automatic = Off. This setting can be overridden if Allow Horizontal Scale to Override Min Sample Rate is selected.
Allow Horizontal Scale to Override Min Sample Rate	Select to set the oscilloscope to automatically change the sample rate to acquire a full waveform record. Changes to the sample rate are shown in the Horizontal badge readout, not in the Minimum Sample rate field of the menu. Only available when Horizontal Mode = Automatic= Off.
Horizontal Scale	Tap to set the Horizontal Scale using the assigned multipurpose knob, double-tap to set the scale using the virtual keypad, or tap the up and down arrows. You can also use the front-panel Horizontal Scale knob to change this value. The horizontal scale determines the size of the acquisition window relative to the waveform. You can scale the window to contain a single waveform edge, a single cycle, several cycles, or thousands of cycles.
Delay	Delay positions the trigger event to a specified time relative to the center of the waveform record. Use delay to focus on events that occur before (pretrigger) or after the trigger point (posttrigger).

Table continued…

Field or control	Description
Position	Tap to set the trigger Position using the assigned multipurpose knob or double-tap to set the Position using the virtual keypad. When horizontal Delay is on, the time from the trigger point to the horizontal reference (center of waveform record) is the horizontal delay. The horizontal position determines the number of pretrigger and posttrigger samples in the waveform record. When horizontal delay is off, the trigger point and the horizontal reference are at the same time in the middle of the waveform record.
Set to 10%	Tap to set the trigger delay to 10% of the waveform record. Only available when Delay = Off.

Mask Badge configuration menu

Use the Mask Badge configuration menu to set the total number of eye diagram mask hits needed to fail the mask test.

Prerequisite: an Eye Diagram plot with a mask enabled. See *Add Mask Testing to an Eye Diagram plot* on page 94.

To open the Mask Badge configuration menu, double-tap a **Mask** badge in the Results column.

Mask Badge configuration menu fields and controls

Field or control	Description
Total	Sets the total number of hits needed to fail an eye mask test.

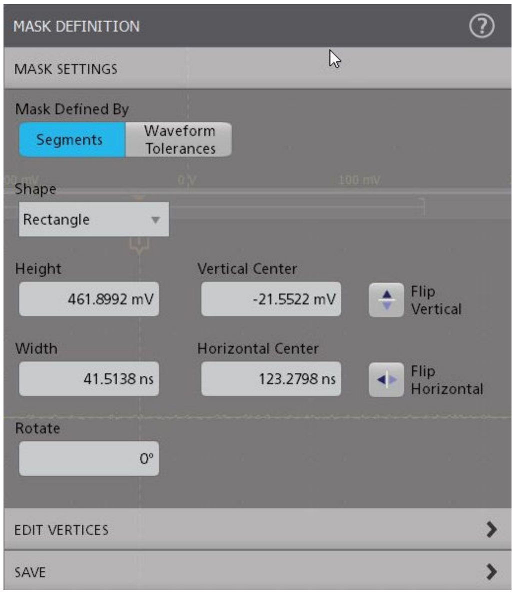
Mask Definition configuration menu

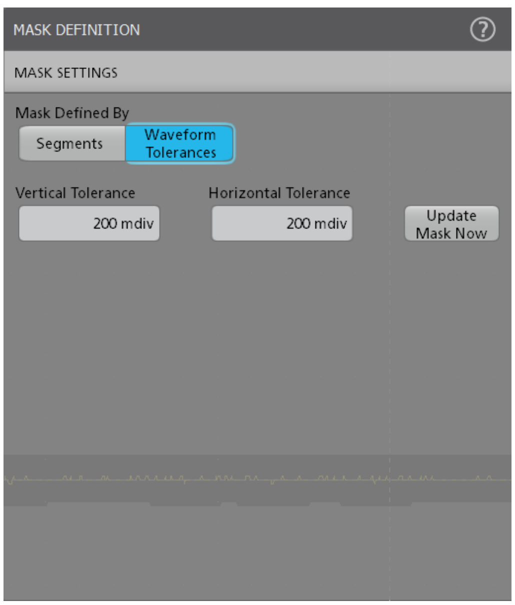
Use the Mask Definition menu to edit mask segment parameters. Double tapping a mask segment opens the Mask Definition configuration menu.

To open the **Mask Definition** menu, double-tap on a mask segment.

To create a mask segment, see *Create a Mask*

Mask Settings panel fields and controls



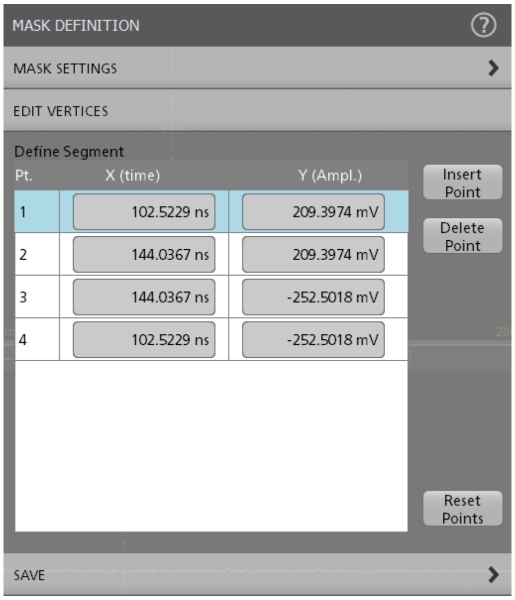


Field or control	Description
Mask Defined By	Define the way you edit the mask. The default option is **Segments**.
Shape	Lists the current shape type, and also lets you change the current shape to a specified shape. Changing a shape defines the minimum rectangle that includes all vertices of the current segment, and then does a best-fit approach to create the specified shape. If you made changes to a default shaped segment that results in an segment that no longer meets the default shape definition, that segment is listed as a Custom shape in the menu.

Table continued…

Field or control	Description
Height	Sets the segment height, in amplitude units, between the topmost vertex and the bottommost vertex.
Vertical Center	Sets the segment vertical center, in amplitude units, as the point halfway between the topmost vertex and bottommost vertex.
Flip Vertical	Flips the segment vertically around its Vertical Center value.
Width	Sets the segment width, in time units, between the leftmost vertex and the rightmost vertex.
Horizontal Center	Sets the segment horizontal center, as time units, as the point halfway between the leftmost vertex and rightmost vertex.
Flip Horizontal	Flips the segment horizontally around its Horizontal Center value.
Rotate	Rotates the segment in units of degrees, from 0° to 360°. The rotation is an absolute angle measurement referenced from 0°, where 0° is the position of the segment when it was first created. For example, if you rotate the segment 40°, and then rotate it again with 20°, the resulting segment rotation is 20°. The segment height and width are relative to the current segment orientation, and automatically change places as needed when the segment is rotated.
Vertical Tolerance	Tap the Vertical Tolerance field and enter the tolerance value as divisions of the current channel settings (volts, amps, and so on). Or double-tap on the field and use the A knob to change the value.
Horizontal Tolerance	Tap the Horizontal Tolerance field and enter the tolerance value as divisions of the current channel settings (volts, amps, and so on). Or double-tap on the field and use the A knob to change the value.
Update Mask Now	Updates the mask as per the configured tolerance values.

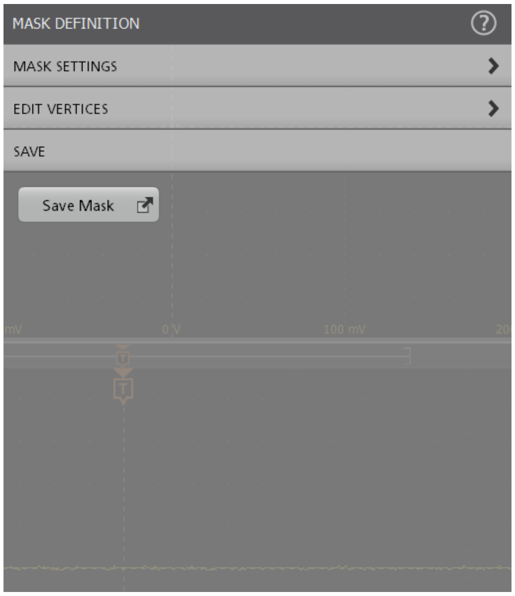
Edit Vertices panel fields and controls



Field or control	Description
Define Segment	A table that lists the X (time) and Y (amplitude) values for each vertex (point) of the segment. Use the scroll bar to show points if there are more than nine points in the segment. Selecting a row in the table highlights the associated vertex on the segment. Use the Multipurpose knobs to change the value of the X or Y settings, or double-tap a setting and enter the value directly.
Insert Point	Inserts a new row above the selected row and creates a new vertex on the segment shape. The new vertex is halfway between the vertices defined in the prior row and the following row in the table.
Delete Point	Deletes the currently selected point, keeps the row selected, and moves all rows below it up one row. The **Delete Point** button is not available when a triangular segment is selected.
Reset Points	Deletes all but three data points from the table. The remaining three data points are set to a default triangle, set to two divisions for height and width, and centered in the waveform area.

Note: Redo and Undo are available for most Edit Vertices panel controls.

Save panel fields and controls



Field or control	Description
Save Mask	Saves the mask in the desired location with the given file name.

Mask Definition right click menu

See *Right click menu functions associated with mask segments* on page 331.

Right click menu functions associated with mask segments

The following mask definition functions are available when you right click on a mask segment.

Mask Segments controls

Field or control	Description
Triangle	Sets the shape of the mask to a triangle.
Rectangle	Sets the shape of the mask to a rectangle
Trapezoid	Sets the shape of the mask to a trapezoid.
Hexagon	Sets the shape of the mask to a hexagon.
Custom	Sets the shape of your choice.
Create Duplicate	Creates a new segment with the same characteristics.
Configure Segment	Opens the Mask Definition configuration menu for the selected segment.
Delete Segment	Deletes the selected segment.
Delete All	Deletes all segments associated with the mask.

Tolerance Mask controls

Field or control	Description
Configure Mask	Opens the Mask Definition configuration menu for the selected mask.
Delete Mask	Deletes the selected mask.

Mask Test badge configuration menu

Use the Mask Test badge menu to edit the settings of a mask test and define the actions to be taken according to the results. Double tapping the badge opens the Mask Test badge configuration menu.

To open the **Mask Test** badge menu, double-tap on the badge.

A Mask Test badge is created when the first segment of a mask is defined.

Test Settings panel fields and controls

Field or control	Description
Mask Test	Turn the mask test On or Off.
Mask Display	Turn the mask display On (default) or Off.
Label	The text field to add a label to the mask test badge. By default, this field is blank. When you recall a mask from a file, the name of the file is automatically displayed in the label field.
Source	Lists the valid source signals for mask testing. Valid sources do not include other sources that are already being used in mask tests.
Show segment hits in badge	A checkbox to display the hits on each segment in the badge. By default, the setting is unchecked.
Number of Waveforms	The numeric field to define the number of waveforms to test against. This value is shared amongst all mask tests. Changing the value in one badge changes it in all mask test badges. This field can be used along with the **Single / Seq Stops After** control to run a mask test with 100 waveforms ten times.
Failure Threshold	The numeric field to set the failure threshold in number of acquisitions.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a Mask event occurs (Pass, Fail, or Hits). See *Act On Event configuration menu* on page 128 for the available fields and controls.

Mask Test badge right click menu

See *Right click menu functions associated with the Mask Test badge* on page 332.

Right click menu functions associated with the Mask Test badge

The following functions are available when you right click on a mask test badge.

Field or control	Description
Configure Mask Test	Opens the Mask Test badge configuration menu for the selected badge.
Delete Mask Test	Deletes the selected mask test badge.

Plot configuration menus

Use plot configuration menus to change settings of a displayed plot. Double-tap a plot view to open its configuration menu.

Use the following links to access information on a specific Plot configuration menu.

Eye Diagram plot configuration menu

Use this menu to change settings of a displayed Eye Diagram plot.

To open the Eye Diagram plot menu, double-tap anywhere in the **Eye Diagram** plot view.

To display an eye diagram plot, see *Add Plot configuration menu* on page 241.

Settings panel (Eye Diagram plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale** on or off. Turn AutoScale off to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is **Off**, a small Zoom window appears in the plot. Drag the blue vertical bar in the **Zoom** window to view that area in the main Plot view. Use the **Zoom** and **Pan** front-panel knobs to change the zoom area and position.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. Only available when **AutoScale** is **Off** (unchecked).
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. Only available when **AutoScale** is **Off** (unchecked).
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Eye Rendering	**Fast** Eye Rendering shows the **UIs** (Unit Intervals) that define the boundaries of the eye along with a user specified number of surrounding UIs for added visual context. **Complete** Eye Rendering shows all valid **UIs** (Unit Intervals).

Mask panel (Eye Diagram plot configuration menu) fields and controls

Field or control	Description
Mask Test	Toggles mask display on or off. Setting **Mask Test** to **Off** clears and stops mask test calculations.

Table continued…

Field or control	Description
Autofit Mask to Minimize Hits	**Autofit Mask** adjusts the horizontal value of the central mask segment to minimize the mask hits in the central segment. This adjustment is then applied to all 3 segments. This control is disabled if the mask does not contain three segments. **Autofit Mask** assumes that segment 2 (Seg2) is the central mask.
Mask File	Lists the path and file name of the mask file to load. You can directly enter the path of the mask file, or use the **Browse** button to navigate to a mask file. The field also has a drop-down list of recently selected files from which you can also select a mask file to load. The supported mask file types are: • Legacy base firmware mask files (.msk). Legacy base firmware mask files contain commands that define mask segment geometry and other commands to configure the instrument controls. Only the commands that define the segment geometry are processed. • DPOJET mask files (.msk). DPOJET mask files use the same format as legacy base firmware mask files but only contain commands that define the segment geometry. • 5/6 Series MSO mask files (.xml).
Browse	Opens the **Recall Mask File** menu that lets you navigate to and select a mask file to load. See *Recall Mask File configuration menu* on page 337.
Bit Type	Sets which waveform bit types (**Transition**, **Non-Transition**, or**All**) to include for eye height analysis. **All** does eye analysis using both transition and non-transition bits. **Transition** does eye analysis only on transition bits. A transition bit is a bit that is changing from low to high or high to low. **Non-Transition** does eye analysis only on nontransition bits. A nontransition bit is a bit that is not changing state.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the Save As menu. Navigate to the location where you want to save the plot image. Enter a file name in the File Name field. Select the image file format (PNG, BMP, or JPG). Tap OK to save the plot image.
Save Plot Data	Tap to open the Save As menu. Navigate to the location where you want to save the plot data. Enter a file name in the File Name field. Tap OK to save the plot data as a comma-separated values (.csv) file.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

Math FFT plot configuration menu (Math waveform)

Use the Math FFT plot menu to change settings of a displayed FFT plot, including source, FFT window type, plot type, and gating.

To open the FFT plot configuration menu, double-tap anywhere in the **Math FFT** Plot view.

Plot Settings panel (Math FFT plot configuration menu) fields and controls

Field or control	Description
Auto Scale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot (Horizontal, Vertical, or Both).
X-Axis Scale	Sets the horizontal frequency scale to either Log or Linear. A **Log** scale is useful when the frequency component magnitudes cover a wide dynamic range, letting you show both lesser and greater- magnitude frequency components on the same display. A **Linear** scale is useful when the frequency component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Y-Axis Scale	Sets the vertical amplitude scale to either Log (dBm) or Linear. A **Log dB** scale is useful when the frequency component magnitudes cover a wide dynamic range, letting you show both lesser and greater magnitude frequency components on the same display. A **Linear** scale is useful when the frequency component magnitudes are all close in value, enabling direct comparison of their magnitudes.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scale. These values also define the horizontal zoom area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scale. These values also define the vertical zoom area shown in the small Zoom window. Available when AutoScale is off.

FFT Settings panel (Math FFT plot configuration menu) fields and controls

Field or control	Description
FFT Type	**Magnitude** plots the magnitude values of the frequency components. **Phase** plots the phase of the signal as a function of frequency.
Window	Sets the FFT window type to use for the waveform plot. See *FFT windows* on page 336.
Gating	Sets the region of the waveform to analyze for the FFT plot. **None** uses the entire waveform record to create the FFT plot. **Screen** uses the part of the waveform record displayed on the screen (such as in Zoom mode) to create the FFT plot. **Cursors** uses the waveform data between the cursors to create the FFT plot.
Vertical Units	Sets the vertical scale to Degrees, Radians, or Group Delay. **Note:** The Vertical Units set when FFT Type = Phase override the Y-Axis scale setting in the Plot Settings panel. Only available when FTT Type set to Phase.
Phase Wrap	When checked, the phase trace is unwrapped where the trace jumps more than the number of degrees set in the adjacent field. Only available when FTT Type set to Phase.
Squelch	When checked, the phase trace excludes points that have a voltage smaller than the specified voltage. The squelch voltage should be set to the expected noise voltage level. Minimum value is 100 mV, and increments in units of 100 mV. When unchecked, the phase trace includes all values. Only available when FTT Type set to Phase.

Save panel (Math FFT plot configuration menu) fields and controls

Field or control	Description
Save Plot Image	Opens the *Save As* configuration menu to specify the location and name at which to save an image file of the Plot view.
Save Plot Data	Opens the *Save As* configuration menu to specify the location and name at which to save the Plot view data to a CSV file.

FFT windows

Each FFT window is a trade-off between frequency resolution and magnitude accuracy. What you want to measure and your source signal characteristics help determine which window to use. Use the following guidelines to select the best window.

FFT window	Characteristics	Best for measuring
Hanning	Better frequency, poorer magnitude accuracy than Rectangular. Hanning has slightly poorer frequency resolution than Hamming.	Sine, periodic, and narrow-band random noise. Transients or bursts where the signal levels before and after the event are significantly different.
Rectangular	Best frequency, worst magnitude resolution. This is essentially the same as no window.	Transients or bursts where the signal levels before and after the event are nearly equal. Equal-amplitude sine waves with frequencies that are very close.
Hamming	Better frequency, poorer magnitude accuracy than Rectangular. Hamming has slightly better frequency resolution than Hanning.	Sine, periodic, and narrow-band random noise. Transients or bursts where the signal levels before and after the event are significantly different.
Blackman-Harris	Best magnitude, worst frequency resolution.	Predominantly single frequency signals to look for higher order harmonics.
Kaiser-Bessel	Less spectral leakage than the Hanning, Hamming, or Rectangular windows.	Predominantly single frequency signals to look for higher order harmonics.
Gaussian	The time-domain shape of an exponential Gaussian function transforms into a Gaussian exponential shape in the frequency domain.	Optimal localization in both the time and frequency domain.
Flattop2	Wider resolution bandwidth but lower side lobe attenuation. Also, it is unique because the time domain shape has negative values.	Useful for high accuracy magnitude measurements for signals that do not require very narrow bandwidth.
TekExp	The Tek Exponential window was invented at Tektronix. In the time domain, it is not asymmetrical bell shape as is the case with the other windows. Instead, it is exponential with a peak at the 20% position of the time domain gate. The frequency domain shape is triangular. More of the acquired data record length is used to capture the impulse response.	Use this window for impulse-response testing where the 20% position is the zero phase reference point.

You can also determine the best window empirically by first selecting the Rectangular window, and then selecting (in the following order) the Hamming, Hanning, and Blackman-Harris windows until the frequency components merge. Use the window just prior to where the frequencies emerge for the best compromise between resolution and amplitude accuracy.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

Recall Mask File configuration menu

Use this menu to recall (load) an eye diagram mask definition file to apply to an eye diagram plot.

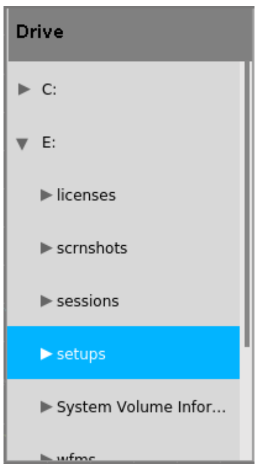
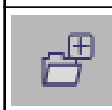
To open the **Recall Mask File** configuration menu:

1. Double-tap anywhere in the **Eye Diagram** plot view to open its configuration menu.
2. Tap the **Mask** panel.

3. Tap **Browse.**

To display an eye diagram plot, see *Add Plot configuration menu* on page 241.

Browse Mask File Location configuration menu fields and controls

Field or control	Description
Look in:	Shows the current directory path to the location of a file. Tap on the file path and use a keyboard to enter the location location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recalled mask file locations, up to a maximum of 20 locations.
	The **Drive** column lists the accessible directories, opening at the root (/) level. Use to quickly navigate to a location. Tap a name in the **Drive** list to show the contents of the directory in the Name pane. Double-tap an item to display the directory and any subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
	Navigation buttons. Use the arrow buttons to navigate the file directory. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder. The Up arrow navigates up one level from the current folder.
	Create new Directory icon. Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File name	Shows the selected file path and name.
Files of type	Use to select the file format you want to recall. The drop-down list shows all file extension types that the instrument can read for the selected file type.
Cancel	Cancels the file open action and closes the menu.
Open	Opens (loads) the selected file. Recalling a mask file adds a Mask badge to the Settings bar and displays the masks in the eye diagram plot.

Histogram plot configuration menu

Use this menu to change settings of a displayed Histogram plot.

To open the Histogram plot configuration menu, double-tap anywhere in a **Histogram** Plot view.

Settings panel (Histogram plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles AutoScale on or off. Turn AutoScale off to set the X and Y-axis range to view an area of interest. When AutoScale is off, the Plot view zoom mode enables and a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Selects which gridlines to show in the plot.
X-Axis Number of Bins	Sets the resolution by the number of bins into which the X axis is divided.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis Scale	Sets the Y axis scale to Linear or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the Save As menu. Navigate to the location where you want to save the plot image. Enter a file name in the File Name field. Select the image file format (PNG, BMP, or JPG). Tap OK to save the plot image.
Save Plot Data	Tap to open the Save As menu. Navigate to the location where you want to save the plot data. Enter a file name in the File Name field. Tap OK to save the plot data as a comma-separated values (.csv) file.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

Save As configuration menu (plot Save panel, Save Plot Data button)

Use this menu to specify the name and location at which to save a comma separated value (csv) file of the data for a specific plot.

To access the Save As configuration menu for plots:

1. Double-tap anywhere in a Plot view top open the plot configuration menu.
2. Tap the **Save** panel.
3. Tap the **Save Plot Data** button.

Save As configuration menu (plot Save panel, Save Plot Data button) fields and controls

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Opens the *Browse Save As Location* configuration menu to navigate to and select a location at which to save the file.
File name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Cancel	Cancels the file save action and closes the configuration menu.
OK	Saves the file to the specified location, closes the Save As menu, and displays a confirmation message.

Save As configuration menu (plot Save panel, Save Plot Image button)

Use this menu to specify the name and location at which to save an image file for the selected plot.

To access the **Save As** configuration menu to save an image of a plot to a file:

1. Double-tap anywhere in a Plot view top open the plot configuration menu.
2. Tap the **Save** panel.
3. Tap the **Save Plot Image** button.

Save As configuration menu (plot Save panel, Save Plot Image button) fields and controls

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Opens the *Browse Save As Location* configuration menu to navigate to and select a location at which to save the file.
File name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save As Type	Lists the available graphic formats to which you can save files. Tap the field and select the graphic save format.
Cancel	Cancels the file save action and closes the configuration menu.
OK	Saves the file to the specified location, closes the Save As menu, and displays a confirmation message.

Spectrum plot configuration menu

Use this menu to change settings of a displayed Spectrum plot.

To open the Spectrum plot configuration menu, double-tap anywhere in the **Spectrum** plot view.

Settings panel (Spectrum plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles Autoscale on or off. Turn AutoScale off to set the X and Y-axis range to view an area of interest. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Use the grid style best suited to your measurement needs.
X-Axis Scale	Sets the X axis scale to Linear or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

Table continued…

Field or control	Description
Y-Axis Scale	Sets the Y axis scale to Linear or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Dynamic Range	Sets the vertical scale dynamic range. Available when Y-Axis Scale is set to Log.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the Save As menu. Navigate to the location where you want to save the plot image. Enter a file name in the File Name field. Select the image file format (PNG, BMP, or JPG). Tap OK to save the plot image.
Save Plot Data	Tap to open the Save As menu. Navigate to the location where you want to save the plot data. Enter a file name in the File Name field. Tap OK to save the plot data as a comma-separated values (.csv) file.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

Plot XY configuration menu

Use this menu to change settings of a displayed XY plot.

To open the XY plot menu, double-tap anywhere in the **XY** Plot view.

Settings panel (Plot XY configuration menu) fields and controls

Field or control	Description
X-Axis	Sets the source for the X-axis signal.
Y-Axis	Sets the source for the Y-axis signal.

- Use the vertical scale and position controls of the waveform connected to the X- and Y-axis sources to set horizontal scale and position of the XY waveform.
- XY format is particularly useful for studying phase relationships between two similar signals, creating Lissajous Patterns.
- XY format is a dot-only display, although it can have persistence. The Vector style selection has no effect when you select XY format.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the Save As menu. Navigate to the location where you want to save the plot image. Enter a file name in the File Name field. Select the image file format (PNG, BMP, or JPG). Tap OK to save the plot image.
Save Plot Data	Tap to open the Save As menu. Navigate to the location where you want to save the plot data. Enter a file name in the File Name field. Tap OK to save the plot data as a comma-separated values (.csv) file.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

XYZ plot configuration menu

Use this menu to change settings of a displayed XYZ plot.

To open the XYZ plot menu, double-tap anywhere in the **XYZ** plot view.

Settings panel (Plot XYZ configuration menu) fields and controls

Field or control	Description
X-Axis	Sets the source for the X-axis signal.
Y-Axis	Sets the source for the Y-axis signal.
Z-Axis	Sets the source for the Z-axis signal.

- XYZ format compares the voltage levels of the X and Y channel waveform records point-by-point, as in XY format. The displayed waveform intensity is modulated by the Z channel waveform amplitude.

- XYZ format is a dot-only display, although it can have persistence. The Vector style selection has no effect when you select XYZ format.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the Save As menu. Navigate to the location where you want to save the plot image. Enter a file name in the File Name field. Select the image file format (PNG, BMP, or JPG). Tap OK to save the plot image.
Save Plot Data	Tap to open the Save As menu. Navigate to the location where you want to save the plot data. Enter a file name in the File Name field. Tap OK to save the plot data as a comma-separated values (.csv) file.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

SOA plot configuration menu

Use this menu to change settings of a displayed SOA (XY) plot.

To open the SOA (XY) plot menu, double-tap anywhere in the **SOA** Plot view to open the configuration.

Settings panel (SOA plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** off to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
X-Axis (Voltage) Scale	To change the scale from linear to log or vice versa, click on the button (linear or log). This scale applies on the plot data as well as on the mask data.

Table continued…

Field or control	Description
Y-Axis (Current) Scale	To change the scale from linear to log or vice versa, click on the button (linear or log). This scale applies on the plot data as well as on the mask data.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap OK to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap OK to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed values.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

Acq (Acquisition) Trend plot configuration menu

Use the Acq Trend configuration menu to set the acq trend plot waveform vertical scale and position, as well as add a label.

To open the Acq Trend configuration menu, double-tap the **Acq Trend** badge in the Measurement badge. You can also double-tap the Acq Trend Plot handle.

Note: Acq Trend plot is available for Power Quality and Efficiency measurements.

Acq Trend represents plotting of a selected measurement for each acquisition on a separate plot.

Note:

- Measurements that can use the Acq trend plot as a source will list the plot in the Source menu list.
- Acq trend plot resets when scope acquisition resets because of change in scope settings and measurement sources.

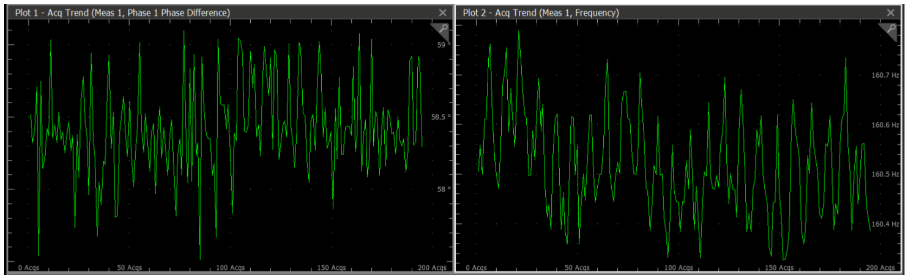


Figure 27: Phase Frequency

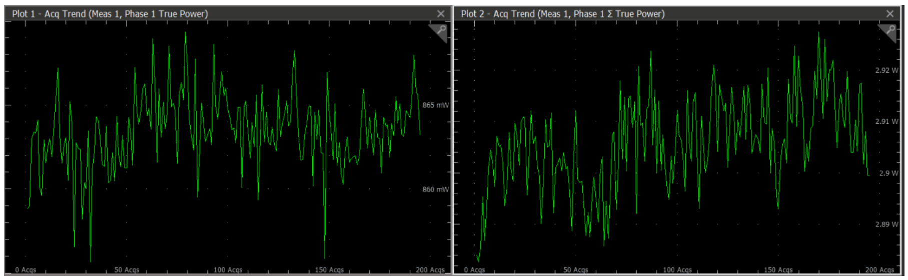


Figure 28: True Power

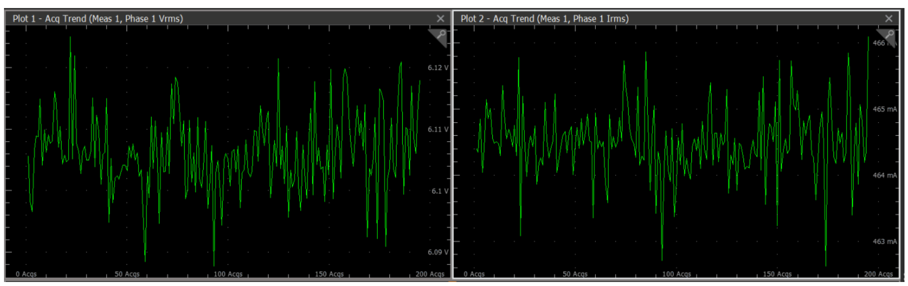


Figure 29: Vrms and Irms

Acq Trend configuration menu fields and controls

Field or control	Description
Auto Scale	Enables or disables the Auto Scale mode. Auto Scale uses the signal data to dynamically set the vertical scale units.
Plot Display	Available values are • Phase 1(V, I) • Phase 2(V, I) • Phase 3(V, I)

Table continued…

Field or control	Description
Measurements	Available measurements are: • Vrms • Irms • Phase Difference • Frequency • True Power • Apparent Power • Reactive Power • Sum of True Power • Sum of Apparent Power • Sum of Reactive Power
Save	Saves the plot data along with time stamp at the specified location.

Deleting a trend plot

To delete a Acq Trend plot, touch and hold the **Acq Trend** badge to open the right-click menu, and select **Delete Acq Trend**. Deleting the Measurement badge that enabled the plot also closes the plot.

Impedance plot configuration menu

Use this menu to change the settings of a displayed Impedance plot (Impedance measurement).

To open the Impedance configuration menu, double-tap anywhere in a **Impedance** Plot view.

Settings panel (Impedance plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** of to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is Off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the **Zoom** and **Pan** front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	**Vectors** draws waveforms with lines between record points. **Dots** draws waveform record points as dots on the screen.
X-Axis	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed only when **AutoScale** is Off (deselected).
Y-Axis	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when **AutoScale** is Off (deselected).

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap **OK** to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed Harmonic values.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

Trend Plot configuration menu

Use the Trend configuration menu to set the trend plot waveform vertical scale and position, as well as add a label.

To open the Trend configuration menu, double-tap the **Trend** badge in the Settings bar. You can also double-tap the Trend Plot handle.

Note: Measurements that can use the trend plot as a source will list the plot in the Source menu list.

Trend configuration menu fields and controls

Field or control	Description
Display	Toggles displaying the Trend plot **On** or **Off**.
Vertical Scale	Shows the vertical scale setting (when Auto Scale is on), or sets the vertical scale (when Auto Scale is off). Use the up and own arrow buttons to set the vertical scale value when Auto Scale is turned off. You can also use the multipurpose knob or screen keypad to change the value.
Auto Scale	Enables or disables the AutoScale mode. Auto Scale uses the signal data to dynamically set the vertical scale units.
Label	Adds a label to the Trend waveform (appears next to the Trend plot handle).

Table continued…

Field or control	Description
Position	Shows the vertical position setting (when Auto Scale is on), or sets the vertical position (when Auto Scale is off).
Set to 0	Sets the trend plot to the center (zero) of the graticule (zero).
Save Plot Data	Saves the plot data to the specified location.

Deleting a trend plot

To delete a Trend plot, touch and hold the **Trend** badge to open the right-click menu, and select **Delete Trend**. Deleting the Measurement badge that enabled the plot also closes the plot.

Time Trend plot configuration menu

Use the Time Trend configuration menu to set the time trend plot waveform vertical scale and position, as well as add a label.

To open the Time Trend configuration menu, double-tap the **Time Trend** badge in the Settings bar. You can also double-tap the Time Trend Plot handle.

Note: Measurements that can use the trend plot as a source will list the plot in the Source menu list.

Time Trend configuration menu fields and controls

Field or control	Description
Display	Toggles displaying the **On** or **Off**.
Vertical Scale	Shows the vertical scale setting (when Auto Scale is on), or sets the vertical scale (when Auto Scale is off). Use the up and own arrow buttons to set the vertical scale value when Auto Scale is turned off. You can also use the multipurpose knob or screen keypad to change the value.
Auto Scale	Enables or disables the Auto Scale mode. Auto Scale uses the signal data to dynamically set the vertical scale units.
Label	Adds a label to the Time Trend waveform (appears next to the Time Trend plot handle).
Position	Shows the vertical position setting (when Auto Scale is on), or sets the vertical position (when Auto Scale is off).
Set to Center	Sets the time trend plot to the center (zero) of the graticule (zero).
Phase Display	Available values are • Phase 1(Vxy, Ix) • Phase 2(Vyz, Iy) • Phase 3(Vzx, Iz)

Table continued…

Field or control	Description
Measurements	Available measurements are • Vrms • Irms • Frequency • True Power • Apparent Power • Reactive Power • ∑ True Power • ∑ Reactive Power • ∑ Apparent Power
Save Plot Data	Saves the plot data to the specified location.

Note: In IMDA Time Trend is displayed only for Power Quality measurements.

Deleting a time trend plot

To delete a Time Trend plot, touch and hold the **Time Trend** badge to open the right-click menu, and select **Delete Time Trend**. Deleting the Measurement badge that enabled the plot also closes the plot.

Control Loop Response (Bode) plot configuration menu

Use this menu to change the settings of a displayed Control Loop Response plot (Control Loop Response measurement).

To open the Control Loop Response plot configuration menu, double-tap anywhere in a **Control Loop Response** Plot view.

Settings panel (Control Loop Response plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** of to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is Off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the **Zoom** and **Pan** front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	**Vectors** draws waveforms with lines between record points. **Dots** draws waveform record points as dots on the screen.
Interpolation	Select the method to display the record points between sampled points. **Sin(x)/x** connects using a Sin(x)/x curve between the computed values. This makes the gain or phase curve smoother than linear interpolation. **Linear** connects between computed points (Gain or Phase value) using a straight-line fit. Measurement annotations are displayed for Linear mode interpolation only.

Table continued…

Field or control	Description
X-Axis (Frequency) Scale	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed only when **AutoScale** is Off (deselected).
Y-Axis (Gain) Scale	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when **AutoScale** is Off (deselected).
Y-Axis (Phase) Scale	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when **AutoScale** is Off (deselected).

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap **OK** to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed Harmonic values.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

Power Supply Rejection Ratio plot configuration menu

Use this menu to change the settings of a displayed Power Supply Rejection Ratio plot (Power Supply Rejection Ratio measurement).

To open the Power Supply Rejection Ratio plot configuration menu, double-tap anywhere in a **Power Supply Rejection Ratio** Plot view.

Settings panel (Power Supply Rejection Ratio plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** of to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is Off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the **Zoom** and **Pan** front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	**Vectors** draws waveforms with lines between record points. **Dots** draws waveform record points as dots on the screen.
Interpolation	Select the method to display the record points between sampled points. **Sin(x)/x** connects using a Sin(x)/x curve between the computed values. This makes the gain or phase curve smoother than linear interpolation. **Linear** connects between computed points (Gain or Phase value) using a straight-line fit. Measurement annotations are displayed for Linear mode interpolation only.
X-Axis	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed only when **AutoScale** is Off (deselected).
Y-Axis	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when **AutoScale** is Off (deselected).

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap **OK** to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed Harmonic values.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

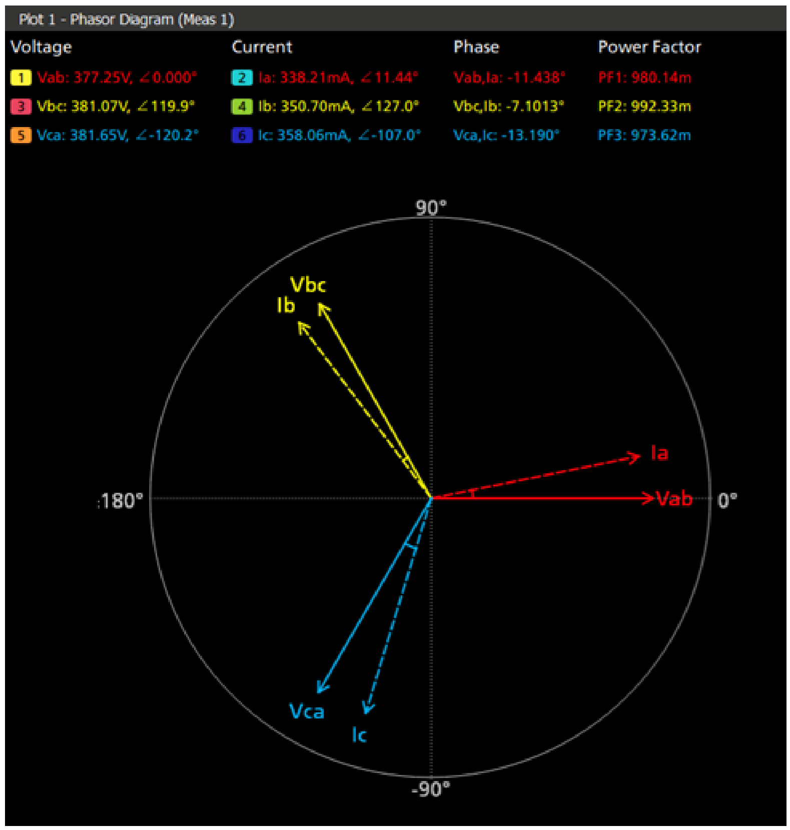
Phasor Diagram plot configuration menu (IMDA-Power Quality)

Use this menu to change settings of a displayed phasor plot (IMDA Power Quality measurement).

Settings panel (Phasor Diagram configuration menu) fields and controls

The Phasor diagram displays the magnitude and phase angle between the voltage and current. The number of V and I depends on the wiring configuration. The Phasor plot includes:

- All voltage magnitudes with phase values
- All current magnitudes with phase values
- Phase angle between the V and I pairs
- Power Factor for all voltages and currents



Field or control	Description
Wiring	Allows to measure 2V2I or 3V3I for 3P3W and 3P4W based on motor configuration. They can also use Line-to-Line and Line-to-Neutral for mathematical conversion. 1 Phase-2 Wire (1V1I) 1 Phase-3 Wire (2V2I) 3 Phase-3 Wire (2V2I) 3 Phase-3 Wire (3V3I) 3 Phase-4 Wire (3V3I)

Table continued…

Field or control	Description
Plot Display	Select the magnitude and the phase angle. The number of V and I depends on the wiring configuration. All One pair V and I One pair V One pair I
V and I Pair	Vac (Math 2), Ia (Ch 1) Vbc (Ch2), Ib (Ch 3) Available when **Plot Display** = **One pair V and I**
Voltage Pair	Vac (Math 2), Vbc (Ch 2) Available when **Plot Display** = **One pair V**
Current Pair	Ia (Ch 1), Ib (Ch 3) Available when **Plot Display** = **One pair I**
Phasor Type	Allows to select amplitude computation method either RMS or Magnitude. RMS is computed on the time domain waveform. Magnitude values are computed at the fundamental or operating frequency of the signal.

Note: Change in vector naming is based on Line-to-Line and Line-to-Neutral selection in the high level.

The Vector displays RMS values of Voltage and Current and not the magnitude. The voltage and current values are RMS vector results, while Phase uses DFT method.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

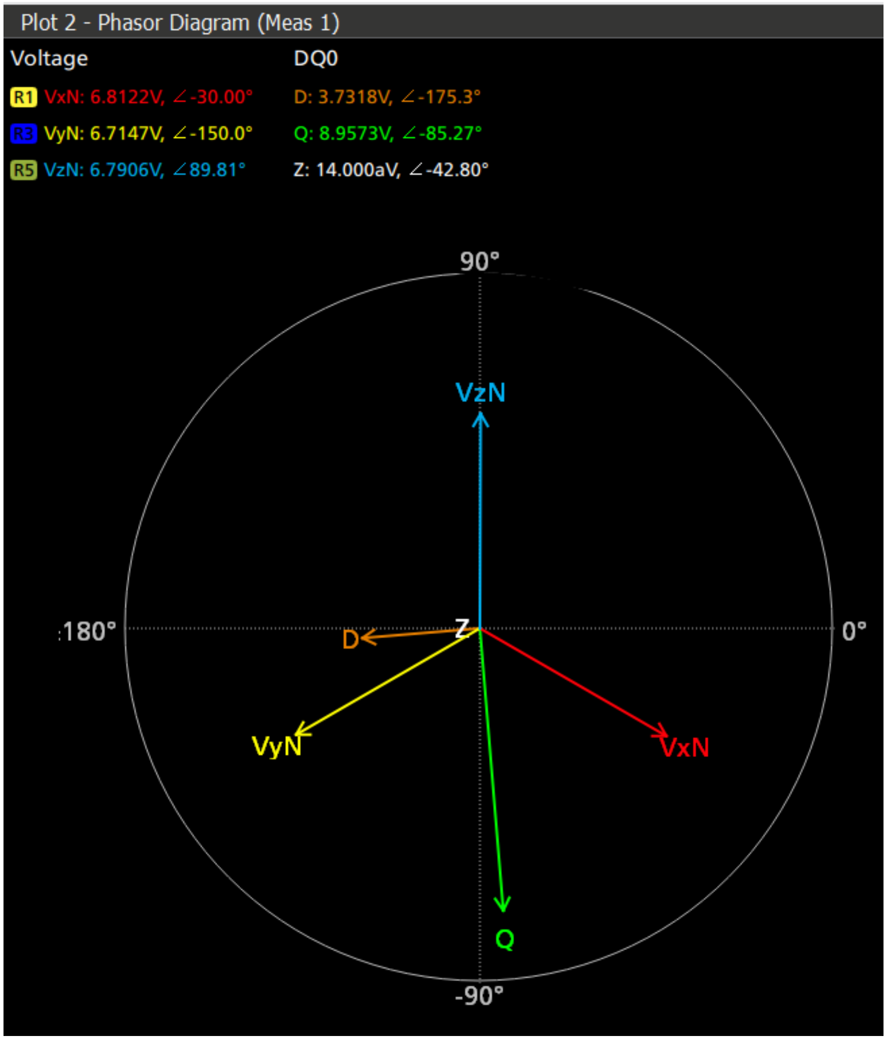
Deleting the Measurement badge that opened the plot also closes the plot.

Phasor Diagram plot configuration menu (IMDA-DQ0)

Use this menu to change settings of a displayed phasor plot (IMDA-DQ0 measurement).

Settings panel (Phasor Diagram configuration menu) fields and controls

The Phasor diagram displays the magnitude and phase angle between the voltage or current. The number of V and I is always 3 phasor. The Phasor plot includes either voltage or current magnitudes with phase values.



Field or control	Description
Wiring	Allows to measure selected input or output wiring.
Plot Display	Select the plot display. All: Displays both DQ0 and ABC components. ABC: Displays 3 phase voltage components. XYZ: Displays 3 phase current components.

Table continued…

Field or control	Description
Phasor Type	Select RMS or Magnitude. RMS: Magnitude:

Note: Change in vector naming is based on Line-to-Line and Line-to-Neutral selection in the source setup panel.

The plot configuration has phasor type to choose RMS or Magnitude. The voltage or current values are RMS vector results, while Phase uses DFT method.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

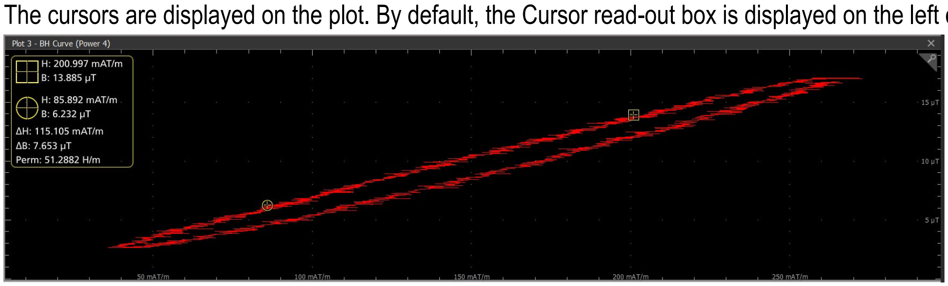
Power plots and cursors

Use cursors in I vs. ∫V and B-H curve plots to take measurements at any point on these power measurement plot waveforms.

Cursor readouts display the voltage and current values at their position and the difference (delta) between the cursors. You can display cursors in **I** vs. ∫V and B-H curve plots.

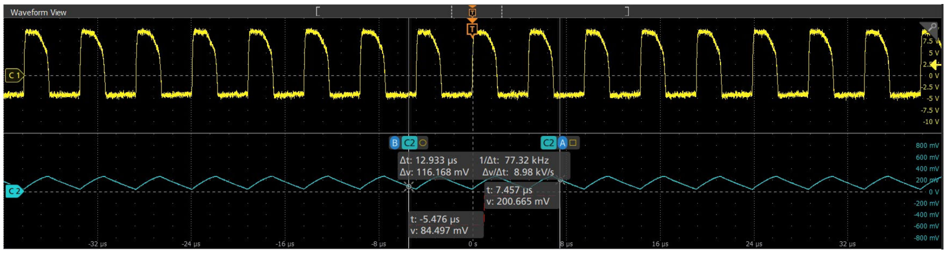
1. Select the plot to which you want to add cursors.
2. Tap the **Add New**…**Cursors** button, or push the front-panel **Cursors** button.

   The cursors are displayed on the plot. By default, the Cursor read-out box is displayed on the left corner of the plot view.



3. Use Multipurpose Knobs **A** and **B** to move the cursors on the waveform, or touch and drag a cursor.

   Note: Moving the waveform cursor in the Waveform View moves the respective cursors in the plot view.



Cursors readouts show the position with B and H coordinates, the difference of B and H between the cursors, and the permeability value.

4. To further configure cursors, double-tap on either cursor line or the cursor readouts to open the **Cursors** configuration menu.
5. Tap the Help icon on the menu title for more information on the menu settings.
6. To stop showing cursors, push the front panel **Cursor** button, or press and hold on a cursor to open the right-click menu and turn cursors off, or open the Cursors configuration menu and set Display to **Off**.

I vs (integral of) V plot configuration menu (Magnetic Analysis power measurement)

Use this menu to change settings of a displayed I vs. ∫V plot (I vs. ∫V measurement).

To open the I vs. ∫V plot configuration menu, double-tap anywhere in the **I vs. ∫V** Plot view.

Settings panel (I vs. ∫V plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** off to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the **Zoom** and **Pan** front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
X-Axis	Sets the beginning and end scale range to display in the plot for the X-Axis scales. It is displayed only when **AutoScale** is off (deselected).
Y-Axis	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. It is displayed only when **AutoScale** is off (deselected).

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap **OK** to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed Harmonic values.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

Inductance Curve configuration menu (Magnetic Analysis power measurement)

Use this menu to change settings of a displayed Inductance curve plot (Inductance measurement).

To open the Inductance Curve plot configuration menu, double-tap anywhere in an **Inductance Curve** Plot view.

Settings panel (Inductance Curve configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** off to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is **Off**, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the **Zoom** and **Pan** front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
X-Axis	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed when **AutoScale** is **Off** (deselected).
Y-Axis	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed when **AutoScale** is **Off** (deselected).

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap **OK** to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed Harmonic values.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

BH curve configuration menu (Magnetic Analysis power measurement)

Use this menu to change the settings of a displayed BH curve plot (Magnetic Property measurement).

To open the BH curve plot configuration menu, double-tap anywhere in a **BH curve** Plot view.

Settings panel (BH curve configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** of to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is Off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the **Zoom** and **Pan** front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
X-Axis	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed only when **AutoScale** is Off (deselected).
Y-Axis	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when **AutoScale** is Off (deselected).
Displayed Cycles	Select either **All** or $\mathbf{B_{Peak}}$.

Note:

- You can move the cursor symbols in the BH curve plot by placing waveform cursors at the correct positions in Waveform view. However, to place cursor symbols correctly at exact positions in Waveform view with precision, you need to enable zoom scale on the Waveform view and increase the zoom scale if you need higher resolution, and then move the waveform cursors using MPH knob in fine mode.
- When you configure a BH plot to Bpeak from All cycles, the waveform cursors does not get placed automatically on the peak cycle in the time domain waveform. You have to manually move the waveform cursors until you view the cursor symbols value in the Bpeak plot. This cycle relates to Bpeak.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap **OK** to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed Harmonic values.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

Trajectory plot configuration menu (Switching Loss power measurement)

Use this menu to change settings of a displayed SWL (Switching Loss) Trajectory plot.

To open the Trajectory plot menu, double-tap anywhere in the Plot view.

Settings panel (Trajectory plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** off to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and position.

Table continued…

Field or control	Description
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
X-Axis (Voltage) **From, To**	Sets the beginning and end scale range to display in the plot for the X-Axis scales. It is displayed only when AutoScale is off (unchecked).
Y-Axis (Voltage) **From, To**	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. It is displayed only when AutoScale is off (unchecked).

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap **OK** to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed values.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

Bathtub plot configuration menu

Use this menu to change settings of a displayed jitter versus BER (bathtub) plot.

To open the Bathtub plot configuration menu, double-tap anywhere in the **Bathtub** plot view.

To close a plot view, tap the X in the upper right corner of the view. Deleting the Measurement badge that opened the plot also closes the plot.

Settings panel (Bathtub plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Units	Sets the X-axis units to be unit intervals or seconds.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in BER units. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis scale	Sets the Y axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the Save As menu. Navigate to the location where you want to save the plot image. Enter a file name in the File Name field. Select the image file format (PNG, BMP, or JPG). Tap OK to save the plot image.
Save Plot Data	Tap to open the Save As menu. Navigate to the location where you want to save the plot data. Enter a file name in the File Name field. Tap OK to save the plot data as a comma-separated values (.csv) file.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

Harmonics Bar Graph plot configuration menu

Use this menu to change settings of a displayed harmonics bar plot (Power Harmonics measurement).

To open the Bar Graph plot configuration menu, double-tap anywhere in a **harmonics** Plot view.

Settings panel (Harmonics Bar Graph configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** off to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is **Off**, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the **Zoom** and **Pan** front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Unit panel	Set the unit to Linear or Log. A Log scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater- magnitude components on the same display. A Linear scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Harmonics (Inside the Unit panel)	Select All/Odd/Even.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **OK** to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap **OK** to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed Harmonic values.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that opened the plot also closes the plot.

Math configuration menu overview

Math waveforms are created by combining and/or mathematically transforming source waveforms into a new waveform for analysis. Use this menu to create math waveforms (basic or advanced) or add an FFT (Fast Fourier Transform) waveform to the screen.

To access a Math configuration menu, double-tap a **Math** waveform badge on the Settings bar. If no Math badge is present, tap the **Add New Math** button to add a Math waveform and open the configuration menu.

Use the following links to access information on the Math waveform menus and settings.

Math configuration menu

Use this menu to set math waveform parameters, create basic and advanced math waveforms, or add an FFT (Fast Fourier Transformation) plot to analyze frequency components of a waveform.

To access the Math menu, double-tap a **Math** waveform badge. If no Math badge is present, tap the **Add New Math** button to add a math waveform and open the menu.

Math configuration menu fields and controls

Field or control	Description
Display	Turns the math waveform or FFT plot **On** or **Off**.
Vertical Scale	Sets the vertical graticule scale units. Tap the arrows to change the value, tap and use the assigned multipurpose knob to change values, or double-tap to open the virtual keypad to enter a specific value. Only available when **Auto Scale** is disabled (unchecked) and **Math Type** = **Basic** or **Advanced**.
Auto Scale	Toggles Auto Scale mode on or off. Auto Scale calculates the vertical scale and position to center and display the entire waveform. Available when **Math Type** = **Basic** or **Advanced**.
Label	Enter a label for the math waveform.
Position	Sets the vertical position of the math waveform. Available when **Math Type** = **Basic** or **Advanced**.
Set to 0	Sets the vertical position of the math waveform to zero (vertical center of a slice (Stacked mode) or the screen (Overlay mode). Available when **Math Type** = **Basic** or **Advanced**.
Units	Sets user-specified units to display on vertical scale readouts and measurement badges.
Average	Toggles averaging the waveform **On** and **Off**. Use averaging to reduce noise on the waveform.
Number of Averages	Sets the number of waveform acquisitions to average. Tap and use the assigned multipurpose knob to set the value. Available when **Average** = **On**.

Table continued…

Field or control	Description
Math Type	Sets the type of math waveform to display. **Basic** creates a math waveform by adding, subtracting, multiplying, or dividing two analog waveforms. **Note:** You cannot mix time and frequency sources for basic math waveforms. To create a spectrum basic math waveform, both sources must be spectrum waveforms. The spectrum math waveform is added in a new slice in the Spectrum View window. **FFT** opens an FFT view of the specified signal to display the frequency components of that signal. Double-tap on the FFT view to open its configuration menu. See *Math FFT plot configuration menu (Math waveform)* on page 335. **Note:** You cannot create an **FFT** math waveform from a spectrum trace waveform source. **Advanced** displays a drop-down from which to select the 20 last-accessed equations created by the Equation Editor. This mode also provides access to the Equation Editor. See *Equation Editor (Math configuration menu)* on page 366. **Note:** You cannot create an advanced math waveform using a spectrum trace waveform source.
Source, Source1, Source 2	Defines the signal source or sources for a **Basic** or **FFT** math waveform. Basic and FFT math waveforms are created from analog channels only (Ch, Math, Ref, or). Available when Math Type = Basic or FFT. **Note:** You cannot mix time and frequency sources for basic math waveforms. To create a spectrum basic math waveform, both sources must be spectrum waveforms.
Basic math operation list	Located between the **Source 1** and **Source 2** fields. A drop-down list to select a basic math operation (add, subtract, multiply, divide) to apply to the two sources. Available when **Math Type** = **Basic**.
Math n =	Lists the last-accessed advanced equation. Tap the down arrow to display a list of the last-accessed equations (up to a maximum of 20) created by the **Equation Editor**. Select an equation to display that math waveform. Tap **Edit** to open the **Equation Editor** to edit the displayed equation. You can also double-tap on an equation in this field and directly edit the equation using the virtual keyboard. See *Equation Editor (Math configuration menu)* on page 366. Available when **Math Type** = **Advanced**.
Edit	Opens the **Equation Editor** to create advanced math waveforms from analog channels, reference, math waveform, measurement, filter, and variable sources. Tap the **Edit** button to open the **Equation Editor**. See *Equation Editor (Math configuration menu)* on page 366. Available when **Math Type** = **Advanced**.

Math waveform guidelines

- Digital channels are not valid in math waveforms.

- You can take measurements on math waveforms in the same way as on channel waveforms.
- Math waveforms derive their horizontal scale and position from the sources in their math expressions. Adjusting these controls for the source waveforms also adjusts the math waveform.
- You cannot create an **FFT** math waveform from a spectrum trace waveform source.
- You cannot mix time and frequency sources for basic math waveforms. To create a spectrum basic math waveform, both sources must be spectrum waveforms.
- You cannot create an advanced math waveform using a spectrum trace waveform source.
- You can Zoom on math waveforms.

Equation Editor (Math configuration menu)

Use the Equation Editor to build your advanced math waveform expression using sources, operators, constants, measurements, variables, filters, and functions.

To access the math Equation Editor:

1. Double-tap a **Math** waveform badge. If no Math badge is present, tap the **Add New Math** button to add a Math waveform and open the configuration menu.
2. Set Math Type to **Advanced**.
3. Tap **Edit** to open the Equation Editor.

Equation Editor menu fields and controls

Field or control	Description
Sources	Lists all available sources that you can add to an equation. Tap a source icon to add it to the cursor position in the Math x = input box. Drag the sources field up or down to scroll through selections. If a source column is empty, there are no active or defined sources for that category.
Add Filter	Tap to open the **Add Filter** menu, to create a filter definition to add to the **Filters** source column in the **Equation Editor** menu. See *Add Filter menu (math Equation Editor)* on page 367.
Add Variable	Tap to open the **Add Variable** menu, to add a defined variable to the **Variables** source column in the **Equation Editor** menu. See *Add Variable menu (math Equation Editor)* on page 369.
Functions	Select the math functions to apply to your signal or signals. See *Add Functions (math Equation Editor)* on page 368.
Keypad	Use to enter numeric, basic math operations, and logic conditions.
Math <x> =	The field that lists the equation components that you add with the equation controls for the numbered Math waveform. You can also directly edit this field.
Clear	Clears the math equation field.
Arrows	Use to move the cursor position in the **Math <x> =** equation field.
Delete	Deletes the selected part of the equation (recommend that you use a mouse to more easily select equation text).
Backspace	Deletes the character to the left of the cursor.

Table continued…

Field or control	Description
Apply	Tap to apply the math expression shown in the **Math <x> =** field and display the math waveform. The first ~ eight characters of the equation also appears in the Math badge on the Settings bar. When the editor detects a logic or syntax error with the equation, the editor displays an error message, along with the character position in the equation where the editor considers the error to have occurred.
Cancel	Closes the editor menu, does not update the equation list if you made any changes from the last time you Applied or OK'd an equation.
OK	Applies the equation to the math waveform, closed the Advanced Editor window, and adds the equation to the available equation list.

Equation editor guidelines

- Math definitions are not applied if the sources are not valid.
- Syntax or other equation or source errors display a short error message on the Math configuration menu.
- Use parentheses to group terms in the expression to control execution order, for example, 5*(Ch1 + Ch2).

Add Filter menu (math Equation Editor)

Use the Add Filter menu to add a high pass, low pass, or arbitrary filter expression to the Filters column of the Equation Editor Sources table.

To access the Add Filter menu:

1. Double-tap a Math waveform badge on the Settings bar. If no Math badge is present, tap the **Add New Math** button to add a Math waveform and open the configuration menu.
2. Set Math Type to **Advanced**.
3. Tap **Edit** to open the Equation Editor.
4. Tap **Add Filter** to open the Add Filter menu.

Add Filter menu fields and controls

Field or control	Description
Filter Type	Sets the filter type: **High Pass**, **Low Pass**, or **ArbFlt** (arbitrary filter). ArbFlt requires you to load a FLR-format filter file. If loading a filter file, tap Load to navigate to and select the FLR file to load.
Cutoff Frequency	Sets the filter cutoff frequency as a predefined fraction of the sample rate (SR). Default is 0.25 * SR.
Label	Shows the filter selections as A:B, where A is the filter type (HP or LP) and B is cutoff frequency setting. Arbitrary filter files use the file name as the label.

Add Functions (math Equation Editor)

Use the Add Functions controls to add predefined math operations to your math waveform equation.

Button	Description
∫	Integral. Inserts the text INTG(into the math expression. Enter an argument to the function. The integral function produces the integral of the argument.
Max	Inserts the text MAX(into the math expression. The MAX function accumulates, over time, the maximum value at each point in the vector.
Log	Base 10 logarithm. Inserts the text LOG(into the math expression. Enter an argument to the function. The log function produces the base 10 logarithm of the argument.
e^y	Natural antilog. Inserts the text EXP(into the math expression. The EXP function produces the natural antilog of the argument.
Ceil	Ceiling function. Inserts the text CEIL(into the math expression. CEIL returns the smallest integer that is ≥ the expression within the parenthesis.
Inv	Inserts the text INV(into the math expression. Enter an argument or channel to the function. The invert function inverts the argument within the parentheses.
FFT Mag	FFT Magnitude. Inserts the text FftMag(into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT waveform that shows the magnitude components of the source signal.
FFT Phs	FFT Phase. Inserts the text FftPhase(into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT math waveform that shows the phase components of the source signal.
d/dx	Derivative. Inserts the text DIFF(into the math expression. Enter an argument to the function. The derivative function produces the derivative of the argument.
Min	Inserts the text MIN(into the math expression. The MIN function accumulates over time the minimum value at each point in the vector.
Ln	Natural logarithm. Inserts the text LN(into the math expression. The natural logarithm function produces the natural logarithm of the argument.
√x	Square root. Inserts the text SQRT(into the math expression. Enter an argument to the function.

Table continued…

Button	Description
Floor	Floor function. Inserts the text FLOOR(into the math expression. FLOOR returns the largest integer that is ≤ the expression within the parenthesis.
|x|	Absolute. Inserts the text FABS(into the math expression. The FABS function takes the absolute value of the expression.
FFT Real	FFT Real. Inserts the text FftReal(into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT math waveform that displays only the real part of the source signal.
FFT Img	FFT Imaginary. Inserts the text FftImaginary(into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT math waveform that displays only the imaginary part of the source signal.
SIN	Inserts the text SIN(into the math expression.
COS	Inserts the text COS(into the math expression.
TAN	Inserts the text TAN(into the math expression.
SIN^{-1}	Arc sine. Inserts the text ASIN(into the math expression.
COS^{-1}	Arc cosine. Inserts the text ACOS(into the math expression.
TAN^{-1}	Arc tangent. Inserts the text ATAN(into the math expression.
Deg	Degrees. Inserts the text DEG(into the math expression. The function expresses the value of the expression in degrees.
Rad	Radians. Inserts the text RAD(into the math expression. The function expresses the value of the expression in Radians.

Add Variable menu (math Equation Editor)

Use the Add Variable menu to add a defined variable source to the Equation Editor Sources table, which you can then add to your math waveform expression.

To open the Add Variable menu:

1. Double-tap a Math waveform badge on the Settings bar. If no Math badge is present, tap the **Add New Math** button to add a Math waveform and open the configuration menu.
2. Set Math Type to **Advanced**.
3. Tap **Edit** to open the Equation Editor menu.
4. Tap **Add Variable** to open the Add Variable menu.

Add Variable menu fields and controls

Field or control	Description
Variable Type	Selects a predefined variable to add to the Variables column of the Equation Editor. **Sample Rate** creates a variable with the value of the current sample rate. **Sample Interval** creates a variable with the value of the current sample interval (1/sample rate) **Record Length** creates a variable with the value of the current record length.
Cancel	Closes the menu without adding a variable to the **Equation Editor** Variables column.
OK	Closes the menu and adds the current variable selection to the Variables column of the **Equation Editor's Sources** table.

Menu bar overview

The Menu bar provides access to file, utility, and help functions.

The Menu bar

Field or control	Description
File	Provides typical system file management operations such as opening, saving, moving, and renaming files. See the links after this table for more information. **Autoset** executes an immediate Autoset operation. See *Quickly display a waveform (Autoset)* . **Default Setup** immediately restores the oscilloscope to factory default settings. See *Using Default Setup*. **Restart** powers off the oscilloscope and restarts the oscilloscope. **Shutdown** powers off the oscilloscope.
Edit	Provides a menu to Undo or Redo the last operation. See the links after this table for more information.
Utility	Use to set user preferences, configure input, output, and network settings, run self tests, verify calibration status and run signal path compensation, and erase nonvolatile memory. See the links after this table for more information.
Application	Use to access installed software applications. This menu only appears when testing applications are installed on the oscilloscope. Only valid for Windows OS installations. See the links after this table for more information.
Help	Opens the Help viewer, displays current instrument software, See the links below for more information.

Use the following links to access information on the Menu bar menus.

Save As configuration menu (File menu)

Use this menu to configure saving screen captures, waveforms, oscilloscope setups, reports, and session files.

To access the **Save As** configuration menu, tap **File** on the menu bar and select **Save As...**.

Note: Selecting **File** > **Save** the first time after powering up the oscilloscope opens the **Save As** configuration menu. This lets you set or verify the save locations for all types of information you can save.

Once the **Save As** configuration menu has been opened and closed, the next time you select **Save** causes the instrument to automatically save the file type last selected in the **Save As** configuration menu. This lets you quickly save files with a simple menu selection.

Saving files with the front-panel User button

Pushing the front-panel **User** button automatically saves the file type last selected in the **Save As** configuration menu. If no saves have been performed since the instrument power-up, pushing the User button opens the **Save As** configuration menu. Select the type of save operation you want to perform and tap OK. After that, pushing the User button automatically saves the file type.

Note: The User button is not assigned a specific save type by default; it retains whatever save action was last selected in the **Save As** configuration menu.

File operations and Microsoft Windows 10 Operating System SSD

Instruments with Windows 10 SSD will display the standard Windows file tools to navigate to and select files and folders.

The Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. This is different from standard instruments (without Windows 10 installed), which assign a non-changing drive letter to each USB port.

Save As configuration menu fields and controls

The following fields and controls are common to all Save As actions.

Field or control	Description
File save type	Tabs on the left let you set which type of file to save (**Screen Capture**, **Waveform**, **Setup**, **Report**, or **Session**). Selecting a file type sets the file extensions in the Save As Type field to the correct value.
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Tap to open the **Browse Save As Location** configuration menu, to navigate to and select the location to which to save the file. See *Browse Save As Location configuration menu*.

Table continued…

Field or control	Description
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Auto Increment File Name	Select to enable auto-incrementing of the file name. Auto increment adds a number count to the end of the file name, and increments it on each subsequent saving of the same file name.
Count	Sets the increment count start number. Default is 000.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the save format.
Cancel	Cancels the file save action and closes the configuration menu.
OK	Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.

Screen Capture tab fields and controls

The following settings are specific for saving a screen capture

Field or control	Description
File save type	Tap the **Screen Capture** tab to save a screen image to a file. Selecting Screen Capture sets the file extensions in the Save As Type field to available graphic file formats.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the graphic save format.

To save screen captures with waveforms on a white background, tap **Utility**>**User Preferences** and set **Screen Capture** Colors to **Inverted.** When you save the next screen capture, the display inverts colors for just a moment, saves the displayed image to the file, and returns the screen to normal colors. This capability us useful to save ink on printouts of screen captures. See the **Display** panel settings in *User Preferences (Utility menu)* on page 381.

Waveform tab fields and controls

The following settings are specific for saving a waveform.

Field or control	Description
File save type	Tap the **Waveform** tab to save waveform(s) to a file. Selecting **Waveform** sets the file extensions in the **Save As Type** field to available waveform file formats.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the graphic save format.

Table continued…

Field or control	Description
Source	Sets the waveform source to save. You can save a single waveform, or save all active (displayed) waveforms. Waveform file names include the source channel or waveform appended to the end of the file name.
Display as next available Ref waveform	Sets the waveform to be saved and automatically opened (displayed) as the next available reference waveform. It doesn't matter what type of file is saved, that waveform will be opened as a Ref waveform. If gating is in use, a smaller, gated waveform is saved and displayed. This control is not present when source is set to **All**.
Gating	Sets the method to save a specified part of the waveform data. **None** saves the full waveform data (default). **Cursors** saves the waveform data located between the vertical cursors. If cursors aren't on when selecting cursor gating, the cursors are activated. **Screen** saves the waveform data that is on the screen. **Resample** saves the waveform data at a sample interval set by the user. The resulting saved waveform is a resampled version of the original waveform with fewer data points. Gating notes: • Default Setup restores Gated Save to its default setting (None). • The state of Gated Save gets saved in Setup and Session files. • Gated saves do not work on plot waveform data. • Gating cannot be used to save waveform data while in Fast Frame mode.

Setup tab fields and controls

The following settings are specific for saving an instrument setup.

Field or control	Description
File save type	Tap the **Setup** tab to save the instrument setup and measurement settings to a file. Selecting Setup sets the file extension in the Save As Type field to .set.
Include Reference Waveforms	Include the instrument waveforms.

Report tab fields and controls

The following settings are specific for saving an instrument report.

A report can include information on instrument settings, measurement results, screen images, individual measurement configuration, source input settings, and error and warning information for measurements. Reports are either a PDF file or a single file Web page.

Field or control	Description
File save type	Tap the **Report** tab to save a report file. Selecting Report sets the file extensions in the Save as Type field to available report formats.

Table continued…

Field or control	Description
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the report save format (PDF or Single File Web Pages).
Append Report	Appends a report to an existing report file.
Comments	Add comments to clarify the contents or purpose of the report, or specifics of the signals being measured.
Include Images and Annotations	Add Waveform and Plot images to the report.
Include Setup Configuration	Add instrument and measurement configuration information to the report.

Session tab fields and controls

The following settings are specific for saving an instrument session.

A session is a zipped file that contains an instrument setup file and all acquired waveform data. Use session file contents to move analysis activities to a PC and free up the instrument for others to use.

Field or control	Description
File save type	Tap the **Session** tab to enable saving a session file. Selecting Session sets the file extensions in the Save As Type field to .tss.
Save As Type	A session file can only be saved to format .tss.

Recall configuration menu (File menu)

Use this menu to recall (load) reference waveforms, instrument setups, mask settings, and sessions (setup plus waveforms).

To open the file Recall configuration menu:

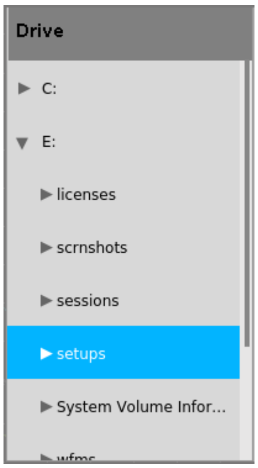
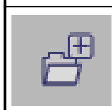
1. Tap **File** on the menu bar.
2. Tap **Recall** to open the Recall configuration menu.

File operations and Microsoft Windows 10 Operating System SSD

Instruments with Windows 10 SSD will display the standard Windows file tools to navigate to and select files and folders.

The Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. This is different from standard instruments (without Windows 10 installed), which assign a non-changing drive letter to each USB port.

Recall configuration menu fields and controls

Field or control	Description
Look in:	Shows the current directory path to the location of a file. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, up to a maximum of 20 locations.
File type to open (tabs)	Tabs on the left let you set which type of file to recall (Waveform, Setup, Mask, or Session). Selecting a file type sets the file extensions in the Files of Type field to the correct value.
	The Drive column lists the directory structure, opening at the root (/) level. Use to quickly navigate to a location. Tap to list the contents of the directory in the Name pane. Double-tap an item to display the directory and any subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
	Use the arrow buttons to navigate the file directory. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder. The Up arrow navigates up one level from the current folder.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File name	Lists the selected file name.
Files of type	Use to select the file format you want to open. The drop-down list shows all file extension types that the instrument can read for the selected file type.
Cancel	Cancels the file open action and closes the configuration menu.
Recall	Recalls the selected file. Recalling a waveform file adds a Reference waveform badge to the Settings bar and displays the waveform as it fits in the current Horizontal setting. Recalling a Setup or Session immediately sets the oscilloscope to the settings in the file and displays any waveforms that were part of the recalled session.

USB port drive names and locations

Use the following table to determine which drive to select when navigating to and/or selecting a file on system memory or a connected USB memory device.

Drive name	Drive letter	Drive or physical USB port location
MSO44, MSO46		
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**E**	USB 2.0 (top)
	F	USB 2.0 (middle)
	G	USB 2.0 (bottom)
Rear panel	**H**	USB 2.0 (left)
	I	USB 2.0 (right)

Drive name	Drive letter	Drive or physical USB port location
MSO54, MSO56, MSO58, MSO64, LPD64 without Windows OS		
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**E**	USB 3.0 (left)
	F	USB 2.0 (center)
	G	USB 2.0 (right)
Rear panel	**H**	USB 2.0 (top)
	I	USB 2.0 (bottom)
	J	USB 3.0 (top)
	K	USB 3.0 (bottom)
Instruments with Windows OS		
Root drive	**C**	User-accessible memory on the oscilloscope.
USB ports	**Dynamic port letter assignment**	If Windows operating system is installed, the Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. Use standard Windows procedures to mount and access network drives.

Drive name	Drive letter	Drive or physical USB port location
MSO58LP without Windows OS		
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**E**	USB 3.0 (left)
	F	USB 2.0 (right)

Table continued…

Drive name	Drive letter	Drive or physical USB port location
Rear panel	**G**	USB 2.0 (top)
	H	USB 2.0 (bottom)
	I	USB 3.0 (top)
	J	USB 3.0 (bottom)
Instruments with Windows OS		
Root drive	**C**	User-accessible memory on the oscilloscope.
USB ports	**Dynamic port letter assignment**	If Windows operating system is installed, the Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. Use standard Windows procedures to mount and access network drives.

Drive name	Drive letter	Drive or physical USB port location
LPD64 without Windows OS		
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**F**	USB 3.0 (left)
	E	USB 2.0 (center)
	G	USB 2.0 (right)
Rear panel	**H**	USB 2.0 (top)
	I	USB 2.0 (bottom)
	J	USB 3.0 (top)
	K	USB 3.0 (bottom)
Instruments with Windows OS		
Root drive	**C**	User-accessible memory on the oscilloscope.
USB ports	**Dynamic port letter assignment**	If Windows operating system is installed, the Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. Use standard Windows procedures to mount and access network drives.

Undo, Redo (Edit menu)

The Edit menu lets you Undo or Redo recent actions. Tap Undo or Redo. Not all oscilloscope actions can be undone. If the Undo or Redo menu items are grayed out, then the last action cannot be undone or redone.

About (Help menu)

Use the About configuration menu to show instrument information and installed options, and to install or uninstall analysis or feature options.

To open the About menu:

1. Tap **Help** on the menu bar.
2. Select **About** from the menu to open the About configuration menu.

About configuration menu fields and controls

Field or control	Description
System information	Provides system-related information such as model, bandwidth, serial number, Host ID, and installed firmware version. Provide this information when communicating with Tektronix to purchase option licenses or communicate with Customer Support.
Probes Detected	Lists probes connected to the instrument. TekVPI probes will list the probe model, serial number, and installed probe firmware version. Non-TekVPI probes may show their attenuation factor. **Note:** Connecting or disconnecting probes while the About menu is open does not update the Probes Detected list. The Probes Detected list is not dynamic.
Installed Options	Lists installed options. Click on an item in the list to show details of the option in the Option Details area below the list.
Option Details	Lists details of the option selected in the list, including license type, when the license was checked out, when it was installed on the instrument, and the date the license expires.
Remove License	Opens the Location to Save the Exit Key dialog. Use this to uninstall a floating license option for use by others. See *Location to Save Exit Key configuration menu*.
Install License	Tap this button to open the Browse License Files dialog to navigate to and select an option license file to install. See *Browse License Files menu (Help > About)* on page 379.

Help... (Help menu)

Tap this item to open the Help viewer. This Help viewer is similar in operation to a traditional Microsoft Windows help viewer.

Demo (Utility menu)

Use this menu to access demonstrations of key oscilloscope features.

To open the Demo configuration menu, select **Utility > Demo...** in the Menu bar.

Demo menu fields and controls

Field or control	Description
Demo overview panel	The upper half of the menu shows an overview of the demonstration available in the selected panel. This pane may also contain a screen shot showing the waveforms and capability being demonstrated.

Table continued…

Field or control	Description
Connection Details	Tap this button to open a dialog box that shows how to connect the oscilloscope to a signal source to perform the selected demonstration.
Recall Demo Session	Tap this button to load the session file for the selected demonstration.
Demo panels	Each panel provides a demonstration of oscilloscope capabilities within a category. The categories are Miscellaneous and Serial Bus. Each panel will have two or more demonstration buttons in them. Selecting a button updates the upper half of the menu to show the relevant content (and image if available) for the selected demonstration.

I/O Utility menu for TekScope

This configuration menu is to set up Incoming communication and Remote scope communication signal parameters. Use the following socket server settings to set up and use a socket server between your oscilloscope and a remote terminal

To open the I/O menu:

- Tap the Utility menu.
- Tap I/O....

Table 12: I/O panel fields and controls

Field or control	Description
Incoming communication	
Socket Server	Tap to toggle the socket server On or Off.
Protocol	Tap to select a protocol, either None or Terminal. A communication session run by a user at a keyboard typically uses a terminal protocol. An automated session might handle its own communications without such protocol from the oscilloscope.
Port	Enter the port number using the virtual keypad.
Remote Scope Communication	
TekScope Remote Control	Tap to toggle the TekScope Remote Control On or Off.
Protocol	Tap to select a protocol, either VISA or Scoket.

User Interface Tutorial (Help menu)

Tap this menu item to run a screen tutorial to learn the basics of the touch screen user interface.

To start the tutorial animation, tap **Help > User Interface Tutorial**. While the tutorial is running, you do not have access to any scope actions.

The tutorial closes automatically when it completes. You can also close the tutorial anytime by tapping on the Close button in the text box.

Browse License Files menu (Help > About)

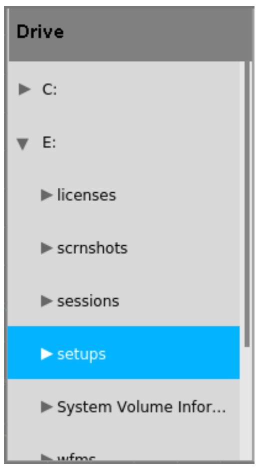
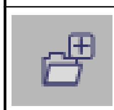
Use this menu to select and install an option license file to enable new functions.

To access the Browse License Files menu:

1. Tap **Help > About...** on the menu bar.

2. Tap **Install License** button to open the Browse License Files configuration menu.

Browse License Files configuration menu fields and controls

Field or control	Description
Look in	Shows the current directory path and file name. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.
	The Drive column lists the directory structure, opening at the root level. Installed USB memory devices are listed. Use to quickly navigate to a location. Tap to list the contents of the directory in the files listing. Double-tap an item, or tap the small arrow to the left of the directory, to display the subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
	Use the arrow buttons to navigate the directory structure. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File Name	Lists the selected file name. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the on-screen keyboard and enter the new name. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.
Files of Type	Use to select the file format you want to open. Tap the field to show a list of all file extension types that the instrument can read for the selected file type. The files pane only lists files of the specified type.
Cancel	Cancels the configuration menu changes, closes the menu, and returns to the prior menu without making any changes.
Open	Closes the configuration menu, returns to the About configuration menu, and installs the license. Follow any instructions that may be shown during the installation.

User Preferences (Utility menu)

Use this menu to set global display, Autoset, measurement, jitter and eye analysis settings, custom waveform colors, and other user preferences.

To open the **User Preferences** menu:

1. Tap **Utility** menu.
2. Tap **User Preferences** to open the configuration menu.
3. Tap a panel.

Display panel fields and controls

Field or control	Description
Default Waveform View Display Mode	Sets the default waveform view display mode, either **Overlay** or **Stacked**.
Waveform View Graticule	Sets the graticules to be **Movable** or **Fixed** for both stacked and overlay modes. **Movable** (default) sets graticules to move along with waveforms, and displays graticule units. **Fixed** sets graticules so that they do not move when you change a waveform's vertical or horizontal position. **Fixed** graticule mode also does not display the Vertical and Horizontal graticule labels. Use **Cursors** to take on-screen measurements.
Displayed Colors	Tap and select either **Normal** or **Inverted** colors to set how the instrument displays waveforms and plots. **Normal** shows waveforms and plots in color with a back background. **Inverted** makes the waveform background white, with graticule markings in black. **Inverted** does not change the Menu bar, Results bar, or Settings bar colors. Use this setting to save ink on printed screen captures.
Define Custom Colors	Opens the *Define Custom Colors* menu. This menu lets you change channel, math, and reference waveform colors for both **Normal** and **Inverted** display modes.
Screen Capture Colors	Sets how the instrument saves screen images (**Normal** or **Inverted** colors). **Normal** saves waveforms and plots in the colors as shown on the screen. **Inverted** makes the waveform background white, with graticule marking in black, just for screen saves. **Inverted** does not change the Menu bar, Results bar, or Settings bar colors. Also, if **Displayed Colors** is set to **Inverted**, selecting **Inverted** for screen saves does not invert the colors back to normal for the screen save; to save normal colors for a screen save, select **Normal.**
Backlight	Sets the intensity of the backlight. **Note:** Performing an Autoset resets the backlight value to High.
Auto-Dim	Select On to automatically dim the screen backlight after a specified time.

Table continued…

Field or control	Description
Time	Sets the amount of time to wait before the display is dimmed. Tap in the field and use the knob to change the time value, or double-tap to open the virtual keypad and set a time value. Only available when **Auto-Dim** is **On**.

Autoset panel fields and controls

Field or control	Description
Autoset Adjusts	Selects which controls to change as part of the Autoset operation (**Vertical Settings**, **Horizontal Settings**, **Trigger Settings**, and **Acquisition Settings**). The default is for all adjustments to be enabled.
Autoset in Overlay Display Mode Optimizes	Sets whether to optimize waveform resolution or visibility when doing an Autoset while in the Overlay display mode. **Resolution** uses as much of the ADC's range as possible to provide the best vertical resolution and measurement accuracy, but waveforms will overlap each other. **Visibility** scales and positions waveforms so they are visually separated from each other at the expense of vertical resolution and measurement accuracy.

Autoset guidelines:

- Autoset uses the current trigger source as its reference for various operations such as setting horizontal scale.
- If the current trigger type is **Bus** or **Sequence**, or if trigger type is **Edge** and the trigger source is **AC Line**, Autoset uses the lowest-numbered active channel as its reference.
- Autoset changes the acquisition mode to **Sample**.
- Doing a **Default Setup** operation does not change Autoset panel settings.
- All analysis and measurements abort immediately when an Autoset is started, and then restart upon Autoset completion.

Measurements panel fields and controls

Field or control	Description
Shared Reference Levels	Tap to select either Global or Per Source reference levels. **Global** applies the same reference levels to all measurement sources. **Per Source** allows selection of a different reference levels for all measurement sources.
Calculate Reference Levels	Tap to select how often to calculate reference levels. **First Acq** only calculates the reference levels on the first acquisition and re-uses them for all subsequent acquisitions. Pushing the Clear front-panel button, or making measurement changes (adding, deleting, or reconfiguring) also clears the reference levels which are then recalculated on the next acquisition. **Every Acq** calculates the reference levels with every acquisition.

Table continued…

Field or control	Description
Measurement Interpolation	Interpolation defines how the waveform is drawn between waveform record data points. Tap and select the measurement interpolation method. **Auto** selects the best interpolation method. **Sin(x)/x** interpolation computes record points using a curve fit between the actual samples acquired. **Linear** interpolation computes record points between actual acquired samples by using a straight line fit.
Measurement Annotations	Annotations show the exact segment of the waveform from which the measurement is derived. The annotation types consist of horizontal bars, vertical bars, or cross-hatch marks. **Auto** sets annotations to display if valid for the measurement. To view annotations for a measurement, select that measurement badge. If annotations are valid for that measurement, they are added to the waveform source for that measurement. **Off** turns off display of measurement annotations.

Jitter and Eye Analysis panel fields and controls

Field or control	Description
Calculate Reference Levels	Tap to select calculating the reference levels on the first acquisition or on every acquisition.
Horizontal Measurement Units	Tap to select horizontal measurement units of seconds or unit intervals.
Jitter Separation Model	Tap to select the jitter separation model (**Spectral Only** or **Spectral + BUJ**).
Lock RJ Value	Selecting **Lock RJ Value** calculates the measurements at the specified random jitter value. The checkbox is unchecked by default. Selecting the checkbox displays a text box where you can enter the RJ value. The default value is 1 ps. ⚠ **Note:** Lock RJ Value cannot be configured when Jitter Separation Model is Spectral + BUJ.

DDR Analysis panel fields and controls

Field or control	Description
Pass/Fail Limits	Turns **On** or **Off** the validation of the DDR measurements as Pass/Fail in the measurement results, based on the values in the limits file. Tap a measurement row in the Results table to view the limit setting and the Pass/Fail status for that measurement.
Limit File Location	Shows the location of the currently-loaded pass/fail limits file.
Browse	Tap **Browse** to open the *Browse Save As Location* configuration menu to navigate to and select a location at which to save the file.

Other panel fields and controls

Field or control	Description
Language	Enables displaying the user interface and Embedded Help files in an available language. Tap the field and select the language. The user interface changes immediately to the specified language. ⚠ **Note:** Not all UI terms or Help topics may be translated for new UI or menu items at the time of a firmware release. These items will be translated for a future firmware release. ⚠ **Note:** The Embedded Help files support English, Japanese, Simplified Chinese, and Russian. All other language selections will display the Help topics in English.
Font Size	Sets the font size for UI text elements. Font size range is 12 to 20.
Right Clicks via Touch	Turns **On** or **Off** the ability to use touch and hold method to open right-click menus on badges and other screen items.
Time	Sets the time it takes to respond to a touch and hold before opening a right-click menu.
Programmatic Interface Backward Compatibility	Allows you to choose an XML file that ensures backward compatibility of the commands defined in the XML file. Toggle button turns **On** or **Off** the backward compatibility option. Default is Off. For more information refer to *4/5/6 Series Mixed Signal Oscilloscopes Programmer Manual*.
Load	Tap to select the XML file. Appears when **Programmatic Interface Backward Compatibility** = **On**.
Connected Scope Preferences	Enables the connected oscilloscope with additional capabilities and improves the overall experience while using the oscilloscope. ⚠ **Note:** The oscilloscope should be connected to the Internet.
Oscilloscope tracks feature usage to help reproduce and fix software bugs. User identifiable information, waveform data, analysis results or screenshots are never saved	Enables oscilloscope tracks feature usage to help reproduce and fix software bugs. It will not save user-identifiable information, waveform data, analysis results or screen shots.

DDR Measurement limits file

The following content provides details about the limits file and a sample limit file for DDR.

A limits file contains the measurements and associated limits used to determine the Pass or Fail status for tests. Each limits file includes a list of one or more measurements, and the ranges of acceptable values for any or all statistics for each measurement. The measurement includes combinations of all measurements and statistical characteristics, and an appropriate range of values for each combination.

The application provides preconfigured limits files for many combinations of standards and speed grades. These preconfigured DDR limits files are located at `C:\Users\Public\Tektronix\TekScope\Applications\DDR\Limits`. You can create a limit file by specifying limits for any of the result parameters such as Mean, Std Dev, Max, Min, peak-to-peak, population, MaxPosDelta, and MinPosDelta. For each of these result parameters, you can specify the Upper Limit Equality (UL), and the Lower Limit Equality (LL).

To include pass/fail status in the result statistics, create a custom limits file in the following format using an XML editor or any other editor. If the file is created in any other editor such as Notepad, it should be saved in Unicode format.

The following is a sample limit file of `DDR3_800MHz_Limits.xml`.

```
<?xml version="1.0" encoding="utf-8"?>
<Main>
   <!-- DDR3 800 MHz Limits -->
 <Measurement>
   <NAME>tDH-Diff(base)</NAME>
   <STATS>
     <STATS_NAME>Min</STATS_NAME>
     <LIMIT>LL</LIMIT>
     <UL>0</UL>
     <LL>150e-12</LL>
   </STATS>
  </Measurement>
  <Measurement>
   <NAME>tDS-Diff(base)</NAME>
   <STATS>
     <STATS_NAME>Min</STATS_NAME>
     <LIMIT>LL</LIMIT>
     <UL>0</UL>
     <LL>75e-12</LL>
   </STATS>
  </Measurement>
</Main>
```

Font Color menu (Text Settings configuration)

Use this menu to change the label color. Touch and hold on callout or label text, select **Format Text** in the right-click menu, and select **Color** to open this menu. Click on a color to change the text color.

Text Settings configuration menu (Callout and Waveform labels text)

Use this menu to change and format existing Callout or Waveform labels (font type and size, color, bold, italic, and underline).

Prerequisite: There is callout or waveform label on the screen.

To open the **Text Settings** configuration menu, double-tap on the Callout or Waveform label. You can also touch and hold on the Callout/label text and select **Format Text** from the right-click menu.

To delete a Callout or label, touch and hold on the Callout/label and select **Delete** from the right-click menu.

Text Settings configuration menu fields and controls

Field or control	Description
Text	Double-tap and enter the desired text using the virtual keyboard.
Font	Tap and select the desired font from the drop-down list.

Table continued…

Field or control	Description
Size	Tap and select the desired font size from the drop-down list.
Color	Tap and select the desired font color from the color palette.
B	Tap to toggle text bolding On or Off.
I	Tap to toggle text italics On or Off.
U	Tap to toggle text underlining On or Off.

Reference waveform configuration menu

Use this menu to configure display settings for a reference waveform.

To open a reference waveform configuration menu, double-tap a **Ref** badge on the Settings bar.

Reference waveform configuration menu fields and controls

Field or control	Description
Display	Turns On or Off displaying the waveform.
Vertical Scale	Set the vertical scale by using the assigned multipurpose knob, a virtual keypad, or tap the up or down arrows.
Label	Adds a label to the waveform. Tap and enter text using a keyboard, or double-tap to open the virtual keyboard. The label text is the same color as the waveform. Once you have entered the label, close the menu and double-tap the label text to open the Text Settings menu to change the font color, size, and other characteristics.
Units	Set the units label that you want to display on the vertical scale.
Position	Set the vertical position of the waveform using the assigned multipurpose knob or the virtual keypad.
Set to 0	Tap to set the vertical position to 0 (vertical center of the graticule).
Reference File	Shows the path and file name of the current Reference waveform. Double-tap the field to open the on-screen keyboard to enter or edit the path to open a different waveform file. Tap the down arrow icon to list the 20 most recently accessed reference waveform files.
Browse	Opens the Browse Waveform File dialog. Use this dialog to navigate to and select a waveform file to load.
Sample Rate, Record Length	Readout-only text that show the sample rate and record length values of the reference waveform. These fields are replaced with **Center Frequency**, **Span**, and **RBW** readouts for spectrum reference waveforms.
Deskew	Changes the horizontal position of the reference waveform. Use this function to align the reference waveform to meet your measurement needs. This field is not displayed for spectrum reference waveforms.

Table continued…

Field or control	Description
Set to 0	Sets the reference waveform deskew value to zero. This field is not displayed for spectrum reference waveforms.
Selected Frame	Sets the frame to view of a recalled Fast Frame waveform file. This control is only available if the recalled waveform includes Fast Frame acquisitions. ⚠ **Note:** This control has nothing to do with the selected frame of current acquired data. Adjusting this field does not change the selected frame in acquisition data. Conversely, adjusting the selected frame in the acquisition data does not change the selected frame of the reference waveform. Navigating through the frames of a reference waveform does not require the Fast Frame feature to be active. If more than one fast frame reference waveform is active, the Selected Frame in each waveform's configuration menu is independent of the others.

Recall configuration menu (Ref waveform configuration menu)

Use this menu to locate and load a reference waveform file.

Prerequisite: a Ref badge must be present on the Settings bar. See *Add a math, reference, or bus waveform*.

To open the Recall configuration menu:

1. Double-tap a **Ref** badge on the Settings bar.
2. Tap **Browse** to open the Recall configuration menu.

Recall configuration menu (Ref configuration menu) fields and controls

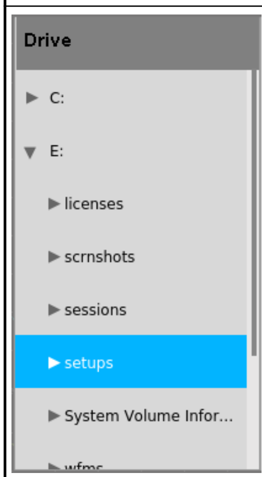
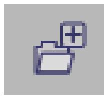
Field or control	Description
Look in	Shows the current directory path and file name. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.
	The Drive column lists the directory structure, including any network-connected drives. Use to quickly navigate to a location. Tap to list the contents of the directory in the file list area. Double-tap an item, or tap the small arrow to the left of the directory, to display the subdirectories under it. Double-tap again to close that directory structure. If there is a scroll bar, drag the list up and down to show more entries.

Table continued…

Field or control	Description
	Use the arrow buttons to navigate the directory structure. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder. The Up arrow navigates up one level from the current folder.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File Name	Lists the selected file name. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.
Files of Type	Use to select the file format you want to open. Tap the field to show a list of all file extension types that the instrument can read for the selected file type. The files column only lists files of the specified type.
Cancel	Cancels any changes, closes the menu, and returns to the prior menu without loading a file.
OK	Closes the dialog, returns to the prior menu, and loads and displays the specified file.

Search configuration menu

Use the Search configuration menu to define conditions that you want to search for on a channel or waveform signal. Each occurrence of the search condition is marked on the signal with a triangle along the top of the waveform slice or view.

To open the Search menu, double-tap on a **Search** badge in the Results bar.

See the following links for information on the search type menus.

Bus Search configuration menus on page 250

Edge Search configuration menu on page 305

Logic search configuration menu on page 306

Pulse Width Search configuration menu on page 308

Rise/Fall Time Search configuration menu on page 310

Runt Search configuration menu on page 311

Setup and Hold Search configuration menu on page 312

Timeout Search configuration menu on page 314

Window Search configuration menu

DDR Read Search configuration menu on page 316

DDR Write Search configuration menu on page 318

DDR Read & Write Search configuration menu on page 320

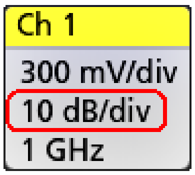
Spectrum View menus

Use the Spectrum View menus to activate and configure frequency-domain spectrum traces for analog channel time-domain signals.

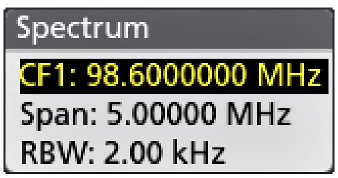
Spectrum View and badges

Spectrum View makes changes to channel badges and adds a new Spectrum badge.

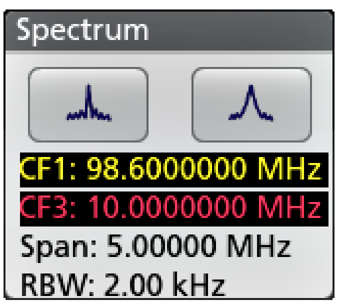
When a channel is set to show a **Spectrum View** trace, the second row of the channel badge changes from the probe symbol or input impedance readout to the vertical units of that channel's spectrum trace.



When the first spectrum trace is enabled for any analog channel, the instrument opens the **Spectrum View** window and adds the **Spectrum** badge to the **Settings Bar**. The first line of the Spectrum badge shows the center frequency of the selected spectrum trace.

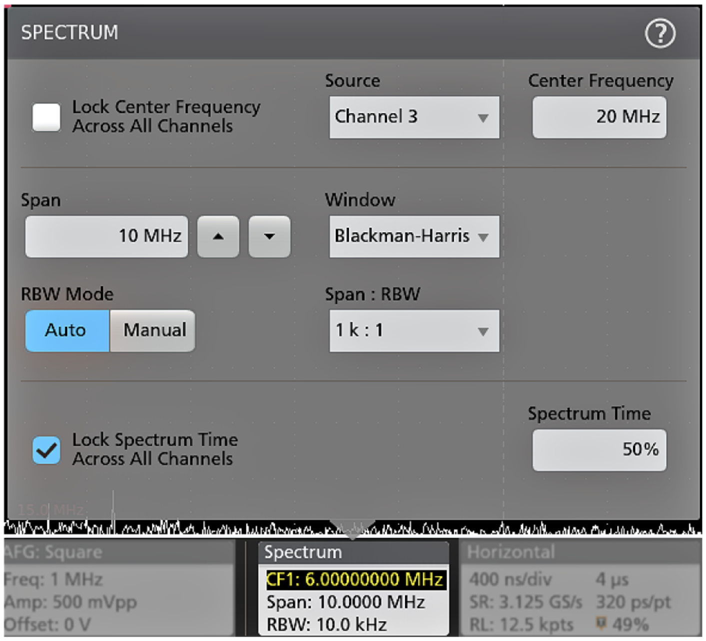


Tap the **Spectrum** badge to show the **Span** buttons to quickly increase or decrease the Spectrum trace span. Changing the span for one Spectrum trace changes the span for all traces.



When more than one Spectrum trace is enabled and at least two center frequencies are defined, the Spectrum badge lists the center frequency for each enabled Spectrum trace.

Double-tap the **Spectrum** badge to open the **Spectrum** menu to set Span, RBW, and other parameters.

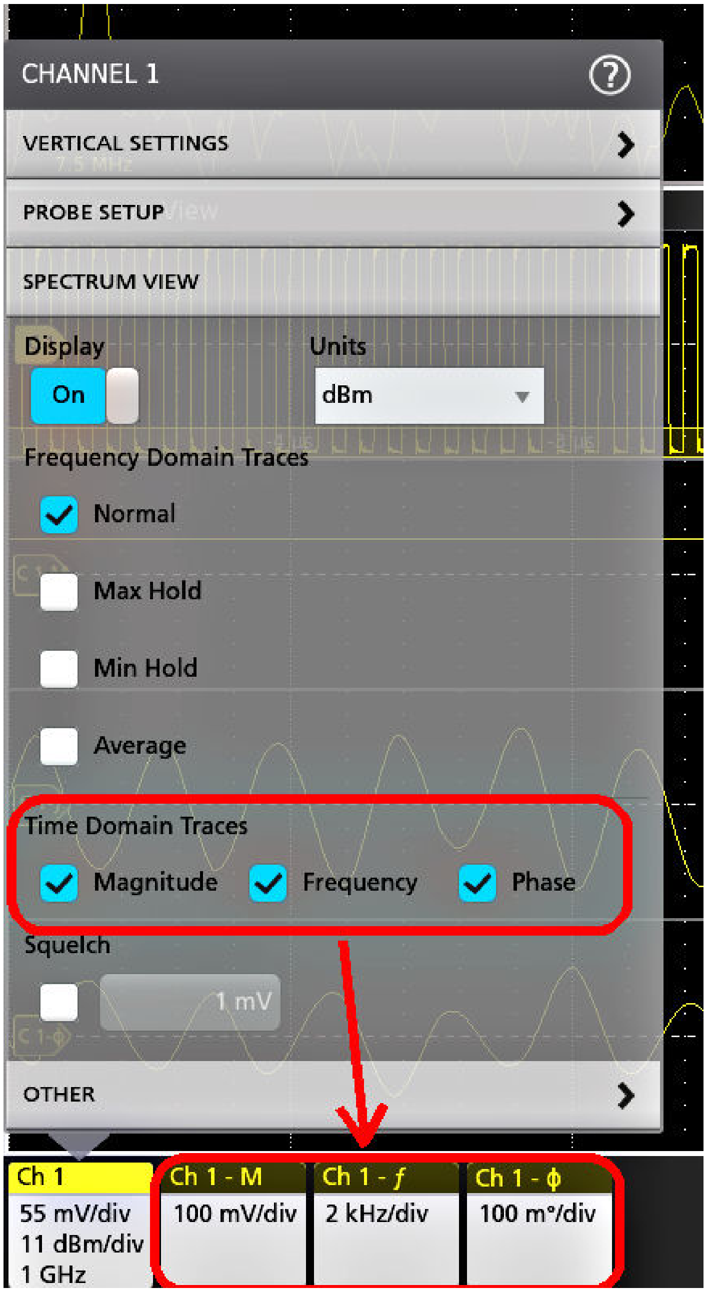


1303-010

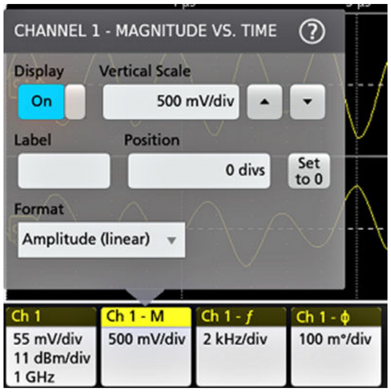
The spectrum **Span**, **RBW** (resolution bandwidth), and (FFT) **Window** settings apply to all spectrum traces. The **Center Frequency** and **Spectrum Time** can be set for each Spectrum trace. See the *Spectrum badge menu*.

Spectrum View Time Domain trace badges

The **Spectrum View Time Domain** traces (**Magnitude Vs. Time**, **Frequency Vs. Time**, **Phase Vs. Time**) are shown when you enable these functions in the **Spectrum View** panel of a **Channel** badge configuration menu.



Each time domain trace has its own badge for each channel, and each badge has its own configuration menu. Open a badge menu by double-tapping on the badge.



Spectrum View Cursors menu

Use the Spectrum View cursors to take manual measurements on spectrum signals.

To open the **Spectrum View Cursors** menu, double-tap on a cursor or its readout in the **Spectrum View** window.

Cursors menu (Spectrum View), fields and controls

Field or control	Description
Display	Tap to toggle the spectrum cursor display **On** or **Off**.
Source Waveform	Sets the cursor trace mode (the trace(s) on which to display the cursors). **Same** places both cursors on the same trace. **Split** allows each cursor to be on a different trace.
Source	Moves the spectrum trace cursors and readouts to the selected source (default) or a specific selected from the list. **Selected Waveform** lets you tap the screen or use a mouse to quickly move the cursors to a selected trace (**Overlay** mode) or slice (**Stacked** mode). Only available when **Source Waveform** = **Same**.
Cursor A Source, Cursor B Source	Sets the waveform sources for Cursor A and B when in split cursor mode. Only available when **Source Waveform** = **Split.**
Cursor A X-Position	Sets a specific x-axis (frequency) position for Cursor A using the multipurpose knob or the virtual keypad.
Cursor B X-Position	Sets a specific x-axis (frequency) position for Cursor B using the multipurpose knob or the virtual keypad.
Readout	Sets the cursor readouts to show **Absolute** or **Delta** values. The default value is **Absolute.** **Delta** readouts are relative to the Reference Marker.

Spectrum cursors guidelines

- **Spectrum View** only supports waveform cursors.
- Cursor readouts only appear in the selected spectrum slice. The cursor bar extends through all slices (similar to waveform cursors in **Stacked Waveform View** slices).

- Cursors maintain their screen position if you change the center frequency or span.
- Cursor measurements are taken on the currently selected trace (**Normal**, **Average**, **Max Hold** or **Min Hold**) which is indicated by the MANm trace handle. See *Spectrum trace handle and trace types* on page 117.
- Cursor A becomes the Reference marker. The cursor A readout shows Frequency, Amplitude and Noise Density values. The automatic peak markers still show the same number of peaks as before, but they use the cursor A reference marker for delta measurements.
- The cursor B readout depends on the Absolute/Delta mode:
  - When cursor readouts are set to **Absolute** mode, the cursor B readout shows Frequency, Amplitude, and Noise Density values.
  - When cursor readouts are set to **Delta** mode, the cursor B readout shows the delta Frequency from cursor A, the delta Amplitude from cursor A, and the Phase Noise of cursor B relative to cursor A (dBc/Hz).
  - Split cursors readouts only display frequency readouts when the cursor sources are using different vertical units.

Spectrum trace handle and trace types

The spectrum trace handle provides details on the trace reference level, the source channel for the trace, and which trace types are displayed and selected.

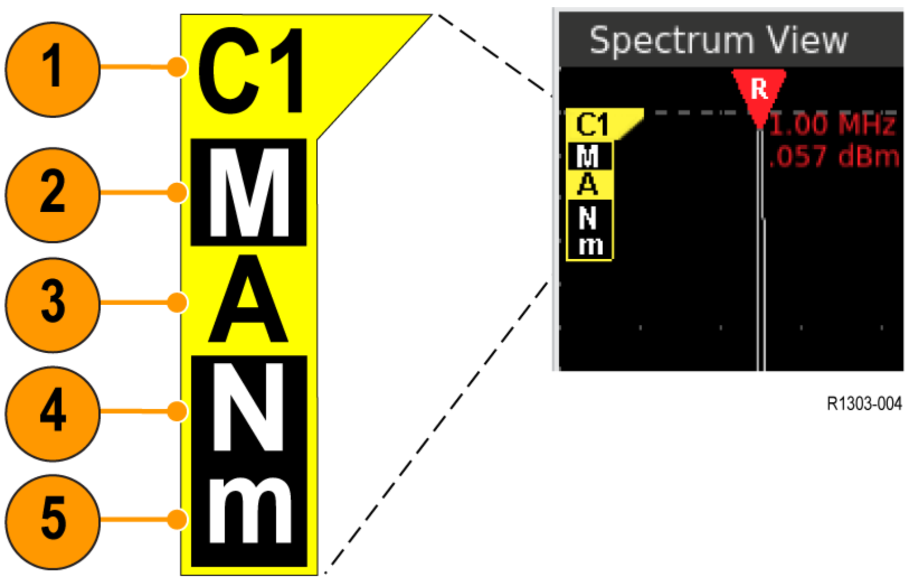


Figure 30: The Spectrum View trace handle elements

1. The trace's source channel (at top of the handle) and trace reference level (top edge of the handle, with dashed line extending across the screen). Spectrum View shows the spectrum traces relative to a Reference Level reference point. The color of the handle is the same as the source channel.

   If the Reference level is outside of the displayed trace (above or below the graticule), the handle is drawn with the channel source text pointing toward the Reference level.
2. A capital **M** indicates that the maximum trace is enabled. If the maximum trace is not enabled, this letter is not shown.
3. A capital **A** indicates that the average trace is enabled. If the average trace is not enabled, this letter is not shown.
4. A capital **N** indicates that the normal trace is enabled. If the normal trace is not enabled, this letter is not shown.
5. A small **m** indicates that the minimum trace is enabled. If the maximum trace is not enabled, this letter is not shown.

The above image shows all traces enabled and the **A** (**Average**) trace selected. Highlighting around a letter indicates that that trace type is selected.

Trace handle and trace type guidelines

There are important distinctions between enabled (displayed) and selected traces:

- An enabled trace is a trace type that is enabled in the **Spectrum View** panel of the source channel badge and is shown in the spectrum view window. If the trace type is not enabled, its corresponding letter is not shown in the trace handle and there is no corresponding trace shown on the screen.
- A selected trace (highlighted around the letter) is the trace that is used for marker and cursor readouts. Only one trace type can be selected at a time. Double-tap the trace handle to change the selected trace type.
- You can enable and display any combination of trace types for each spectrum trace.
- Disabling all trace types causes the spectrum view for that channel to close.
- If there is only one spectrum trace shown in the **Spectrum View**, and you disable all trace types or turn off the spectrum function for that channel, the **Spectrum View** window closes.

Frequency Vs. Time badge menu

Use this menu to set the vertical scale, label, and the trace offset position of the Spectrum View Frequency vs Time trace.

To open the **Frequency Vs. Time** badge menu, double-tap the badge.

Frequency Vs. Time menu fields and controls

Field or control	Description
Display	Tap to toggle the trace display **On** or **Off**. Each Frequency Vs. Time trace is drawn in its own slice (stacked mode).
Vertical Scale	Sets the vertical scale. Use the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Label	Adds a label to the trace. Double-tap the field to use the virtual keyboard. Drag the label to move it.
Position	Sets the vertical position of the trace. Use the multipurpose knob, or double-tap to bring up the virtual keypad, to change the trace position.

Magnitude Vs. Time badge menu

Use this menu to set the vertical scale, label, trace offset position, and the vertical scale format of the Spectrum View Magnitude Vs. Time trace.

To open the **Magnitude Vs. Time** badge menu, double-tap the badge.

Magnitude Vs. Time menu fields and controls

Field or control	Description
Display	Tap to toggle the trace display **On** or **Off**. Each Magnitude Vs. Time trace is drawn in its own slice (stacked mode).
Vertical Scale	Sets the vertical scale. Use the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Label	Adds a label to the channel display using the virtual keypad. Drag the label to move it.

Table continued…

Field or control	Description
Position	Sets the vertical position of the trace. Use the multipurpose knob, or double-tap to bring up the virtual keypad, to change the trace position.
Set to 0	Sets the channel signal vertical offset to 0 units (centers the 0 unit level vertically in the trace).
Format	Sets the vertical scale mode to **Amplitude (linear)**, **Power (linear)**, or **Power (log)**.

Phase Vs. Time badge menu

Use this menu to set the vertical scale, label, trace ofset position, and the vertical scale format of the Spectrum View Phase Vs. Time trace.

To open the **Phase Vs. Time** badge menu, double-tap the badge.

Phase Vs. Time menu fields and controls

Field or control	Description
Vertical Settings panel	
Display	Tap to toggle the trace display **On** or **Off**. Each Phase Vs. Time trace is drawn in its own slice (stacked mode).
Vertical Scale	Sets the vertical scale. Use the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Label	Adds a label to the trace. Double-tap the field to use the virtual keyboard. Drag the label to move it.
Position	Sets the vertical position of the trace. Use the multipurpose knob, or double-tap to bring up the virtual keypad, to change the trace position.
Phase Wrap	Enables the **Phase Wrap** mode and opens a field in which to set the Phase Wrap value in degrees (default value is 180°). Phase Wrap **On** constrains the phase values in the **Phase vs. Time** waveform to be within ± the specified limit. Calculated phase values that are below (or above) the limit are *wrapped* by repeatedly adding (or subtracting) 360° until they are within range. This can create a discontinuity in the output waveform but all the phase values will be in the default display position and scaling range. Phase Wrap **Off** would be for applications where the discontinuity is undesirable.
Phase Reference panel	
Master Reference	Sets the channel used for this trace as a master phase reference for all other Phase Vs. Time traces. This function is only available when all source spectrum channels use the same center frequency.

Table continued…

Field or control	Description
Set Phase Reference Using	Sets the phase reference point to either the **Trigger Point** or the **Cursor A** position. When **Set Phase Reference Using** is set to **Trigger Point**, the reference source is based on the phase at the trigger point. When **Set Phase Reference Using** is set to **Cursor A**, use Cursor A to set the **Phase Reference** field value. This is intended for the user who can use a cursor to visually identify a phase reference in the waveform and set the reference relative to the cursor position.
Phase Reference	Sets or shows the reference phase angle to use to draw the Phase Vs. Time trace. When **Set Phase Reference Using** is set to **Trigger Point**, this field shows the phase angle at the signal trigger point. When **Set Phase Reference Using** is set to **Cursor A**, use Cursor A to set the **Phase Reference** field value, or directly enter a phase value in the field.
Set Now	Sets the **Phase Reference** to the specified value. You must tap this button to enter and set the phase value to update the trace with this setting. This control is only present when **Set Phase Reference Using** is set to **Cursor A**.

Trigger configuration menu overview

Use the Trigger menu to define the channel or waveform signal conditions on which to trigger the oscilloscope. The trigger event establishes the time-reference point in the waveform record. All waveform record data is located in time with respect to the trigger point.

To quickly change the trigger source from one channel to another when using a single source trigger type, single-tap on the **Trigger** badge. Select the new trigger source from the list. Single source trigger types include **Edge**. When the trigger type is **Edge** selecting a new source also initiates a **Set to 50%** operation on the new source.

Edge Trigger configuration menu

Use the edge **Trigger** menu to trigger the instrument when a signal rises and/or falls through a specified level.

To open the Edge trigger menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Edge**.

Settings panel (Edge Trigger configuration menu) fields and controls

Field or control	Description
Trigger Type	Set to **Edge**.

Table continued…

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control. **AC Line** source (**Edge** trigger): The instrument uses the AC line to generate a trigger based on the power line frequency. The trigger level is fixed at zero volts. This source is often used to look at signals related to the power line frequency (for example, signals from devices such as lighting equipment and power supplies). Because the instrument generates the trigger from the power line, you do not have to use a channel input. **AUX In**: The MSO58LP and 6 Series MSO also have an **AUX In** (auxiliary) edge trigger source. **AUX In** provides an extra trigger signal source when you need to use the input channels for other signals. For example, you might want to trigger on a clock while displaying related logic signals. The **AUX In** input connector is not compatible with most probes, and you cannot display the auxiliary trigger signal in the Waveform view.
Coupling	Set the conditioning to apply to the source signal trigger circuit from the source signal. **DC** coupling passes all input signals directly to the trigger circuitry. **HF Reject** coupling attenuates signals above 50 kHz before passing the signal to the trigger circuitry. **LF Reject** coupling attenuates signals below 50 kHz before passing the signal to the trigger circuitry. **Noise Reject** coupling provides stable triggering by increasing the trigger hysteresis. Increased hysteresis reduces the trigger sensitivity to noise so may require greater signal amplitude.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Slope	Sets the signal transition direction to detect. (rising, falling, or either direction).

Mode & Holdoff panel (Edge Trigger configuration menu) fields and controls

<table>
<tr><th>Field or control</th><th>Description</th></tr>
<tr><td>Trigger Mode</td><td>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:

Auto trigger mode enables the instrument to acquire and display a waveform even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting.

When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen.

If valid triggers occur, the display will become stable.

Normal trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.</td></tr>
<tr><td>Force Trigger</td><td>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</td></tr>
<tr><td>Holdoff</td><td>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.

Random sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal.

Time sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.</td></tr>
<tr><td>Holdoff Time</td><td>Tap the Holdoff Time field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.</td></tr>
<tr><td>Act on Trigger</td><td>Tap the Act on Trigger button to configure the actions the instrument must take when a trigger event occurs. See Act On Event configuration menu on page 128 for the available fields and controls.</td></tr>
<tr><td>Trigger Frequency Counter</td><td>Turn On to display the trigger event frequency in the Trigger badge.

The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT.

Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</td></tr>
</table>

Virtual Keyboard

Use the onscreen virtual keyboard to enter textual information such as a file path, file name, label text, or on-screen callout.

To access the virtual keyboard, double-tap in a menu or dialog text input box. Enter your text and tap **Enter** to close the keyboard and add your text to the menu or dialog field.

Tap **ESC**, **Cancel**, or anywhere outside the keyboard, to dismiss it without adding text to the input box.

Single-tap in the keyboard text field to position the insertion cursor at that location. Double-tap to select individual words. Triple-tap to select all text in the field.

Touch and drag the title bar to move the keyboard on the screen.

Virtual Keypad

Use the virtual Keypad to enter numeric values and units for settings.

To open the virtual keypad, double-tap inside a field that requires numeric values.

Virtual Keypad fields and controls

Field or control	Description
Clear	Clears all values from the input entry field.
Exp	Lets you enter exponential notation entries.
Max	Enters the maximum value allowed for this setting.
Min	Enters the minimum value allowed for this setting.
Bksp ←	Deletes characters to the left of the insert text marker position.
Enter	Closes the number pad and assigns the entered value to the field.
±	Tap the button to set a numeric value to a positive (default) or negative value.
Unit buttons	Use to set the units of the entered value.

Visual Trigger Area configuration menu

Use the Visual Trigger Area menu to edit visual trigger area parameters. Double tapping a Visual Trigger area opens the Area configuration menu and makes the related source the selected source.

To open the Visual Trigger **Area** menu, double-tap on a visual trigger area.

To create a visual trigger area, see *Create Visual Trigger areas*

Area Settings panel fields and controls

Field or control	Description
Source	Lists the available source signals for which to create an area. The default value is the signal source of the selected area.
Shape	Lists the current shape type, and also lets you change the current shape to a specified shape. Changing a shape defines the minimum rectangle that includes all vertices of the current area, and then does a best-fit approach to create the specified shape. If you made changes to a default shaped area that results in an area that no longer meets the default shape definition, that area is listed as a Custom shape in the menu.

Table continued…

Field or control	Description
Waveform Must Be	Sets the logic condition for how the area logic equation interprets when a waveform intersects an area. **In (✓)**: A waveform must intersect this area for the area logic to be true. **Out (X)**: A waveform must not intersect this area for the area logic to be true. **Don't Care (?)**: Don't care if a waveform intersects the area. This setting lets you ignore an area during visual trigger development, instead of having to delete and recreate areas.
Height	Sets the area height, in amplitude units, between the topmost vertex and the bottommost vertex.
Vertical Center	Sets the area vertical center, in amplitude units, as the point halfway between the topmost vertex and bottommost vertex.
Flip Vertical	Flips the area vertically around its Vertical Center value.
Width	Sets the area width, in time units, between the leftmost vertex and the rightmost vertex.
Horizontal Center	Sets the area horizontal center, as time units, as the point halfway between the leftmost vertex and rightmost vertex.
Flip Horizontal	Flips the area horizontally around its Horizontal Center value.
Rotate	Rotates the area in units of degrees, from 0° to 360°. The rotation is an absolute angle measurement referenced from 0°, where 0° is the position of the area when it was first created. For example, if you rotate the area 40°, and then rotate it again with 20°, the resulting area rotation is 20°. The area height and width are relative to the current area orientation, and automatically change places as needed when the area is rotated.

Edit Vertices panel fields and controls

Field or control	Description
Define Area	A table that lists the X (time) and Y (amplitude) values for each vertex (point) of the area. Use the scroll bar to show points if there are more than nine points in the area. Selecting a row in the table highlights the associated vertex on the area. Use the Multipurpose knobs to change the value of the X or Y settings, or double-tap a setting and enter the value directly.
Insert Point	Inserts a new row above the selected row and creates a new vertex on the area shape. The new vertex is halfway between the vertices defined in the prior row and the following row in the table.
Delete Point	Deletes the currently selected point, keeps the row selected, and moves all rows below it up one row. The **Delete Point** button is not available when a triangular area is selected.

Table continued…

Field or control	Description
Reset Points	Deletes all but three data points from the table. The remaining three data points are set to a default triangle, set to two divisions for height and width, and centered in the waveform area.

Note: Redo and Undo are available for most Edit Vertices panel controls.

Visual trigger right click menu

See *Right click menu functions associated with visual trigger areas* on page 401.

Right click menu functions associated with visual trigger areas

The following functions are available when you right click on a visual trigger area.

Field or control	Description
In	Sets the area's Waveform Must Be control to **In**.
Out	Sets the area's Waveform Must Be control to **Out**.
Don't Care	Sets the area's Waveform Must Be control to **Don't Care**.
Rectangle	Sets the area's shape to a rectangle
Triangle	Sets the area's shape to a triangle.
Trapezoid	Sets the area's shape to a trapezoid.
Hexagon	Sets the area's shape to a hexagon.
Create Duplicate	Creates a new area with the same characteristics as the current area, but offset from the original area by X+50 and Y+50.
Configure Area	Opens the Area configuration menu for the selected area.
Delete Area	Deletes the selected area.

Visual Trigger Area Combinatorial Logic menu

Use the Area Combinatorial Logic menu to describe the logic conditions required for a true condition of all associated areas.

To open the Visual Trigger **Area Combinatorial Logic** menu:

1. Double-tap the **Trigger** badge.
2. Tap the **Visual Trigger** panel.
3. Double-tap the **Area Combinatorial Logic** field to open the **Area Combinatorial Logic** menu.

Note: There must be at least two areas before you can open the **Area Combinatorial Logic** menu.

Area Combinatorial Logic menu fields and controls

Field or control	Description
Logic expression field	Displays the area combination logic for all areas associated with the same signal source. All areas are ANDed by default, including areas set as Don't Care. Use the buttons in the menu to edit or define the area combination logic, or double-tap the field and enter the logic expression directly. You must tap the **Enter** button to check and save the logic expression to make it valid.
Clear	Deletes all text in the logic expression field.
Move insertion point	Moves the insertion point left or right in the logic expression field.
Logic action buttons	Inserts the selected logic function or grouping parenthesis at the insertion point in the logic expression field.
Bksp	Deletes the character to the left of the insertion point.
Insert Area buttons	Inserts the selected area text into the logic expression. Only available areas are listed. If more than nine areas are defined, use the scroll bar to view area buttons greater than A9.
Cancel	Closes the menu without saving any changes.
Enter	Checks the syntax of the logic expression. If it is valid, the changes are saved and the menu is closed. If the logic expression syntax is not valid, or the logic expression field is empty, the instrument displays an error message, the menu remains open, and no changes are saved.

Waveform View configuration menu

Use this menu to configure the Waveform View to set display mode (Stacked or Overlay), waveform interpolation method, persistence, and other parameters.

To open the Waveform View menu, double-tap anywhere in the Waveform View screen.

Waveform View menu fields and controls

Fields or controls	Description
Display Mode	Sets how waveforms are shown on the screen. **Overlay** mode displays all waveforms on the screen, overlaid on each other. This is the traditional way that oscilloscopes displayed waveforms, and lets you overlay waveforms to do direct comparisons of waveform shape. The grid vertical position and scale values change for each selected waveform. **Stacked** mode draws each waveform in an individual section, or slice, of the screen, stacked one on top of the other. This lets you view each waveform uncluttered by overlaying waveforms. Each waveform slice displays its own vertical scale units. A trigger level indicator in a slice indicates that that waveform is the trigger source.

Table continued…

Fields or controls	Description
Interpolation	Selects the method used to display record points between sampled points. **Sin(x)/x** computes record points along a curve between the actual acquired samples. This form of interpolation is useful when acquiring rounded waveforms such as sine waves. It is good for general-purpose uses but may introduce overshoot or undershoot in signals with fast rise times. This interpolation is also useful for looking at high-frequency signals, especially where the frequency components are just below the Nyquist frequency. **Linear** computes record points between actual acquired samples using a straight-line fit. This interpolation is useful for measuring waveforms with fast rise times, such as pulse trains.
Persistence	Sets the length of time data points are displayed on screen before being erased. **Off** sets the record points to appear for the current acquisition only. **Infinite** continuously accumulates record points on the waveform until you change one of the acquisition display settings or clear the acquisition memory. Use infinite persistence for displaying record points that may occur outside the normal acquisition envelope. **Variable** lets you specify a time length to retain data points on screen. Each record point decays independently according to the time interval. **Auto** sets the **Waveform Intensity** field to control the persistence time.
Variable Persistence Time (Persistence = Variable)	Sets the length of time data points remain displayed. Tap the field and use the A knob to adjust, or double-tap and use the keypad to set a time.
Waveform Style	Sets how waveforms are drawn on the screen. **Vectors** draws waveforms with lines between record points. **Dots** draws waveform record points as dots on the screen, and adds crosshair markers to real sampled points.
Waveform Intensity	Sets the brightness of the waveform. Tap the field and use the A knob to set the waveform intensity.
Graticule Style	Sets the type of graticule to display. **Grid** shows a traditional grid on the instrument display. This is the default grid mode. **Time** displays vertical graticules for just the horizontal (time) scale units. **Full** displays the same style graticule lines for both horizontal and vertical scale units **None** turns off the graticule, including the vertical scale readouts.
Graticule Intensity	Sets the brightness of the graticule. Tap the field and use the A knob to set the graticule intensity.

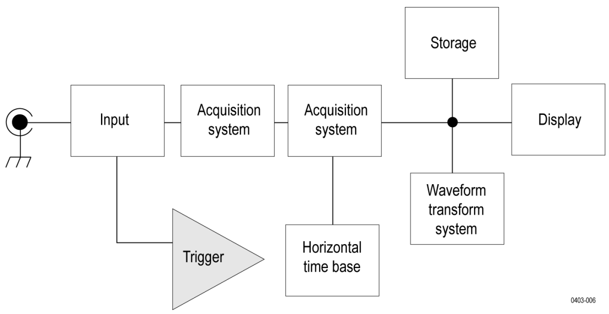
Trigger concepts

Triggering concepts

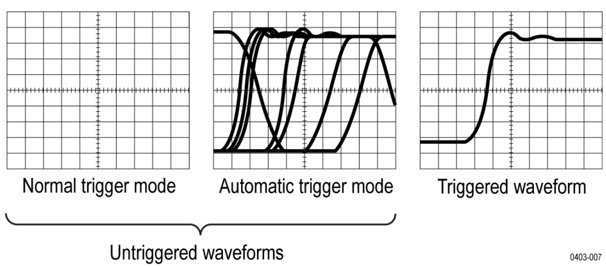
Overview

User selected trigger conditions are used to capture waveforms for measurement and analysis.

The next figure shows how triggers fit into the overall instrument operation.



Triggers help you capture meaningful waveforms to display on screen. This instrument has simple edge triggers as well as a variety of advanced triggers.



The trigger event

The trigger event establishes the time-zero point in the waveform record. All waveform record data are located in time with respect to that point. The instrument continuously acquires and retains enough sample points to fill the pretrigger portion of the waveform record (that part of the waveform that is displayed before, or to the left of, the triggering event on screen).

When a trigger event occurs, the instrument starts acquiring samples to build the posttrigger portion of the waveform record (displayed after, or to the right of, the trigger event). Once a trigger is recognized, the instrument will not accept another trigger until the acquisition is complete and the holdoff time has expired.

Trigger sources

The trigger source provides the signal that triggers acquisition. Use a trigger source that is synchronized with the signal that you are acquiring and displaying.

You can derive your trigger from the following sources:

- **Input channels**. Analog input channels are the most commonly used trigger sources. You can select any of the input channels. The channel that you select as a trigger source will function whether it is displayed or not.
- **Digital channels**. These sources are available if you have a digital probe connected to a FlexChannel. You can select any combination of digital channels.

- **Bus**. This source is used to trigger a parallel bus or a serial bus. You can include any combination of analog, math, or digital channels to build a parallel bus, or use any channel as a component in a serial bus.

Trigger types

The trigger types include:

Edge. This is the simplest and most commonly used trigger type, used with both analog and digital signals. An edge trigger event occurs when the trigger source passes through a specified voltage level in the specified direction (rising or falling signal voltage). When the edge trigger source is set to **AC Line**, the instrument uses the instrument's AC power line signal to generate a trigger based on the power line frequency. The 6MSO series and the MSO58LP instruments also have an **AUX In** connector that is available as another source for the Edge trigger.

Trigger modes

The trigger mode determines how the instrument behaves in the absence of a trigger event:

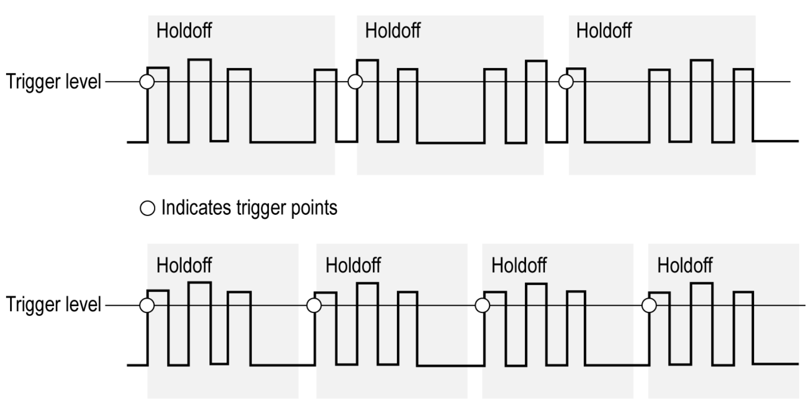
- **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the instrument does not acquire a waveform, and the last waveform record acquired remains "frozen" on the display. If no last waveform exists, no waveform is displayed.
- **Auto** trigger mode enables the instrument to acquire a waveform even if a trigger does not occur. Auto mode uses a timer that starts after a trigger event occurs. If another trigger event is not detected before the time out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting.

Auto mode, when forcing triggers in the absence of valid triggering events, does not synchronize the waveform on the display. In other words, successive acquisitions are not triggered at the same point on the waveform; therefore, the waveform will appear to roll across the screen. If valid triggers occur, the display will become stable.

Trigger holdoff

Trigger holdoff can help stabilize triggering. When the instrument recognizes a trigger event, it disables the trigger system until acquisition is complete. In addition, the trigger system remains disabled during the holdoff period that follows each acquisition. Adjust holdoff to obtain stable triggering when the instrument is triggering on undesired trigger events.

A digital pulse train is a good example of a complex waveform. Each pulse looks like any other, so many possible trigger points exist. Not all of these will result in the same display. The holdoff period allows the instrument to trigger on the correct edge, resulting in a stable display.



At the longer holdoff time for the top waveform, unstable triggering occurs. With a shorter holdoff set for the bottom waveform, triggers all occur on the first pulse in the burst to remedy the unstable trigger.

The Holdoff setting range is 0 s (minimum holdoff available) to 10 s (maximum holdoff available). For more information on how to set holdoff, see *add hyperlink* . If you select Auto holdoff, the instrument selects a holdoff value for you. When

Trigger Holdoff is set to Random, the instrument delays the trigger a random amount of time between triggers. This means that successive acquisitions are unrelated to the previous trigger signal.

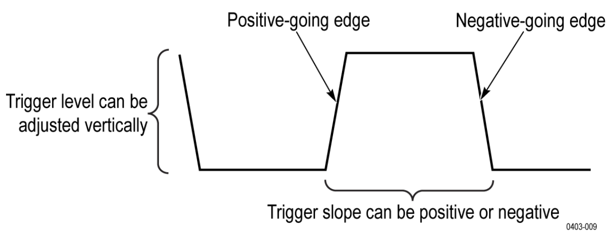
Trigger coupling

Trigger coupling determines what part of the signal is passed to the trigger circuit. Edge triggering can use all available coupling types: DC, Low Frequency Rejection, High Frequency Rejection, and Noise Rejection. All of the advanced trigger types use DC coupling only.

- **DC**. This coupling passes all input signals to the trigger circuitry.
- **HF Reject**. This coupling attenuates signals above 50 kHz before passing the signal to the trigger circuitry.
- **LF Rej**. This coupling attenuates signals below 50 kHz before passing the signal to the trigger circuitry.
- **Noise Rej**. This coupling provides stable triggering by increasing the trigger hysteresis. Increased hysteresis reduces the trigger sensitivity to noise but may require greater signal amplitude.

Trigger slope and level

The slope control determines whether the instrument finds the trigger point on the rising or the falling edge of a signal. The level control determines where on that edge the trigger point occurs. See the next figure.



Trigger position in waveform record

Trigger position is an adjustable feature that defines where the trigger occurs on the waveform record. It lets you choose how much the instrument acquires before and after the trigger event. The part of the record that occurs before the trigger is the pretrigger portion. The part that occurs after the trigger is the posttrigger portion. A longer posttrigger period may be useful when you want to see the effects an event has on your system under test.

Pretrigger data can be valuable when troubleshooting. For example, if you are trying to find the cause of an unwanted glitch in your test circuit, you can trigger on the glitch and make the pretrigger period large enough to capture data before the glitch. By analyzing what happens before the glitch, you may uncover information that helps you find the source of the glitch.

Trigger delay

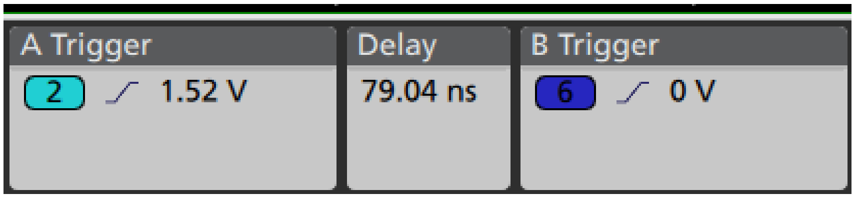
Use the Trigger Delay to trigger the instrument a specified period of time after the A trigger. After the A trigger arms the trigger system, the instrument triggers on the next B trigger event that occurs after the time that you specify.

You can trigger with the A trigger system alone or you can combine the A trigger with the B (Delayed) trigger to trigger on sequential events. When using sequential triggering, the A trigger event arms the trigger system, and the B trigger event triggers the instrument when the B trigger conditions are met.

A and B triggers can (and typically do) have separate sources. The B trigger condition can be based on a time delay or a specified number of counted events.

Advanced triggering

You can check the advanced trigger status in the settings bar. The readout indicates the trigger type and then shows sources, levels, or any other parameters that are important for the particular trigger type.



Bus triggering concepts

A bus trigger occurs when a supported instrument detects a bus pattern that you specify for a parallel bus, or a bus cycle you select for a serial bus.

You can set the instrument to trigger on a parallel bus when the instrument detects a match to the bus pattern, or when the instrument detects that the value on the bus is < or > the value of the bus pattern. The pattern can be in Binary or Hex format.

You can set the instrument to trigger on an SPI bus when the instrument detects an SS Active bus cycle or Data.

You can set the instrument to trigger on an I2C bus when the instrument detects a Start, Stop, Repeated Start, Missing Ack, Address, Data, or Addr + Data bus cycle or activity.

You can set the instrument to trigger on a USB bus when the instrument detects a Sync, Reset, Suspend, Resume, End of Packet, Token (Address) Packet, Data Packet, Handshake Packet, Special Packet, or Error bus cycle or activity.

You can set the instrument to trigger on an RS232 bus when the instrument detects a Start, End of Packet, Data, or Parity Error bus cycle or activity.

You can set the instrument to trigger on an CAN bus when the instrument detects a Start of Frame, Type of Frame, Identifier, Data, Id and Data, End of Frame, Missing Acq, or Bit Stuffing Error bus cycle or activity.

You can set the instrument to trigger on an LIN bus when the instrument detects a Sync, Identifier, Data, Identifier & Data, Wakeup Frame, Sleep Frame, or Error bus cycle or activity.

You can set the instrument to trigger on an FlexRay bus when the instrument detects a Start of Frame, Indicator Bits, Frame Id, Cycle Count, Header Fields, Data, Identifier & Data, End of Frame, or Error bus cycle or activity.

You can set the instrument to trigger on an Ethernet bus when the instrument detects a Start of Frame, MAC Address, MAC Length/Type, IP Header, TCP Header, Client Data, End of Packet, Idle, or FCS (CRC) Error bus cycle or activity.

You can set the instrument to trigger on an AUDIO bus when the instrument detects a Word Select, or Data bus cycle or activity.

For all the serial standard buses, you can also set the component threshold levels through the Bus Setup menu

Pulse width trigger concepts

A pulse width trigger occurs when the instrument detects a pulse that is inside or outside some specified time range. The instrument can trigger on positive or negative width pulses. Pulse width triggers can also be qualified by the logical state of other channels.

Timeout trigger

A timeout trigger occurs when the instrument does not detect an expected pulse transition within a user specified period of time, such as when a signal gets stuck either high or low. If the pulse transition occurs prior to a specified timeout time (the expected case), then no trigger results.

Runt trigger

A runt trigger occurs when the instrument detects a short pulse that crosses one threshold but fails to cross a second threshold before recrossing the first.

- You can set the instrument to detect any positive or negative runt pulse, or only those wider than a specified minimum width.
- Runt pulses can also be qualified by the logical state of other channels.

Window trigger

Use the Window trigger to trigger the instrument when the input signal rises above an upper threshold level or falls below a lower threshold level.

After setting these levels, you can specify whether you want to trigger the instrument as the signal is entering or leaving the threshold window. You can further qualify the trigger event in terms of time, or by the logical state of other channels.

Logic trigger concepts

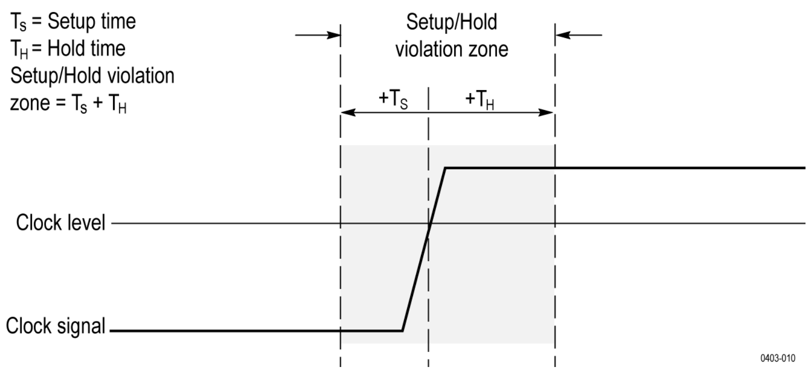
For various trigger types, touch Logic Qualification to toggle logic qualification On to trigger the instrument when the logic patterns are true. You can set each bit to be active High, Low, or Don't Care. You can also set the logic thresholds and define the logic (AND, OR, NOR, or NAND).

Setup and Hold trigger concepts

A setup/hold trigger occurs when a data signal changes state inside of the user specified setup and hold times relative to the clock. When you use setup/hold triggering, you define:

- The channel containing the logic input (the data source) and the channel containing the clock (the clock source)
- The direction of the clock edge to use
- The clocking level and data threshold that the instrument uses to determine if a clock or data transition has occurred
- The setup and hold times that together define a time range relative to the clock

Data that changes state within the setup/hold violation zone triggers the instrument. The next figure shows how the setup and hold times that you choose position the violation zone relative to the clock.



Setup/hold triggering uses the setup/hold violation zone to detect when data is unstable too near the time it is clocked. Each time trigger holdoff ends, the instrument monitors the data and clock sources. When a clock edge occurs, the instrument checks the data stream it is processing (from the data source) for transitions occurring within the setup/hold violation zone. If any occur, the instrument triggers with the trigger point located at the clock edge.

The setup/hold violation zone spans the clocking edge as shown above. The instrument detects and triggers on data that does not become stable long enough before the clock (setup time violation) or that does not stay stable long enough after the clock (hold time violation).

Rise/Fall time trigger concepts

Rise/Fall time triggering is based on the slope (change in voltage/change in time) of a pulse edge.

Use the Rise/Fall trigger to trigger the instrument on pulse edges that traverse between two thresholds at faster or slower rates than the specified time. You can set up the instrument to trigger on positive or negative edges. The trigger can be logic qualified.

Sequential (A B) trigger concepts

In applications that involve two or more signals, you may be able to use sequential triggering to capture more complex events. Sequential triggering uses the A (Main) trigger to arm the trigger system, and then uses the B (Delayed) trigger to trigger the instrument if a specific condition is met.

You can choose one of two trigger conditions:

- Trigger after a Delay. After the A trigger arms the trigger system, the instrument triggers on the next B-trigger event that occurs after the trigger delay time. You can set the trigger delay time with the keypad or a multipurpose knob.
- Trigger on the Nth event. After the A trigger arms the trigger system, the instrument triggers on the Nth B event. You can set the number of B events with the keypad or a multipurpose knob.

Note: The traditional delayed trigger mode called "Runs After" is controlled by the Horizontal Delay feature. You can use horizontal delay to delay acquisition from any trigger event, whether from the A trigger alone or from a sequential trigger that uses both the A and B triggers.

Visual Trigger concepts

Visual Trigger makes the identification of the desired waveform events quick and easy by scanning through all acquired analog waveforms and graphically comparing them to geometric shapes on the display. By discarding acquired waveforms that do not meet the graphical definition, Visual Triggering extends the oscilloscope's trigger capabilities beyond the traditional hardware trigger system.

Although Visual Trigger is similar in appearance to mask testing, where acquired waveforms are graphically compared to mask regions on the display, there is an important difference. Visual Trigger only displays waveforms that conform to the specified shape, so only conforming waveforms are measured.

Creating a visual trigger always begins with setting up the oscilloscope's hardware trigger system to acquire the waveforms in a standard trigger mode. The trigger can be as simple as an edge trigger or as complex as a pulse width, runt, logic multi-state trigger.

Once a standard trigger is set up, create visual trigger areas to refine the trigger condition. Each visual trigger areas is associated with a specific analog input channel, and you can define areas for multiple channels. By default, newly-created areas are rectangular, are associated with the selected channel, and are assigned the logic condition In when created.

Once areas are created on the oscilloscope's display, they can be re-positioned, re-sized, and/or edited interactively on the screen to create the required shape to define the visual trigger condition.

The visual trigger **Area** menu enables more precise editing of each Visual Trigger area. You can set exact coordinates for the vertices of each area in amplitude and time values, and assign the area to a specified signal source (channel). The menu also lets you set the logic condition for each shape (the waveform must be inside the area, outside the area, or don't care). Double-tap on an area to open the Area menu.

Finally, a logical equation is automatically generated when two or more areas are added to the display. This equation describes how Visual Trigger uses the areas to determine which acquired waveforms are displayed and which are discarded. For example, the equation A1 & A2 & A3 specifies that the waveform must meet the area logic condition for all three areas to display a waveform.

Visual trigger areas

How an area is shown depends on the instrument display mode. In Stacked mode, areas are outlined in cyan and have a semi-transparent cyan fill. The areas display the area number and the logic condition setting icon. Areas in Stacked mode do not include a channel label.

In Overlay mode, areas are outlined in the source channel's color and have a semi-transparent fill in the same color. The areas display the area number, the logic condition setting icon, and the channel with which the area is associated.

When zoom is active, areas are drawn in both the overview and zoomed view.

Area vertices all have specific time and amplitude values associated with them. These values are maintained as you change other controls. This means that the size of the areas change to when you change horizontal or vertical scale settings. The areas will also move up or down with vertical position changes. Any text that is displayed as part of the area is still displayed in the standard size, so you can still easily identify the location visual trigger areas.

If a visual trigger area is off the display, the instrument displays a directional arrow that points to the location of the visual trigger area. If zoom is on, the directional arrows are drawn in both the zoom overview and the zoomed view as needed.

Moving visual trigger areas

To move a visual trigger area, just touch and drag the area to a new position.

When dragging a visual trigger area while in Stacked or Overlay display mode, the area source remains the same.

When dragging a visual trigger area while in Stacked display mode, you cannot drag a visual trigger area from one waveform slice to another.

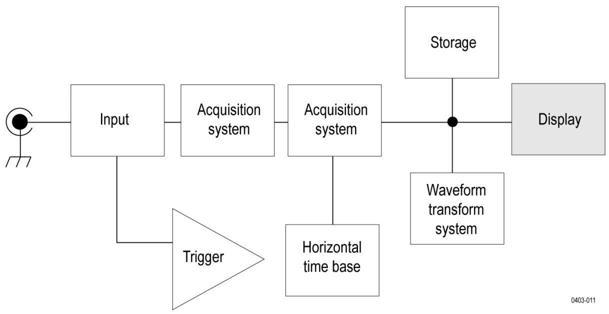
Visual trigger caveats

- Visual triggering is not compatible with Fast Acq or Fast Frame modes. If visual triggering is active and you turn on one of these modes, then visual triggering is turned off and vice versa.
- You cannot save or recall visual trigger definitions as individual items. However, you can save visual trigger definitions as part of **Session** and **Setup** files.

Waveform display concepts

Waveform display overview

This instrument includes a flexible, customizable display that lets you control how waveforms appear. The figure shows how the display features fit into the overall instrument operation.



The display shows analog, digital, math, reference and bus waveforms. The waveforms include channel markers, individual waveform graticule readings, and trigger source and level indicators. You can set the display to stack each waveform vertically in separate graticules, called 'slices' (the default mode), or overlay all the waveforms on the screen (traditional waveform view).

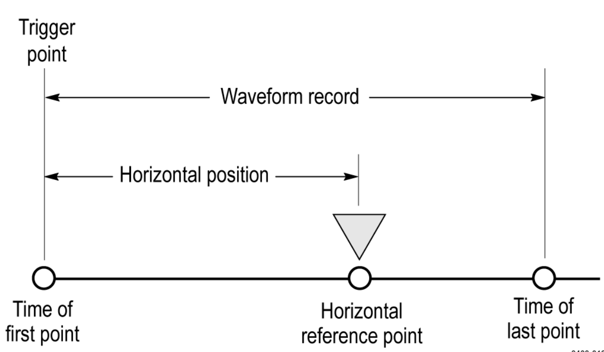
You can also add histogram, spectral, eye, and measurement results views (plots) for individual measurements. These plot views are contained within their own view window and can be moved within the overall view area.

Waveform preview mode

The waveform preview attempts to show what the next acquisition will look like when the acquisition is delayed due to slow triggers or long acquisition duration, or when the acquisitions have stopped. Waveform preview recalculates math waveforms, but does not represent changes in trigger levels, trigger modes, or different acquisition modes.

Horizontal position and the horizontal reference point

The time value you set for horizontal position is measured from the trigger point to the horizontal reference point. This is not the same as the time value from the trigger point to the start of the waveform record, unless you set the horizontal reference to 0%. See the next figure.



Measurement concepts

Measurement variables

By knowing how the instrument makes calculations, you may better understand how to use your instrument and how to interpret your results. The instrument uses a variety of variables in its calculations. These include:

Definition of Base and Top

Base is the value used as the 0% level in measurements such as fall time and rise time.

Top is the value used as the 100% level in measurements such as fall time and rise time. For example, if you set the 10% to 90% rise time, then the instrument calculates 10% and 90% as percentages of Top and Base, with Top representing 100%.

The exact value of Base and Top depends on which Base Top Method you select in the Reference Levels panel of a Measurement configuration menu. It also depends on if you set the reference level to be Global (applies to all measurements set as Global in the Reference Levels panel), or Local (just applies to the measurement that is set to Local).

Base, Top calculation methods

The Base Top calculation method is set in the Reference Levels panel of a Measurement configuration menu.

Auto is the default method, and automatically determines the best Base Top method to use. Most commonly sets the Base Top method to Histogram Mode.

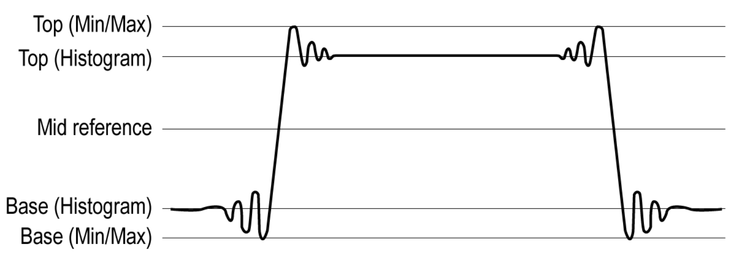
MinMax defines the 0% and the 100% waveform levels as the lowest value and the highest value samples of the waveform record. This setting is best for examining waveforms that have no large, flat portions at a common value, such as sine waves and triangle waves - almost any waveform except for pulses.

The MinMax method calculates the Top and Base values as follows:

Top = Max

and

Base = Min



Histogram Mean uses histogram analysis to calculate the mean or average value using all values above and below the waveform midpoint. Top is set to the mean high value, and Base is set to the mean low value. This setting is best for examining eye patterns and optical signals.

Histogram Mode uses histogram analysis to select the most common values either above or below the midpoint. Since this statistical approach ignores short-term aberrations (overshoot, ringing, and so on), Mode is the best setting for examining pulses.

The oscilloscope calculates the histogram-based *Top* and *Base* values as follows:

1. It makes a histogram of the record with one bin for each digitizing level.

2. It splits the histogram into two sections at the halfway point between *Min* and *Max* (also called *Mid*).
3. The level with the most points in the upper histogram is the *Top* value, and the level with the most points in the lower histogram is the *Base* value.

   If *Mid* gives the largest peak value within the histogram, the oscilloscope returns the *Mid* value for both *Top* and *Base* (this is probably a very low amplitude waveform).

   If more than one histogram level (bin) has the maximum value, the oscilloscope chooses the bin farthest from *Mid*.

This algorithm does not work well for two-level waveforms with greater than about 100% overshoot.

Histogram Eye Center uses histogram analysis of the amplitudes in the center of each bit (unit interval) while ignoring the waveform during bit transitions. The histogram sets the Top at the nominal high level and Base at the nominal low level. This is similar to the Histogram Mode, except it is less influenced by the shape of the waveform during transitions between bits.

HighRef, MidRef, LowRef

You set the various reference levels, through the Reference Levels tab of the Measure menu. They include:

High is the waveform high reference level (also HighRef). Used in all measurements. Typically set to 90%. You can set it from 0% to 100% or to a voltage level.

Mid is the waveform middle reference level (also MidRef). Mid reference levels are used in all measurements that need to find edges. Typically set to 50%. You can set it from 0% to 100% or to a voltage level.

Low is the waveform low reference level (also LowRef). Used in all measurements. Typically set to 10%. You can set it from 0% to 100% or to a voltage level.

High, mid and low reference levels can be set uniquely for each measurement source. Reference levels can also be set differently for rising edge detection and falling edge detection.

Other variables

The instrument also measures several values itself that it uses to help calculate measurements.

Record Length is the number of data points in the time base. You set it with the Horizontal menu Record Length item.

Start is the location of the start of the measurement zone (X-value). It is 0.0 samples unless you are making a gated measurement. When you use cursor gated measurements, it is the location of the left vertical cursor.

End is the location of the end of the measurement zone (X-value). It is (*RecordLength* – 1.0) samples unless you are making a gated measurement. When you use cursor gated measurements, it is the location of the right vertical cursor.

Hysteresis Is the hysteresis band of the waveform amplitude.

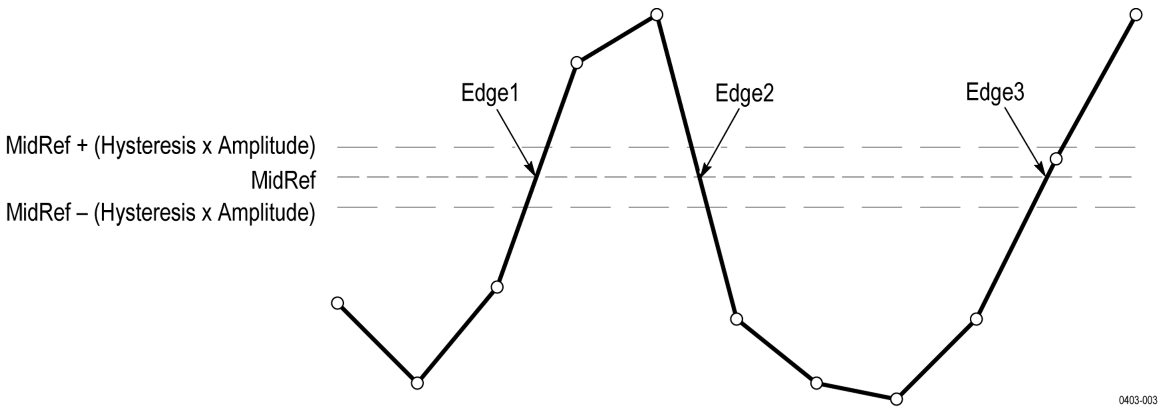
For example, once a crossing has been measured in a negative direction, the waveform data must fall below the hysteresis value of the amplitude from the *Mid* reference point before the measurement system is armed and ready for a positive crossing. Similarly, after a positive *Mid* reference crossing, waveform data must go above the hysteresis value of the amplitude before a negative crossing can be measured. Hysteresis is useful when you are measuring noisy signals, because it allows the oscilloscope to ignore minor fluctuations in the signal.

Edge calculations

Edge1, Edge2, and Edge3 refer to the first, second, and third *Mid* reference edge times, respectively.

An edge can be detected when the waveform is either rising or falling past Midref. The direction of the edges alternates, that is, if Edge1 is rising, Edge2 will be falling.

A rising edge has positive polarity. A falling edge has negative polarity.



The instrument calculates these values as follows:

1. Find the first *Mid* reference edge in the waveform record or the gated region. This is *Edge1*.
2. Continuing from *Edge1*, find the next *Mid* reference edge in the waveform record (or the gated region) of the opposite direction of *Edge1*. This is *Edge2*.
3. Continuing from Edge2, find the next *Mid* reference edge in the waveform record (or the gated region) of the same direction as Edge1. This is *Edge3*.

Cycle-cycle measurements are made on each cycle of the waveform. In the diagram above a cycle starts at Edge1 and ends at Edge3.

TPOS is the location of the sample just before the trigger point (the time reference zero sample). In other terms, it contains the domain reference location. This location is where time = 0.

TSOFF is the offset between *TPOS* and the actual trigger point. In other words, it is the trigger sample offset. Values range between 0.0 and 1.0 samples. This value is determined by the instrument when it receives a trigger. The actual zero reference (trigger) location in the measurement record is at (*TPOS* + *TSOFF*).

Missing or out-of-range samples

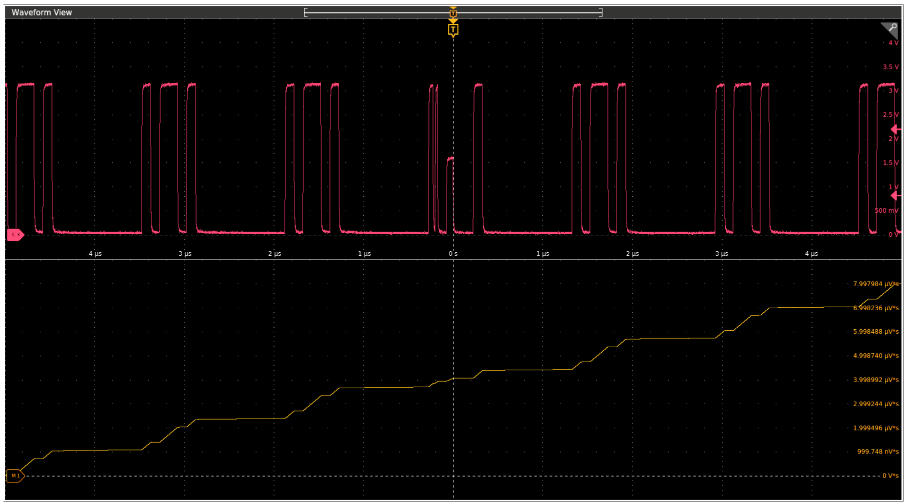
If some samples in the waveform are missing or off-scale, the measurements will interpolate between known samples to make an appropriate guess as to the sample value. Missing samples at the ends of the measurement record will be assumed to have the value of the nearest known sample. The interpolation method can be changed in User Preferences.

When samples are out of range, the measurement will give a warning to that effect (for example, CLIPPING) if the measurement could change by extending the measurement range slightly. The algorithms assume the samples recover from an overdrive condition instantaneously.

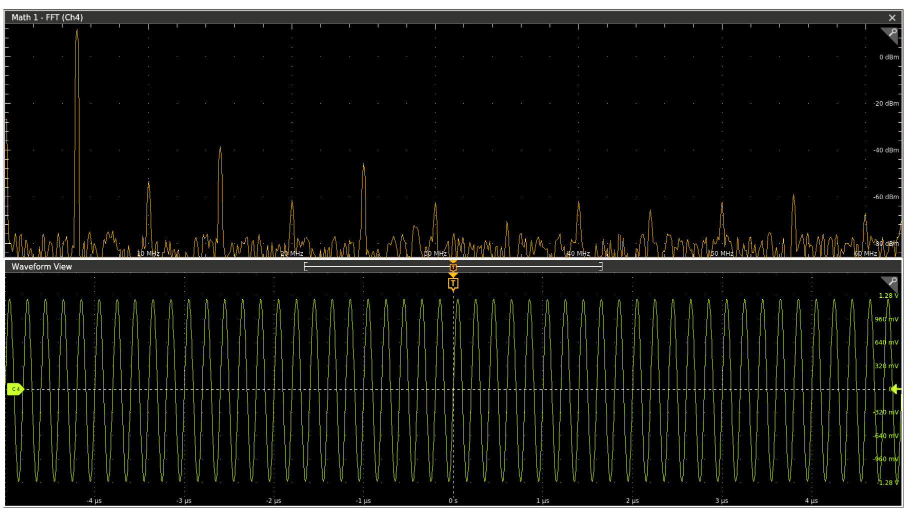
For example, if the *Mid* reference level is set directly, then *Mid* would not change even if samples were out of range. However, if *Mid* was chosen using the % choice from the **Set Levels in %** selection of the Measure menu Reference Levels tab, then *Mid* could give a CLIPPING warning.

Math waveforms

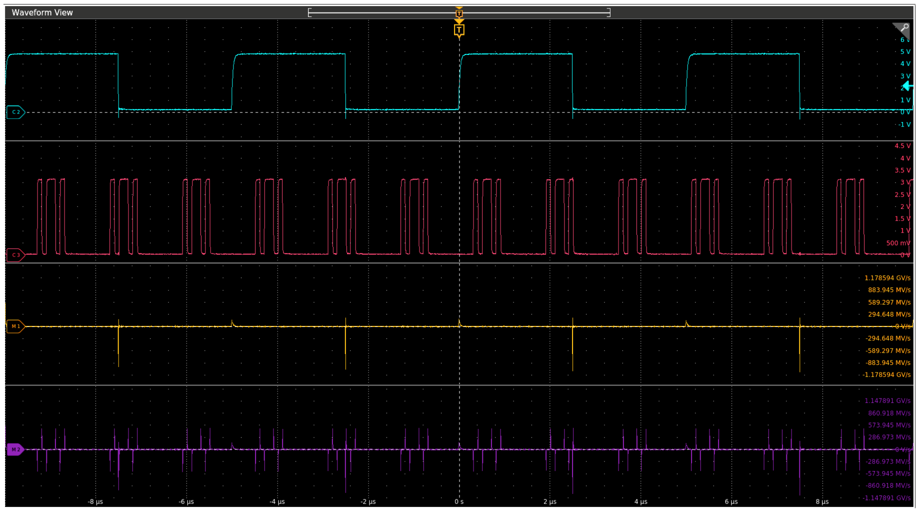
Once you have acquired waveforms or taken measurements on waveforms, the instrument can mathematically combine them to create a waveform that supports your data-analysis task. For example, you might have a waveform clouded by background noise. You can obtain a cleaner waveform by subtracting the background noise from your original waveform. Or, you can integrate a single waveform into an integral math waveform as shown below.



With spectral analysis you can analyze waveforms in the frequency domain. See the next figure.



This instrument supports mathematical combination and functional transformations of waveforms it acquires. The next figure shows this concept:



You create math waveforms to support the analysis of your channel and reference waveforms. By combining and transforming source waveforms and other data into math waveforms, you can derive the data view that your application requires. Create math waveforms that result from:

- Mathematical operations on one or several waveforms: add, subtract, multiply, and divide.
- Functional transformations of waveforms, such as integration, differentiation, and so on.
- Spectral analysis of waveforms, such as testing impulse response.

Measurement scalars can be used in math expressions. For example, you can measure the average of a waveform (using the measurement capabilities of the instrument) and subtract it from the original waveform to define a new math waveform. For example: Ch1 - mean(amplitude(Ch1)).

Math waveform elements

You can create Math waveforms from the following:

- Channel waveforms
- Reference waveforms
- Measurement scalars (automated measurements) that measure channel, reference, or math waveforms, or histograms.
- Other math waveforms
- Variables
- Filters

Dependencies

In general, math waveforms that include sources as operands are affected by updates to those sources:

- Shifts in amplitude or DC level of input sources that cause the source to clip also clip the waveform data supplied to the math waveform.
- Changes to the vertical offset setting for a channel source that clips its data also clips the waveform data supplied to the math waveform.
- Changes to the acquisition mode globally affects all input channel sources, modifying any math waveforms using them. For example, with the acquisition mode set to Envelope, a Ch1 + Ch2 math waveform will receive enveloped channel 1 and channel 2 data, and will also be an envelope waveform.
- Clearing the data in a waveform source causes a baseline (ground) to be delivered to any math waveform that includes that source until the source receives new data.

Math waveform sources

You can create Math waveforms from the following:

- Channel waveforms
- Reference waveforms
- Measurement scalars (automated measurements) that measure channel, reference, or math waveforms, or histograms.
- Other math waveforms
- Variables

Dependencies

In general, math waveforms that include sources as operands are affected by updates to those sources:

- Shifts in amplitude or DC level of input sources that cause the source to clip also clip the waveform data supplied to the math waveform.
- Changes to the vertical offset setting for a channel source that clips its data also clips the waveform data supplied to the math waveform.
- Changes to the acquisition mode globally affects all input channel sources, modifying any math waveforms using them. For example, with the acquisition mode set to Envelope, a Ch1 + Ch2 math waveform will receive enveloped channel 1 and channel 2 data, and will also be an envelope waveform.
- Clearing the data in a waveform source causes a baseline (ground) to be delivered to any math waveform that includes that source until the source receives new data.

Guidelines for working with math waveforms

Use the following guidelines when working with math waveforms:

- Keep math waveforms simple. If the math expression becomes too complex, try separating the expression into more than one math waveform and then combining the waveforms (for example, Math1 = Math2 + Math4).
- Math calculations are not available on digital channels.
- Math waveforms cannot be turned on without a math expression.
- To avoid syntax errors in a math expression, verify the use of operators, parentheses, operands, and the spelling of functions.
- If one or more reference waveforms are used in a math waveform, the record length is equal to the smallest of all the source waveforms (reference, math, or channel waveforms). The math is calculated using the first point from each source, followed by the next point, and so forth. This is true even if the sources have different times between points in the record.

Math waveform editor syntax

You can build math waveforms using the predefined expressions or the equation editor. To help you create valid math waveforms, the following tools will block most illegal entries by disabling any window element that would create an invalid entry in the math waveform expression.

Predefined expressions are accessible using the FFT or Basic Math Types.

The following syntax describes the valid math expressions you can use with the Equation Editor for the Advanced Math Type:

A math expression is composed of settings, functions, scalars and sources.

Settings have the syntax [settingName=settingValue] and are generally applied to measurements. The setting applies to everything to the right of the closing square brackets.

Example: [CoefFileName="highpass_0.25bw.flt"]HighPass(Ch1)

CoefFileName is the setting and is used as the high pass filter on channel 1.

Example: [CoefFileName="highpass_0.25bw.flt"] HighPass(Ch1) + [CoefFileName="lowpass_0.05bw.flt"] LowPass(Ch2)

The high pass filter file is applied to channel 1 and lowpass_0.05bw.flt is applied to channel 2.

Functions, except for basic and logic functions, have the syntax function(source).

In the previous examples the functions are HighPass and LowPass.

Basic and logic functions have the syntax

source1 function source2.

Examples: Ch1 * Ch2

Ch1 AND Ch2

Ch1 >= Ch2

Logic functions, ==|<|>|!=|<=|>=|AND|OR|NAND|NOR|XOR|EQV result in a waveform consisting of binary 0 and 1 values.

Scalars can be integers, floating point values, PI or meas<x>.

Sources can be Ch<x>, Ref<x>, Math<x>

Math waveform differentiation

The math capabilities of the instrument include waveform differentiation. This allows you to display a derivative math waveform that indicates the instantaneous rate of change of the waveform acquired.

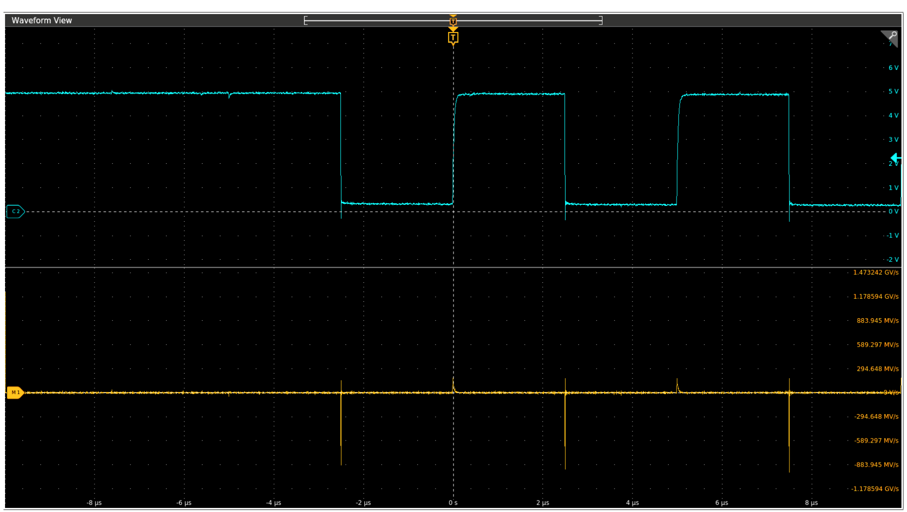
Derivative waveforms are used in the measurement of slew rate of amplifiers and in educational applications. You can create a derivative math waveform and then use it as a source for another derivative waveform. The result is the second derivative of the waveform that was first differentiated.

The math waveform, derived from the sampled waveform, is computed based on the following equation:

Yn = (X(n + 1) - Xn) * 1/T

Where: **X** is the source waveform, **Y** is the derivative math waveform, and **T** is the time between samples.

Since the resultant math waveform is a derivative waveform (see the next figure), its vertical scale is in volts/second (its horizontal scale is in seconds). The source signal is differentiated over its entire record length; therefore, the math waveform record length equals that of the source waveform.



Math waveform offset, position, and scale

The settings that you make for offset, scale, and position affect the math waveform you obtain. Here are some tips for obtaining a good display:

- Scale and position the source waveform so that it is contained on the screen. (Off-screen waveforms may be clipped, resulting in errors in the derivative waveform).
- Use vertical position and vertical offset to position your source waveform. The vertical position and offset will not affect your derivative waveform unless you position the source waveform off screen so that it is clipped.

Waveform integration

The math capabilities of the instrument include waveform integration. This allows you to display an integral math waveform that is an integrated version of the acquired waveform.

Use integral waveforms in the following applications:

- Measuring power and energy, such as in switching power supplies.
- Characterizing mechanical transducers, as when integrating the output of an accelerometer to obtain velocity.

The integral math waveform, derived from the sampled waveform, is computed based on the following equation:

$$y(n) = scale \sum_{i=1}^{n} \frac{x(i) + x(i-1)}{2} T$$

Where: **x(i)** is the source waveform, **y(n)** is a point in the integral math waveform, **scale** is the output scale factor, and **T** is the time between samples.

Since the resultant math waveform is an integral waveform, its vertical scale is in volt-seconds (its horizontal scale is in seconds). The source signal is integrated over its entire record length; therefore, the math waveform record length equals that of the source waveform.

Offset and position

When creating integrated math waveforms from live channel waveforms, consider the following:

- You should scale and position the source waveform so that it is contained on screen. (Off screen waveforms may be clipped, which will result in errors in the integral waveform.)
- You can use vertical position and vertical offset to position your source waveform. The vertical position and vertical offset will not affect your integral waveform unless you position the source waveform off screen so that it is clipped.

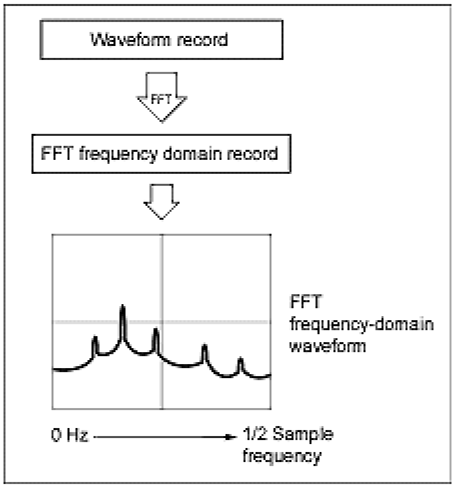
DC offset

The source waveforms that you connect to the instrument often have a DC offset component. The instrument integrates this offset along with the time-varying portions of your waveform. Even a few divisions of offset in the source waveform may be enough to ensure that the integral waveform saturates (clips), especially with long record lengths.

FFT process

The FFT process mathematically converts the standard time-domain signal (repetitive or single-shot acquisition) into its frequency components.

The FFT function processes the waveform record and displays the FFT frequency domain record, which contains the input signal frequency components from DC (0 Hz) to ½ the sample rate (also called the *Nyquist frequency*).



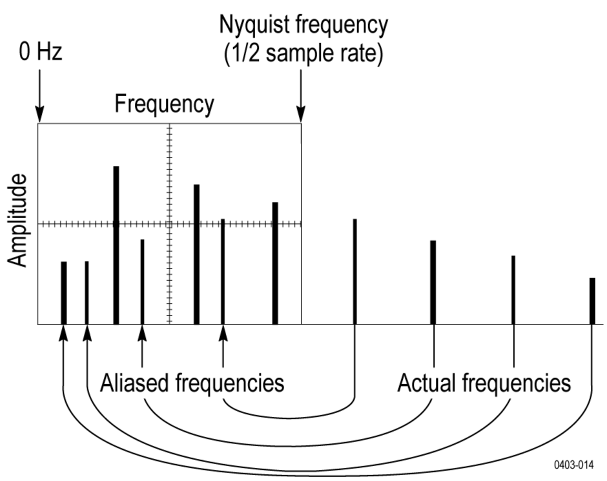
FFT and aliasing

Aliasing occurs when the input frequency of a signal is greater than one half of the sampling frequency (the sample rate).

Set the sample rate high enough so that the signals in the spectrum appear at their correct frequency as opposed to a lower aliased frequency value. Also, complex signal shapes that have many harmonics in them, such as a triangle or square wave, can appear to be OK in the time domain when in fact many of the harmonics in that signal are aliased.

One way to check for aliasing is to increase the sample rate and observe whether any of the harmonics unwrap to different frequency locations.

Another way to recognize aliasing is to realize that higher order harmonics usually have decreasing magnitudes compared to lower order harmonics. Thus, if you see a series of increasing harmonic magnitude values as frequency increases then you can suspect that they may be aliased. In the spectral math waveform, the actual higher frequency components are under sampled, and therefore they appear as lower frequency aliases that "fold back" around the Nyquist point. (See the next figure.) You may test by increasing the sample rate and observing if aliases unwrap to different frequency positions.



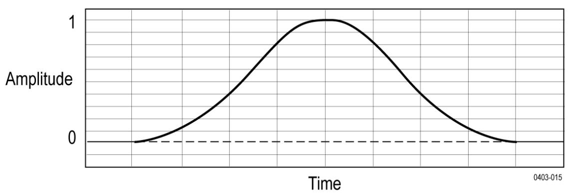
If you have a variable-frequency signal source, another way to observe aliasing is to adjust the frequency slowly while watching the spectral display. If some of the harmonics are aliased, you will see the harmonics decreasing in frequency when they should be increasing or vice versa.

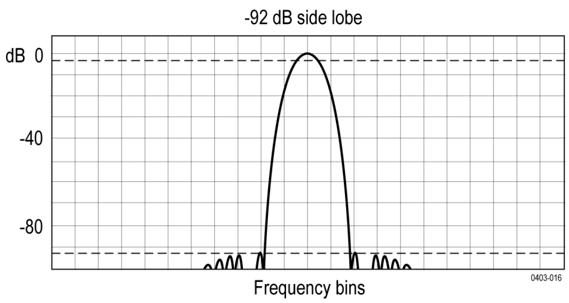
Blackman-Harris FFT window concepts

FFT windows have various resolution bandwidths and scallop losses (see the figure below). Choose the one that best allows you to view the signal characteristics you are interested in.

The Blackman-Harris window has a low amount of energy leakage compared to the other windows. Its best use is for single frequency signals to look for higher order harmonics.

Blackman-Harris window

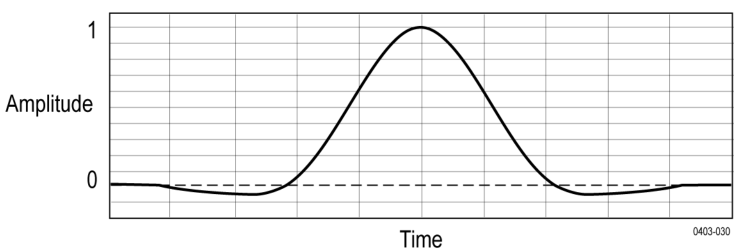


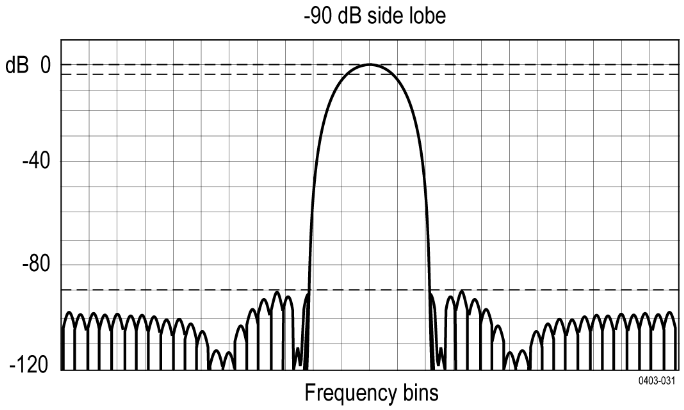


Flattop2 window

This window has the lowest scallop loss of any of the windows. It also has a wider resolution bandwidth but lower side lobe attenuation. Also, it is unique because the time domain shape has negative values.

Note: The Flattop2 window is useful for high accuracy magnitude measurements for signals that do not require very narrow bandwidth.

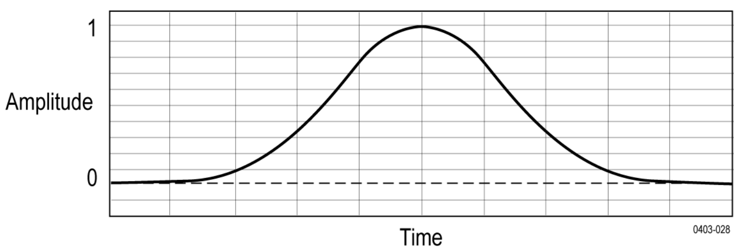


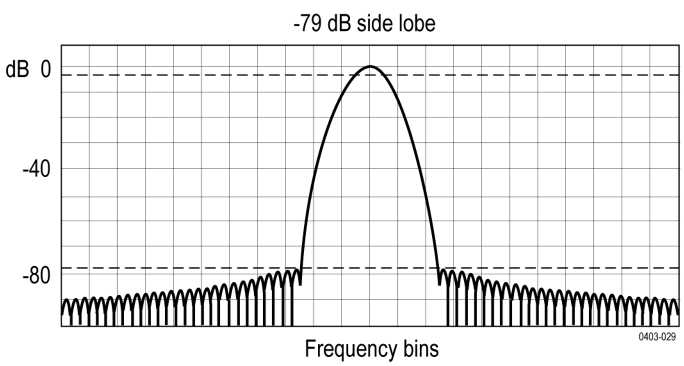


Gaussian window

This is the default window function (see the next figure). It is unique in that the time-domain shape of an exponential Gaussian function transforms into a Gaussian exponential shape in the frequency domain.

Note: This window provides optimal localization in both the time and the frequency domain.



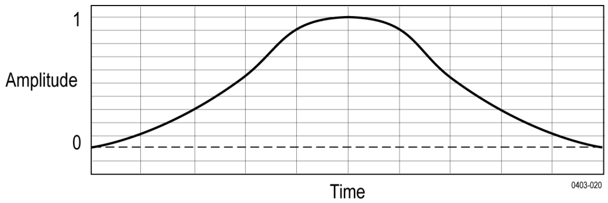


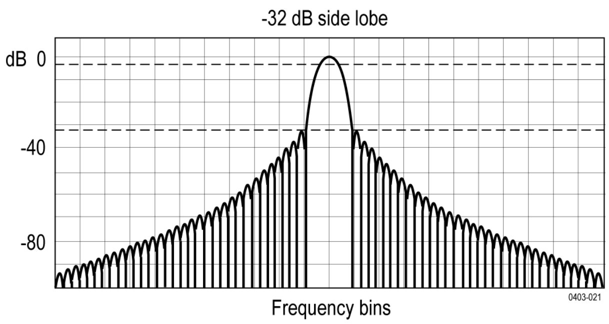
Hanning FFT window

FFT windows have various resolution bandwidths and scallop losses (see the figure below). Choose the one that best allows you to view the signal characteristics you are interested in.

The Hanning window has the narrowest resolution bandwidth, but higher side lobes. Hanning has slightly poorer frequency resolution than Hamming. Hanning is best for measuring sine, periodic, and narrow-band random noise, and transients or bursts where the signal levels before and after the event are significantly different.

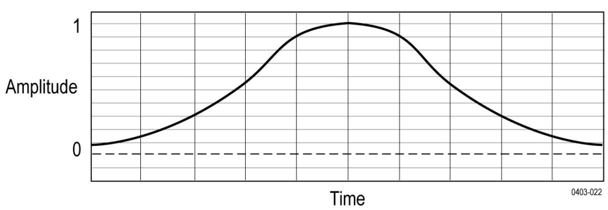
Hanning window

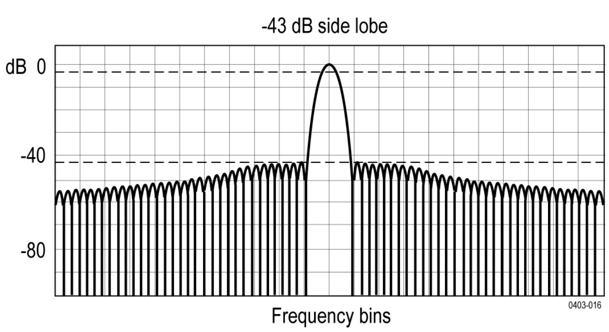




Hamming window

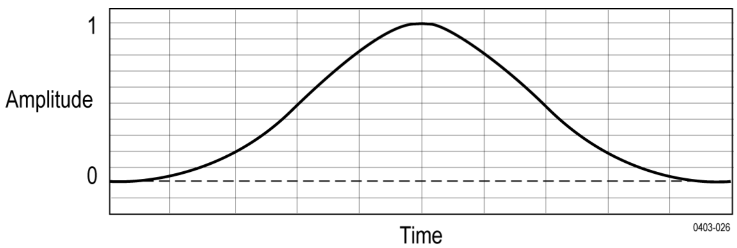
This window is unique in that the time domain shape does not taper all the way to zero at the ends. This makes it a good choice if you wanted to process the real and imaginary parts of the spectrum off line and inverse transform it back to the time domain. Because the data does not taper to zero, you can remove the effect of the window function from the result.

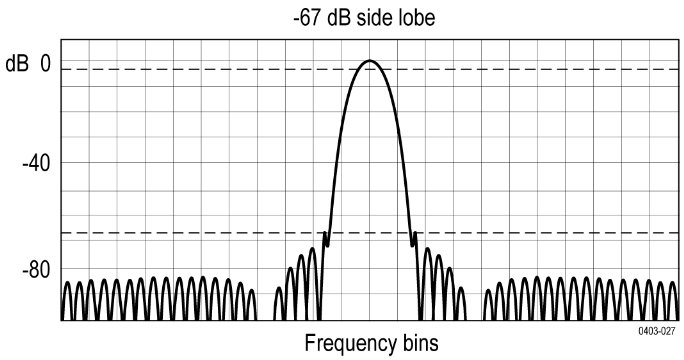




Kaiser-Bessel FFT window

A Kaiser-Bessel window balances amplitude accuracy, side lobe distance, and side lobe height. Although similar to the Blackman-Harris window, the near side lobes in a Kaiser- Bessel window tend to be higher for the same main lobe width, while the further-out side lobes are lower. This window can reveal signals close to the noise floor, and analyze two tones with close frequencies but different amplitudes.

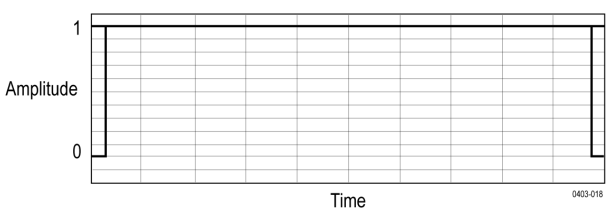


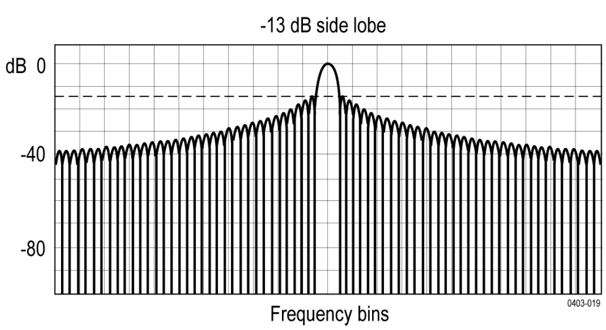


Rectangular window

This window is equal to unity (see the next figure). This means the data samples in the gate are not modified before input to the spectral analyzer. Rectangular windows are best for measuring transients or bursts where the signal levels before and after the event are nearly equal.

Note: This window has the narrowest resolution bandwidth of any of the windows, but it also has the most spectral leakage and the highest side lobes.



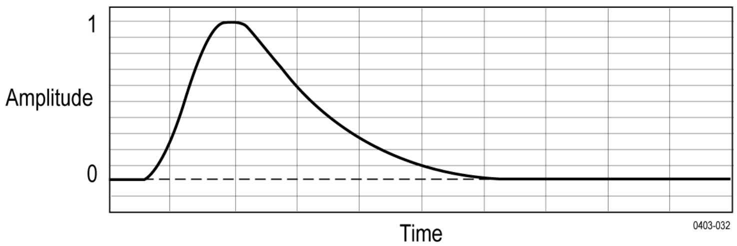


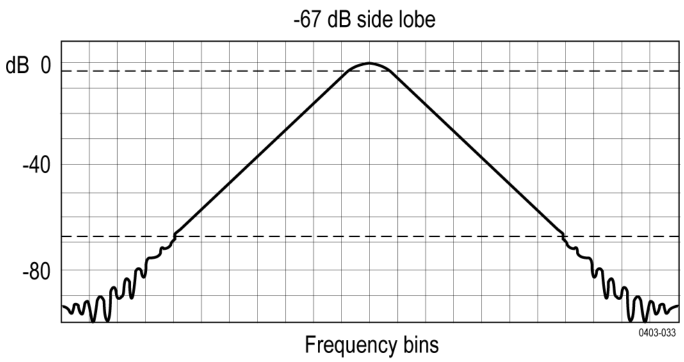
Tek-Exponential window

In the time domain, it is not a symmetrical bell shape as is the case with the other windows. Instead, it is exponential with a peak at the 20% position of the time domain gate. The frequency domain shape is triangular.

Note: Use this window for impulse-response testing where the 20% position is the zero phase reference point. More of the acquired data record length is used to capture the impulse response.

Exact details of how to compute its values were published in the article; *Impulse-Response Testing Lets a Single Test Do the Work of Thousands*, by John Pickerd, EDN magazine, April 27, 1995.





Measurement algorithms

Amplitude measurement algorithms

AC RMS measurement algorithm

AC RMS is the true Root Mean Square of the data points about the Mean (μ). This measurement can be made across the entire record or on each cycle in the record.

Area measurement algorithm

Area is the arithmetic area for one waveform. The area measured above ground is positive. The area measured below ground is negative. Remember that one waveform is not necessarily equal to one cycle. For cyclical data you may prefer to use the cycle area rather than the arithmetic area.

$$Area = \int_{Start}^{End} Waveform(t)\,\mathrm{dt}$$

0020-012

Details of the integration algorithm are given later *Integration Algorithm*.

Amplitude measurement algorithm

Amplitude is the difference between the Top value and the Base value.

Amplitude = Top - Base

Base measurement algorithm

Base is calculated using the selected Base Top method. Base is the most common data value below the midpoint of the waveform, when the default Base Top method Histogram Mode is selected. This measurement can be made across the entire record or on each cycle in the record.

Integration algorithm

The integration algorithm used by the instrument is as follows:

$$\int_A^B W(t)dt \text{ is approximated by } \int_A^B \hat{W}(t)dt \text{ where:}$$

0020-025

W(t) is the sampled waveform

Ŵ(t) is the continuous function obtained by interpolation of *W(t)*

A and *B* are numbers between 0.0 and *RecordLength* – 1.0

If *A* and *B* are integers, then:

$$\int_A^B \hat{W}(t)dt = s \times \sum_{i=A}^{B-1} \frac{W(i) + W(i+1)}{2}$$

0020-026

where *s* is the sample interval.

Similarly,

$$\int_A^B (W(t))^2 dt \text{ is approximated by } \int_A^B \left(\hat{W}(t)\right)^2 dt \text{ where:}$$

0020-027

W(t) is the sampled waveform

Ŵ(t) is the continuous function obtained by interpolation of *W(t)*

A and *B* are numbers between 0.0 and *RecordLength* – 1.0

If *A* and *B* are integers, then:

$$\int_A^B \left(\hat{W}(t)\right)^2 dt = s \times \sum_{i=A}^{B-1} \frac{(W(i))^2 + W(i) \times W(i+1) + (W(i+1))^2}{3}$$

0020-028

where *s* is the sample interval.

Maximum measurement algorithm

Maximum is the maximum data point. Typically the most positive peak voltage.

Negative Overshoot measurement algorithm

Negative Overshoot is the difference between Minimum and Base, divided by the amplitude. It is the percent that the waveform goes below base.

$$\text{Negative Overshoot} = \frac{\text{Base} - \text{Min}}{\text{Amplitude}} \text{X} 100\,\%$$

Note that overshoot values should never be negative (unless *Top* or *Base* are set out-of-range).

Positive Overshoot measurement algorithm

Positive Overshoot is the difference between Maximum and Top, divided by the amplitude. It is the percent that the waveform goes above top.

$$\text{Positive Overshoot} = \frac{\text{Max} - \text{Top}}{\text{Amplitude}} \text{X} 100\,\%$$

Note that this value should never be negative.

Peak-To-Peak measurement algorithm

Peak to peak is the difference between Maximum and Minimum.

PeaktoPeak = *Max* – *Min*

RMS measurement algorithm

RMS is the true root mean square of the data points.

$$\text{RMS} = \sqrt{\frac{(\text{data1}^2 + \text{data2}^2 \ldots)}{(\text{end - start})}}$$

Top measurement algorithm

Top is calculated using the selected Base Top method. Top is the most common data value above the midpoint of the waveform, when the default Base Top method Histogram Mode is selected. This measurement can be made across the entire record or on each cycle in the record.

Timing measurement algorithms

Burst Width measurement algorithm

Burst Width is the duration of a series of adjacent crossings of the mid reference level. The duration of a burst. Bursts are separated by a user-defined idle time.

Data Rate measurement algorithm

Data Rate is the reciprocal of Unit Interval. This measurement is made on each bit in the record.

Delay measurement algorithm

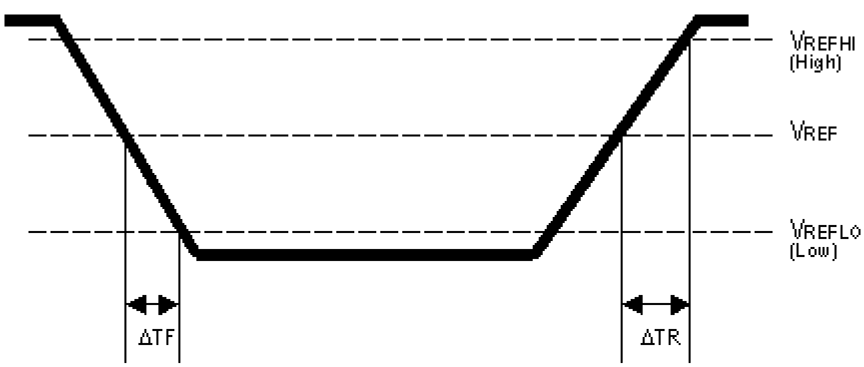
Delay is the time between a mid reference level edge on one source to a mid reference level edge on a second source. The direction of each edge can be configured by the user.

Falling slew rate

Falling Slew Rate is the rate of change in value as an edge transitions from the high or mid reference level to the mid or low reference level. The levels are configurable.

In the diagram below, the Falling Slew Rate from mid ref to low ref is calculated using the following equation:

Falling Slew Rate = (VREF - VREFLO)/ΔTF



Fall Time measurement algorithm

Fall Time is the time required for an edge to fall from the high reference level to the low reference level. By default the measurement is from reference level 90% amplitude to 10% amplitude.

The following figure shows a falling edge with the two edges necessary to calculate a Fall measurement. The figure shows the default high reference level which is 90% of Top and the default low reference level which is 10% of Base.

1. Searching from *Start* to *End*, find the first sample in the measurement zone greater than *HighRef*.

2. From this sample, continue the search to find the first (negative) crossing of edge of *HighRef*. The time of this edge is *THF*. (Use interpolation if necessary.)

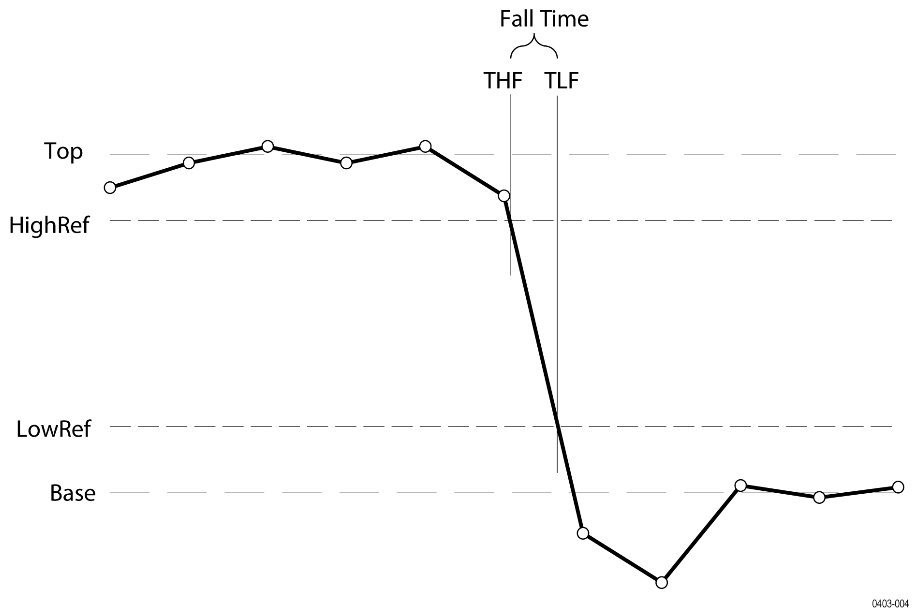


Figure 31: Fall Time

3. From *THF*, continue the search, looking for a crossing of *LowRef*. Update *THF* if subsequent *HighRef* crossings are found. When a *LowRef* crossing is found, it becomes *TLF*. (Use interpolation if necessary.)
4. *FallTime* = *TLF* – *THF*

Frequency measurement algorithm

Frequency is the reciprocal of the period. Frequency is typically measured in Hertz (Hz) where 1 Hz = 1 cycle per second.

Frequency = 1 / Period

High Time measurement algorithm

High Time is the amount of time that a waveform cycle is above the High reference voltage level.

The application calculates the measurement using the following equation:

$$T_N^{High} = T_N^{Hi-} - T_N^{Hi+}$$

Where:

T High is the high time.

T Hi- is the High reference crossing on the falling edge.

T Hi+ is the High reference crossing on the rising edge.

Hold Time measurement algorithm

Hold Time is the time between the mid reference level crossing of the clock source (Source1) and the next mid reference level crossing of the data source (Source2). The crossings (edges) may be configured to be rising, falling or either.

The application calculates this measurement using the following equation:

$$T_n^{Hold} = T_n^{2nd} - T_i^{Main}$$

Where:

T Hold is the hold time.

T Main is the source 1 (clock) Mid reference edge time in the configured direction.

T 2nd is the source 2 (data) Mid2 reference edge time in the configured direction.

Low Time measurement algorithm

Low Time is the amount of time that a waveform cycle is below the Low reference voltage level.

The application calculates this measurement using the following equation:

$$T_n^{Low} = T_n^{Lo+} - T_n^{Lo-}$$

Where:

T Low is the low time.

T Lo+ is the Low reference crossing on the rising edge.

T Lo- is the Low reference crossing on the falling edge.

N-Periods Duration measurement algorithm

Duration N-Periods is the time required to complete N cycles.

The source can be configured to be treated as either a clock or data waveform. Given a voltage waveform, the N-Period is calculated as follows:

If the Signal Type is Clock

The N–Period measurement calculates the elapsed time for N consecutive crossings of the mid reference voltage level in the direction specified.

The application calculates this measurement using the following equation:

$$NP_n^{Clock} = T_{n+N}^{Clock} - T_n^{Clock}$$

Where:

NP Clock is the accumulated period for N clock cycles.

T Clock is the VRefMid crossing time for the configured edge direction.

If the Signal Type is Data

The N–Period measurement calculates the elapsed time for N consecutive unit intervals.

The application calculates this measurement using the following equation:

$$NP_n^{Data} = T_{n+N}^{Data} - T_n^{Data}$$

Where:

NP Data is the duration for N unit intervals.

T Data is the VRefMid crossing time in either direction.

If *T n+N Data* does not exist for a given n, no measurement is recorded for that position.

Negative Duty Cycle measurement algorithm

Negative Duty Cycle is the ratio of the negative pulse width to the signal period expressed as a percentage.

NegativeWidth is defined in *Negative Pulse Width*, below.

If *Period* = 0 or undefined then return an error.

$$NegativeDutyCycle = \frac{NegativeWidth}{Period} \times 100\%$$

0020-018

Negative Pulse Width measurement algorithm

Negative Pulse Width is the time (or distance) the signal remains below the mid reference level. It is the distance from a falling edge to the next rising edge.

Period measurement algorithm

Negative Pulse Width is the time (or distance) the signal remains below the mid reference level. It is the distance from a falling edge to the next rising edge. Period is measured in horizontal units, typically seconds.

Period = Edge3 - Edge1

Phase measurement algorithm

Phase is the ratio of the Skew between two sources to the Period of the first source. It is the amount of phase shift between edges of the two waveforms. The phase shift is expressed in degrees of the Source1 waveform cycle, where 360 degrees is one complete cycle (Period) of Source1. For best results, Source1 and Source2 should be of the same frequency or one waveform should be a harmonic of the other.

Phase is determined in the following manner for each cycle of the record:

1. The first two adjacent edges of the configured 'from' edge type are found in Source1.
2. The period of Source1 is calculated (see *Period* above). If the "from" edge is set to either, the half-period of Source1 is calculated.
3. The first edge in Source2 in the configured 'to' edge direction is found.
4. The Skew from the first Source1 edge to the next Source2 edge is calculated.

   (Skew/Period)*360
5. When "from" edge is either, the calculation is (Skew/half-Period)*180.

Positive Duty Cycle measurement algorithm

Positive Duty Cycle is the ratio of the positive pulse width to the signal period, expressed as a percentage.

PositiveWidth is defined in *Positive Pulse Width*, following.

$$PositiveDutyCycle = \frac{PositiveWidth}{Period} \times 100\%$$

0020-021

Positive Pulse Width measurement algorithm

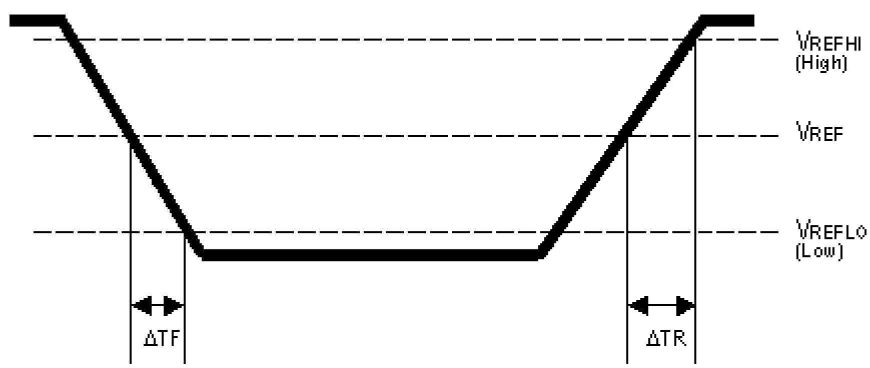
Positive Pulse Width is the time the signal remains above the mid reference level. It is the distance from a rising edge to the next falling edge.

Rising Slew Rate measurement algorithm

Rising Slew Rate is the rate of change in value as an edge transitions from the low or mid reference level to the mid or high reference level. The levels are configurable.

In the diagram below, the Rising Slew Rate from mid ref to high ref is calculated using the following equation:

(VREFHI - VREF)/ΔTR

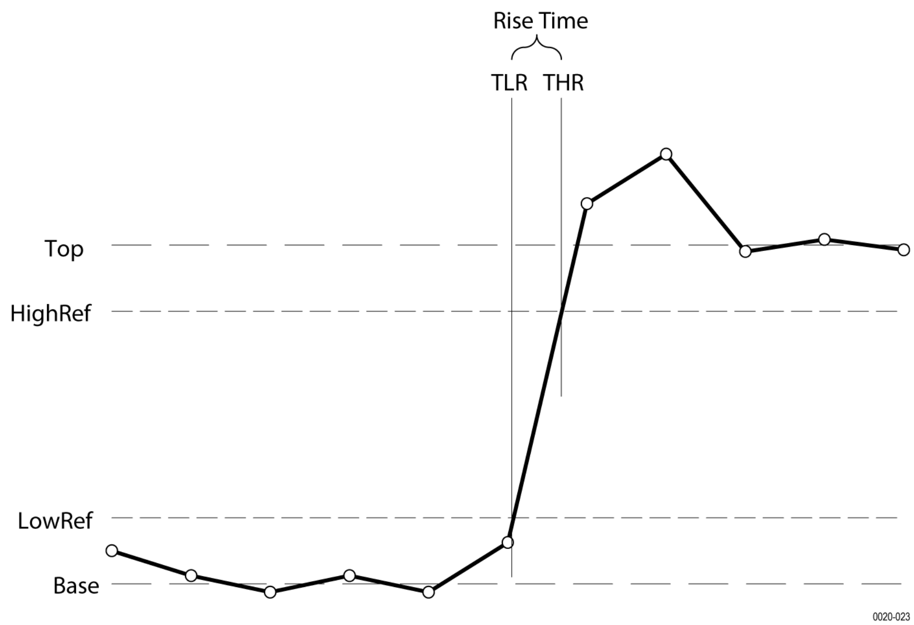


Rise Time measurement algorithm

Rise Time is the time required for an edge to rise from the low reference level to the high reference level. By default the measurement is from reference level 10% amplitude to 90% amplitude.

The following figure shows a rising edge with the two crossings necessary to calculate a Rise Time measurement.

1. Searching from *Start* to *End*, find the first sample in the measurement zone less than *LowRef*.
2. From this sample, continue the search to find the first (positive) crossing of *LowRef*. The time of this crossing is the low rise time or *TLR*. (Use linear or sin interpolation if necessary.)
3. From *TLR*, continue the search, looking for a crossing of *HighRef*. Update *TLR* if subsequent *LowRef* crossings are found. If a *HighRef* crossing is found, it becomes the high rise time or *THR*. (Use linear or sin interpolation if necessary.)
4. *RiseTime* = *THR* – *TLR*



Setup

Setup Time is the time between the mid reference level crossing of the clock source (Source1) and the closest previous mid reference level crossing of the data source (Source2). The crossings (edges) may be configured to be rising, falling or either.

The application calculates this measurement using the following equation:

$$T_n^{Setup} = T_i^{Main} - T_n^{2nd}$$

Where:

T Setup is the setup time.

T Main is the Main input (clock) Mid reference crossing time in the specified direction.

T 2nd is the 2nd input (data) Mid2 reference crossing time in the specified direction.

Skew

Skew is the time between the mid reference level crossing on Source1 to the mid reference level crossing on Source2. The direction of the edge crossing is configurable.

The application calculates this measurement using the following equation:

$$T_n^{Skew} = T_n^{Main} - T_n^{2nd}$$

Where:

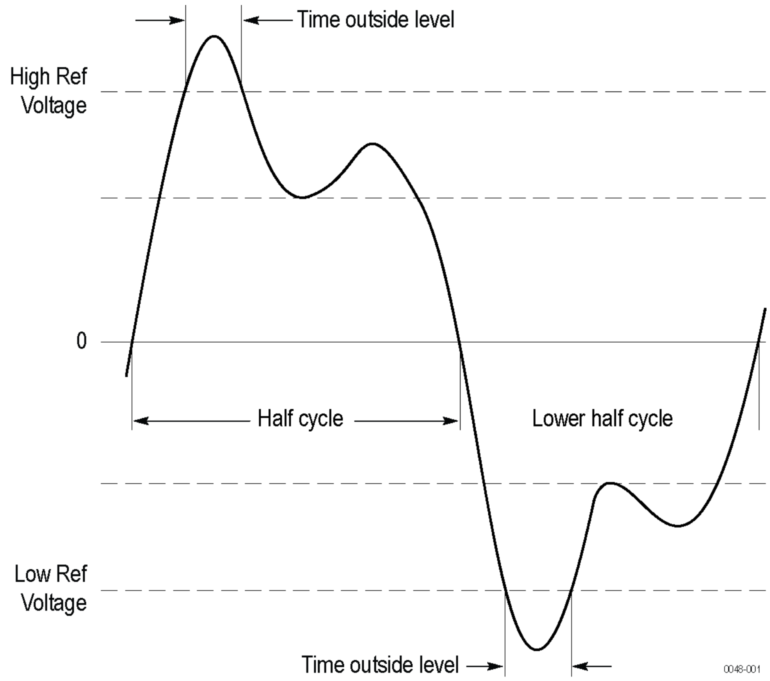
T Skew is the timing skew.

T Main is the Main input Mid reference crossing time in the configured direction.

T 2nd is the 2nd input Mid2 reference crossing time in the configured direction.

Time Outside Level measurement algorithm

Time Outside Level is the time the signal remains above the high reference level and/or below the low reference level.



Unit Interval measurement algorithm

Unit Interval is the time difference between two successive bits. This measurement is made on each bit in the record. The bits are calculated using clock recovery method constant mean.

Jitter measurement algorithms

AC Common Mode

AC Common Mode (Pk-Pk) is the peak-to-peak amplitude of the common mode of the sources. Voltage sources are typically filtered to include only the frequency components above the cutoff frequency (30 kHz). The filter can be disabled in the measurement configuration. The measurement can be configured to take one or two sources, and the cutoff frequency may be enabled or disabled. The application calculates this measurement using the following equation:

Peak-to-Peak(High Pass Filter((Source1 + Source2)/2))

Bit Amplitude measurement algorithm

Bit Amplitude is the difference between the levels of the "1" and "0" bits surrounding each transition, measured over a specified range at the center of the recovered unit interval. This measurement is made on each transition bit (Mean) or across the entire record (Mode).

Bit High measurement algorithm

Bit High is the amplitude of a "1" bit. The amplitude is measured over a user-specified portion at the center of the recovered unit interval. This measurement is made on each high bit in the record (Mean) or across the entire record (Mode).

Bit Low measurement algorithm

Bit Low is the amplitude of a "0" bit. The amplitude is measured over a user-specified portion at the center of the recovered unit interval. This measurement is made on each low bit in the record (Mean) or across the entire record (Mode).

DC Common Mode measurement algorithm

DC Common Mode is the arithmetic mean of the common mode of two sources. This measurement is made across the entire record.

Differential Crossover measurement algorithm

Differential Crossover is the voltage level of a differential signal pair at the crossover point(s). This measurement is made at each crossover point in the record.

SSC Freq Dev measurement algorithm

SSC Freq Dev is the spread spectrum clock frequency deviation from the nominal frequency in ppm (parts per million).

Clock recovery is used on the measurement. The method is constant clock mean.

SSC Modulation Rate measurement algorithm

SSC Modulating Rate is the modulating frequency of a spread spectrum clock. It is the rate that the clock frequency changes.

Clock recovery is used on the measurement. The method is constant clock mean.

TIE

TIE (Time Interval Error) is the difference in time between an edge in the source waveform and the corresponding edge in a reference clock. The reference clock is usually determined by a clock recovery process performed on the source waveform. For Explicit-Clock clock recovery, the process is performed on an explicitly identified source.

If the Signal Type is Clock

The application calculates Clock TIE measurement using the following equation:

$$TIE_n^{Clock} = T_n^{Clock} - T'^{Clock}_n$$

Where:

TIE Clock is the clock time interval error.

T Clock is the Mid reference crossing time for the specified clock edge.

T 'Clock is the corresponding edge time for the specified reference clock.

If the Signal Type is Data

The application calculates Data TIE measurement using the following equation:

$$TIE_k^{Data} = T_k^{Data} - T'^{Data}_k$$

Where:

TIE Data is the data time interval error.

T Data is the Mid reference crossing time in either direction.

T 'Data is the corresponding edge time for the specified reference clock.

The subscript k is used to indicate that there is one measurement per actual edge.

T/nT Ratio measurement algorithm

T/nT Ratio is the ratio in dB of the midpoint level of the 1st bit after each transition to the midpoints of the 2nd and subsequent non-transition bit levels prior to the next transition. This measurement is made for each non-transition bit in the record.

DCD

Duty Cycle Distortion (DCD) is the peak-to-peak amplitude for that portion of the deterministic jitter directly correlated with signal polarity, that is the difference between the mean positive edge displacement versus that on negative edges. A single DCD value is determined for each acquisition, by means of RJ-DJ separation analysis.

DDJ

Data-Dependent Jitter (DDJ) is the peak-to-peak amplitude for that portion of the deterministic jitter directly correlated with the data pattern in the waveform. A single DDJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

DJ

Deterministic Jitter (DJ) is the peak-to-peak amplitude for all timing errors that follow deterministic behavior. A single DJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

Dual Dirac deterministic jitter

Dual Dirac Deterministic Jitter (DJ–δδ) the peak-to-peak magnitude for all timing errors exhibiting deterministic behavior, calculated based on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal magnitude dirac functions (impulses). A single DJ–δδ value is determined for each acquisition, by means of RJ-DJ separation analysis.

F/2 measurement algorithm

F/2 is the peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by 2. This measurement is made across the entire record.

F/4 measurement algorithm

F/4 is the peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by 4. This measurement is made across the entire record.

F/8 measurement algorithm

F/8 is the peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by 8. This measurement is made across the entire record.

J2

J2 is Total Jitter at a Bit Error Rate (BER) value of 2.5E-3. This statistical value predicts a peak-to-peak jitter that will only be exceeded with a probability equal to the BER.

J9

J9 is Total Jitter at a Bit Error Rate (BER) value of 2.5E-10. This statistical value predicts a peak-to-peak jitter that will only be exceeded with a probability equal to the BER.

Jitter Summary measurement

The Jitter Summary measurement is a predefined set of jitter measurements displayed in a single badge. The measurements include TIE, TJ@BER, Eye Width@BER, RJ-66, DJ-66, PJ, DDJ, and DCD. This measurement also adds Eye Diagram, TIE Histogram, Tie Spectrum, and Bathtub plots to the screen.

NPJ

Non-Periodic Jitter (NPJ) is the dual-dirac magnitude of that portion of Bounded Uncorrelated Jitter (BUJ) that is not periodic. Since it is not periodic and is not correlated with the data pattern, NPJ is frequently difficult to distinguish from (Gaussian) RJ.

This component of jitter is not analyzed by default, but you can enable it by switching the jitter analysis mode to Spectral + BUJ. Since it typically requires high populations to distinguish, you may need to acquire multiple waveforms before jitter results are available when Spectral + BUJ mode is enabled.

Phase noise

The Phase Noise measurement performs a jitter measurement, converts the result into the frequency domain, and reports the rms jitter integrated between two specific frequencies selected by the user.

The phase noise measurement is defined only for clock signals. If the source waveform appears to be a data signal, a warning message will be produced but the measurement will proceed.

PJ

Periodic Jitter (PJ) is the peak-to-peak amplitude for that portion of the deterministic jitter which is periodic, but for which the period is not correlated with any data pattern in the waveform. A single PJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

RJ

Random Jitter (RJ) is the rms magnitude of all timing errors not exhibiting deterministic behavior. A single RJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

RJ 66 measurement algorithm

RJDIRAC (dual-dirac random jitter) is random jitter based on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal-magnitude Dirac functions. This measurement is made across the entire record.

SRJ

Sub-Rate Jitter is periodic jitter at a rate that integrally divides the data rate. For example, if the data rate is F bits/second, sub-rate jitter components could occur at F/2 or F/4. It typically occurs when a serial data stream is formed by multiplexing (interleaving) an integral number of lower-rate bit streams together, although there can be other causes. Sub-rate jitter is a sub-component of PJ.

The SRJ measurement is the peak-to-peak amplitude for the sum of all F/N jitter components that are tracked by DPOJET. Since different F/N components are correlated with each other, the peak-to-peak SRJ depends on relative phases and is not simply the sum of the individual F/N components.

The SRJ measurement always tracks and accounts for N = 2, 4 and 8 regardless of whether the corresponding F/N measurements have been selected.

TJ@BER

Total Jitter at a specified Bit Error Rate (BER). This extrapolated value predicts a peak-to-peak jitter that will only be exceeded with a probability equal to the BER. It is generally not equal to the total jitter actually observed in any given acquisition. A single TJ@BER value is determined for each acquisition, by means of RJ-DJ separation analysis.

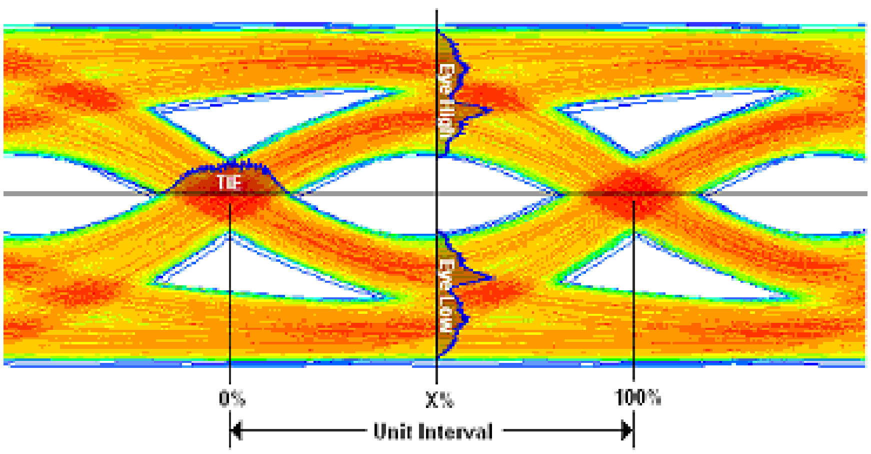
Eye measurement algorithms

Eye Height measurement algorithm

Eye Height is the minimum vertical eye opening at the mid of the unit interval. This measurement is made across the entire record.

Eye high

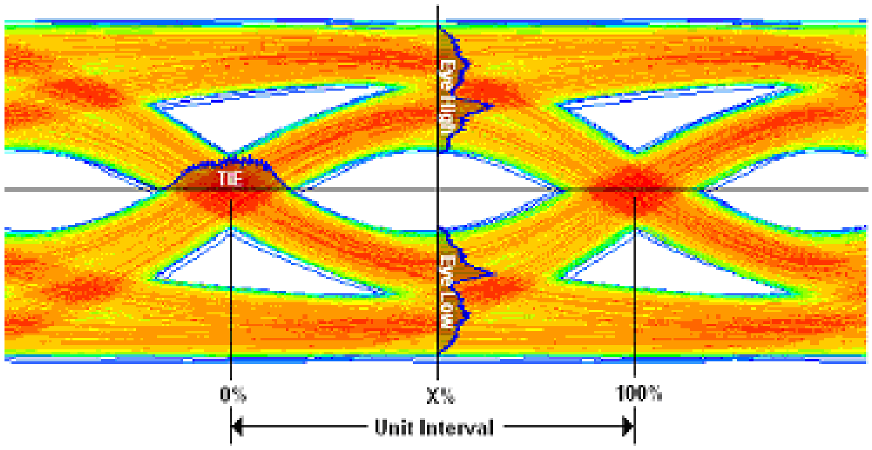
Eye High calculates the voltage at a selected horizontal position across the unit interval, for all High bits in the waveform. You specify the offset at which the measurement takes place from 0% to 100% of the unit interval. Configure the measurement to include all bits, only transition bits, or only non-transition bits. (Note that some of the waveform can be omitted from the measurement due to initialization of clock recovery or filtering.) A histogram of the Eye High measurement corresponds to a vertical slice through the upper half of a three-dimensional eye diagram.



Note: This illustration shows how the measurement is made, and does not represent how the oscilloscope actually displays an eye diagram or histograms on an eye diagram plot.

Eye low

Eye Low calculates the voltage at the selected horizontal position across the unit interval, for all Low bits in the waveform. A histogram of the Eye Low measurement corresponds to a vertical slice through the lower half of a three-dimensional eye diagram.



Note: This illustration shows how the measurement is made, and does not represent how the oscilloscope actually displays an eye diagram or histograms on an eye diagram plot.

Eye Width measurement algorithm

Eye Width is the minimum horizontal eye opening at the user-specified reference level. This measurement is made across the entire record.

Height@BER

Height@BER is the Eye Height at a specified Bit Error Rate (BER). This extrapolated value predicts a vertical eye opening that will be violated with a probability equal to the BER. It is generally not equal to the eye height actually observed in any given acquisition. A single Height@BER value, in the given interval, is determined for each acquisition by means of Q-scale extrapolation.

Q-factor

Quality Factor is the ratio of eye size to noise.

The final measurement value would be computed according to the equation below:

```
Q-factor = [mean(EyeHigh) - mean(EyeLow)] / [stddev(EyeHigh) +
stddev(EyeLow)]
```

Where:

Eye High: the sample values of positive UI at x%.

Eye Low: the sample values of negative UI at x%.

For more details refer Eye Height

Width@BER

Width@BER is the Eye Width at a specified Bit Error Rate (BER). This extrapolated value predicts a horizontal eye opening that will be violated with a probability equal to the BER. It is generally not equal to the eye width actually observed in any given acquisition. A single Width@BER value is determined for each acquisition, by means of RJ-DJ separation analysis.

Power measurements: Input Analysis algorithms

Power Quality measurement algorithm

The Power Quality measurement calculates the Frequency and RMS values of the voltage and current, Crest Factors of the voltage and current, True Power (TrPwr), Reactive Power (RePwr), Apparent Power (ApPwr), Power Factor (PF), and Phase Angle (θ) of the AC signal.

- **RMS Voltage**: The application calculates the RMS voltage using the following equation:

  $$V_{RMS} = \sqrt{\frac{1}{N}\sum_{n-0}^{N-1} v^2(n)}$$

  Where:

  V_{RMS} is the RMS voltage in Volts.

  N is the number of samples.

  n is the data point.

  v(n) is the absolute value of the voltage at the particular data point.

  Note: *The voltage RMS is for all the time domain cycles in the acquisition.*

- **RMS Current**: The application calculates the RMS current using the following equation:

  $$I_{RMS} = \sqrt{\frac{1}{N}\sum_{n-0}^{N-1} i^2(n)}$$

  Where:

  I_{RMs} *is the RMS current in Amps*.

  N is the number of samples.

  n is the data point.

  i(n) is the absolute value of the current at the particular data point.

- **Apparent Power (ApPwr)**: It is the product of the RMS voltage and current (mathematically, the absolute value of the vector sum of the true and reactive power), measured in Volt-Amperes or VA. The application calculates the **Apparent Power** (ApPwr) using the following equation:

  $$\text{ApPwr} = V_{RMS} \times I_{RMS}$$

  Where:

  ApPwr is the Apparent Power, Volt-Amperes (VA).

  V_{RMS} *is the root mean square of the voltage*.

  I_{RMS} *is the root mean square of the current*.

- **Reactive Power (RePwr)**: The reactive power or the imaginary power delivered to and temporarily stored in the reactive (inductive or capacitive) elements of the load, measured in units of Volt-Amperes-Reactive or VAR. The application calculates the Reactive power using the following equation:

  $$RePwr = \sqrt{ApPwr^2 - TpPwr^2}$$

  Where, *RePwr* is the Reactive Power, Volt-Amperes-Reactive or VAR.
- **Power Factor (PF)**: It is calculated using the following equation:

  $$PF = \frac{TrPwr}{ApPwr}$$

  Where, *PF* is the Power Factor.

  In PQ measurement True, Apparent and Reactive Power's and Power Factor will be computed only when connection type is Line-to- Neutral.
- **Crest Factor (CF)**: It is the ratio of the peak voltage value of the signal to the RMS value of the signal. Use the following equation to calculate the crest factor for the voltage and current:

  $$VCF = \frac{V_{pk}}{V_{RMS}}$$

  Where:

  VCF is the Voltage Crest Factor.

  V_{PK} *is the peak value of the voltage*.

  V_{RMS} *is the Root Mean Square of the voltage*.

  $$ICF = \frac{I_{pk}}{I_{RMS}}$$

  Where:

  ICF is the Current Crest Factor.

  I_{RMS} *is the Root Mean Square of the current*.
- **Phase Angle (σ)**: It is the angle (-90 to +90) whose cosine is the true power factor. Unit of Phase Angle is degrees. The angle is positive if the Ch1 waveform (typically voltage) leads the Ch2 waveform (typically current). The angle is negative if the Ch1 waveform lags behind the Ch2 waveform. The application calculates the phase angle using the following equation:

  $$\emptyset = Cos^{-1}\frac{TrPwr}{ApPwr}$$

  Where, *σ* is the Phase Angle, Degree.

Harmonics algorithm

Harmonics are the sinusoidal voltages or currents having frequencies that are integer multiples of the frequency at which the supply system is designed to operate (termed the fundamental frequency). Distorted waveforms can be decomposed into sum of the fundamental frequency and its harmonics.

- The measurement uses the Discrete Fourier Transform (DFT) to calculate the Real component (Re(k)) and Imaginary component Im(k). The Real component (Re(k)) and Imaginary component Im(k) are calculated using the following equation:

$$Re[k] = \sum_{i=0}^{N-1} x[i]\cos(2\pi ki/N)$$

$$Im[k] = -\sum_{i=0}^{N-1} x[i]\sin(2\pi ki/N)$$

  Where:

  Re[k] is the Real component of kth harmonic.

  Im[k] is the Imaginary component of kth harmonic.

  i is the Index of the input data value.

  k is the Index of the harmonics, k index is calculated using the harmonics number.

  x[i] is the discrete set of acquire time samples.

- **Harmonics Fk** is calculated using the following equation:

$$F[k] = \sqrt{2\left\{\left(\frac{Re(k)}{N}\right)^2 + \left(\frac{Im(k)}{N}\right)^2\right\}}$$

  Where:

  F[k] is the kth harmonic, in Amp/Volt.

  Unit of F[k] for voltage signal is Volt and for current signal is Ampere.

  Harmonics is converted into dB using the following equation:

$$F[k]^{dB} = 20 \times log_{10}(F[k]) + 120dB$$

  $F[k]^{dB}$ is the k^{th} harmonic, dBA or dBV based on the harmonics input source configuration.

  Note: In case of 'AM 14' standard, acquired signal is divided into 15 chunks and all harmonics are calculated for each chunk. For each individual harmonics maximum is taken out of 15 values.

- **RMS**: The RMS of harmonics is calculated in unit of volt or ampere using the following equation:

$$RMS = \sqrt{F1^2 + F2^2 + \cdots .. + FOH^2}$$

  F1 is the fundamental harmonics or line harmonics.

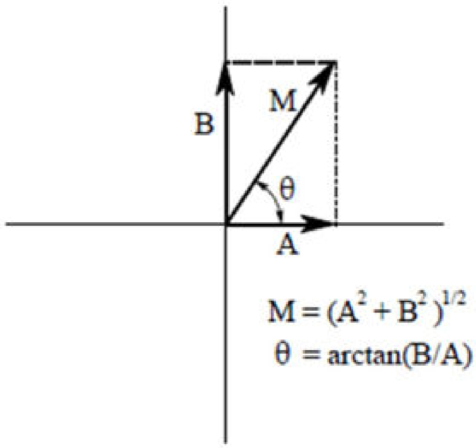
- **Total Harmonic distortion (THD-F)**: It is measured as the ratio to the RMS value of the fundamental component of the source waveform. Reported as a percentage and calculated using the following equation:

$$THD - F = \sqrt{\frac{RMS^2 - F^2[k]}{F1}} \times 100\%$$

- **Total Harmonic Distortion (THD-R)**: It is measured as a ratio to the RMS value of the source waveform. Reported as a percentage and calculated using the following equation:

$$THD - R = \sqrt{\frac{RMS^2 - F^2[k]}{RMS}} \times 100\%$$

- **Phase calculation**: The frequency domain can be expressed in polar form. In this notation, real (Re[ ])& imaginary (Im[ ]) component in frequency domain are replaced with two other arrays, called the Magnitude is Mag[i], and the Phase is written as: σ. The magnitude and phase are a pair-for-pair replacement for the real and imaginary parts.



$$Mag[i] = \sqrt{Rel[i]^2 + Img[i]^2}$$

Where:

i is the Index of harmonics.

Rel[i] is the Real component of the harmonics (Cos frequency).

Img[i] is the Imaginary component of the harmonics (Sin frequency).

$$Phase[k] = tan\frac{Img[k]}{Rel[k]}$$

If, Rel [k] < 0 & Im[k] < 0 Phase[k] = Phase[k] - π

If, Rel[k] < 0 & Im[k] > 0 Phase[k] = Phase[k] + π

- **Partial Odd Harmonics Current (POHC(M))**:

For the 21st standard higher odd order harmonics, the average values obtained for each individual odd harmonic over the full observation period, are calculated from the acquired waveform. The measured partial odd harmonic current does not exceed the partial odd harmonic current which can be calculated from the applicable limits.

$$\mathrm{POHC(M)} = \sqrt{\sum_{k=21,23}^{OH} F[k]^2}$$

Where:

M is the measured value.

POHC(S) and POHC(L) is the (S) is pass and fail status based on (L) limits.

Input Capacitance algorithm

Input capacitance measures the input capacitance of a DUT using the input voltage and current signals. Supports annotation where you can navigate between Previous and Next regions from the Results badge.

The equation for input capacitance is c = q/v. where:

- c - capacitance in farads
- q - accumulated charge, which is the integration of the current waveform
- v - peak-peak voltage

Inrush Current algorithm

Inrush Current measures the peak value of the inrush current of the DUT. It is a single source current measurement. Supports annotation where you can navigate between Previous and Next regions from the Results badge.

Power measurements: Amplitude Analysis algorithms

Cycle Base measurement algorithm

Cycle Base is calculated using the selected Base Top method. Cycle Base is the most common data value below the midpoint of the waveform, when the default Base Top method Histogram Mode is selected. This measurement is made across each cycle in the record.

Cycle Maximum measurement algorithm

Cycle Maximum is the maximum data point. Typically the most positive peak voltage.

Measurements are calculated on each cycle within the record.

Cycle Minimum measurement algorithm

Cycle Minimum is the minimum data point. Typically the most negative peak voltage.

Measurements are calculated on each cycle within the record.

Cycle Pea-to-Peak measurement algorithm

Cycle Peak to peak is the difference between Cycle Maximum and Cycle Minimum calculated for each cycle.

$PeaktoPeak = Max - Min$

Measurements are calculated on each cycle within the record.

Cycle Top measurement algorithm

Cycle Top is calculated using the selected Base Top method. Cycle Top is the most common data value above the midpoint of the waveform, when the default Base Top method Histogram Mode is selected. This measurement is made across each cycle in the record.

Cycle Amplitude measurement algorithm

Cycle Amplitude is the difference between the Top value and the Base value. This is applicable for each cycle.

Cycle Amplitude = Top - Base

Power measurements: Timing Analysis algorithms

Frequency measurement algorithm

Frequency is the reciprocal of the period. Frequency is typically measured in Hertz (Hz) where 1 Hz = 1 cycle per second.

Frequency = 1 / Period

Negative Duty Cycle measurement algorithm

Negative Duty Cycle is the ratio of the negative pulse width to the signal period, expressed as a percentage.

$$NegativeDutyCycle = \frac{NegativeWidth}{Period} \times 100\%$$

0020-018

NegativeWidth is defined in *Negative Pulse Width measurement algorithm*.

If *Period* = 0 or undefined then return an error.

Negative Pulse Width measurement algorithm

Negative Pulse Width is the time the signal remains below the mid reference level. This measurement is made on each cycle in the record.

Period measurement algorithm

Period is the time required to complete a cycle. This measurement is made on each cycle in the record.

Positive Duty Cycle measurement algorithm

Positive Duty Cycle is the ratio of the positive pulse width to the signal period, expressed as a percentage.

$$PositiveDutyCycle = \frac{PositiveWidth}{Period} \times 100\%$$

0020-021

PositiveWidth is defined in *Positive Pulse Width algorithm*.

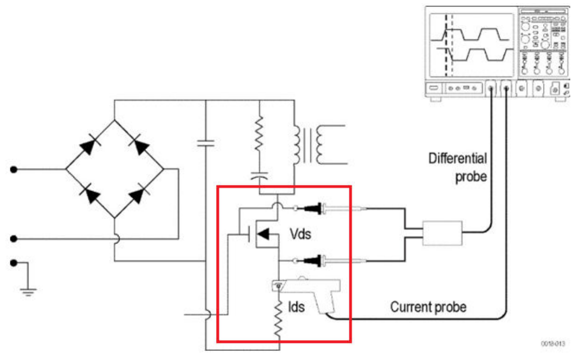
Positive Pulse Width measurement algorithm

Positive Pulse Width is the time the signal remains above the mid reference level. It is the distance from a rising edge to the next falling edge.

Power measurements: Switching Analysis algorithms

Switching Loss algorithm

Switch-Mode Power Supply (SMPS) design has three types of losses, they are Turn-On (T_{on}), Turn-off (T_{off}), and Conduction loss (Cond). To achieve the maximum efficiency, losses should be reduced. This section details about the basics of Switching Loss Analysis. A simplified SMPS schematic is shown below:



SMPS circuit diagram shows the points where switching loss can be measured. After full wave rectification, the current signal should pass through the harmonic standard and enters for DC conversation. MOSFIT plays an important role to meet the design of SMPS.

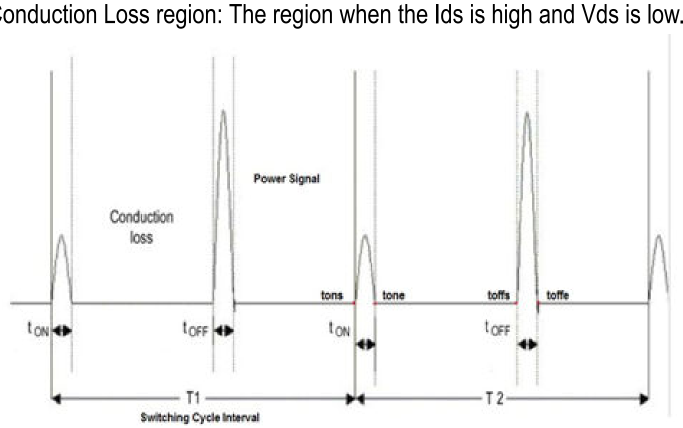
Regions of T_{on}, T_{off}, and Conduction loss (Cond) with voltage source (Vds) and current source (Ids) are shown in Switching Loss T_{on}, T_{off}, and Conduction loss regions.

All the switching losses are measured on Power signal, based on Vds and Ids transition.

T_{on} Loss region: When Vds starts rolling towards zero, the Ids starts to roll upward.

T_{off} Loss region: When Vds starts rolling upwards, the Ids start to roll towards zero.

Conduction Loss region: The region when the Ids is high and Vds is low.



T1 is the First switching cycle.

T2 is the Second switching cycle.

- **T $_{on}$ and T $_{off}$ losses**

  T_{on} and T_{off} losses per switching cycle are computed as in the following equations:

$$T_{oni} = f_{swi} \times \int_{T_{on}-Start_i}^{T_{on}-Stop_i} (V_{ds} \times I_{ds})\, dt\ Watt$$

$$T_{offi} = f_{swi} \times \int_{T_{off}-Start_i}^{T_{off}-Stop_i} (V_{ds} \times I_{ds})\, dt\ Watt$$

  Where,

  T_{oni} is the Turn on loss of the i^{th} switching cycle, in watt.

  T_{OFFi} is the Turn off loss of the i^{th} switching cycle, in watt.

fswi is the Switching frequency of the ith switching cycle, in Hz.

(T_{off}-$Stop_i$) is the stop point of T_{off} region of the i^{th} switching cycle, in time unit.

(T_{on}-$Stop_i$) is the stop point of T_{on} region of the i^{th} switching cycle, in time unit.

(T_{on}-$Start_i$) is the start point of a power T_{on} region of the i^{th} switching cycle.

(T_{off}-$Start_i$) is the start point of TOffi region, in time unit Vds - Voltage drain current, in Volts.

Ids is the drain current, in Amps.

- **Energy Loss Computation**

  Energy loss computation for T_{on} and T_{off} are calculated using the following equation:

$$T_{Eoni} = f_{swi} \times \int_{T_{on}-Start_i}^{T_{on}-Stop_i} (V_{ds} \times I_{ds})\, dt\ Watt$$

$$T_{Eoffi} = f_{swi} \times \int_{T_{off}-Start_i}^{T_{off}-Stop_i} (V_{ds} \times I_{ds})\, dt\ Watt$$

  Where:

  T_{Eoni} is the i^{th} switching cycle turn on energy loss in joule.

  T_{Eoffi} is the i^{th} switching cycle turn off energy loss in joule.

- **Computation of Conduction**

  Conduction is computed as RDS(on) value for the MOSFET is used to calculate total loss in the application. To measure conduction loss and energy in a MOSFET, using the following equation:

$$Cond_i = f_{swi} \times \int_{T_{on}-Stop_i}^{T_{off}-Start_i} \left(R_{ds}(on) \times I_{ds}^2\right) dt\ Watt$$

$$Cond_{Ei} = \int_{T_{on}-Stop_i}^{T_{off}-Start_i} \left(R_{ds}(on) \times I_{ds}^2\right) dt\ Joule$$

  Where:

  R_{DS}(on) is the Dynamic resistance, in Ω.

  $Cond_i$ is the Conduction Loss in Watt, $Cond_{Ei}$ is the Conduction Energy Loss in Joule.

  $Cond_i$ is the i^{th} switching cycle conduction loss in watt. $Cond_{Ei}$ is the i^{th} switching cycle conduction energy in joules.

  f_{swi} is the i^{th} switching cycle frequency, in Hz.

  I is the Cycle number.

  Nc is the Number of conduction cycles.

  To measure conduction loss in a BJT/IGBT, using the following equation:

$$Cond_i = f_{swi} \times \int_{T_{on}-Stop_i}^{T_{off}-Start_i} (V_{ce}(SAT) \times I_{ds})\, dt\ watt$$

$$Cond_{ei} = \int_{T_{on}-Stop_i}^{T_{off}-Start_i} (V_{ce}(SAT) \times I_{ds})\, dt\ joule$$

  Where, V_{CE}(sat) is the voltage in volt, which should be configure in application.

- **Computation of Average Loss and Total Loss**

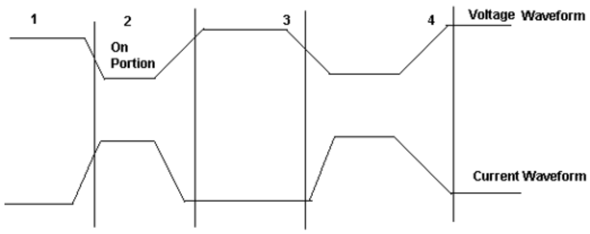
  Average and Total loss are calculated using the following equation:

$$T_{on_{Avg}} = \frac{1}{Nc}\sum_{i=1}^{Nc} T_{on_i}$$

$$T_{off_{Avg}} = \frac{1}{Nc}\sum_{i=1}^{Nc} T_{off_i}$$

$$Cond_{Avg} = \frac{1}{Nc}\sum_{i=1}^{Nc} Cond_i$$

$$TotalLoss = T_{on_{Avg}} + T_{off_{Avg}} + Cond_{Avg}$$



Total Switching Loss=Ton Loss+Toff Loss+Conduction Loss

Concept to identify Ton and Toff using gate voltage for edge analysis:

Use gate voltage for edge analysis with default 50% edge level and hysteresis 10%.

- To find the start of Ton: The start of the Ton is 5% or 1.5 V whichever is lower on the rise slope of the gate voltage.
- To find the stop of Ton: The start index on the switch voltage is 5% or 1.5 V of the rise slope gate voltage. Move forward on the switch voltage from the start index until 5% or the configured level is met.
- To find the start of Toff: The start index is 80% of the gate voltage. From the Stop index, search for 5% of the switch voltage (on rise slope).
- To find the stop of Toff: The 80% of the gate voltage is start index. From this start index on switch current (fall slope) move forward until 5% of the max (switch current) is met.

dv/dt algorithm

dv/dt represents the rate at which the voltage changes during switching. The application uses the math feature to provide a differentiation waveform of the voltage input.

When you run the measurement, the application calculates dv/dt for the first edge by taking the default levels as 10% and 90% and displays the results.

Select a specific section of the waveform on the live signal by providing inputs for high and low levels in terms of percentage and absolute value of voltage and current. Select the edge of interest by viewing it visually on the oscilloscope. You can also enter the edge number on the results panel. The application displays the results for the selected edge and levels on the results panel.

The application calculates dv/dt using the following equation:

$$\frac{dv}{dt} = \frac{v(R_t) - v(R_B)}{R_T - R_A}$$

Where:

X is the timing values.

Y is the vertical (voltage) values of the waveform data between the cursors.

di/dt algorithm

di/dt measurement represents the rate at which the current changes during switching. The application uses the oscilloscope's built in math feature to provide a differentiation waveform of the current input.

When you run the measurement, the application calculates di/dt for the first edge by considering the default levels as 10% and 90% and displays the results.

Select a specific section of the waveform on the live signal by providing inputs for high and low levels in terms of percentage and absolute value of voltage and current. Select the edge of interest by viewing it visually on the oscilloscope. You can also enter the edge number on the results panel. The application displays the results for the selected edge and levels on the results panel.

The application calculates di/dt using the following equation:

$$\frac{di}{dt} = \frac{i(R_t) - i(R_B)}{R_T - R_A}$$

Where:

I is the timing value.

t is the vertical values of the waveform data.

SOA algorithm

SOA plots the graph of the voltage and current waveform. You can configure SOA mask by creating voltage and current co-ordinates in the mask table. You can save and recall mask files. The extension of the mask file name is .pwrmsk.

SOA has two outputs:

1. SOA X-Y plot with mask showing hits on the mask.
2. SOA summary on the measurement badge displays the number of hits with Pass/Fail status.

$R_{DS(on)}$ algorithm

Dynamic resistance ($R_{DS(on)}$) is the resistance offered by a switching device when it is in the ON condition. Power helps to monitor the dynamic resistance using a Time Trend plot. You can calculate $R_{DS(on)}$ by using the below formula:

$$R_{DS(on)} = v/i = \mathrm{v(t)/i(t)}$$

where, v/i is the ratio of voltage to current sample points. It is applicable for switching semiconductors.

Power measurements: Magnetic Analysis algorithms

The Magnetic measurements include I vs. ∫V, Inductance, Magnetic Property, and Magnetic Loss. When using I vs. ∫V measurement, take care to check that the voltage ‘V’ does not have any DC components.

Use AC coupling on the input signal to avoid any DC shifts on the integral of the voltage waveform.

Magnetic Analysis is supported for 4*/5/6 series.

* For 4 Series Oscilloscopes the record length is limited to 5 M.

Inductance algorithm

It measures the inductance (the integral of the voltage divided by the current) of a magnetic component during in-circuit operation. The application creates a single cycle by averaging multiple cycles of current and integrated voltage.

An electric circuit has electromotive force created by a change of current in the same circuit known as self-inductance or in a neighboring circuit as mutual-inductance. Inductance displays the Inductance plot. The unit of Inductance is Henry.

The Inductance provides a view of the core behavior while it is under operation.

I vs (integral of) V algorithm

The XY plot of integral of voltage against current. The integral of the voltage is proportional to B. The integral of current is proportional to H.

Magnetic Loss algorithm

The average value of the product of the voltage times current through the inductor. This represents the total loss of the magnetic device and consists of resistive and eddy current losses during circuit operation.

Magnetic Property algorithm

The following diagram shows a plot of Hysteresis in a typical magnetic material (magnetic field strength (H) versus saturation flux density (B)).

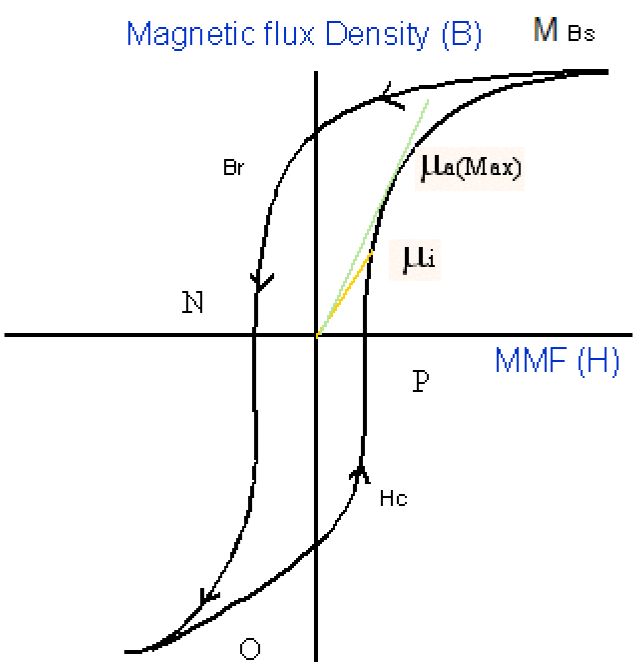


Figure 32: B-H curve

B_s is the Saturation Flux Density

B_r is the Remanence Flux Density

H_c is the Coercive Force (Hc)

U_i is the Initial Permeability

U_a is the Max Amplitude permeability

H is the magnetic field used to induce Magnetic Flux in the magnetic material

MMF is Magneto Motive Force, it is also known as Magnetic Field Strength

Note: The data waveform starts from the Max value of H, decreases, and then increases again (M-N-O-P).

Magnetic Field Strength (H)

The previous figure shows the hysteresis in a typical magnetic material. The magnetic field induces a magnetic flux in the DUT. The units of measurement are Ampere per meter in SI unit, and Oersted in CGS unit.

Saturation Flux Density (B_s)

The Saturation Flux Density represents maximum magnetic flux density that can be induced in the magnetic material regardless of the magnitude of the externally applied field H. This is represented in the B-H curve were B value is considered, when H is maximum.

B_s = Max (Bk)

The Magnetic Field Intensity H is also calculated on the maximum flux density cycle Bk.

B_s = Max (B)

Index I where H is maximum

I = Index of (Max(H))

B_s = B(I) (2)

Remanence (B_r):

Remanence is the Induced magnetic flux density that remains in the material after the externally applied magnetic field (H) is returned to zero during the generation of the hysteresis loop. This represents max value of B for all values of zero value of H in the B waveform.

Find the index at zero value of H on the H waveform and calculate the maximum value of B from these indices.

Let 'q' be the index at zero value of H on H waveform. Let the q1 and q2 be the indices of the waveform. Calculate the value of B at the Indices q1 and q2 on the K^{th} cycle. The maximum magnitude value of B is the remanence Flux density.

Coercive Force(H_c)

Coercivity is the value of H found at the intersection of the H-axis with the hysteresis loop. This represents the external field required to cause the induced flux density (B) to reach zero during the measurement cycle of a hysteresis loop. H_C is symmetrical with the positive and negative axis.

Coercivity H_c is calculated on the cycle where the maximum flux density occurs in the entire acquired waveform.

Finding the index at zero B value on the B waveform: Let 'q' be the index at zero value of B on B waveform. Let q1 and q2 be the indices on the B waveform where B is zero.

Coercivity is the maximum magnitude of the H data at the indices of q1 and q2 is the coercivity.

Permeability

The ratio of B and H calculated on the B_k cycle. Select the points on the B-H plot using the cursor and calculate the slope of the B-H curve using the data selected between the cursor. You can choose the portion of the plot using the cursor to obtain the results.

In the B-H curve plot there is a provision to select the points using the waveform cursors 1 and 2. This is used to calculate the slope of B-H curve using the data between the cursors 1 and 2. The computed slope as explained below displays the scalar value of the permeability (μ) in the results badge and table.

In slope calculation, there are N points between the cursors:

Find

H_{av} = (H1+H2+…Hn)/N

B_{av} = (B1+B2+…Bn)/N

H_{normi} = Hi-Hav, i=1..N

B_{normi} = Bi-Bav, i=1..N

B/H = SUM ($H_{normi}1*B_{normi}1+H_{normi}2*B_{normi}2+…+H_{normi}N*B_{normi}N$) / SUM($H_{normi}1*H_{normi}1+H_{normi}2*H_{normi}2+…$
$H_{normi}N*H_{normi}N$)

Where,

B/H is the Permeability (µ)

Magnetic Loss

Magnetic property computes total magnetic loss and reports hysteresis loss (Hys)

The total magnetic loss and its components account for multiple secondary current windings along with primary windings for the computation.

Power measurements: Output Analysis algorithms

Line Ripple measurement algorithm

The Line Ripple measures RMS at configured Line Ripple frequency and peak to peak of time domain waveform for the configured phases.

Switching Ripple measurement algorithm

Switching Ripple measures RMS at configured Switching frequency and peak to peak of time domain waveform for the configured phases.

Efficiency algorithm

Efficiency measures the ratio of output power to input power for a power conversion circuit. The measurement supports three outputs, the maximum and efficiency is computed at each output. If there are more than one output, then the total efficiency value will be computed. You can configure AC/DC type at input and output sides.

Note: This measurement uses a minimum of four sources and a maximum of eight sources. Two sources are used to measure the input voltage and current to the power supply. Two to six sources are used to measure the power supply output voltage and current.

Math is used to compute Input Power waveform using V * I equation. Computation happens over an exact integer number of cycles to get an accurate result, that is, from the first zero crossing to the last zero crossing.

Math calculates the Input Power, using

$$Math1 = V(t)_{in} * I(t)_{in}$$

Similarly another math calculates the Output Power using V * I at the output side

$$Math2 = V(t)_{o1} * I(t)_{o1}$$

Power Efficiency = Output Power / Input Power

$$Efficiency\ in\ \% = \frac{\sum(Math1)}{\sum(Math2)} * 100$$

Note:

- If more cycles are captured, then the power solution will automatically consider all the integral cycles, starting from first (mid) crossing to the last valid (mid) crossing.
- For multiple outputs, the total efficiency is computed as the ratio of input power to the sum of output(s) power. The total efficiency is expected to be within 100%, since all the outputs are from a same input power. If the efficiency is greater than 100 %, then the application displays a warning message.

Turn on time

Turn On Time measures the time delay between load current going high to other rail outputs going high when power on happens. Annotation will happen on input and corresponding output waveforms.

Turn off time

Turn Off Time measures the time delay between current going low to other rail outputs going low when power down happens.

Power measurements: Frequency Response Analysis algorithms

Control loop response measurement algorithm

Control Loop Response computes and plots gain as 20 log (V_{out}/V_{in}) and phase difference between V_{in} and V_{out} at each frequency within the swept band. The resulting plot is commonly referred to as a Bode Plot.

Points per decade is 10 by default, maximum is 100.

Start frequency is 100 Hz

Stop frequency is 10 MHz

$$Number\ of\ frequency\ points = ppd * (6 - 2)$$

where:

- 6 is $10^6 = 10\,MHz$
- 2 is $10^2 = 100\,Hz$

The gain and phase margin are two metrics to tell the stability of the system.

Gain for $$Bode = 20 * LOG10(\frac{V_{out}}{V_{in}})$$

where:

- V_{out} is the amplitude of the input signal
- V_{in} is the input signal

Phase for Bode is the time shift between the output and input signals.

Power supply rejection ratio measurement algorithm

Power Supply Rejection Ratio (PSRR) measures both the modulated input and output AC voltage levels and then computes the rejection ratio as 20 log (V_{in}/V_{out}) at each frequency within the swept band.

Impedance measurement algorithm

Impedance is computed as the channel ratio of V_{out}/V_{in} for each frequency.

IMDA measurements: Electrical Analysis algorithms

Power Quality measurement algorithm

Refer to *Power Quality measurement algorithm* on page 440.

Power Quality is available under Input Analysis. This computes sub-measurements for all phases (3 phases) based on the wiring configuration. Phase of each voltage and current is found using DFT at operation frequency. Phase angle is calculated as difference between current phase and voltage phase. Voltage is always considered as reference. Power Factor is computed based on cos (Phase difference). Reactive power is computed based on formula.

$$RePwr = \sqrt{ApPwr^2 - TpPwr^2}$$

The sign of the RePwr is derived from sign of V and I Phase difference

The results for all the configured phases are displayed.

Harmonics measurement algorithm

Refer to *Harmonics algorithm* on page 442Harmonics is available under Input Analysis. This computes sub-measurements for all phases (3 phases) based on the wiring configuration. The algorithm is same as in 5-PWR Harmonics measurement where as in 5-PWR computation happens for one pair of Voltage and Current.

Input Voltage measurement algorithm

Refer to *Base measurement algorithm* on page 426.

The V_{RMS} Input Voltage is available under Input Analysis. This computes RMS value for all Voltage phases (3 phases) based on the wiring configuration. The algorithm is same as in standard RMS measurement. Here results are shown for configured phase voltages.

Input Current measurement algorithm

Refer to *Base measurement algorithm* on page 426.

The I_{RMS} Input Current is available under Input Analysis. This computes RMS value for all Current phases (3 phases) based on the wiring configuration. The algorithm is same as in standard RMS measurement. Here results are shown for configured phase currents.

Input Power measurement algorithm

This is a sub-set of Power Quality measurement. Input Power is available under Input Analysis. This computes sub-measurements for all phases (3 phases) based on the wiring configuration. The algorithm is same as in Power Quality. Refer to *Power Quality measurement algorithm* on page 454

Ripple Analysis algorithms

The Ripple Analysis measures RMS at configured Ripple frequency and peak to peak of time domain waveform for the configured phases.

Line Ripple measurement algorithm

The Line Ripple measures RMS at configured Line Ripple frequency and peak to peak of time domain waveform for the configured phases.

Switching Ripple measurement algorithm

Switching Ripple measures RMS at configured Switching frequency and peak to peak of time domain waveform for the configured phases.

IMDA measurements: Output Analysis algorithms

Efficiency measurement algorithm

Refer to *Efficiency algorithm* on page 452

Efficiency measures the ratio of output power to input power for a Motor and Drive per phase. Displays efficiency results for each phase and total system efficiency.

Note: Efficiency may not report proper results with zero input signals

Table 13: Invaild Input and Ouput Wiring for Efficiency measurement

Input	Output
DC 1V1I	3V3I (3P3W)
1V1I (1P2W)	3V3I (3P3W)
3V3I (3P3W)	1V1I (1P2W)
3V3I (3P3W)	2V2I (3P3W)
3V3I (3P3W)	3V3I (3P3W)
2V2I (1P3W)	3V3I (3P4W)
2V2I (3P3W)	3V3I (3P3W)
3V3I (3P4W)	3V3I (3P3W)
2V2I (3P3W)	1V1I (1P2W)

Table 14: Not Supported Input and Ouput Wiring for Efficiency measurement

Input	Output
3V3I (3P4W)	2V2I (3P3W)
3V3I (3P4W)	3V3I (3P3W)
3V3I (3P4W)	3V3I (3P4W)
2V2I (1P3W)	3V3I (3P4W)
2V2I (3P3W)	3V3I (3P4W)

Line-to-Line and Line-to-Neutral: When selected converts mathematically from Line-to-Line results to Line-to-Neutral. This is available only for 3V3I wiring. The formula applied is for all Voltage Line to Line results divide by sqrt(3).

Phasor Diagram measurement algorithm

The measurement displays the magnitude and phase angle between Voltage(V) and Current(I) vectors in a Phasor plot. The V and I vectors depend on the wiring configuration. The Phasor plot is applicable to the input analysis measurement of Power Quality. Each Vector is represented by RMS value and phase is computed using DFT method.

DQ0 measurement algorithm

Direct Quadrature Zero Transformation (DQ0) transforms from three phase output voltage or current signals into D-Q-0 components using matrix coefficients.

DDR Amplitude Measurement algorithms

AOS (DDR)

AOS is the total area of the signal above specified reference level.

AOS Per tCK (DDR)

AOS Per tCK is the total area of the signal that crosses the specified reference level, calculated over consecutive periods. It is applicable to clock and address/command waveforms.

AOS Per UI (DDR)

AOS Per UI is the total area of the signal that crosses specified reference level, calculated over consecutive unit intervals. It is applicable to data and data strobe waveforms.

AUS (DDR)

AUS is the total area of the signal below specified reference level.

AUS Per tCK (DDR)

AUS Per tCK is the total area of the signal that crosses the specified reference level, calculated over consecutive periods. It is applicable to clock and address/command waveforms.

AUS Per UI (DDR)

AUS Per UI is the total area of the signal that crosses specified reference level, calculated over consecutive unit intervals. It is applicable to data and data strobe waveforms.

Vix(ac) (DDR)

Vix(ac) is the differential input cross-point voltage measured from the actual crossover voltage and its complement signal to a specified reference voltage. This is measured on the single ended signal.

DDR Time Measurement algorithms

tRPRE (DDR)

tRPRE is the width of the Read burst' preamble. This is measured from the exit of tristate to the first driving edge of the differential strobe.

tWPRE (DDR)

tWPRE is the width of Write burst preamble. It is measured from the exit of tristate to the first driving edge of the differential strobe.

tPST (DDR)

tPST is the width of Read or Write burst postamble. It is measured from the last falling edge crossing mid reference level to the start of an undriven state (judged by a rising trend as per JEDEC specification).

Hold Diff (DDR)

Hold Diff is the elapsed time between the designated edge of the single ended waveform and the designated edge of a differential waveform. The closest single-ended waveform edge to the differential waveform edge that falls within the range limits is used for the measurement.

Setup Diff (DDR)

Setup Diff is the elapsed time between the designated edge of a single ended waveform and when the differential waveform crosses its own voltage reference level. The closest single ended waveform edge to the differential waveform edge that falls within the range limits is used for the measurement.

tCK(avg) (DDR)

tCK(avg) is the average clock period across a sliding 200-cycle window.

$$tCK(avg) = \left[\sum_{j=1}^{N} tCK_j\right] / N$$

Where,

N=200 (configurable)

tCH(avg) (DDR)

tCH(avg) is the average high pulse width calculated across a sliding 200 cycle window of consecutive high pulses.

$$tCH(avg) = \left[\sum_{j=1}^{N} tCH_j\right] / (N * tCK(avg))$$

Where,

N=200 (configurable)

tCL(avg) (DDR)

tCL(avg) is the average low pulse width calculated across a sliding 200 cycle window of consecutive low pulses.

$$tCL(avg) = \left[\sum_{j=1}^{N} tCL_j\right] / (N * tCK(avg))$$

Where,

N=200 (configurable)

tCH(abs) (DDR)

tCH(abs) is the high pulse width of the differential clock signal. It is the time duration, the waveform remains above the mid reference voltage level.

$$tCH(abs) = tCH/tCK(avg)$$

tCL(abs) (DDR)

tCL(abs) is the low pulse width of the differential clock signal. It is the amount of time the waveform remains below the mid reference voltage level.

$$tCL(abs) = tCL/tCK(avg)$$

tJIT(duty) (DDR)

tJIT(duty) is the largest elapsed time between tCH from tCH(avg) or tCL from tCL(avg) for a 200-cycle window.

$$tJIT(duty) = Min/max\ of\ \{tJIT(CH), tJIT(CL)\}$$

Where,

tJIT(CH) = {tCHi - tCH(avg)}

tJIT(CL) = {tCLi- tCL(avg)}

i =1 to N

N=200 (configurable)

tJIT(per) (DDR)

tJIT(per) is the largest elapsed time between the tCK from tCK(avg) for a 200-cycle sliding window.

$$tJIT(per) = Min/Max\ of\ \{tCK_i - tCK_{(avg)}\}$$

Where,

i =1 to N

N=200 (configurable)

tJIT(cc) (DDR)

tJIT(cc) is the absolute difference in clock period between two consecutive clock cycles.

$$tJIT(cc) = Max\ of\ |tCK_{i+1} - tCk_i|$$

tERR(n) (DDR)

tERR(n) is the cumulative error across multiple consecutive cycles from tCK(avg). It measures time difference between the sum of clock period for a 200-cycle window to n times tCK(avg).

$$tERR(nper) = \left(\sum_{j-1}^{i+n-1} tCK_j \right) - n * tCK(avg)$$

Where,

n = 2 for tERR(2 per)

n = 3 for tERR(3 per) and so on.

tERR(m-n) (DDR)

tERR(m-n)

tERR(m-n) is the cumulative error across multiple consecutive predefined cycles from tCK(avg). This is measured similar to tERR(n per).

$$tERR(nper) = \left(\sum_{j-1}^{i+n-1} tCK_j \right) - n * tCK(avg)$$

Where,

6 ≤ n ≤ 10 for tERR (6-10 per)

11 ≤ n ≤ 50 for tERR (11-50 per)

tDQSCK (DDR)

tDQSCK is the strobe output access time from differential clock. tDQSCK is measured between the rising edge of clock before or after the differential strobe Read preamble time. The edge locations are determined by the mid-reference voltage levels.

tCKSRE

tCKSRE is the valid clock cycles required after Self Refresh Entry (SRE) command. Changing the input clock frequency or the supply voltage is permissible only after tCKSRE time when the SRE command is registered.

tCKSRX

tCKSRX is the valid clock cycles required before the Self Refresh Exit (SRX) command. Changing the input clock frequency or the supply voltage is permissible provided the new clock frequency or supply voltage is stable for the tCKSRX time prior to SRX command.

tCMD-CMD

tCMD-CMD is the elapsed time between two logic states.

DPM Ripple analysis measurement

Ripple

Ripple computes the DC rail output by setting up the horizontal and vertical parameters for the power rail probe. The power rail autoset sets the horizontal and vertical parameters for the power rail probe. Ripple computes peak-to-peak and RMS values and the RMS is computed at configured ripple frequency. You can also select Record or Cycle mode. Select Cycle mode to specify the clock edges.

DPM Power sequence analysis algorithm

Overshoot

Overshoot is the difference between signal Maximum and a user-configured Reference voltage. This measurement can be made across the entire record or on each cycle in the record. For a cycle measurement, the user can specify to use the CLK source as an edge qualifier.

Undershoot

Undershoot is the difference between signal Minimum and a user-configured Reference voltage. This measurement can be made across the entire record or on each cycle in the record. For a cycle measurement, the user can specify to use the CLK source as an edge qualifier.

DC Rail Voltage

DC Rail Voltage is the mean value of each rail output. This can be performed per cycle or for the entire record. In cycle mode, the user can specify Edge qualifier as the source to find edges. Annotation and navigation available in cycle mode. One can traverse in navigation mode using Prev and Next buttons.

Turn on Overshoot

Turn on Overshoot computes the maximum amplitude value of the dc rail output voltage during turn ON of the DUT. This supports multiple rail outputs. User has to provide input signal level to trigger properly and measurement happens on output rail. Annotation of max value on the output is shown as cross hair per rail output.

DPM Transient analysis measurement algorithm

Turn on time

Turn On Time measures the time delay between load current going high to other rail outputs going high when power on happens. Annotation will happen on input and corresponding output waveforms.

Turn off time

Turn Off Time measures the time delay between current going low to other rail outputs going low when power down happens.

References

DPM badge error and warning messages

These tables provide information to help resolve error or warning messages that appear on the DPM measurement badges.

Error messages displayed on the DPM measurement badges

Error Message	Cause	Suggestion
Empty input	The oscilloscope is waiting for a Single Sequence trigger.	Check that the oscilloscope has a valid input waveform.
Input Trigger Level is greater than Maximum Input Voltage	The specified Input Trigger Level is higher than the Maximum Input Voltage in the DPM Turn On/Turn Off measurement.	Adjust the Input Trigger Level to be within the range of the Maximum Input Voltage waveform.

Warning messages displayed on the DPM measurement badges

Warning Message	Cause	Suggestion
Incorrect Input Trigger level	Not able to trigger on the specified Input Trigger Level as set in the DPM Turn On/ Turn Off measurement.	Adjust the Input Trigger Level to be within the range of the input source waveform.
Output<x> Voltage Level reached before input trigger.	Turn On Time value is negative for the given configuration in the DPM Turn On/Turn Off measurement.	Adjust the Output Voltage Level given for Rail<x> or Adjust the Input Trigger Level for a positive Turn On Time.
Incorrect Output<x> Voltage level.	Not able to find the specified Output Voltage level for Rail<x> output source as set in the DPM Turn On/Turn Off measurement.	Adjust the Output Voltage Level given for Rail<x> such that voltage is within the span of the Rail<x> source waveform of the measurement.

Power badge error and warning messages

These tables provide information to help resolve error or warning messages that appear on the power measurement badges.

Error messages displayed on the power measurement badges

Error Message	Cause	Suggestion
Empty input	The oscilloscope is waiting for a Single Seq trigger.	Check that the oscilloscope has a valid input waveform.

Table continued…

Error Message	Cause	Suggestion
Input source mismatch	Incorrect combination of live (active) and Ref source waveforms. In the case of two Ref waveforms, the two waveforms recalled must have the same record length and have been acquired with the same timebase setting.	Acquire two waveforms of the same type (active or ref) Reacquire the Ref waveforms with the same time base and record length.
Too few edges	No edges found on the input waveform as the waveform could be very noisy.	Increase the Hysteresis band and rerun the measurement. The Hysteresis values are set in the measurement configuration menu Reference Levels panel. Change the Acquisition mode and BW limit to reduce the noise in the waveform.
Error from Frequency measurement	Not able to measure frequency value on the input waveform due to noisy waveform.	Adjust the measurement Hysteresis band to compute edges properly so that frequency can be measured. The Hysteresis values are set in the measurement configuration menu Reference Levels panel.
Error from RMS measurement	Not able to measure RMS value on the input waveform due to noisy waveform.	Adjust the measurement Hysteresis band to compute edges properly so that RMS can be measured. The Hysteresis values are set in the measurement configuration menu Reference Levels panel.
Error from MaxElement measurement	Not able to measure MIN/MAX value on the input waveform due to noisy waveform.	Adjust Hysteresis band to compute edges properly so that MIN/MAX can be measured. The Hysteresis values are set in the measurement configuration menu Reference Levels panel.
Invert probing points	Polarity mismatch between the probe to test points on the DUT	Check and inverse the probe polarity to match with DUT test points.
Too few edges	The oscilloscope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Use **Power Autoset** to optimize the oscilloscope settings for power measurements.
Insufficient sampling rate	Not enough sampling rate used to capture signal	Increase the sample rate proportional to input signal frequency. Use **Power Autoset** to optimize the oscilloscope settings for power measurements.
Too few cycles	Not enough sampling rate used to capture signal	Increase the sample rate proportional to input signal frequency. Use **Power Autoset** to optimize the oscilloscope settings for power measurements.
Not enough data	The oscilloscope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Use **Power Autoset** to optimize the oscilloscope settings for power measurements.

Table continued…

Error Message	Cause	Suggestion
Input source mismatch. Current signal expected in current source.	Mismatch between input waveform and wrong source set. For example, if current waveform is used for voltage configuration, this error is shown.	Make sure a Current waveform is set as the input to Current Harmonics measurement.
Input source mismatch. Voltage signal expected in voltage source.	Mismatch between input waveform and wrong source set. For example if voltage waveform is used for current configuration, this error is shown.	Make sure a Voltage waveform is set as the input to Voltage Harmonics measurement.
Invalid input	Incorrect combination of live (active) and Ref source waveforms. In the case of two Ref waveforms, the two waveforms recalled must have the same record length and have been acquired with the same timebase setting.	Acquire two waveforms of the same type (active or ref) Reacquire the Ref waveforms with the same time base and record length.
Insufficient horizontal resolution	Not enough sampling rate used to capture signal	Increase the sample rate proportional to input signal frequency. Use **Power Autoset** to optimize the oscilloscope settings for power measurements.
Invalid signal frequency	Not able to measure frequency value on the input waveform due to noisy waveform	Adjust Hysteresis band to compute edges properly so that frequency can be measured. The Hysteresis values are set in the measurement configuration menu Reference Levels panel.
Not enough data	The oscilloscope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Use **Power Autoset** to optimize the oscilloscope settings for power measurements.
Warning from RMS measurement	Not able to measure RMS value on the input waveform due to noisy waveform	Adjust Hysteresis band to compute edges properly so that RMS can be measured. The Hysteresis values are set in the measurement configuration menu Reference Levels panel.
Too few cycles	The oscilloscope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Use **Power Autoset** to optimize the oscilloscope settings for power measurements.
Not enough data	Table results will not show values.	Check input waveforms and configuration. Recommended to run Power Autoset and then observe the measurement results.
Too few edges	No of edges found in the gated region is less than the number of edges required for the algorithm for calculation.	Increase the record length.
Pos clipping	Vertical scale is not set properly.	Use **Power Autoset** to set the vertical scale automatically.

Table continued…

Error Message	Cause	Suggestion
Neg clipping	Vertical scale is not set properly.	Use **Power Autoset** to set the vertical scale automatically.
Invalid range Indices	No data in the gated region.	Make sure that the input waveforms are valid.
Empty Input	Input waveform has no data.	Make sure to set all the valid inputs.
Input all DC	Input signal is pure DC.	Change the input signal to AC or add some AC component to DC signal.
No data in range	No data in the gated region.	Make sure that the input waveforms are valid.
Unable to connect to generator	Unable to connect to the generator at the specified IP address.	Check the IP address, verify that the generator is powered on, and try to run the measurement again.
No AFG option	AFG option is disabled.	An AFG license is needed.
No Results	Results are not available	An Internal error has occurred. Results are not available. Check the connections and try again.
Ampl set error	Measurement Runtime Error.	The generator amplitude and impedance values do not match. Restart the measurement.
V/H scale error	Unable to perform vertical or horizontal scaling	Restart the measurement.
Runtime error	Measurement Runtime Error.	Windows OS: Exit the TekScope application and relaunch again. Standard instrument (not Windows OS): Restart the oscilloscope.
Incompatible meas	When any other measurement is added in addition to FRA measurement.	Delete all the measurements other than FRA measurement. There should be only one measurement at a time.
Input Trigger Level is greater than Maximum Input Voltage	Input Trigger Level provided is more than the Maximum Input Voltage in Turn On/ Turn Off measurement.	Adjust the Input Trigger Level such that it is within the range of Maximum Input Voltage waveform.
Not Supported	Selected configurations is not applicable for the IMDA measurement	Change Input or Output Wiring
Invalid Wiring	Selected wiring is not applicable for IMDA Efficiency measurement	Change Input or Output Wiring

Warning messages displayed on the Power measurement badges

Warning Message	Cause	Suggestion
No data in range	There are no data between the two cursors.	Place the cursors appropriately.

Table continued…

Warning Message	Cause	Suggestion
Warning from Frequency measurement	Not able to measure frequency value on the input waveform due to noisy waveform.	Adjust Hysteresis band to compute edges properly so that frequency can be measured. The Hysteresis values are set in the measurement configuration menu Reference Levels panel.
Warning from RMS measurement	Not able to measure RMS value on the input waveform due to noisy waveform.	Adjust Hysteresis band to compute edges properly so that RMS can be measured. The Hysteresis values are set in the measurement configuration menu Reference Levels panel.
Warning from MaxElement measurement	Not able to measure MIN/MAX value on the input waveform due to noisy waveform.	Adjust Hysteresis band to compute edges properly so that MIN/MAX can be measured. The Hysteresis values are set in the measurement configuration menu Reference Levels panel.
Voltage source expected	Mismatch between input waveform and measurement. For example if a current waveform is used for dv/dt, this warning is shown.	Make sure a voltage waveform is set as the voltage source input.
Current source expected	Mismatch between input waveform and measurement. For example if a voltage waveform is used for di/dt, this warning is shown.	Make sure a current waveform is set as the current source input.
Warning from RMS measurement	Not able to measure RMS value on the input waveform due to noisy waveform	Adjust the Hysteresis band to compute edges properly so that RMS can be measured. The Hysteresis values are set in the measurement configuration menu Reference Levels panel.
Invalid Mask	Mask is not of the standard format, cannot have closed mask coordinates or intersection of the coordinate points	Recreate the mask from the SOA mask configure table such that inner mask coordinates do not intersect.
No data in range	No data in the gated region.	Make sure that the input waveforms are valid.
Input > 80% DC	Input signal has higher DC component.	Change the input signal to AC or add more AC component to the DC signal.
Trigger Level is greater than maximum voltage	Trigger level set is greater then the maximum voltage.	Set the maximum voltage higher than the trigger level.
Output Voltage Level reached before input trigger	Output is turned on before the input or there is a spike in the output voltage.	Make sure that the input and output voltage are not interchanged, output is dependent on the input, and run the measurement again.
No PM value	No zero dB crossing on the Gain curve.	Check the test setup connections and try again.
No GM value	No zero degree crossing on the Phase curve.	Check the test setup connections and try again.
Incorrect Input Trigger level	Not able to find the specified Input Trigger Level provided in input source of Turn On/ Turn Off measurement .	Adjust the Input Trigger Level such that voltage is within the span of input source waveform of measurement.

Table continued…

Warning Message	Cause	Suggestion
Actual Input Level used in calculation is <n>V	Input Trigger Level provided is over ridden to value<n> by Turn On or Turn Off measurement.	Information about automation of feature of Turn On or Turn Off measurement. Increase the sample rate for better accuracy.
Actual Output<x> Voltage Level used in calculation is <n>V.	Output<x> Voltage Level provided is over ridden to value<n> by Turn On or Turn Off measurement.	Information about automation of feature of Turn On or Turn Off measurement. Increase the sample rate for better accuracy.

Information messages displayed on the Power measurement badges

Information messages	Description
Actual Input Level used in calculation is n V.	In Turn On Time and Turn Off Time measurements, if the input level is not equal to the maximum of input waveform (n), the maximum of the input waveform is used for computation and the same is displayed as the information message.
Actual Output x Level used in calculation is n V.	In Turn On Time and Turn Off Time measurements, if the output x level is not equal to the maximum output x waveform (x), the maximum of the output x waveform is used for computation and the same is displayed as the information message.

Index

C

D

E

F

J

K

L

M

Q

R

S

T

U

V

W

X

Z