LECROY

WAVEEXPERT OSCILLOSCOPE

OPERATOR'S MANUAL

JULY 2005

LeCroy Corporation
700 Chestnut Ridge Road
Chestnut Ridge, NY 10977–6499
Tel: (845) 578 6020, Fax: (845) 578 5985

Internet: www.lecroy.com

Manufactured under an ISO 9000 Registered Quality Management System

Visit www.lecroy.com to view the certificate.

This electronic product is subject to disposal and recycling regulations that vary by country and region. Many countries prohibit the disposal of waste electronic equipment in standard waste receptacles.

For more information about proper disposal and recycling of your LeCroy product, please visit www.lecroy.com/recycle.

WE-OM-E Rev A

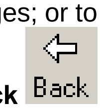
913294-00

BLANK PAGE

How to Use On-line Help

Type Styles

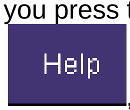
Activators of pop-up text and images appear as green, underlined, italic: *Pop-up*. To close pop-up text and images after opening them, touch the pop-up text again.

Link text appears blue and underlined: Link. Links jump you to other topics, URLs, or images; or to another location within the same Help window. After making a jump, you can touch the **Back** 

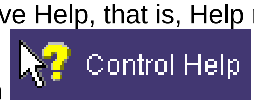
icon in the toolbar at the top of the Help window to return to the Help screen you just left. With each touch of the **Back** icon, you return to the preceding Help screen.

Instrument Help

When you press the front panel Help button 

(if available), or touch the on-screen **Help** button 

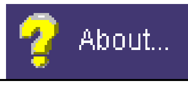
, you will be presented with a menu: you can choose either to have information found for you automatically or to search for information yourself.

If you want context-sensitive Help, that is, Help related to what was displayed on the screen when you requested Help, touch 

in the drop-down menu, then touch the on-screen control (or front panel button or knob) that you need information about. The instrument will automatically display Help about that control.

If you want information about something not displayed on the screen, touch one of the buttons inside the drop-down menu to display the on-line Help manual:

	**Contents** displays the Table of Contents.
	**Index** displays an alphabetical listing of keywords.
	**Search** locates every occurrence of the keyword that you enter.

	**www.LeCroy.com** connects you to LeCroy's Web site where you can find Lab Briefs, Application Notes, and other useful information. This feature requires that the instrument be connected to the internet through the Ethernet port on the scope's rear panel. Refer to Remote Communication for setup instructions.
	**About** opens the Utilities "Status" dialog, which shows software version and other system information.

Once opened, the Help window will display its navigation pane: the part of the window that shows the Table of Contents and Index. When you touch anywhere outside of the Help window, this navigation pane will disappear to reveal more of your signal. To make it return, touch the **Show**

icon at the top of the Help window or touch inside the Help information pane.

Windows Help

In addition to instrument Help, you can also access on-line Help for Microsoft® Windows®. This help is accessible by minimizing the scope application, then touching the **Start** button in the Windows task bar at the bottom of the screen and selecting **Help**.

Returning a Product for Service or Repair

If you need to return a LeCroy product, identify it by its model and serial numbers. Describe the defect or failure, and give us your name and telephone number.

For factory returns, use a Return Authorization Number (RAN), which you can get from customer service. Write the number clearly on the outside of the shipping carton.

Return products requiring only maintenance to your local customer service center.

If you need to return your scope for any reason, use the original shipping carton. If this is not possible, be sure to use a rigid carton. The scope should be packed so that it is surrounded by a minimum of four inches (10 cm) of shock absorbent material.

Within the warranty period, transportation charges to the factory will be your responsibility. Products under warranty will be returned to you with transport prepaid by LeCroy. Outside the warranty period, you will have to provide us with a purchase order number before the work can be done. You will be billed for parts and labor related to the repair work, as well as for shipping.

You should prepay return shipments. LeCroy cannot accept COD (Cash On Delivery) or Collect Return shipments. We recommend using air freight.

Technical Support

You can get assistance with installation, calibration, and a full range of software applications from your customer service center. Visit the LeCroy Web site at http://www.lecroy.com for the center nearest you.

Staying Up-to-Date

To maintain your instrument's performance within specifications, have us calibrate it at least once a year. LeCroy offers state-of-the-art performance by continually refining and improving the instrument's capabilities and operation. We frequently update both firmware and software during service, free of charge during warranty.

You can also install new purchased software options in your scope yourself, without having to return it to the factory. Simply provide us with your instrument serial number and ID, and the version number of instrument software installed. We will provide you with a unique option key that consists of a code to be entered through the Utilities' Options dialog to load the software option.

Copyright

LeCroy Corporation

700 Chestnut Ridge Road

Chestnut Ridge, NY 10977-6499

Tel: (845) 578 6020

Fax: (845) 578 5985

Internet: www.lecroy.com

Manufactured under an ISO 9000 Registered Quality Management System

Windows License Agreement

LeCroy's agreement with Microsoft prohibits users from running software on LeCroy X-Stream oscilloscopes that is not relevant to measuring, analyzing, or documenting waveforms.

END-USER LICENSE AGREEMENT FOR LECROY® X-STREAM SOFTWARE

IMPORTANT-READ CAREFULLY: THIS END-USER LICENSE AGREEMENT ("EULA") IS A LEGAL AGREEMENT BETWEEN THE INDIVIDUAL OR ENTITY LICENSING THE SOFTWARE PRODUCT ("YOU" OR "YOUR") AND LECROY CORPORATION ("LECROY") FOR THE SOFTWARE PRODUCT(S) ACCOMPANYING THIS EULA, WHICH INCLUDE(S): COMPUTER PROGRAMS; ANY "ONLINE" OR ELECTRONIC DOCUMENTATION AND PRINTED MATERIALS PROVIDED BY LECROY HEREWITH ("DOCUMENTATION"); ASSOCIATED MEDIA; AND ANY UPDATES (AS DEFINED BELOW) (COLLECTIVELY, THE "SOFTWARE PRODUCT"). BY USING AN INSTRUMENT TOGETHER WITH OR CONTAINING THE SOFTWARE PRODUCT, OR BY INSTALLING, COPYING, OR OTHERWISE USING THE

SOFTWARE PRODUCT, IN WHOLE OR IN PART, YOU AGREE TO BE BOUND BY THE TERMS OF THIS EULA. IF YOU DO NOT AGREE TO THE TERMS OF THIS EULA, DO NOT INSTALL, COPY, OR OTHERWISE USE THE SOFTWARE PRODUCT; YOU MAY RETURN THE SOFTWARE PRODUCT TO YOUR PLACE OF PURCHASE FOR A FULL REFUND. IN ADDITION, BY INSTALLING, COPYING, OR OTHERWISE USING ANY MODIFICATIONS, ENHANCEMENTS, NEW VERSIONS, BUG FIXES, OR OTHER COMPONENTS OF THE SOFTWARE PRODUCT THAT LECROY PROVIDES TO YOU SEPARATELY AS PART OF THE SOFTWARE PRODUCT ("UPDATES"), YOU AGREE TO BE BOUND BY ANY ADDITIONAL LICENSE TERMS THAT ACCOMPANY SUCH UPDATES. IF YOU DO NOT AGREE TO SUCH ADDITIONAL LICENSE TERMS, YOU MAY NOT INSTALL, COPY, OR OTHERWISE USE SUCH UPDATES.

THE PARTIES CONFIRM THAT THIS AGREEMENT AND ALL RELATED DOCUMENTATION ARE AND WILL BE DRAFTED IN ENGLISH. LES PARTIES AUX PRÉSENTÉS CONFIRMENT LEUR VOLONTÉ QUE CETTE CONVENTION DE MÊME QUE TOUS LES DOCUMENTS Y COMPRIS TOUT AVIS QUI S'Y RATTACHÉ, SOIENT REDIGÉS EN LANGUE ANGLAISE.

1. GRANT OF LICENSE.

1.1 License Grant. Subject to the terms and conditions of this EULA and payment of all applicable fees, LeCroy grants to you a nonexclusive, nontransferable license (the "License") to: (a) operate the Software Product as provided or installed, in object code form, for your own internal business purposes, (i) for use in or with an instrument provided or manufactured by LeCroy (an "Instrument"), (ii) for testing your software product(s) (to be used solely by you) that are designed to operate in conjunction with an Instrument ("Your Software"), and (iii) make one copy for archival and back-up purposes; (b) make and use copies of the Documentation; provided that such copies will be used only in connection with your licensed use of the Software Product, and such copies may not be republished or distributed (either in hard copy or electronic form) to any third party; and (c) copy, modify, enhance and prepare derivative works ("Derivatives") of the source code version of those portions of the Software Product set forth in and identified in the Documentation as "Samples" ("Sample Code") for the sole purposes of designing, developing, and testing Your Software. If you are an entity, only one designated individual within your organization, as designated by you, may exercise the License; provided that additional individuals within your organization may assist with respect to reproducing and distributing Sample Code as permitted under Section 1.1(c)(ii). LeCroy reserves all rights not expressly granted to you. No license is granted hereunder for any use other than that specified herein, and no license is granted for any use in combination or in connection with other products or services (other than Instruments and Your Software) without the express prior written consent of LeCroy. The Software Product is licensed as a single product. Its component parts may not be separated for use by more than one user. This EULA does not grant you any rights in connection with any trademarks or service marks of LeCroy. The Software Product is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The Software Product is licensed, not sold. The terms of this printed, paper EULA supersede the terms of any on-screen license agreement found within the Software Product.

1.2 Upgrades. If the Software Product is labeled as an "upgrade," (or other similar designation) the License will not take effect, and you will have no right to use or access the Software Product unless you are properly licensed to use a product identified by LeCroy as being eligible for the upgrade ("Underlying Product"). A Software Product labeled as an "upgrade" replaces and/or supplements the Underlying Product. You may use the resulting upgraded product only in accordance with the terms of this EULA. If the Software Product is an upgrade of a component of a package of software programs that you licensed as a single product, the Software Product may be used and transferred only as part of that single product package and may not be separated for use on more than one computer.

1.3. Limitations. Except as specifically permitted in this EULA, you will not directly or indirectly (a) use any Confidential Information to create any software or documentation that is similar to any of the Software Product or Documentation; (b) encumber, transfer, rent, lease, time-share or use the Software Product in any service bureau arrangement; (c) copy (except for archival purposes), distribute, manufacture, adapt, create derivative works of, translate, localize, port or otherwise modify the Software Product or the Documentation; (d) permit access to the Software Product by any party developing, marketing or planning to develop or market any product having functionality similar to or competitive with the Software Product; (e) publish benchmark results relating to the Software Product, nor disclose Software Product features, errors or bugs to third parties; or (f) permit any third party to engage in any of the acts proscribed in clauses (a) through (e). In jurisdictions in which transfer is permitted, notwithstanding the foregoing prohibition, transfers will only be effective if you transfer a copy of this EULA, as well as all copies of the Software Product, whereupon your right to use the Software product will terminate. Except as described in this Section 1.3, You are not permitted (i) to decompile, disassemble, reverse compile, reverse assemble, reverse translate or otherwise reverse engineer the Software Product, (ii) to use any similar means to discover the source code of the Software Product or to discover the trade secrets in the Software Product, or (iii) to otherwise circumvent any technological measure that controls access to the Software Product. You may reverse engineer or otherwise circumvent the technological measures protecting the Software Product for the sole purpose of identifying and analyzing those elements that are necessary to achieve Interoperability (the "Permitted Objective") only if: (A) doing so is necessary to achieve the Permitted Objective and it does not constitute infringement under Title 17 of the United States Code; (B) such circumvention is confined to those parts of the Software Product and to such acts as are necessary to achieve the Permitted Objective; (C) the information to be gained thereby has not already been made readily available to you or has not been provided by LeCroy within a reasonable time after a written request by you to LeCroy to provide such information; (D) the information gained is not used for any purpose other than the Permitted Objective and is not disclosed to any other person except as may be necessary to achieve the Permitted Objective; and (E) the information obtained is not used (1) to create a computer program substantially similar in its expression to the Software Product including, but not limited to, expressions of the Software Product in other computer languages, or (2) for any other act restricted by LeCroy's intellectual property rights in the Software Product. "Interoperability" will have the same meaning in this EULA as defined in the Digital Millennium Copyright Act, 17 U.S.C. §1201(f), the ability of computer programs to exchange information and of such programs mutually to use the information which has been exchanged.

1.4 PRERELEASE CODE. Portions of the Software Product may be identified as prerelease code ("Prerelease Code"). Prerelease Code is not at the level of performance and compatibility of the final, generally available product offering. The Prerelease Code may not operate correctly and may be substantially modified prior to first commercial shipment. LeCroy is not obligated to make this or any later version of the Prerelease Code commercially available. The License with respect to the Prerelease Code terminates upon availability of a commercial release of the Prerelease Code from LeCroy.

2. SUPPORT SERVICES.

At LeCroy's sole discretion, from time to time, LeCroy may provide Updates to the Software Product. LeCroy shall have no obligation to revise or update the Software Product or to support any version of the Software Product. At LeCroy's sole discretion, upon your request, LeCroy may provide you with support services related to the Software Product ("Support Services") pursuant to the LeCroy policies and programs described in the Documentation or otherwise then in effect, and such Support Services will be subject to LeCroy's then-current fees therefor, if any. Any Update or other supplemental software code provided to you pursuant to the Support Services will be considered part of the Software Product and will be subject to the terms and conditions of this EULA. LeCroy may use any technical information you provide to LeCroy during LeCroy's provision of Support Services, for LeCroy's business purposes, including for product support and development. LeCroy will not utilize such technical information in a form that personally identifies you.

3. PROPRIETARY RIGHTS.

3.1 Right and Title. All right, title and interest in and to the Software Product and Documentation (including but not limited to any intellectual property or other proprietary rights, images, icons, photographs, text, and "applets" embodied in or incorporated into the Software Product, collectively, "Content"), and all Derivatives, and any copies thereof are owned by LeCroy and/or its licensors or third-party suppliers, and is protected by applicable copyright or other intellectual property laws and treaties. You will not take any action inconsistent with such title and ownership. This EULA grants you no rights to use such Content outside of the proper exercise of the license granted hereunder, and LeCroy will not be responsible or liable therefor.

3.2 Intellectual Property Protection. You may not alter or remove any printed or on-screen copyright, trade secret, proprietary or other legal notices contained on or in copies of the Software Product or Documentation.

3.3 Confidentiality. Except for the specific rights granted by this EULA, neither party shall use or disclose any Confidential Information (as defined below) of the other party without the written consent of the disclosing party. A party receiving Confidential Information from the other shall use the highest commercially reasonable degree of care to protect the Confidential Information, including ensuring that its employees and consultants with access to such Confidential Information have agreed in writing not to disclose the Confidential Information. You shall bear the responsibility for any breaches of confidentiality by your employees and consultants. Within ten (10) days after request of the disclosing party, and in the disclosing party's sole discretion, the receiving party shall either return to the disclosing party originals and copies of any Confidential Information and all information, records and materials developed therefrom by the receiving party, or destroy the same, other than such Confidential Information as to which this EULA expressly provides a continuing right to the receiving party to retain at the time of the request. Either party may only disclose the general nature, but not the specific financial terms, of this EULA without the prior consent of the other party; provided either party may provide a copy of this EULA to any finance provider in conjunction with a financing transaction, if such provider agrees to keep this EULA confidential. Nothing herein shall prevent a receiving party from disclosing all or part of the Confidential Information as necessary pursuant to the lawful requirement of a governmental agency or when disclosure is required by operation of law; provided that prior to any such disclosure, the receiving party shall use reasonable efforts to (a) promptly notify the disclosing party in writing of such requirement to disclose, and (b) cooperate fully with the disclosing party in protecting against any such disclosure or obtaining a protective order. Money damages will not be an adequate remedy if this Section 4.3 is breached and, therefore, either party shall, in addition to any other legal or equitable remedies, be entitled to seek an injunction or similar equitable relief against such breach or threatened breach without the necessity of posting any bond. As used herein, “Confidential Information” means LeCroy pricing or information concerning new LeCroy products, trade secrets (including without limitation all internal header information contained in or created by the Software Product, all benchmark and performance test results and all Documentation) and other proprietary information of LeCroy; and any business, marketing or technical information disclosed by LeCroy, or its representatives, or you in relation to this EULA, and either (i) disclosed in writing and marked as confidential at the time of disclosure or (ii) disclosed in any other manner such that a reasonable person would understand the nature and confidentiality of the information. Confidential Information does not include information (A) already in the possession of the receiving party without an obligation of confidentiality to the disclosing party, (B) hereafter rightfully furnished to the receiving party by a third party without a breach of any separate nondisclosure obligation to the disclosing party, (C) publicly known without breach of this EULA, (d) furnished by the disclosing party to a third party without restriction on subsequent disclosure, or (e) independently developed by the receiving party without reference to or reliance on the Confidential Information.

4. TERMINATION.

This EULA will remain in force until termination pursuant to the terms hereof. You may terminate this EULA at any time. This EULA will also terminate if you breach any of the terms or conditions of this EULA. You agree that if this EULA terminates for any reason, the License will immediately terminate and you will destroy all copies of the Software Product (and all Derivatives), installed or otherwise, the Documentation, and the Confidential Information (and all derivatives of any of the foregoing) that are in your possession or under your control. The provisions of Sections 1.3, 4, 6, 7, 8, and 9 will survive any termination or expiration hereof.

5. U.S. GOVERNMENT RESTRICTED RIGHTS.

If any Software Product or Documentation is acquired by or on behalf of a unit or agency of the United States Government (any such unit or agency, the “Government”), the Government agrees that the Software Product or Documentation is “commercial computer software” or “commercial computer software documentation” and that, absent a written agreement to the contrary, the Government’s rights with respect to the Software Product or Documentation are, in the case of civilian agency use, Restricted Rights, as defined in FAR §52.227.19, and if for Department of Defense use, limited by the terms of this EULA, pursuant to DFARS §227.7202. The use of the Software Product or Documentation by the Government constitutes acknowledgment of LeCroy’s proprietary rights in the Software Product and Documentation. Manufacturer is LeCroy Corporation, 700 Chestnut Ridge Road, Chestnut Ridge, NY 10977 USA.

6. EXPORT RESTRICTIONS.

You agree that you will not export or re-export the Software Product, any part thereof, or any process or service that is the direct product of the Software Product (the foregoing collectively referred to as the "Restricted Components"), to any country, person, entity or end user subject to U.S. export restrictions. You specifically agree not to export or re-export any of the Restricted Components (a) to any country to which the U.S. has embargoed or restricted the export of goods or services, which currently include, but are not necessarily limited to Cuba, Iran, Iraq, Libya, North Korea, Sudan and Syria, or to any national of any such country, wherever located, who intends to transmit or transport the Restricted Components back to such country; (b) to any end user who you know or have reason to know will utilize the Restricted Components in the design, development or production of nuclear, chemical or biological weapons; or (c) to any end-user who has been prohibited from participating in U.S. export transactions by any federal agency of the U.S. government. You warrant and represent that neither the BXA nor any other U.S. federal agency has suspended, revoked or denied your export privileges. It is your responsibility to comply with the latest United States export regulations, and you will defend and indemnify LeCroy from and against any damages, fines, penalties, assessments, liabilities, costs and expenses (including reasonable attorneys' fees and court costs) arising out of any claim that the Software Product, Documentation, or other information or materials provided by LeCroy hereunder were exported or otherwise accessed, shipped or transported in violation of applicable laws and regulations.

7. RISK ALLOCATION.

7.1 No Warranty. THE SOFTWARE PRODUCT IS NOT ERROR-FREE AND THE SOFTWARE PRODUCT AND SUPPORT SERVICES IS/ARE BEING PROVIDED "AS IS" WITHOUT WARRANTY OF ANY KIND. LECROY, FOR ITSELF AND ITS SUPPLIERS, HEREBY DISCLAIMS ALL WARRANTIES, WHETHER EXPRESS OR IMPLIED, ORAL OR WRITTEN, WITH RESPECT TO THE SOFTWARE PRODUCT OR ANY SUPPORT SERVICES INCLUDING, WITHOUT LIMITATION, ALL IMPLIED WARRANTIES OF TITLE OR NON-INFRINGEMENT, MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, ACCURACY, INTEGRATION, VALIDITY, EXCLUSIVITY, MERCHANTABILITY, NON-INTERFERENCE WITH ENJOYMENT, FITNESS FOR ANY PARTICULAR PURPOSE, AND ALL WARRANTIES IMPLIED FROM ANY COURSE OF DEALING OR USAGE OF TRADE. YOU ACKNOWLEDGE THAT NO WARRANTIES HAVE BEEN MADE TO YOU BY OR ON BEHALF OF LECROY OR OTHERWISE FORM THE BASIS FOR THE BARGAIN BETWEEN THE PARTIES.

7.2. Limitation of Liability. LECROY'S LIABILITY FOR DAMAGES FOR ANY CAUSE WHATSOEVER, REGARDLESS OF THE FORM OF ANY CLAIM OR ACTION, SHALL NOT EXCEED THE GREATER OF THE AMOUNT ACTUALLY PAID BY YOU FOR THE SOFTWARE PRODUCT OR U.S.$5.00; PROVIDED THAT IF YOU HAVE ENTERED INTO A SUPPORT SERVICES AGREEMENT WITH LECROY, LECROY'S ENTIRE LIABILITY REGARDING SUPPORT SERVICES WILL BE GOVERNED BY THE TERMS OF THAT AGREEMENT. LECROY SHALL NOT BE LIABLE FOR ANY LOSS OF PROFITS, LOSS OF USE, LOSS OF DATA, INTERRUPTION OF BUSINESS, NOR FOR INDIRECT, SPECIAL, INCIDENTAL,

CONSEQUENTIAL OR EXEMPLARY DAMAGES OF ANY KIND, WHETHER UNDER THIS EULA OR OTHERWISE ARISING IN ANY WAY IN CONNECTION WITH THE SOFTWARE PRODUCT, THE DOCUMENTATION OR THIS EULA. SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF INCIDENTAL OR CONSEQUENTIAL DAMAGES, SO THE ABOVE EXCLUSION OR LIMITATION MAY NOT APPLY TO YOU. THESE LIMITATIONS ARE INDEPENDENT FROM ALL OTHER PROVISIONS OF THIS EULA AND SHALL APPLY NOTWITHSTANDING THE FAILURE OF ANY REMEDY PROVIDED HEREIN.

7.3 Indemnification. You will defend, indemnify and hold harmless LeCroy and its officers, directors, affiliates, contractors, agents, and employees from, against and in respect of any and all assessments, damages, deficiencies, judgments, losses, obligations and liabilities (including costs of collection and reasonable attorneys' fees, expert witness fees and expenses) imposed upon or suffered or incurred by them arising from or related to your use of the Software Product.

8. GENERAL PROVISIONS.

8.1 Compliance with Laws. You will comply with all laws, legislation, rules, regulations, and governmental requirements with respect to the Software Product, and the performance by you of your obligations hereunder, of any jurisdiction in or from which you directly or indirectly cause the Software Product to be used or accessed.

8.2 No Agency. Nothing contained in this EULA will be deemed to constitute either party as the agent or representative of the other party, or both parties as joint venturers or partners for any purpose.

8.3 Entire Agreement; Waiver; Severability. This EULA constitutes the entire agreement between the parties with regard to the subject matter hereof. No provision of, right, power or privilege under this EULA will be deemed to have been waived by any act, delay, omission or acquiescence by LeCroy, its agents, or employees, but only by an instrument in writing signed by an authorized officer of LeCroy. No waiver by LeCroy of any breach or default of any provision of this EULA by you will be effective as to any other breach or default, whether of the same or any other provision and whether occurring prior to, concurrent with, or subsequent to the date of such waiver. If any provision of this EULA is declared by a court of competent jurisdiction to be invalid, illegal or unenforceable, such provision will be severed from this EULA and all the other provisions will remain in full force and effect.

8.4 Governing Law; Jurisdiction; Venue. This EULA will be governed by and construed in accordance with the laws of the State of New York, USA, without regard to its choice of law provisions. The United Nations Convention on Contracts for the International Sale of Goods will not apply to this EULA. Exclusive jurisdiction and venue for any litigation arising under this EULA is in the federal and state courts located in New York, New York, USA and both parties hereby consent to such jurisdiction and venue for this purpose.

8.5 Assignment. This EULA and the rights and obligations hereunder, may not be assigned, in whole or in part by you, except to a successor to the whole of your business, without the prior written consent of LeCroy. In the case of any permitted assignment or transfer of or under this EULA, this EULA or the relevant provisions will be binding upon, and inure to the benefit of, the successors, executors, heirs, representatives, administrators and assigns of the parties hereto.

8.6 Notices. All notices or other communications between LeCroy and you under this EULA will be in writing and delivered personally, sent by confirmed fax, by confirmed e-mail, by certified mail, postage prepaid and return receipt requested, or by a nationally recognized express delivery service. All notices will be in English and will be effective upon receipt.

8.7 Headings. The headings used in this EULA are intended for convenience only and will not be deemed to supersede or modify any provisions.

8.8 Acknowledgment. Licensee acknowledges that (a) it has read and understands this EULA, (b) it has had an opportunity to have its legal counsel review this EULA, (c) this EULA has the same force and effect as a signed agreement, and (d) issuance of this EULA does not constitute general publication of the Software Product or other Confidential Information.

Virus Protection

Because your scope runs on a Windows-based PC platform, it must be protected from viruses, as with any PC on a corporate network. It is crucial that the scope be kept up to date with Windows Critical Updates, and that anti-virus software be installed and continually updated.

Visit http://www.lecroy.com/dsosecurity for more information regarding Windows Service Pack compatibility with LeCroy operating software, and related matters.

Warranty

The instrument is warranted for normal use and operation, within specifications, for a period of one year from shipment. LeCroy will either repair or, at our option, replace any product returned to one of our authorized service centers within this period. However, in order to do this we must first examine the product and find that it is defective due to workmanship or materials and not due to misuse, neglect, accident, or abnormal conditions or operation.

LeCroy shall not be responsible for any defect, damage, or failure caused by any of the following: a) attempted repairs or installations by personnel other than LeCroy representatives, or b) improper connection to incompatible equipment or c) for any damage or malfunction caused by the use of non-LeCroy supplies. Furthermore, LeCroy shall not be obligated to service a product that has been modified or integrated where the modification or integration increases the task duration or difficulty of servicing the oscilloscope. Spare and replacement parts, and repairs, all have a 90-day warranty.

The oscilloscope's firmware has been thoroughly tested and is presumed to be functional. Nevertheless, it is supplied without warranty of any kind covering detailed performance. Products not made by LeCroy are covered solely by the warranty of the original equipment manufacturer.

Note: Specifications are subject to change without notice.

Sampling Heads

Module Performance (Guaranteed Values)

Module	Rise time	RMS noise	Aberrations	Bandwidth
ST-20	18 ps	700 µV	First 40 ps +/-10% 40 ps to 200 ps +/-5% 200 ps to 10 ns +/-2%	20 GHz
SE-30	12 ps	1 mV	First 40 ps +/-10% 40 ps to 200 ps +/-5% 200 ps to 10 ns +/-2%	30 GHz
SE-50	8 ps	2 mV	First 40 ps +/-10% 40 ps to 200 ps +/-5% 200 ps to 10 ns +/-2%	50 GHz
SE-70	5 ps	3 mV	First 40 ps +/-10% 40 ps to 200 ps +/-5% 200 ps to 10 ns +/-2%	70 GHz*
SE-100	4 ps	3 mV	First 40 ps +/-10% 40 ps to 200 ps +/-5% 200 ps to 10 ns +/-2%	100 GHz*

* limited by connector moding

Vertical System

Maximum Input Channels: 4

Maximum Input Range: +/- 2 V

Dynamic Range: 2 V_{p-p}

Input Impedance: 50 ohms ±1%

Input Coupling: DC

Max Input Voltage: +/- 2.5 V

Static Sensitivity: The sampling module inputs are highly static sensitive. A grounding strap must be worn at all times when handling the modules.

Installation (Overvoltage) Category: CAT I

Vertical Resolution: 14 bits up to 17 bits with enhanced resolution (ERES)

Sensitivity: 1 mV/div to 1 V/div

Offset Range: -1 V to +1 V

Offset Accuracy:

Horizontal System

Timebases: 2 timebases common to all 4 channels are available; Sequential (SEQ) and Coherent Interleaved Sampling (CIS)

Maximum Sampling Rate: 10 MS/s (CIS), 1 MS/s (SEQ)

Math & Zoom Traces: 4 independent zoom and 4 math/zoom traces standard; 8 math/zoom traces available with XMAP (Master Analysis Package) option

Timebase Jitter: 600 fs rms (CIS); 1.8 ps rms (SEQ)

Acquisition System

Memory Options:

Memory Option	Sequential	Coherent Interleaved
Standard	16k	0.1 ms
M Memory Option	50k	0.4 ms
L Memory Option	100k	0.8 ms
CIS - standard	100k	4M

CIS - M	100k	256M
CIS - L	100k	512M

Acquisition Processing

Averaging: Summed averaging to 1 million sweeps; Continuous averaging to 1 million sweeps

Envelope (Extrema): Envelope, floor, roof for up to 1 million sweeps

Triggering System

Modes: normal, prescale, TDR (internal)

Sources: front panel SMA connectors for direct and prescale, internal 1 MHz clock for TDR

Coupling: DC 50 ohms for direct trigger, AC 50 ohms for prescaled trigger

Minimum Trigger Delay: 12 ns (Sequential timebase)

Trigger Range: ±1 V (direct)

Trigger Jitter: 1.5 ps rms (sequential)

Automatic Setup

Autosetup: Automatically sets timebase, trigger, and sensitivity to display a wide range of repetitive signals.

Vertical Find Scale: Automatically sets the vertical sensitivity and offset for the selected channels to display a waveform with maximum dynamic range.

Color Waveform Display

Type: Color 10.4-inch flat panel TFT LCD with high resolution touch screen

Resolution: SVGA; 800 x 600 pixels

Real-time Clock: Date, hours, minutes, and seconds displayed with waveform; SNTP support to synchronize to precision internet clocks

Number of Traces: Maximum of eight traces; simultaneously displays channel, zoom, memory, and math traces

Grid Styles: Single, Dual, Quad, Octal, XY, Single+XY, Dual+XY

Waveform Display Styles: Sample dots joined or dots only

Analog Persistence Display

Analog and Color-graded Persistence: Variable saturation levels; stores each trace's persistence data in memory

Persistence Selections: Select analog, color, or 3-D

Trace Selection: Activate Analog Persistence on all or any combination of traces

Persistence Aging Time: From 500 ms to infinity

Sweeps Displayed: All accumulated or all accumulated with last trace highlighted

Zoom Expansion Traces

Display up to 4 Zoom and 4 Math/Zoom traces; 8 Math/Zoom traces available with XMAP (Master Analysis Package) and XMATH (Advanced Math Package) options.

Rapid Signal Processing

Processor: Intel Pentium 4 @ 2.53 GHz (or better) with MS Windows XP Platform

Processor Memory: Up to 512 Mbyte (up to 2 Gbytes with CIS-L memory option)

Internal Waveform Memory

Waveform: M1, M2, M3, M4 (Store full-length waveforms with 16 bits/data point.) Or save to any number of files (limited only by data storage media).

Setup Storage

Front Panel and Instrument Status: Save to the internal hard drive, floppy drive, or to a USB connected peripheral device.

Interface

Remote Control: Through Windows Automation or LeCroy Remote Command set, supports front panel controls and internal functions via GPIB or Ethernet.

GPIB Port (optional): Supports IEEE-488.2

Ethernet Port: 10/100Base-T Ethernet interface

Floppy Drive: Internal, DOS format, 3.5 inch, high density

USB Ports: 4 USB ports support Windows compatible devices.

External Monitor Port (standard): 15-pin D-Type SVGA compatible

Parallel Port: 1 standard

Math Tools (standard)

Display up to four math function traces (F1 to F4). The easy-to-use graphical interface simplifies setup of up to two operations on each function trace. Function traces can be chained together to perform math-on-math.

absolute value	invert (negate)
average (summed)	ln (log base e)
average (continuous)	log (base 10)

derivative	product (X)
deskew (resample)	ratio (/)
difference ()	reciprocal
enhanced resolution (to 11 bits vertical)	rescale (with units)
envelope	roof
exp (base e)	segment
exp (base 10)	(sinX)/X
fft (basic)	square
floor	square root
histogram of 1,000 events	sum (+)
integral	trend (datalog) of 1,000 events
	zoom (identity)

Measure Tools (standard)

Display any 8 parameters together with statistics, including their average, high, low, and standard deviations. Histicons provide a fast, dynamic view of parameters and wave shape characteristics.

amplitude	number of points
area	overshoot+
base	overshoot-
cycles	peak-to-peak
delay	period
delta delay	phase
delta time @ level	rise time (10-90%, 20-80%, @ level)
duration	rms
duty cycle	std. deviation
fall time (90-10%, 80-20%, @ level)	time @ level
first	top
frequency	width
last	x @ minimum (min.)
level @ x	x @ maximum (max.)

maximum mean minimum	x at max x at min

Pass/Fail Testing

Test multiple parameters against selectable parameter limits at the same time. Pass or fail conditions can initiate actions including: document to local or networked files, email the image of the failure, save waveforms, send a pulse out at the front panel auxiliary BNC output, or (with GPIB option) send a GPIB SRQ.

Master Analysis Package (XMAP)

This package provides a comprehensive set of signal WaveShape Analysis tools that provide insight into the wave shape of complex signals. Additional analysis capability provided by XMAP includes:

- Jitter and Timing Analysis package (JTA2)
- 8 math traces total (4 additional)
- Parameter Math add, subtract, multiply, or divide two different parameter measurements
- User-definable parameter measurements and math functions, using VBScripting with MS Excel and MATLAB
- Histograms expanded with 19 histogram parameters and up to 2 billion events
- Trend (datalog) of up to one million events
- Track graphs of any measurement parameter
- FFT capability expands the basic FFT to include power averaging, power density, real and imaginary components, frequency domain parameters and FFT on up to 25 Mpts.
- Narrow Band power measurements
- Correlation function
- Interpolation
- Sparse

Jitter Analysis Package (WE-JTA)

This package provides tools for analyzing jitter in serial data and clock signals. The primary measurement is Time at Level which is measured on the sampled waveform using an edge model.

- tf@lvl parameter (measures edge jitter using curve fitting)
- Histograms expanded with 19 histogram parameters and up to 2 billion events
- Trend (datalog) of up to one million events
- Persistence Histogram; Persistence Trace
- Htie to BER function (extrapolates a histogram to 10e-16 BER)
- AltNcycle function (plots the mean or standard deviation vs. bit position for repeating data

patterns)

- CIS to Persist function (creates eye pattern of data collected with the CIS timebase)
- 4-quadrant summary view of jitter and eye pattern plots
- “serial data” analysis mode including a user interface that integrates all eye and jitter measurements

General

Auto Calibration: Ensures specified DC and timing accuracy is maintained for 1 year minimum.

Power Requirements: Single phase, 100 to 240 V_{rms} (±10%) at 50/60 Hz (±5%); or single phase, 100 to 120 V_{rms} (±10%) at 400 Hz (±5%); Automatic AC voltage selection

Voltage Range:	90 to 264 V_{rms}	90 to 132 V_{rms}
Frequency Range:	47 to 63 Hz	380 to 420 Hz

Power Consumption: On State: 400 watts (400 VA) depending on accessories installed (sampling modules, PC port plug-ins, etc.). Standby State: 12 watts

Fuse: One 5x20 mm fuse (T6.3 A/250 V)

Battery Backup: Front panel settings retained for two years minimum

Physical Dimensions (HWD): 264 mm x 397 mm x 491 mm (10.4 in. x 15.6 in. x 19.3 in.); height measurement excludes foot pads

Weight: 16 kg (36 lbs.)

Shipping Weight: 22 kg (48 lbs.)

Warranty and Service

1-year warranty; calibration recommended yearly

Optional service programs include extended warranty, upgrades, and calibration services.

Environmental Characteristics

Temperature

Operating: 5 to 40 °C

Storage (non-operating): -20 to +60 °C

Humidity

Operating: 5 to 80% RH (non-condensing) up to 30 °C; upper limit derates linearly to 45% RH (non-condensing) at 40 °C

Storage (non-operating): 5 to 95% RH (non-condensing) as tested per MIL-PRF-28800F

Altitude

Operating: Up to 3048 m (10,000 ft) at or below 25 °C

Storage (non-operating): Up to 12,192 m (40,000 ft)

Random Vibration

Operating: 0.31 grms, 5 Hz to 500 Hz, 15 minutes in each of 3 orthogonal axes

Non-operating: 2.4 grms, 5 to 500 Hz, 15 minutes in each of 3 orthogonal axes

Shock

Functional Shock: 20 g peak, half sine, 11 ms pulse, 3 shocks (positive and negative) in each of 3 orthogonal axes, 18 shocks total

Certifications

EC Declaration of Conformity	Meets intent of the European Council Directives 73/23/EEC for product safety and 89/336/EEC for electromagnetic compatibility. This declaration is based upon compliance of the WaveExpert oscilloscope to the following standards: EN 61326: 1997 +A1:1998 +A2:2001 +A3:2003 EMC requirements for electrical equipment for measurement, control, and laboratory use. Emissions: EN 55011: 1998+A2:2002 Radiated & Conducted Emissions (Class A*) EN 61000-3-2:2000 Harmonic Current Emissions Immunity: EN 61000-4-2:1999 Electrostatic discharge (±4 kV contact discharge; ±8 kV air discharge) EN 61000-4-3: 2002+A1:2003 RF Radiated Fields (3 V/m, 80 MHz to 1 GHz, 80% amplitude modulated) EN 61000-4-4: 2004 Electrical Fast Transient/Burst (1 kV on AC mains) EN 61000-4-5: 1995+A1:2001 Surge (1 kV differential mode, 2 kV common mode) EN 61000-4-6: 1996+A1:2001 RF Conducted Field (3 V, 150 kHz to 80 MHz, amplitude modulated with 1 kHz sine wave)

	EN 61000-4-11: 2004 Mains Dips and Interruptions (100% interruption for 1 full AC cycle) EN 61010-1: 2001 Safety requirements for electrical equipment for measurement control and laboratory use With the following limits: Installation (Overvoltage) Category II (Line voltage in equipment and to wall outlet) Installation (Overvoltage) Category I (All mains isolated terminals) Pollution Degree 2 Protection Class I

* To conform to Radiated Emissions standard, use properly shielded cables on all I/O terminals.

UL and cUL Listed - Conforms to UL 61010-1, 2nd Edition and CAN/CSA C22.2 No. 61010-1-04

SAFETY

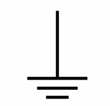
This section contains information and warnings that must be observed to keep the scope operating in a correct and safe condition. You are required to follow generally accepted safety procedures in addition to the safety precautions specified in this section.

Safety Symbols

Where the following symbols appear on the scope's front or rear panels, or in this manual, they alert you to important safety considerations.

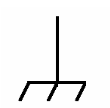
This symbol is used where caution is required. Refer to the accompanying information or documents in order to protect against personal injury or damage to the scope.

This symbol warns of a potential risk of shock hazard.



This symbol is used to denote the measurement ground connection.

This symbol is used to denote a safety ground connection.



This symbol is used to denote a grounded frame or chassis terminal.

This symbol shows that the switch is a Standby (power) switch. When it is pressed, the scope's state toggles between operating and Standby mode. This switch is not a disconnect device. The scope can only be placed in a complete Power Off state by unplugging the power cord from the AC supply.

This symbol is used to denote "Alternating Current."

The ESD symbol indicates a potential hazard. It calls attention to the susceptibility of the equipment to electrostatic discharge (ESD) induced damage if anti-static measures are not taken.

CAUTION The CAUTION sign indicates a potential hazard. It calls attention to a procedure, practice or condition which, if not followed, could possibly cause damage to equipment. If a CAUTION is indicated, do not proceed until its conditions are fully understood and met.

WARNING The WARNING sign indicates a potential hazard. It calls attention to a procedure, practice or condition which, if not followed, could possibly cause bodily injury or death. If a WARNING is indicated, do not proceed until its conditions are fully understood and met.

CAT I Installation (Overvoltage) Category rating per EN 61010-1 safety standard and is applicable for the oscilloscope front panel measuring terminals. CAT I rated terminals must only be connected to source circuits in which measures are taken to limit transient voltages to an appropriately low level.

Operating Environment

The scope is intended for indoor use and should be operated in a clean, dry environment with an ambient temperature within the range of 5 °C to 40 °C.

Note: Direct sunlight, radiators, and other heat sources should be taken into account when assessing the ambient temperature.

WARNING

The scope must not be operated in explosive, dusty, or wet atmospheres.

CAUTION

Protect the scope’s display touch screen from excessive impacts with foreign objects.

CAUTION

Do not exceed the maximum specified front panel terminal voltage levels. Refer to Specifications for more details.

CAUTION

ESD sensitive: The sampling modules inputs are highly static sensitive. A grounding strap must be worn at all times when handling the modules.

Installation (Overvoltage) Category II refers to local distribution level, which is applicable to equipment connected to the mains supply (AC power source).

Installation (Overvoltage) Category I refers to signal level, which is applicable to equipment measuring terminals that are connected to source circuits in which measures are taken to limit transient voltages to an appropriately low level.

Pollution Degree 2 refers to an operating environment where normally only dry non-conductive pollution occurs. Occasionally a temporary conductivity caused by condensation must be expected.

Protection Class 1 refers to a grounded equipment, in which protection against electric shock is achieved by Basic Insulation and by means of a connection to the protective ground conductor in the building wiring.

Cooling

The scope relies on forced air cooling with internal fans and ventilation openings. Care must be taken to avoid restricting the airflow around the apertures (fan holes) at the sides and rear of the scope. To ensure adequate ventilation it is required to leave a 10 cm (4 inch) minimum gap around the sides and rear of the scope.

The scope also has internal fan control circuitry that regulates the fan speed based on the ambient temperature. This is performed automatically after start-up with no manual intervention required.

Note

The design of the instrument has been verified to conform to EN 61010-1 safety standard per the following limits:

Installation (Overvoltage) Categories II (Mains Supply Connector) & I (Measuring Terminals)

Pollution Degree 2

Protection Class I

CAUTION

Do not block the ventilation holes located on both sides and rear of the scope.

CAUTION

Do not allow any foreign matter to enter the scope through the ventilation holes, etc.

AC Power Source

The scope operates from a single-phase, 100 to 240 V_{rms} (+/-10%) power source at 50/60 Hz (+/-5%), or single-phase 100 to 120 V_{rms} (+/-10%) at 400 Hz (+/-5%) power source.

No manual voltage selection is required because the scope automatically adapts to line voltage.

Depending on the accessories installed (sampling modules, PC port plug-ins, etc.), the scope can draw up to 400 W (400 VA).

The power supply of the scope is protected against short circuit and overload by a 5x20 mm fuse (T6.3 A/250 V). See Fuse Replacement section for replacement instructions.

NOTE:
The scope automatically adapts itself to the AC line input within the following ranges:

Voltage Range:	90 to 264 V_{rms}	90 to 132 V_{rms}
Frequency Range:	47 to 63 Hz	380 to 420 Hz

Power and Ground Connections

The scope is provided with a grounded cord set containing a molded three-terminal polarized plug and a standard IEC320 (Type C13) connector for making line voltage and safety ground connection. The AC inlet ground terminal is connected directly to the frame of the scope. For adequate protection against electrical shock hazard, the power cord plug must be inserted into a mating AC outlet containing a safety ground contact.

WARNING

Electrical Shock Hazard!

Any interruption of the protective conductor inside or outside of the scope, or disconnection of the safety ground terminal creates a hazardous situation.

Intentional interruption is prohibited.

In Standby mode the scope is still connected to the AC supply. The scope can only be placed in a complete Power Off state by physically disconnecting the power cord from the AC supply.

The scope should be positioned to allow easy access to the socket-outlet. To disconnect the scope from the AC supply, unplug the scope's power cord from the AC outlet after the scope is placed in Standby state.

See "Standby (Power) Switch and scope Operational States" section for more information.

Standby (Power) Switch and Scope Operational States

The front Standby (Power) switch controls the operational state of the scope. This toggle switch is activated by momentarily pressing and releasing it. The color of the LED below the switch indicates the status of the scope as follows:

On (LED Green)* scope is fully powered and operational

Standby (LED Off)* scope is powered off (except for some “housekeeping” circuits)

Standby (LED Red) scope’s computer subsystems (hard drive, etc.) are in Standby (reduced Power mode). All other scope subsystems are fully powered.

* Factory Settings

The scope’s factory settings result in only two basic scope states: On (LED Green) or Standby (LED Off). In this case of Standby (LED Off), the scope is powered off with the exception of some “housekeeping” circuitry (approximately 12 watts dissipation). The scope can only be placed in a complete power off state by unplugging the scope’s power cord from the primary power source (AC outlet). It is recommended that the power cord be unplugged from the AC outlet if the scope is not being used for an extended period of time.

You have the option to change the scope's original factory settings via the “Power Options Properties” menu in Windows by following the path: Settings Power Options. It is important to note that the Windows Power Option named “Standby” provides control of only the scope’s computer subsystems (CPU, hard drive, etc.) and does not affect the other subsystems within the scope. In general, these other subsystems remain fully powered. For additional information on setting these Power Options, see the Windows Help menu or other related technical documentation. In terms of control buttons, this scope uses only a power button/switch and therefore references to a sleep button are not applicable.

The scope can always be placed in the Standby state (LED Off) Power Off (except for some “housekeeping” circuits) by pressing and holding in the Standby toggle switch for approximately 5 seconds.

Fuse Replacement

Set the scope Standby (power) switch to Standby mode (LED off) and disconnect the power cord before inspecting or replacing the fuse. Open the black fuse holder (located at the rear of the scope directly to the right of the AC inlet) using a small, flat-bladed screwdriver. Remove the old fuse, replace it with a new 5x20 mm “T” rated 6.3 A/250 V fuse, and reinstall the fuse holder.

WARNING

For continued fire protection at all line voltages, replace fuse with the specified type and rating only. Disconnect the power cord before replacing fuse.

Calibration

The recommended calibration interval is one year. Calibration should be performed by qualified personnel only.

Cleaning

Clean only the exterior of the scope, using a damp, soft cloth. Do not use chemicals or abrasive elements. Under no circumstances allow moisture to penetrate the scope. To avoid electrical shock, unplug the power cord from the AC outlet before cleaning.

WARNING

Electrical Shock Hazard!

No operator serviceable parts inside. Do not remove covers.

Refer servicing to qualified personnel.

Abnormal Conditions

Operate the scope only as intended by the manufacturer.

If you suspect the scope's protection has been impaired, disconnect the power cord and secure the scope against any unintended operation.

The scope's protection is likely to be impaired if, for example, the scope shows visible damage or has been subjected to severe transport stresses.

Proper use of the scope depends on careful reading of all instructions and labels.

WARNING

Any use of the scope in a manner not specified by the manufacturer may impair the scope's safety protection. The scope and related accessories should not be directly connected to human subjects or used for patient monitoring.

Alternate Access Methods

The instrument often gives you more than one way to access dialogs and menus.

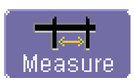
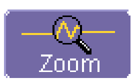
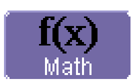
Mouse and Keyboard Operation

In the procedures we focus on touch-screen operation, but if you have a mouse connected to the instrument, you can also click on objects. Likewise, if you have a keyboard connected, you can use it instead of the virtual keyboard provided by the instrument.

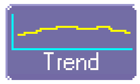
If you want to connect a mouse to the instrument, use only a USB mouse.

Tool Bar Buttons

The procedures also focus on the use of the menu bar at the top of the screen to access dialogs and menus. However, on several dialogs common functions are accessible from a row of buttons that save you a step or two in accessing their dialogs. For example, at the bottom of the Channel Setup dialog, these buttons perform the following functions:

	Calls up the Measure menu. You can then select a parameter from this menu without leaving the Channel Setup dialog. The parameter automatically appears below the grid.
	Creates a zoom trace of the channel trace whose dialog is currently displayed.
	Calls up the Math menu. You can then select a math function from this menu without leaving the Channel Setup dialog. A math trace of the channel whose dialog is currently open is automatically displayed.
	Loads the channel trace into the next available memory location (M1 to M4).
	Automatically performs a vertical scaling that fits the waveform into the grid.
	Automatically moves the channel trace whose dialog is currently open onto the next grid. If you have only one grid displayed, a new grid will be created automatically, and the trace moved.
	Enables you to attach identifying labels to your waveforms. The labels are preserved when the waveform is saved as a LabNotebook entry and when saved to file.

Another example is these buttons that appear at the bottom of the Measure **Px** dialogs. Each button opens a menu from which to choose a math trace (F1 to Fx) to display the functions named in the buttons:

, 

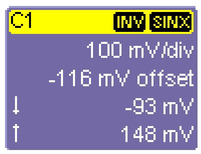
, 

.

By using these buttons you can remain in the Measure dialog to set up other options.

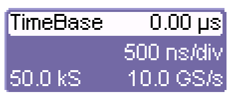
Trace Descriptors

Vertical and horizontal trace descriptors (labels) are displayed below the grid. They provide a summary of your channel, timebase, and trigger settings. To make adjustments to these settings, touch the respective label to display the setup dialog for that function.

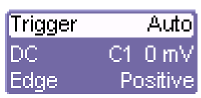


Channel trace labels show the vertical settings for the trace, as well as cursor information if cursors are in use. In the title bar of the label are also included indicators for (SinX)/X interpolation, waveform inversion (INV), deskew (DSQ), and averaging (AVG). These indicators have a long and short form, dependent on available space in the title bar of the label:

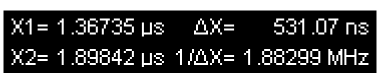
Besides channel traces, math and parameter measurement labels are also displayed. Labels are displayed only for traces that are turned on.



The title bar of the **Timebase** label shows the trigger delay setting. Time per division and sampling information is given below the title bar. The timebase in use is indicated above the record length as sequential (SEQ), coherent interleaved sampling (CIS) or random interleaved sampling (RIS).



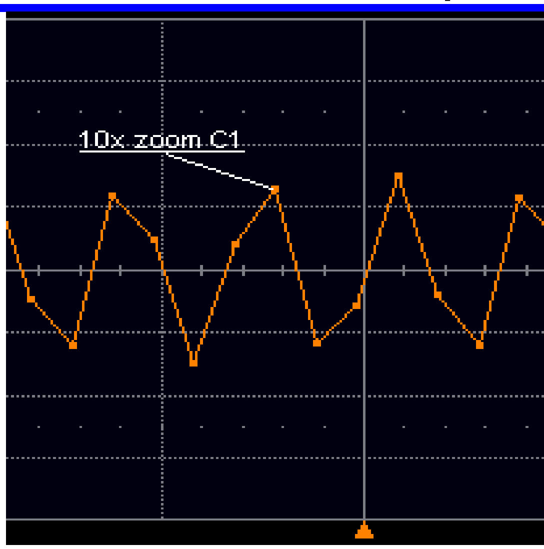
The title bar of the **Trigger** label shows the trigger source: direct, prescale, or TDR. Below the title bar is given the mode (stop, auto, or normal), level (0 mV), and slope (Positive).



Shown below the Timebase and Trigger labels is setup information for horizontal cursors, including the time between cursors and the frequency.

Trace Annotation

The instrument gives you the ability to add an identifying label, bearing your own text, to a waveform display:

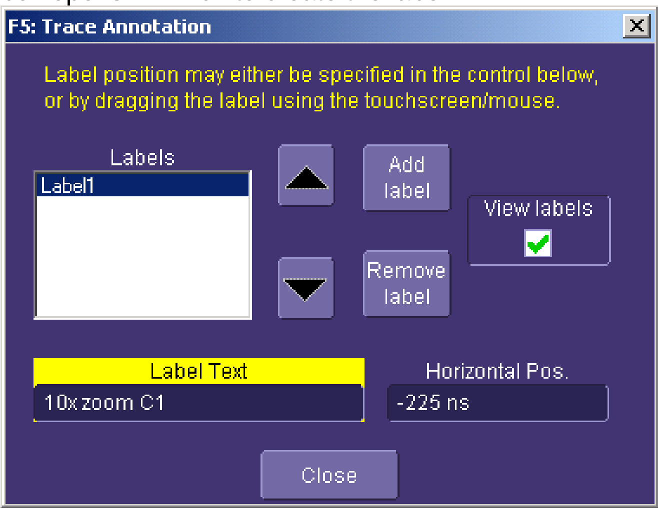


For each waveform, you can create multiple labels and turn them all on or all off. Also, you can position them on the waveform by dragging or by specifying an exact horizontal position.

The labels are preserved when the waveform is saved as a LabNotebook entry and when saved to file.

To Annotate a Waveform

1. Touch the waveform you want to annotate, then **Set label...** in the pop-up menu. A dialog box opens in which to create the label.



If you are creating a label for the first time for this waveform, **Label1** is displayed with default text. If you are modifying an existing label, under **Labels** touch the label you want to change.

Note 1: If the dialog for the trace you want to annotate is currently displayed, you can touch the label button 

at the bottom to display the Trace Annotation setup dialog.

Note 2: You may place a label anywhere you want on the waveform. Labels are numbered sequentially according to the order in which they are added, and not according to their placement on the waveform.

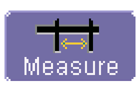
2. If you want to change the label's text, touch inside the **Label Text** field. A pop-up keyboard appears for you to enter your text. Touch **O.K.** on the keyboard when you are done. Your edited text will automatically appear in the label on the waveform.
3. To place the label precisely, touch inside the **Horizontal Pos.** field and enter a horizontal value, using the pop-up numeric keypad.
4. To add another label, touch the **Add label** button. To delete a label, select the label from the list, then touch the **Remove label** button.
5. To make the labels visible, touch the **View labels** checkbox.

To Turn On a Channel Trace Label

Note: If you want to display each trace on its own grid automatically, enable Autogrid by touching **Display** in the menu bar, then **Autogrid** in the drop-down menu.

- On the front panel, press a channel select button, such as 

, to display the trace label for that input channel and turn on the channel.
- To turn on a math function trace, touch **Math** in the menu bar, then **Math Setup...** in the drop-down menu. Touch the **On** checkbox for the trace you want to activate.
- You can also quickly create traces (and turn on the trace label) for math functions and memory traces, without leaving the Vertical Adjust dialog, by touching the icons at the bottom of the Vertical Adjust dialog:



, 

, 

, 

.

Whenever you turn on a channel, math, or memory trace via the menu bar at the top of the screen, the dialog at the bottom of the screen automatically switches to the vertical setup or math setup dialog for that selection. You can configure your traces from here, including math setups.

The channel number appears in the **Vertical Adjust** tab of the "Vertical Adjust" dialog, signifying that all controls and data entry fields are dedicated to the selected trace.

Front Panel Controls

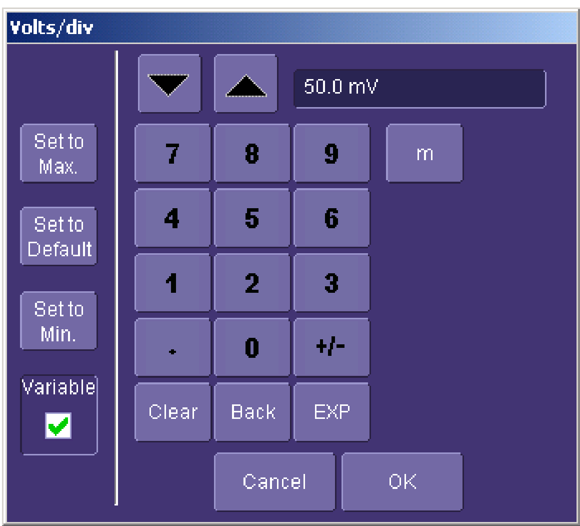
Front Panel Buttons and Knobs

The control buttons of the instrument's front panel are logically grouped into analog and special functional areas. Analog functions are included in the Horizontal, Trigger, and Vertical groups of control buttons and knobs.

Sometimes you may want to change a value without using the numeric keypad. In that case, simply touch once inside the data entry field in the scope dialog area (the field will be highlighted in yellow), then use the Adjust group of buttons and single knob to dial in values into the selected field.

By default, the control knob makes coarse adjustments (that is, digits to the left of the decimal point). Press the **FINE** button to adjust digits to the right of the decimal point. To enter exact values, you can also display a keypad by touching twice inside the data entry field.

Example Data Entry Field

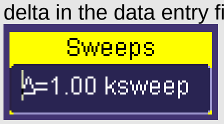


Example Pop-up Numeric Keypad

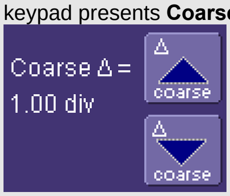
Then use the keypad to type in the value. The **SELECT** button steps through a dialog from one control to the next.

Note: You can set the granularity (delta) of the coarse adjustment in two ways:

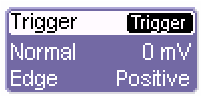
• By pressing and holding the Fine front panel button while turning the Adjust knob. In this case you can read the changing delta in the data entry field that is selected:



• By double-tapping inside the data entry field, then touching the **Advanced** checkbox in the pop-up numeric keypad. The keypad presents **Coarse** delta up/down buttons to set the delta:



In the pop-up keypad, be sure to leave the **Fine** checkbox unchecked to adjust the coarse delta.

Trigger Knobs:	
Level	Selects the trigger threshold level. The Level is indicated in the **Trigger** label: 
Trigger Buttons:	
Setup	Activates the trigger setup menu to select the trigger type and the trigger conditions.
Stop	Prevents the scope from triggering on a signal. If you boot up the instrument with the trigger in Stop mode, the message "no trace available" will be displayed. Press Auto to display your trace.
Auto	Triggers the scope using the internal timebase if no trigger is present, or using the selected trigger input if a trigger signal is present.
Normal	Triggers the scope only when a trigger signal is present at the selected trigger input.
Single	Enables triggering on the selected trigger input until the selected waveform memory is filled, after which the trigger is disabled.

Horizontal Knobs:	
Delay	Horizontally positions the scope trace on the display. The minimum setting is 12 ns in sequential mode and 0 ns in the CIS mode.
Time/Division	Sets the time/division of the scope timebase (acquisition system). The time scale is adjusted from the left edge of the display.
Horizontal Buttons:	
Zero	Sets the horizontal delay to zero. The trigger point is positioned on the left edge of the display. This button sets the delay to 12 ns (minimum) in Sequential mode.
Calibrate	Sets up the automatic calibration mode for the sampling strobes and TDR pulses.
Setup	Activates the TIMEBASE menu to allow you to select acquisition conditions, including the sample mode, maximum memory length, etc.
Vertical Knobs:	
Offset	Adjusts the vertical offset of a channel.
Volts/Div	Adjusts the Volts/Division setting of the channel selected.
Channel Buttons:	
1, 2, 3, 4	Turns a channel on or off. These buttons activate the dialog that lets you change the channel's setup conditions including coupling, gain, and offset. They are used also to select multiple grids, to automatically set the gain (FIND SCALE), or to automatically display a zoom of the signal. Press twice to toggle the trace on and off. The Vertical dialog is context sensitive giving different setup choices for the eye mode and oscilloscope modes.
Wavepilot Control Knobs:	
Position	Adjusts the horizontal position of a zoom trace on the display. The zoom region is highlighted in color on the source trace.
Zoom	Adjusts the horizontal zoom (magnification factor) of the selected zoom trace.
Position	Adjusts the vertical position of the selected zoom trace on the display.

↕ *Zoom*	Adjusts the vertical zoom (magnification factor) of the selected zoom trace on the display.
Wavepilot Control Buttons:	
Reset	Resets the zoom factors.
Math	Provides access to the Math setup dialog.
Measure	Provides access to the Measure setup dialog.
Analysis	Provides access to the Analysis setup dialogs.
Quick Set Buttons:	
Scope	Takes the scope out of TDR or Eye mode.
TDR	Provides access to the TDR setup dialog and sets the scope to TDR mode.
Eye	Provides access to the Eye mode setup dialog and sets the scope to eye mode.
Special Features Buttons:	
Auto Setup	Automatically sets the scope's horizontal timebase (acquisition system), vertical gain and offset, as well as trigger conditions, to display a wide variety of signals.
Cursors	The center button calls up the "Standard Cursors" setup dialog. The other two buttons control the placement of the cursors on your waveform.
Default Setup	Sets the scope's horizontal timebase (acquisition system), vertical gain and offset, and trigger conditions to default settings.
Help	Displays the on-line Help manual. You can choose to receive control help, or to search for the information you need using the Table of Contents and Index. Control Help displays help for a particular button, menu item, data field, etc. contained in the dialogs.
Save/Recall	Calls up the dialogs for saving and recalling waveforms and setups, and for disk utilities.
Analog Persist	Provides a three dimensional view of the signal: time, voltage, and a third dimension related to the frequency of occurrence, as shown by a color-graded

	(thermal) or intensity-graded display.
QuickZoom 	Automatically displays magnified views of up to four signal inputs on multiple grids. With four input signals, the signals are displayed along with four zoom traces, each on its own grid. This button turns off all other traces.
General Control Buttons:	
Print Screen 	Prints the displayed screen to a file, a printer, the clipboard, or attaches it as an e-mail. Select the device and format it in the Utilities --> Hardcopy dialog.
Utilities	For setup of scope features including hardcopy devices and formats, date and time, and remote control interfaces, etc.; or for checking status, options, etc..
Touch Screen (toggle switch)	Activates or deactivates the touch screen.
Clear Sweeps	Clears data from multiple sweeps (acquisitions) including: persistence trace displays, averaged traces, parameter statistics, and Histicons. During waveform readout, cancels readout.
STANDBY Lamp:	The STANDBY lamp indicates when the scope has placed itself in standby mode. In this mode, current settings are retained.

On-screen Toolbars, Icons, and Dialog Boxes

Menu Bar Buttons

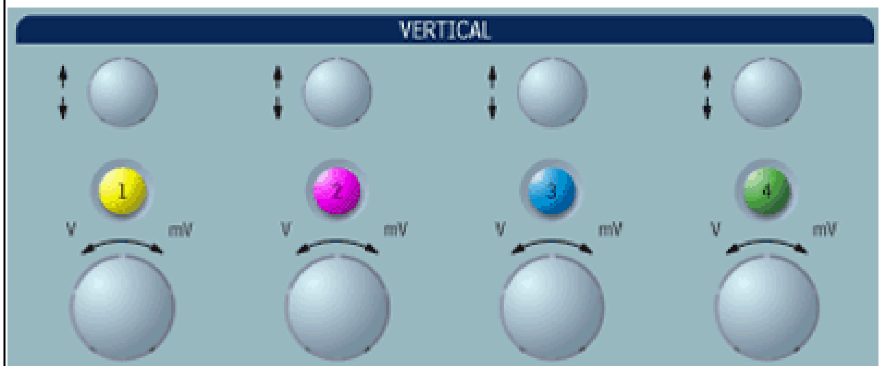
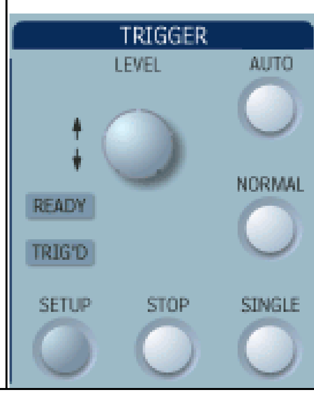
The menu bar buttons at the top of the scope's display are designed for quick setup of common functions. At the right end of the menu bar is a quick setup button that, when touched, opens the setup dialog associated with the trace or parameter named beside it. The named trace or parameter is the one whose setup dialog you last opened:

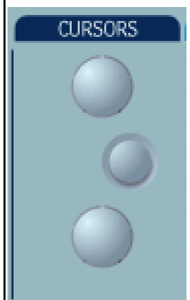
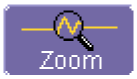
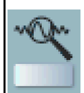
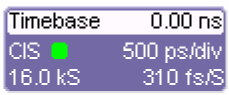
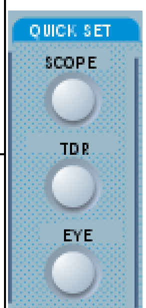
.

This button also appears as an undo button 

after front panel buttons Autosetup and **QUICKZOOM**  are pressed. If you want to perform an Undo operation, it must be the very next operation after you perform the Autosetup or QuickZoom operation.

Many of the menu bar buttons give you access to the same functions as do the front panel buttons. Refer to this Table of Equivalent Functions.

Display Buttons	Front Panel Push Buttons
File	
Vertical	
Timebase	
Trigger	

	
 (icon in Channels dialog zooms one trace)	 (zooms all displayed traces)
 (then TDR button)  (then Eye button)	
 (then Utilities Setup...)	
	

Dialog Boxes

The dialog area occupies the bottom one-third of the screen. To expand the signal display area, you can minimize each dialog box by touching the **Close** tab at the right of the dialog box.

Screen Layout

The instrument's screen is divided into three areas:

- menu bar
- signal display area
- dialog area

Menu Bar

The top of the screen contains a toolbar of commonly used functions. Whenever you touch one of these buttons, the dialog area at the bottom of the screen switches to show the setup for that function.

Signal Display Grid

You can set up the signal display area by touching 

in the toolbar, then the 

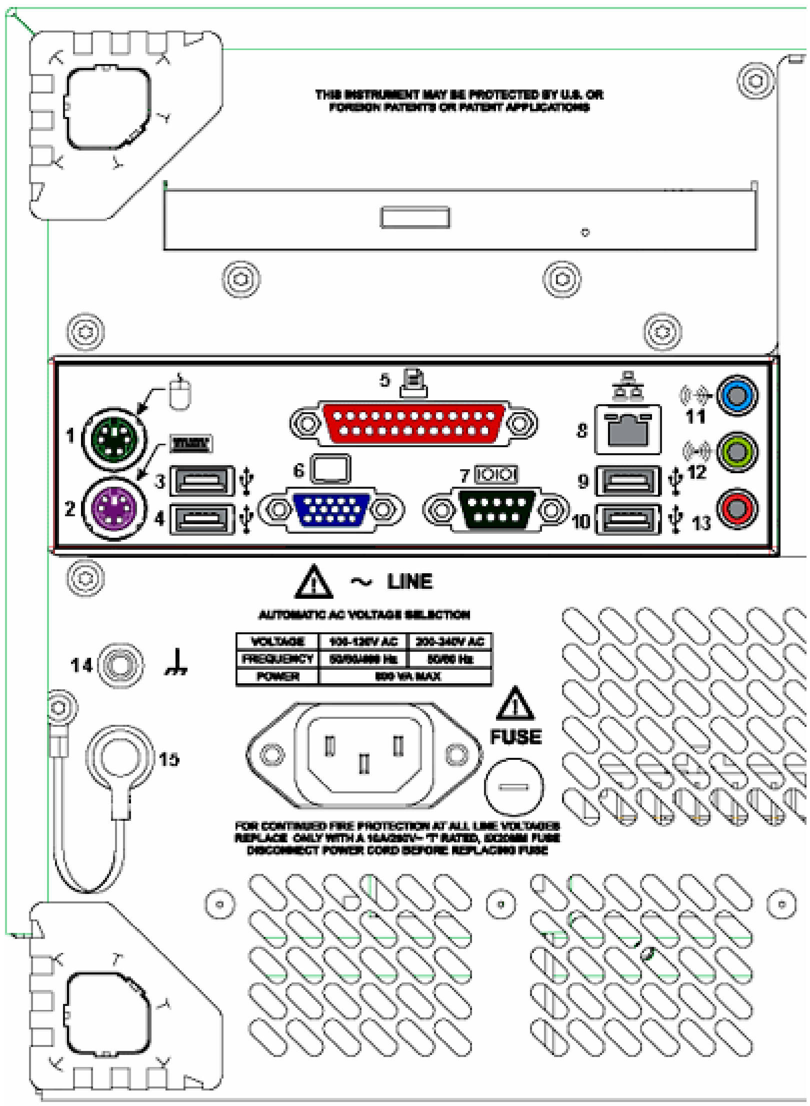
tab. The display dialog offers a choice of grid combinations and a means to set the grid intensity.

Dialog Area

The lower portion is where you make selections and input data. The dialog area is controlled by both toolbar touch buttons and front panel push buttons. Similarities between functions are shown in the Table of Equivalent Functions above.

Hardware

Instrument Rear Panel



(1) Mouse; (2) Keyboard; (3) USB Port; (4) USB Port; (5) Centronics Port; (6) External VGA Monitor; (7) RS-232-C Port; (8) Ethernet Port; (9) USB Port; (10) USB Port; (11) Line In; (12) Speakers; (13) Microphone; (14) Ground Connector; (15) Gated Trigger Input with Grounded EMI Shield installed (required when port is not in use)

Software

Checking the Scope Status

To find out the scope's software and hardware configuration, including software version and installed options, proceed as follows:

1. In the menu bar, touch Utilities.
2. Touch the Status tab.

You can find information related to hard drive memory, etc. as follows:

1. Minimize the instrument application by touching File, then selecting **Minimize** in the drop-down menu.
2. Touch the **Start** taskbar button and, per usual Windows® operation, open Windows Explorer.

Default Settings

You can reset the scope to default settings by simply pressing the **DEFAULT SETUP** push button on the front panel. This feature turns on Channel1 and Channel 2, with no processing enabled.

Other default settings are as follows:

Vertical	Timebase	Trigger
100 mV/div	1.00 ns/div	Internal
0 V offset	10.0 kS/s, -12 ns delay	TDR
	Sequential acquisition	Auto trigger mode

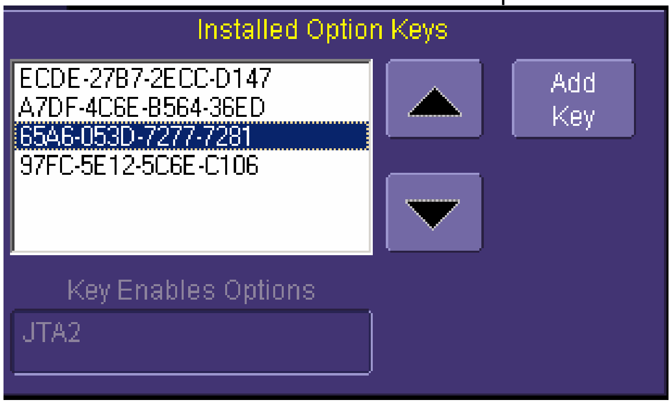
Adding a New Option

To add a software option you need a key code to enable the option. Call LeCroy Customer Support to place an order and receive the code.

To add the software option do the following:

1. In the menu bar, touch Utilities.
2. In the dialog area, touch the Options tab.
3. Touch Add Key.

4. Use the pop-up keyboard to type the key code. Touch **O.K.** on the keyboard to enter the information.
5. The name of the feature you just installed is shown below the list of key codes. You can use the scroll buttons to see the name of the option installed with each key code listed:



The full array of installed software and hardware options is displayed on the left side of the dialog:

Restoring Software

Restarting the Application

Upon initial power-up, the scope will load the instrument application software automatically. If you exit the application and want to reload it, touch the shortcut icon on the desktop:

.

If you minimize the application, touch the appropriate task bar or desktop icon to maximize it:


.

Restarting the Operating System

If you need to restart the Windows® operating system, you will have to reboot the scope by pressing and holding in the power switch for 10 seconds, then turning the power back on.

Removable Hard Drive

The removable hard drive option replaces the standard internal hard drive with a removable hard drive that is installed at the rear of the scope, in the slot normally occupied by the CD-ROM drive. The kit includes two hard drives, which can be used interchangeably. It also includes a USB CD-ROM for loading of new software.

Caution! The Removable Hard Drive Is Not Hot-swappable

To avoid damage to the drive or the oscilloscope, shut off power to the oscilloscope before you insert or remove the hard drive. Ensure that the protective cover is installed over the drive at all times.

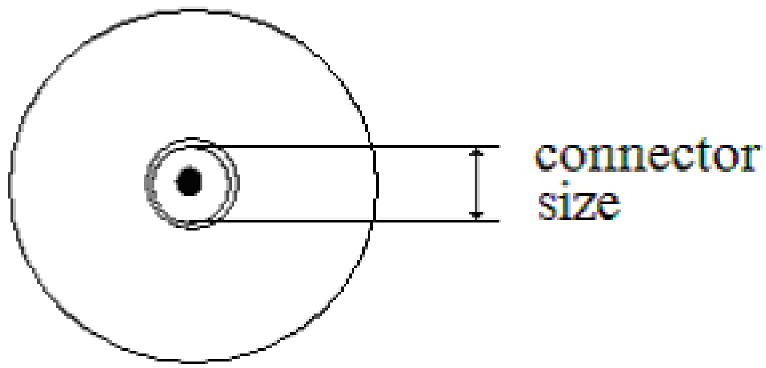
Proper Orientation of Drive

Protective Cover

Electrical Modules

The sampling modules are equipped with RF connectors of different types depending on the bandwidth of the particular module, as indicated in the table below. The modules have male connectors, and each module is shipped with a female-to-female adapter that matches the connector type. This adapter should be attached to the module whenever possible, as it will protect the connector on the module from possible damage caused by incorrect threading of cables or connectors.

The connector types on the modules are named according to the inside diameter of their outer conductor. This diameter must be matched within any connectors or cables attached to the module in order to maintain the specified bandwidth.



Module	Bandwidth	Connector Type
ST-20	20 GHz	2.92 mm
SE-30	30 GHz	2.92 mm
SE-50	50 GHz	2.40 mm
SE-70	70 GHz	1.85 mm
SE-100	100 GHz	1 mm

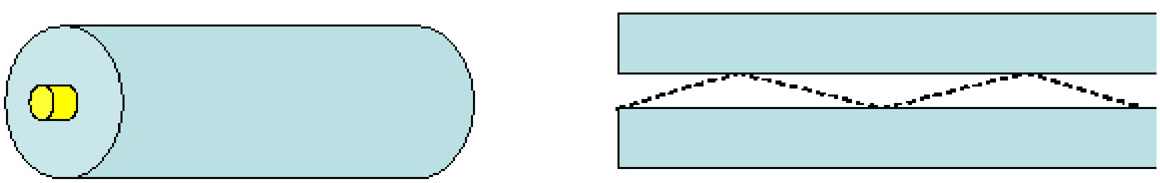
Connector Torque

In order to maintain bandwidth performance and minimize loss, connections to the instrument modules should be torqued to the appropriate tightness. The connectors all have standard 3/8 inch nuts on them and should be torqued to 8 in-lbs. using an appropriate torque wrench. The 100 GHz module has a unique connector that requires a smaller torque value of 4 in-lbs. This module is supplied with the appropriate torque wrench.

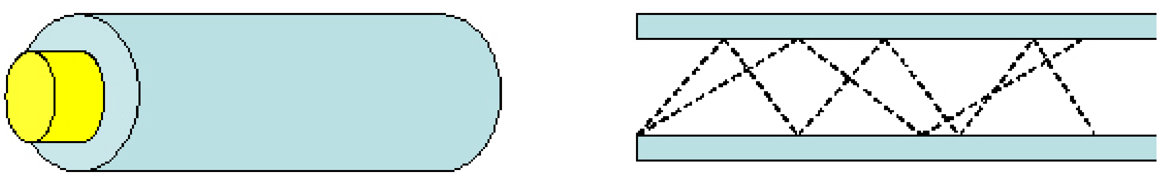
Optical Modules

Several optical modules are also available for use with the WaveExpert mainframe. The modules are outlined in the table below. Optical modules are available in single-mode and multimode configurations. The single-mode modules are designed to operate over wavelengths in the 1280 to 1620 nm range, while the multimode module covers the 750 to 1620 nm range. Single-mode and multimode fibers are characterized by different diameters. Multimode fiber, as its name implies, allows many "modes" or group velocities to propagate, while single-mode allows only one. The diameter of multi-mode fiber is much larger than that of single-mode fiber (50 to 62.5 nm vs. 9.5 nm) so connecting a multimode fiber to a single-mode one will result in a large amount of signal attenuation caused by the difference in area of the connector faces. Since the energy in a multimode signal is spread out into the many modes propagating in the fiber, at the boundary with the single-mode fiber only one of these modes will pass through.

Module	Wavelength Range	Fiber Type	Fiber Diameter
SO50	1280-1620 nm	Single mode	10 µm
SO25	1280-1620 nm	Single mode	10 µm
SO10	750-1650 nm	Multimode	62.5 µm



Single-mode fiber has a narrow core diameter that allows only one mode to propagate. This type of fiber is very low loss, so it is the choice for long-haul communications.



Multimode fiber has a much wider core diameter that allows many modes to propagate. This type of fiber has higher loss, but is less costly, is easier to work with, and is the choice for LAN and other short-haul applications.

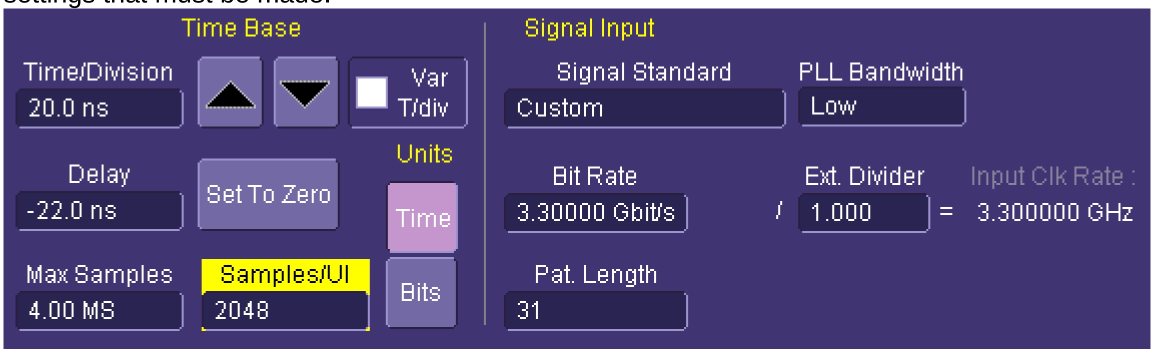
The optical modules come standard with an FC-PC connector, and adapters are available for a variety of other optical connector types. Adapters can be easily swapped on the optical module front panel by pressing the release latch on the bottom of the connector while pulling on it. The exposed fiber end can be cleaned using alcohol and/or an optical fiber cleaner.

SAMPLING MODES

There are two sampling modes available on the NRO9000 and SDA100G; coherent interleaved sampling (CIS) and Sequential (SEQ). The WE9000 only supports the sequential mode (SEQ)

To select a sampling mode

1. In the menu bar, touch **Timebase**, then **Horizontal Setup...** in the drop-down menu.
2. In the "Horizontal" dialog, touch a **Sample Mode** button. The CIS mode includes several settings that must be made.



The signal data rate must be entered into the **Bit Rate** field, and the pattern length must be entered into the **Pat. Length** field. The pattern length is the number of symbols in the data stream after which the pattern repeats. If the incorrect length is entered or the signal does not have a repeating pattern, the display will show an eye diagram. The **Samples/UI** field allows you to set the samples per unit interval (UI). Another way to view the sample density is as an effective sampling rate. The **Ext. Divider** field is used when a clock at a rate lower than the actual symbol rate is being applied to the trigger input. Using this control, you only need to know the symbol rate of the signal and the divide ratio of the clock. The **PLL Bandwidth** sets the loop response 3 dB point of the internal phase locked loop in the CIS strobe generation circuit. The loop bandwidth is approximately 10 kHz in the **Low** setting and 1 MHz in the **High** setting.

Note: the **Low** setting will result in an apparent increase in the observed jitter in the signal under test. This is because the lower jitter in the timebase allows more of the low frequency jitter in the signal under test to be visible.

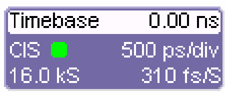
Sequential Sampling Mode

The standard mode for sampling the signal is through a technique known as sequential sampling. In this mode, a trigger signal applied to either the trigger or prescale inputs on the scope front panel is used to start the timebase. Once started, the timebase waits for a minimum amount of time before generating the sampling strobe that causes the sampling modules each to take one voltage sample. On each subsequent trigger, the sampling strobe is taken at a time slightly longer than the previous sample after the timebase. The delay difference between each subsequent sample is controlled by the time/division and record length settings.

Coherent Interleaved Sampling Mode

The coherent interleaved sampling (CIS) mode is available on the NRO 9000 and SDA 100G and uses the signal applied to the trigger input as a reference for an internal phase locked loop (PLL). The PLL generates a sampling strobe at a nominal 10 MHz rate that is phase locked to the trigger signal. The PLL operates in a fractional-N mode, so the sampling strobe is not an integer submultiple of the bit rate. Instead, the sampling rate is set so that the sample point shifts by a fraction of a unit interval for successive sample. This fraction is set by the **Samples/UI** control.

When the instrument is in CIS mode, the colored dot indicator in the Timebase label has the following meanings



Green -- The timebase is locked.

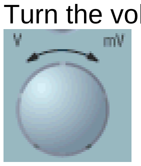
Red -- The timebase is not locked. The dot turns red when the bit rate is set incorrectly or when the trigger is in Auto mode.

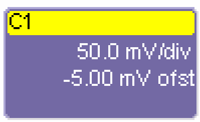
VERTICAL SETTINGS AND CHANNEL CONTROLS

Adjusting Sensitivity and Position

To Adjust Sensitivity

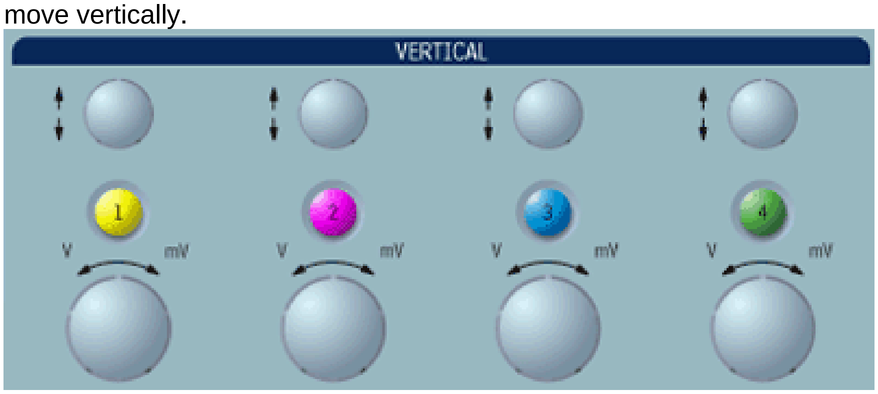
1. Press the appropriate channel push button, for example  to turn on channel 1. Or touch **Vertical** in the menu bar, then **Channel 1** in the drop-down menu.
2. Touch inside the **Trace On** checkbox to display the trace.


3. Turn the volts per division knob for the selected channel.


Or you can touch inside the **Volts/Div** field and type in a value using the pop-up keypad, or use the up/down arrows.
4. The voltage that you set is displayed in the trace descriptor label and in the **Volts/Div** field.


To Adjust the Waveform's Position

Turn the vertical offset adjust knob directly above the channel button whose waveform you want to move vertically.



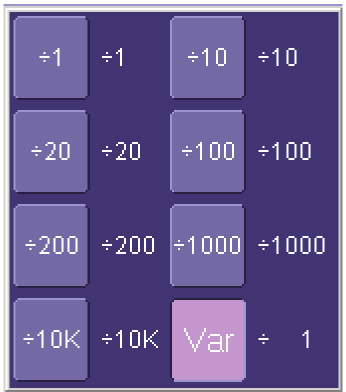
Or you can touch inside the **Offset** field and type in a value on the pop-up keypad. To set the vertical offset to zero, touch the **Zero Offset** button directly below the **Offset** field.

Probe Attenuation

To Set Probe Attenuation

LeCroy's ProBus® system automatically senses probes and sets their attenuation for you. If you want to set the attenuation manually,

1. In the menu bar, touch **Vertical**, then select a channel from the drop-down menu.
2. Touch inside the **Probe Atten.** field. Touch a divide-by menu selection or touch **Var** (variable). If you choose **Var**, type in a value using the pop-up numeric keypad.



Linear and (SinX)/X Interpolation

Linear interpolation, which inserts a straight line between sample points, is best used to reconstruct straight-edged signals such as square waves. (Sinx)/x interpolation, on the other hand, is suitable for reconstructing curved or irregular wave shapes, especially when the sample rate is 3 to 5 times the system bandwidth.

To Set Up Interpolation

1. Touch the button for the channel you want to set up,  for example.
2. In the dialog area, touch inside the **Interpolation** data entry field under **Pre-Processing**. "Pre-Processing" means before Math processing.
3. Touch inside the **Interpolation** data entry field. A pop-up menu appears offering **Linear** or **Sinx/x** interpolation.
4. Touch the button for the type of interpolation you want.

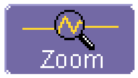
Inverting Waveforms

Touch the **Invert** checkbox to invert the waveform for the selected channel.

QuickZoom

QuickZoom automatically displays a zoom of the channel or trace on a new grid.

To Turn On a Zoom

Touch the **Zoom** button 

in the channel dialog.

Finding Scale

You can access the **Find Scale** button from the channel setup dialog. This feature automatically calculates peak-to-peak voltage, and chooses an appropriate Volts/Div scale to fully display the waveform.

To Use Find Scale

1. Touch the trace label for the waveform you desire.
2. Touch the **Find Scale** icon.

Variable Gain

Variable Gain lets you change the granularity with which the gain is incremented. For example, when **Variable Gain** is disabled, the gain will increase or decrease in preset increments of 10 or 100 mV each time you touch the **Up/Down** buttons.

However, when **Variable Gain** is enabled, you can increase or decrease the gain in increments as small as 1 mV, depending on the scale of the waveform.

To Enable Variable Gain

1. Touch the descriptor label for the waveform whose gain you want to vary.
2. Touch the **Variable Gain** check box.

Channel Deskew

Unlike the Deskew math function, channel Deskew does no resampling, but instead adjusts the horizontal offset by the amount that you enter. The valid range is dependent on the current timebase +/- 9 divisions.

To Set Up Channel Deskew

1. In the menu bar, touch **Vertical**; from the drop-down menu, select a channel to set up.
2. Touch inside the **Deskew** data entry field and enter a value using the pop-up numeric keypad.

Dark Calibration

Dark calibration is a compensation value added to a channel with an optical module. The **Dark Cal Level** is the residual power measured by the optical head with no input applied (dark input), and is used by the extinction ratio measurement. The dark calibration is independent of what type of module is connected to the selected channel, so an external optical-to-electrical converter can be compensated.

To perform dark calibration

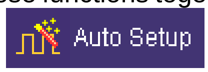
1. In the menu bar, touch **Vertical**, then select a channel from the drop-down menu.

2. Touch the **Dark Cal** button 

.
3. A dialog box will appear instructing you to disconnect any optical signal from the channel. Touch the **Start Dark Calibration** button when this is done.
4. Dark calibration will be performed, and the dark level will appear in the **Dark Cal Level** field.

Autosetup

The autosetup function operates differently depending upon the operating mode of the instrument. The functions included in autosetup are vertical scaling, setting the bit rate of the signal under test ,and setting the horizontal scale.

You can perform an autosetup of all these functions together by simply pressing **AUTO SETUP** on the front panel, or by touching **Autosetup** 

in the Vertical, Timebase, or Trigger drop-down menu. The autosetup operates in the following way for each operating mode:

Scope Mode	Sets the vertical scale to 120% of the signal amplitude, finds the bit rate, and sets the horizontal scale to 1 UI per division (UI = 1/bit rate).
Eye Mode	Sets the vertical scale to 120% of the signal amplitude, finds the bit rate, sets the horizontal scale to 1.66 unit intervals (0.166 UI/division), and sets the UI offset to 0.
TDR Mode	Sets the selected channel to average mode (32 sweeps), the horizontal scale to 100 ps/division, and the trigger to TDR mode (free running)

Timebase Setup and Control

Set up the timebase by using the front panel Horizontal controls, just as for analog scopes. The timebase in a sampling oscilloscope such as the Wave Expert is controlled by the trigger signal. Therefore, the sampling rate is set by the trigger signal frequency and is limited by the maximum acquisition rate of approximately 53 kS/s in Sequential mode. The sampling rate is fixed at 10 MS/s in CIS mode in the NRO 9000 and SDA 100G models. Adjusting the horizontal scale control varies the amount of waveform memory used by the acquisition system.

Touch **Timebase** in the menu bar and then **Horizontal Setup…** in the drop-down menu to display the "Horizontal" menu. The menu consists of 4 sections.

Operation Modes

Operation Modes

The Operation Mode section contains three buttons, which are also available on the instrument front panel in the **Quick Set** control group. Each mode affects the horizontal and vertical menus as well as the display mode, as appropriate, for the selected mode of operation.

Scope Mode

Scope mode is selected by touching the **Scope** button in the "Horizontal" menu or by pressing the Scope button on the instrument front panel in the Quick Set control group. The scope mode displays waveform data directly on the screen either as a voltage vs. time waveform or as an eye pattern, depending upon the trigger type and sampling mode.

TDR Mode

TDR mode is selected by touching the **TDR** button in the "Horizontal" menu or by pressing the TDR button on the instrument front panel in the Quick Set control group. This displays the TDR menu tab that appears with the "Horizontal" menu. The acquisition mode is set to sequential and waveform averaging is enabled (32 sweeps). The "TDR Main" menu provides access to control of the TDR step generators in the ST-20 sampling modules. Only channels that contain ST-20 modules are capable of generating a TDR step. Channels containing other modules or no module will be indicated in the menu with the designation: “Pod has no TDR capabilities” or “Pod is missing.”

1. Touch the **On** check box to activate the TDR step for the selected channel.
2. Select **Pos** or **Neg** step polarity in the **Polarity** control
3. Select the trace scaling in V (volts), (reflectance), (impedance) or jX (reactance). The vertical scale of the trace is interpreted as follows:

V (volts)	Displays the sum of the incident and reflected voltages at the sampling diode within the sampling head.
ρ (reflectance)	The vertical scale is the reflection coefficient: $V_{reflected}/V_{incident}$. varies from 0 (a perfectly matched load with no reflection) to 1 (an open circuit with 100% reflection).
Ω (impedance)	The vertical scale is the characteristic impedance of the network connected to the TDR head. The impedance is determined from the reflectance using the relationship $$Z = Z_0 \frac{(1+\rho)}{(1-\rho)}$$
jX (reactance)	The display is scaled in volts and parameter P1 indicates the reactance (capacitive or inductive) of the section of the trace between the cursors. The cursors are turned on automatically when jX is selected. The reactance is determined by integrating the voltage between the cursors. A negative area results from the capacitive reactance and a positive area from an inductive one. The reactance is computed from

	$L = \frac{1}{2}\int_{t1}^{t2} Z(t)dt$ for inductance, and $C = \frac{1}{2}\int_{t1}^{t2} \frac{1}{Z(t)}dt$ for capacitance. Note that the reactance values are only approximate and do not take into account the affect of multiple reflections from impedance mismatches before the section being measured.

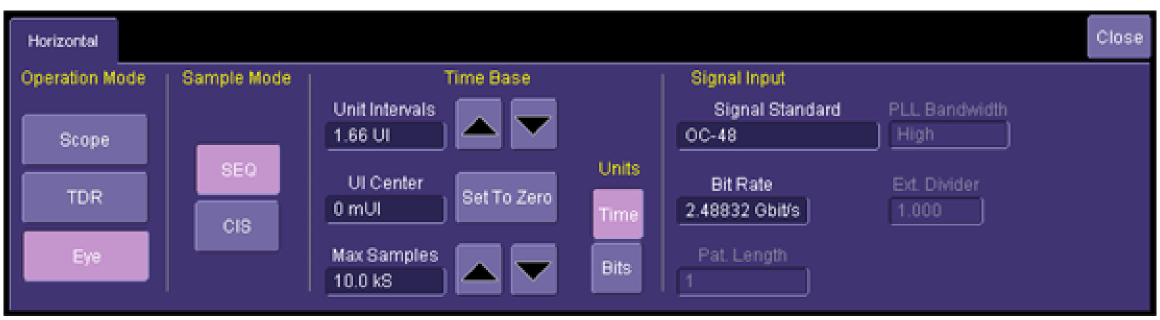
4. Differential TDR is selected by checking the **Differential** checkbox.

Note: differential operation is only possible when two ST-20 modules are placed in channels 1 and 2 or channels 3 and 4. The default configuration is channel 1 positive and channel 2 negative (for Ch 1+2 operation) and channel 3 positive and channel 4 negative (for Ch 3+4 operation).

5. Adjust the timing skew using the **Skew** control. The skew control shifts the waveforms to line them up with, for example, the connection between the cables and the device under test.
6. Select the desired edge rate (rise time) by setting the appropriate value in the **Edge** control. The rise time of the ST-20 TDR step is 20 ps; however, this is often much faster than most signals. It is often desirable to view the TDR trace using the rise time of the signal for which the network under test was designed. The **Edge** control is implemented using a finite impulse response (FIR) low-pass filter applied to the TDR trace. The filter is a Gaussian design with its cutoff frequency set to give the selected rise time.

Eye Mode

Eye Mode is selected by touching the **Eye** button in the "Horizontal" menu, or by pressing the Eye button on the instrument front panel in the Quick Setup group. Eye mode sets the timebase to the sequential mode and sets the display to infinite persistence mode. When in Eye mode, all channels in the instrument are set to display the eye pattern of their respective signals. Up to four eye patterns sharing the same timebase can be displayed. The horizontal menu changes to reflect the unique functions in the eye mode:



The sample mode controls are set to SEQ. CIS is grayed in the SDA 100G and NRO 9000 models.

1. Set the number of unit intervals to be displayed by entering the value in the **Unit Intervals** field. The up and down arrows increment or decrement the number of UI in steps as fine as .01UI.
2. Position the eye pattern in the screen by setting the **UI Center** value. The default value for this control is 0 UI, which places the trigger point at the center of the screen. The trigger point is assumed to be at the center of a unit interval (UI), but this may not be the case if the path lengths for the signal and trigger are different (due to different cable lengths, for example). A positive setting for the UI Center moves the eye diagram to the left on the screen and a negative value moves the eye pattern to the right.
3. Select the number of samples per waveform, using the **Max Samples** control. The minimum value is 500 samples per waveform and the maximum is 100k for the SDA 100G and NRO 9000 models. The maximum for the WE 9000 is 16k with standard memory and 100k with option L. In general, a higher number of samples per waveform will result in faster throughput.
4. Select **Time** or **Bits** for the horizontal display by touching the appropriate button in the menu. The selected scale factor is also applied to all measurements on the eye patterns (i.e., UI for Bits and ps for Time).
5. Select the signal standard being measured in the **Signal Standard** field. The signal standard sets the value in the **Bit Rate** field and loads any eye pattern masks associated with the standard. The specific mask is selected and enabled in the vertical menu for the selected channel.
6. Set the bit rate if a non-standard rate is being measured, or if **Custom** is selected for Signal Standard.

TRIGGERING

Trigger Setup

Trigger Setup Considerations

Trigger Modes

Auto mode causes the scope to sweep even without a trigger. When there is no trigger signal present at the selected trigger input, the internal clock generates a sampling strobe at a rate of 1M samples per second.

In **Normal** mode, the scope acquires data only when a trigger signal is present at the selected trigger input. Otherwise, it continues to display the last acquired waveform.

In **Single** mode, the scope will acquire data if a trigger signal is present at the selected trigger input. The scope acquires one sample per trigger in sequential mode until the number of samples indicated in the horizontal descriptor box has been acquired; then the acquisition stops. On the SDA 100G and NRO 9000 models, the selected number of samples is acquired at a 10 MHz sampling rate before the acquisition stops.

Stop mode inhibits all acquisitions until you select one of the other three modes.

Trigger Types

The triggers available to you are defined as follows:

Trigger	Direct trigger. This button selects the Trigger input on the instrument front panel as the timebase trigger. The trigger signal is DC coupled and includes slope and level controls. The Trigger input will respond to trigger signals up to 5 GHz. This input is also used for the CIS timebase in the SDA 100G and NRO 9000 for trigger signals below 125 MHz rate.
Prescaler	This button selects the Prescale input on the instrument front panel as the trigger signal for the timebase. The prescale input is AC coupled and has no level or slope controls. This is the primary input for the CIS timebase in the SDA 100G and NRO 9000 for trigger rates above 125 MHz. The Prescale input responds to trigger signals up to 14 GHz
TDR	The TDR trigger is an internal, free-running clock that is used to drive the TDR step generator in the ST-20 module at the same time as the sampling strobe. This button cannot be selected in the trigger menu; rather, it is activated when the TDR mode is selected in the timebase "Horizontal" menu.

Determining Trigger Level, Slope, Source, and Coupling

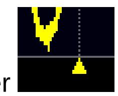
Level -- Defines the voltage at which the trigger circuit will generate an event and cause a sampling strobe to be generated. This control is only available when the **Trigger** source is selected.

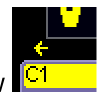
Coupling -- The Trigger input is DC coupled with a 50W load. The Prescale input is AC coupled with a nominal 50W load

Slope -- Determines the direction of the trigger voltage transition used for generating a particular trigger event. You can choose a positive or negative slope. This control is only available when the **Trigger** input is selected.

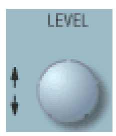
Control Edge Triggering

Horizontal: Turn the Delay knob in the HORIZONTAL control group to adjust the trigger's horizontal position. Or, touch inside the **Delay** field in the timebase setup dialog and enter a value, using the pop-up keypad.

The trigger location is shown by a marker  below the grid.

Post-trigger delay is indicated by a left-pointing arrow 

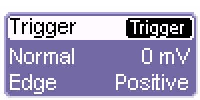
below-left of the grid. Since the sampling system does not acquire data until a trigger signal is present, the delay can never be positive. The minimum delay for the sequential mode is -12 ns. The CIS timebase in the SDA 100G and NRO 9000 runs continuously when a trigger is present and has a minimum delay of 0 ps. In this case, the position arrow appears at the left edge of the screen.

Vertical: Turn the **LEVEL** knob 

in the TRIGGER control group to adjust the trigger's vertical threshold when the trigger signal is applied to the Trigger input. This control knob has no affect when the Prescale input is being used.

Alternatively, in the "Trigger" dialog, you can touch inside the **Level** field and type in a value, using the pop-up numeric keypad. To quickly set a level of zero volts, touch the **Zero Level** button directly below the **Coupling** field.

The trigger source and level are indicated in the trigger descriptor label:

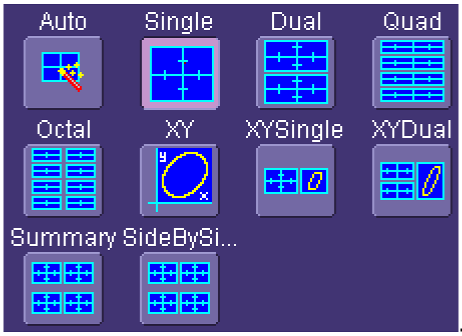


.

DISPLAY FORMATS

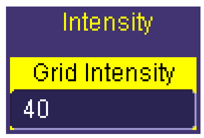
Display Setup

1. In the menu bar, touch **Display;** then touch **Display Setup** in the drop-down menu.
2. Touch one of the **Grid** combination buttons:

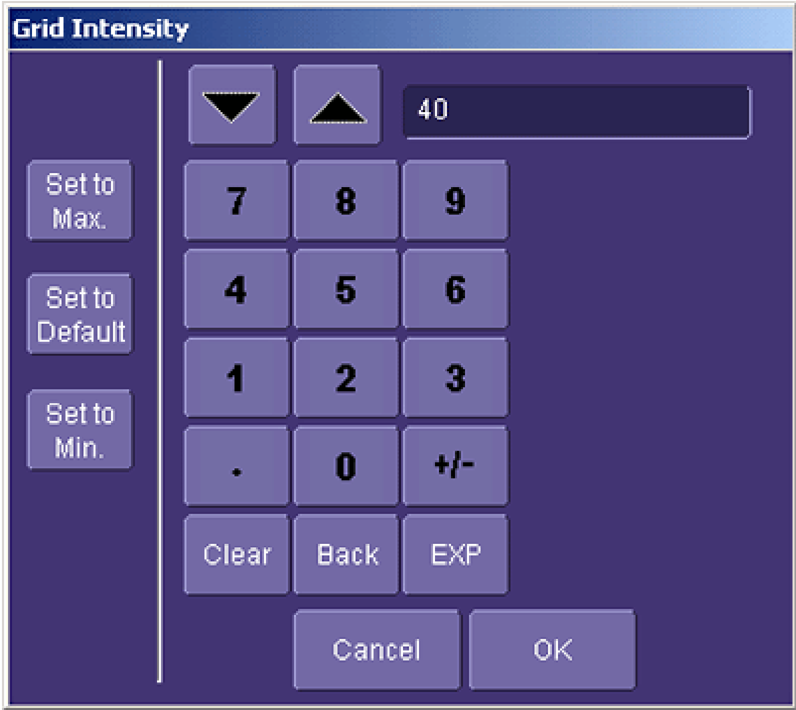


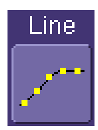
.

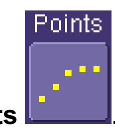
Autogrid automatically adds or deletes grids as you select more or fewer waveforms to display.

3. Touch inside the grid **Intensity** data entry field 

and enter a value from 0 to 100 using the pop-up keypad



4. Touch the **Grid on top** checkbox if you want to superimpose the grid over the waveform. Depending on the grid intensity, some of your waveform may be hidden from view when the grid is placed on top. To undo, simply uncheck **Grid on top**.
5. Touch the **Axis labels** checkbox to permanently display the values of the top and bottom grid lines (calculated from volts/div) and the extreme left and right grid lines (calculated from the timebase).
6. Choose a line style for your trace: solid **Line** 

or **Points** 

.

Persistence Setup

The analog Persistence feature helps you display your waveform and reveal its idiosyncrasies or anomalies for a repetitive signal. Use Persistence to accumulate on-screen points from many acquisitions to see your signal change over time. The instrument persistence modes show the most frequent signal path "three-dimensionally" in intensities of the same color, or graded in a spectrum of colors.

You can show persistence for up to eight inputs for any channel, math function, or memory location

(M1 to M4).

Saturation Level

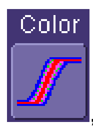
The Persistence display is generated by repeated sampling of the amplitudes of events over time, and the accumulation of the sampled data into "3-dimensional" display maps. These maps create an analog-style display. User-definable persistence duration can be used to view how the maps evolve proportionally over time. Statistical integrity is preserved because the duration (decay) is proportional to the persistence population for each amplitude or time combination in the data. In addition, the instrument gives you post-acquisition saturation control for a more detailed display.

When you select 

mode from the Persistence dialog (with **All Locked** selected), each channel is assigned a single color. As a persistence data map develops, different intensities of that color are assigned to the range between a minimum and a maximum population. The maximum population automatically gets the highest intensity, the minimum population gets the lowest intensity, and intermediate populations get intensities in between these extremes.

The information in the lower populations (for example, down at the noise level) could be of greater interest to you than the rest. The Analog persistence view highlights the distribution of data so that you can examine it in detail.

You can select a saturation level as a percentage of the maximum population. All populations above the saturation population are then assigned the highest color intensity: that is, they are saturated. At the same time, all populations below the saturation level are assigned the remaining intensities. Data populations are dynamically updated as data from new acquisitions is accumulated.

Color mode persistence, selected by touching 

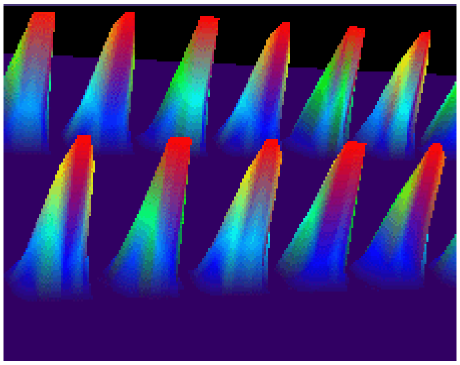
, works on the same principle as the Analog persistence feature, but instead uses the entire color spectrum to map signal intensity: violet for minimum population, red for maximum population. A saturation level of 100% spreads the intensity variation across the entire distribution; at lower saturation levels the intensity will saturate (become the brightest color) at the percentage value specified. Lowering this percentage causes the pixels to be saturated at a lower population, and makes visible those rarely hit pixels not seen at higher percentages.

3-Dimensional Persistence

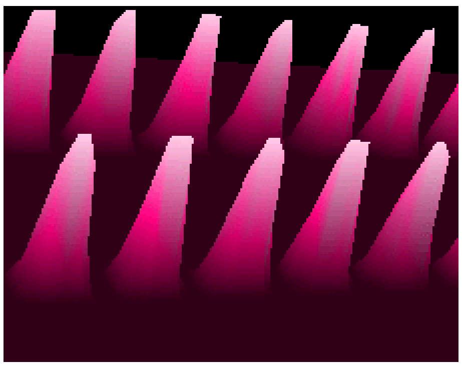
By selecting **3d** 

, you can create a topographical view of your waveform from a selection of shadings, textures, and hues. The advantage of the topographical view is that areas of highest and lowest intensity are shown as peaks and valleys, in addition to color or brightness. The shape of the peaks (pointed or flat) can reveal further information about the frequency of occurrences in your waveform.

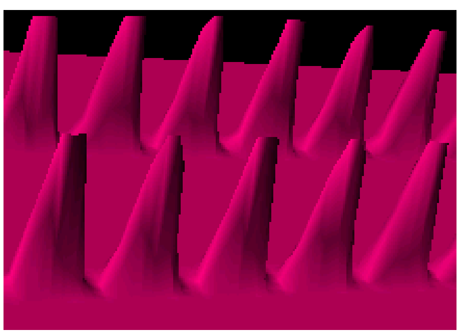
The instrument also gives you the ability to turn the X and Y axes of the waveform through 180° of rotation from -90° to +90°.



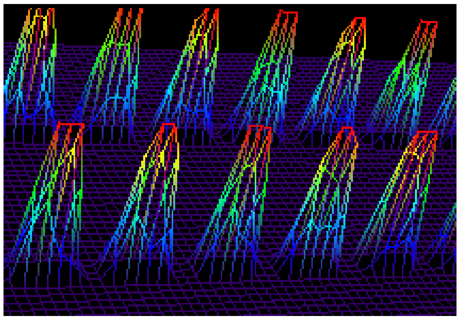
Here is an example of a 3-dimensional view of a square wave using the **solid** view of color-graded persistence. Saturation is set at 50%, with red areas indicating highest intensity. The X-axis has been rotated 60%; the Y-axis has been rotated 15%.



Here is a **monochrome** (analog) view of the same waveform. The lightest areas indicate highest intensity, corresponding to the red areas in the solid view.



Here is a **shaded** (projected light) view of the same waveform. This view emphasizes the shape of the pulses.



Here is a **wire frame** view of the same waveform in which lines of equal intensity are used to construct the persistence map.

Show Last Trace

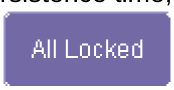
For most applications, you may not want to show the last trace because it will be superimposed on top of your persistence display. In those cases turn off **Show Last Trace** by touching the checkbox. However, if you are doing mask testing and want to see where the last trace is falling, turn **Show Last Trace** on.

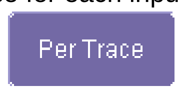
Persistence Time

You can control the duration of persistence by setting a time limit, in seconds, after which persistence data will be erased: 0.5 s, 1 s, 2 s, 5 s, 10 s, 20 s, or infinity.

Locking of Traces

The instrument gives you the choice of constraining all input channels to the same mode, saturation level, persistence time, and last trace display, or setting these for each input channel individually.

Choose 

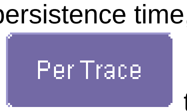
to constrain input channels. Choose 

to set up input channels individually.

To Set Up Persistence

1. In the menu bar touch **Display**, then touch **Persistence Setup...** in the drop-down menu.
2. Touch the **Persistence On** checkbox. If **Per Trace** is selected, touch the **Reset All** button 

to return all input channel setups to their default settings.
3. Touch the **All Locked** button 

if you want to set the same mode, saturation level, persistence time, and last trace display for all input channels. Touch the **Per Trace** button 

to set these for each input channel individually.
   A. If you selected **All Locked**, touch one of the mode buttons:

.

B. Then touch the **Show last trace** checkbox if you want the last trace displayed.

C. Touch inside the **Saturation** data entry field and enter a whole number integer, using the pop-up numeric keypad.

D. Touch inside the **Persistence time** data entry field and make a selection from the pop-up menu.

4. If you selected **Per Trace**, for each input channel touch its tab, then make selections of mode, saturation level, persistence time, and last trace display in the same way as for **All Locked**.

5. To create a 3-dimensional view, touch the 3d button 

. Then

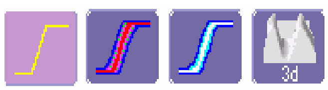
A. Touch inside the **Saturation** data entry field and enter a whole number integer, using the pop-up numeric keypad.

B. Touch inside the **Persistence time** data entry field and make a selection from the pop-up menu.

C. Under "3D settings," touch inside the **Quality** field and select an image quality from the pop-up menu: wire frame, solid, or shaded.

D. For each axis, touch inside the data entry field and enter a value from -90° to +90°.

6. To turn off persistence for an individual channel, touch the left-most persistence mode button:



.

To turn off persistence for all channels, press the front panel **ANALOG PERSIST** button. This button toggles Analog Persistence on and off.

Screen Saver

The Windows screen saver is activated in the same way as for any PC.

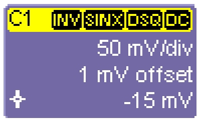
1. Minimize the instrument display by touching **File** in the menu bar, then **Minimize** in the drop-down menu.
2. Touch **Start** down in the task bar.
3. Touch **Settings** in the pop-up menu.
4. Touch **Control Panel**.
5. Touch **Display**.
6. Touch the **Screen Saver** tab.

Moving Traces from Grid to Grid

You can move traces from grid to grid at the touch of a button.

To Move a Channel or Math Trace

1. Touch the descriptor label for the waveform that you want to move.



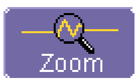
Example Descriptor Label

2. Touch the **Next Grid** button 

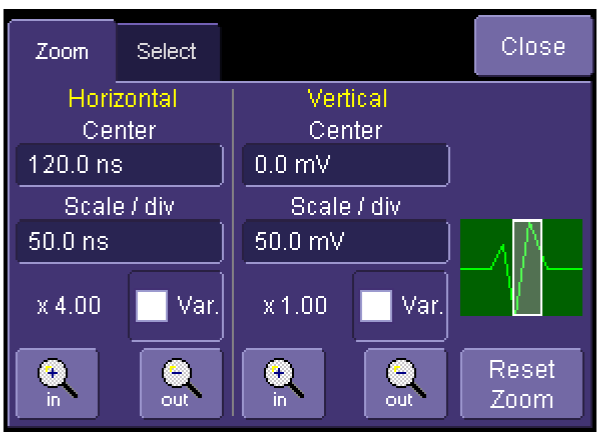
.

> **Note:** If you have more than one waveform displayed on only one grid, a second grid will open automatically when you select **Next Grid**.

Zooming Waveforms

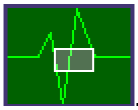
The Zoom button 

appears as a standard button at the bottom of the channel "Cx Vertical Adjust" setup dialog if you want to create a math function zoom trace of your input waveform. On the other hand, you can zoom a memory or math function non-zoom trace directly without having to create a separate zoom trace. For such traces, a zoom control mini-dialog is provided at the right of each math trace "Fx" setup dialog.

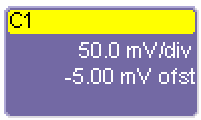


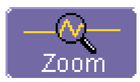
The front panel "QuickZoom" button  creates multiple zooms, one for each displayed input channel.

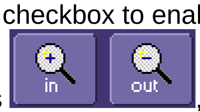
At any time, you can also zoom a portion of a waveform by touching and dragging a rectangle around any part of the input waveform. The zoom trace will size itself to fit the full width of the grid. The degree of magnification, therefore, will depend on the size of the rectangle that you create.

When you zoom a waveform, an approximation of the zoomed area will appear in a thumbnail icon in the "Zoom" dialog:
.
The "Zoom" dialog appears alongside the math setup dialog when Zoom is the math or memory function selected.

To Zoom a Single Channel

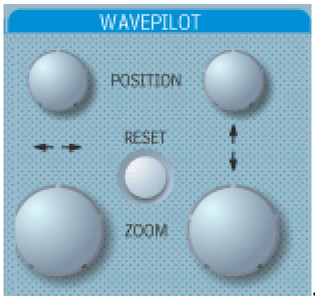
1. In the menu bar, touch **Vertical**; then touch a channel number in the drop-down menu. Alternatively, you can just touch the channel trace label for a displayed channel:


.
Touch the **Zoom** button 

at the bottom of the "Cx Vertical Adjust dialog." A zoom math trace (one of F5 to Fx) will be created of the selected channel.
2. To vary the degree of zoom, touch the newly created Fx trace label. The setup dialog for the math function opens, and the zoom control dialog appears at lower-right. It shows the current horizontal and vertical zoom factors.
3. If you want to increase or decrease your horizontal or vertical zoom in small increments, touch the **Var.** checkbox to enable variable zooming. Now with each touch of the zoom control buttons 

, the degree of magnification will change by a small increment. To zoom in or out in large standard increments with each touch of the zoom control buttons, leave the **Var.** checkbox unchecked. To set exact horizontal or vertical zoom factors, touch inside the Horizontal **Scale/div** data entry field and enter a time-per-div value, using the pop-up numeric keypad. Then touch inside the Vertical **Scale/div** field and enter a voltage value.
4. To reset the zoom to x1 magnification, touch **Reset Zoom** in the dialog or press the front panel zoom button:
.

To Zoom by Touch-and-Drag

1. Touch and drag a rectangle around any part of an input channel waveform, math trace, or memory trace. If you have more than one trace displayed, a pop-up "Rectangle Zoom Wizard" will appear.
2. If more than one trace is displayed, touch the "Source" tab and select a trace to act on.
3. Touch the "Action" tab and select **Create a New Zoom Trace**. You will be offered the choice of creating a new zoom trace or modifying the current trace.
4. Touch the **Zoom** tab and select a math function trace to display the zoom.
5. Turn the front panel Wavepilot position knobs to adjust the vertical and horizontal position of the zoom


.
6. Turn the front panel Zoom knobs to control the boundaries of the zoom.

To Zoom Multiple Waveforms Quickly

Press the QuickZoom button  on the front panel. Math function traces **F5** to **F8** will be used to create a zoom of each displayed input channel waveform. Each zoom will be displayed in its own grid.

To Turn Off Zoom

1. Touch the math function trace label for the zoom you want to turn off.
2. Touch the **Trace On** checkbox to delete the check mark and disable the zoom trace.

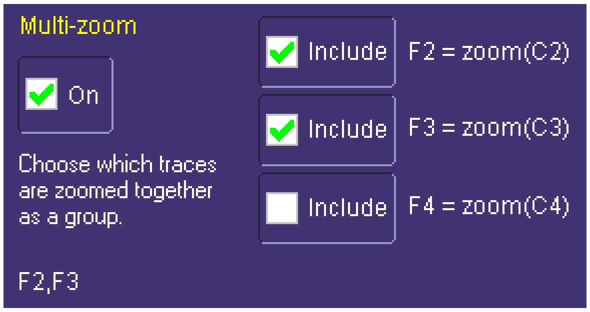
Multi-Zoom

The Multi-zoom feature creates time-locked zoom traces for only the waveforms that you choose to include. The zooms are of the same X-axis section of each waveform. Thus, as you scroll through a waveform, all included zooms scroll in unison.

To Set Up Multi-zoom

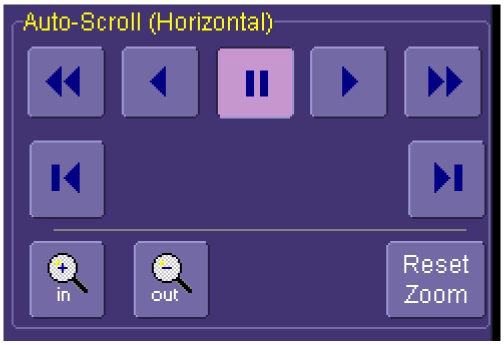
1. In the menu bar, touch **Math**, then **Math Setup...** in the drop-down menu.
2. Verify that the math function selected for each **Fx** position you want to include is **zoom**. If you need to change the math function for any Fx position, simply touch the **Fx** button and select **Zoom** from the **Select Math Operator menu**.

3. Touch the **On** checkbox to display each zoom you want to include in the multi-zoom.
4. Touch the Multi-Zoom **Setup** button. The **Multi-Zoom** dialog opens.
5. Touch the Multi-zoom **On** checkbox to enable Multi-zoom. Then touch the **Include** checkbox for each zoom trace you want to include in the time-locked multi-zoom:



Here the user has chosen to include only F2 and F3 in the Multi-zoom, even though F4 is also a zoom function and is also displayed. Thus, the scrolling feature will not affect zoom F4.

6. Use the **Auto-Scroll** buttons at the right of the Multi-Zoom dialog to control the zoomed section of your waveforms:



To Turn Off Multi-Zoom

1. In the menu bar, touch **Math**, then **Math Setup...** in the drop-down menu.
2. Touch the Multi-Zoom **On** checkbox to turn off Multi-zoom.

XY Display

Use XY displays to measure the phase shift between otherwise identical signals. You can display either voltage on both axes or frequency on both axes. The traces must have the same X-axis. The shape of the resulting pattern reveals information about phase difference and frequency ratio.

To Set Up XY Displays

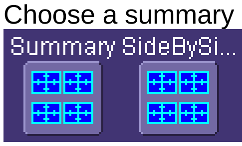
1. In the menu bar, touch **Display**; then touch **Display Setup...** in the drop-down menu.
2. Choose an XY display by touching one of the XY display mode buttons 

.
   You have the choice of showing the two waveforms on just the XY grid, or you can also show the input waveforms on a single or dual grid.
3. Touch inside the **Input X** and **Input Y** data entry fields and select your input sources from the pop-up menus. 

   The inputs can be any combination of channels, math functions, and memory locations.

Summary Displays

The summary displays offer an alternative way to view multiple traces in side-by-side windows. This type of display gives more vertical detail at the cost of some horizontal resolution in the display.

1. In the menu bar, touch **Display**; then touch **Display Setup …** in the drop-down menu.
2. Choose a summary display by touching one of the summary mode buttons 

.
   You have the choice of a four-quadrant display or a dual, side-by-side display.

Saving and Recalling Scope Settings

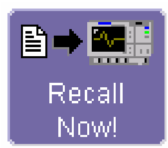
You can save or recall scope settings to or from hard disk, floppy disk, or LAN location.

To Save Scope Settings

1. In the menu bar, touch **File**; then touch **Save Setup...** in the drop-down menu. Or, press the Save/Recall front panel button, then touch the "Save Setup" tab.
2. To **Save To File**, touch inside the **Save Instrument Settings** data entry field and use the pop-up keyboard to enter the path to the destination folder. Or touch **Browse** to navigate to the destination folder. Then touch **Save Now!** 

below the data entry field. To save to folder **Internal Setups** on the scope's hard drive, touch inside a **SetupX** data entry field and use the pop-up keyboard to enter a file name. Touch **Save** alongside the data entry field. The file is deposited in **D:\Internal Setups**, and the current date is displayed above the field.

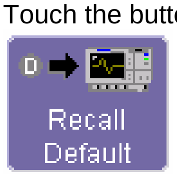
To Recall Scope Settings

1. In the menu bar, touch **File**; then touch **Recall Setup...** in the drop-down menu.
2. To **Recall From File**, touch inside the **Recall panels from file** data entry field and use the pop-up keyboard to enter the path to the source folder. Or touch **Browse** to navigate to the source folder. Then touch **Recall Now!** 

. To recall settings from folder **D:\ Internal Setups** on the scope's hard drive, touch **Recall** alongside the file you want to recall.

To Recall Default Settings

1. In the menu bar, touch **File**; then touch **Recall Setup...** in the drop-down menu.
2. Touch the button under **Recall Default Setup:**



.

The default settings are as follows:

Vertical	Timebase	Trigger
100 mV/div	1.0 ns/div	Prescale
0 V offset	Sequential mode, 10 kS record length	
	-12 ns delay	Auto trigger mode

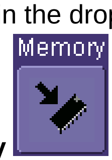
Saving Screen Images

You can send images to a hard copy printer or to storage media. Both types of output are done from the same dialog.

1. In the menu bar, touch **Utilities**, then **Utilities Setup...** in the drop-down menu.
2. Touch the **Hardcopy** tab.
3. Touch the **File** button.
4. Touch inside the **File Format** field and select a file type.
5. Under **Colors**, touch the **Use Print Colors** checkbox if you want your waveforms to print in color with a white background. A white background saves printer toner.
6. Touch inside the **Directory** field and type in the path to the directory where you want the image stored, using the pop-up keyboard. Or you can touch the browse button and navigate there.
7. Touch inside the **File Name** field and type in a name for your image, using the pop-up keyboard.
8. Under **Include On Print**, touch the **Grid Area Only** checkbox if you do not want to include the dialog area in the image.
9. Touch the **Print Now** button.

Saving and Recalling Waveforms

Saving Waveforms

1. In the menu bar, touch **File**; then touch **Save Waveform...** in the drop-down menu.
2. In the "Save Waveform" dialog, touch the **Save To Memory** 

or **File** 

button.
3. Touch inside the **Source** field and select a source from the pop-up menu. The source can be any trace; for example, a channel (C1C4), math function (F1F4), or a waveform stored in non-volatile RAM (M1M4).
4. Touch inside the **Trace Title** data entry field if you want to change the default name of your waveforms. Use the pop-up keyboard to type in the new name.

Note: You can change the name but not the sequence number.

CAUTION

If you use a name that ends in a number instead of a letter, the instrument may truncate the number. This is because, by design, the first waveform is automatically numbered 0, the second 1, etc. For example, if you want to use waveform name "XYZ32" but it is not preceded by waveforms XYZ0 through XYZ31, the waveform will be renumbered with the next available number in the sequence.

If you need to use a number in your waveform's name, it is recommended that you append an alpha character at the end of the number : "XYZ32a" for example.

5. If you are saving to file, touch the **Data Format** field and select a format type from the pop-up menu:


.
If you select **ASCII** or **Excel**, also touch the **SubFormat** field and select either **Time Data** or **Time & Ampl**. Then touch the **Delimiter** field and select a delimiter character from the pop-up menu: comma, space, semicolon, or tab.
6. Touch the **Browse** button for the **Save file in directory** field and browse to the location where you want the file saved. The file name is assigned automatically and is shown below the field.
7. Touch 

.

Auto Save

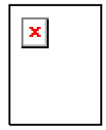
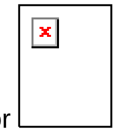
You can also enable Auto Save from this dialog by touching one of the Auto Save buttons:


.
Wrap (old files overwritten) or **Fill** (no files overwritten).

CAUTION

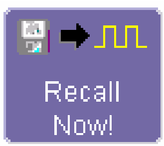
If you select Fill, you can quickly use up all disk space on your hard disk.

Recalling Waveforms

1. In the menu bar, touch **File**; then touch **Recall Waveform...** in the drop-down menu.
2. In the "Recall Waveform" dialog, touch the **Recall From**  or  button.
   A. If you selected **Memory**, touch inside the **Source** field and select a memory location: **M1** to **M4**.
   B. If you selected **File**, touch inside the **Destination** field and select a memory location in which to store the file.

      Touch inside the **Show only files** field and select an area to limit the search to: channels, math functions, or memory.

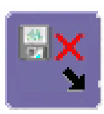
      Touch inside the **Recall files from directory** data entry field and enter the path, using the pop-up keyboard. Or touch the **Browse** button to navigate to the file.

      Touch inside the **Next file will be recalled from** data entry field and enter the path, using the pop-up keyboard. Or touch the **Browse** button to navigate to the file.
3. Touch  .

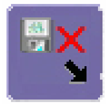
Disk Utilities

Use the Disk Utilities dialog to delete files or create folders.

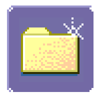
To Delete a Single File

1. Touch **File** in the menu bar, then **Disk Utilities...** in the drop-down menu.
2. Touch the **Delete** button  in the "Disk Utilities" dialog.
3. Touch inside the **Current folder** data entry field and use the pop-up keyboard to enter the path to the folder that contains the file you want to delete. Or touch the **Browse** button and navigate to the folder.
4. Touch inside the **File to be deleted** data entry field and use the pop-up keyboard to enter the name of the file. Or touch the **Browse** button and navigate to the file.
5. Once you have located the file, touch the **Delete File** button.

To Delete All Files in a Folder

1. Touch **File** in the menu bar, then **Disk Utilities...** in the drop-down menu.
2. Touch the **Delete** button  in the "Disk Utilities" dialog.
3. Touch inside the **Current folder** data entry field and use the pop-up keyboard to enter the path to the folder that contains the file you want to delete. Or touch the **Browse** button and navigate to the folder.
4. Once you have located the folder, touch the **Empty Folder** button.

To Create a Folder

1. Touch **File** in the menu bar, then **Disk Utilities...** in the drop-down menu.
2. Touch the **Create** button  in the "Disk Utilities" dialog.
3. Touch inside the **Current folder** data entry field and use the pop-up keyboard to enter the path to the directory you want to create the folder in, and the name of the folder.
4. Touch the **Create Folder** button.

PRINTING AND FILE MANAGEMENT

Print, Plot, or Copy

The instrument gives you the ability to output files to a printer or plotter, to print to file, or to e-mail your files. Any WindowsXP supported printer is supported by your instrument.

Printing

To Set Up the Printer

1. In the menu bar, touch **File**, then **Print Setup...** in the drop-down menu. The Utilities **Hardcopy** dialog opens.
2. In the dialog area, touch the **Printer** icon .
3. Under **Colors**, touch the **Use Print Colors** checkbox if you want the traces printed on a white background. A white background saves printer toner. (You can change the printer colors in the Preference dialog;)
4. Touch inside the **Select Printer** field. From the touch pad pop-up choose the printer you want to print to. Touch the **Properties** button to see your printer setup.
5. Touch the icon for the layout **Orientation** you want: portrait or landscape.
6. Touch the **Grid Area Only** checkbox if you do not need to print the dialog area and you only want to show the waveforms and grids.

To Print

You can print in one of three ways:

- Press the printer button on the front panel:
- In the menu bar, touch **File**, then **Print** in the drop-down menu.
- Touch the **Print Now** button in the "Hardcopy" dialog

Adding Printers and Drivers

Note: If you want to add a printer driver, the driver must first be loaded on the scope.

1. In the menu bar, touch **File**, then **Print Setup...** in the drop-down menu. The Utilities **Hardcopy** dialog opens.
2. In the dialog area, touch the **Printer** icon .
3. Touch the **Add Printer** button. An MS Windows® window with which to add a printer will open.
4. Touch the Properties button to change printer properties such as number of copies.

Changing the Default Printer

1. If you want to change the default printer, minimize the instrument application by touching **File** in the menu bar, then **Minimize** in the drop-down menu.

2. Touch the **Start** button in the task bar at the bottom of the screen.
3. Select **Settings**, then **Printers**.
4. Touch the printer you want to set as the default printer, then touch **File, Set as Default Printer**.

Managing Files

Use the instrument's utilities to create waveform files on floppy disk, internal hard drive or network drives. You can copy files from your hard drive to floppy disk. You also can give your files custom names and create directories for them.

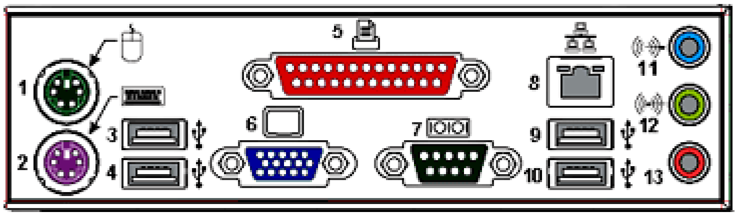
Hard Disk Partitions

The instrument's hard disk is partitioned into drive **C:** and drive **D:**. Drive C: contains the Windows operating system and the instrument application software. Drive D: is intended for data files.

100BASE-T ETHERNET CONNECTION

Connecting to a Network

Use the Ethernet connector (item **8** in the rear panel diagram) to connect the instrument to a network.



Communicating over the Network

In its default configuration the instrument is set up to use the DHCP protocol to retrieve its IP address from the network. In cases where a DHCP server is not available on your network, a static IP address can be configured in the Windows Network Settings dialog.

Windows Setups

Instruments that are required to participate in a Windows Network Domain will need to be "joined" to the domain by a network administrator (the procedure typically requires an administrator username and password).

Domain membership is not required to use the instrument on a network, but will generally make it easier to access network shared drive and printer resources.

Guidelines for Working in Windows

Although the instrument has an open architecture, avoid modifying the Windows operating system, since this may cause problems for the instrument's user interface. Please follow these recommendations:

- Do not load any version of Windows not provided by LeCroy. Windows service packs and critical updates are generally safe, and LeCroy does encourage you to install them to keep your scope safe from network-borne viruses and worms. However, LeCroy cannot guarantee that any update distributed by Microsoft will not adversely affect the operation of your instrument. Any compatibility issues detected by LeCroy will be posted on our DSO Security Web site at http://www.lecroy.com/dsosecurity. It is advisable to check this site before applying updates.
- If the instrument powers up in Windows Safe Mode, the touch screen will not function. You may need a mouse or keyboard to restore normal operation.
- Avoid modifying Control Panel settings.
- Do not change the color resolution (24 bit) or screen size (800 x 600 pixel) settings.
- After you load third-party software applications, if your scope does not work properly try

reloading the instrument software from the CD shipped with the scope. If your instrument is not equipped with a CD drive, you will need a USB CD-ROM to do this (not supplied by LeCroy). This does not apply to WaveSurfer or WaveRunner 6000A models, which can be recovered from their internal hard drives.

- Do not modify or remove any system fonts; doing so may affect the readability of the dialogs.
- Do not change any display properties like Background, Appearance, Effects, or Settings. Functionality of the scope or screen saver may be affected.
- Do not make any changes to the Windows folder.
- Do not make any changes to the BIOS settings.
- Do not make any changes to the Windows power management system.

System Restore

Although the scope creates regularly scheduled restore points automatically, before you install any hardware or software on your instrument LeCroy strongly recommends that you manually create a restore point. The restore point resides on the scope's hard drive, so no external storage medium (floppy disk, USB memory stick, etc.) is required.

To Create a Restore Point

From the **File** menu, minimize or Window the scope display to reveal the task bar.

In the task bar, select **Start**, **Programs**, **Accessories**, **System Tools**, **System Restore**.

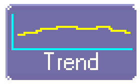
Touch the **Create a restore point** radio button, then touch **Next**.

In the **Restore point description** box, indicate what software or hardware is going to be added after the restore point is created, then touch **Next**.

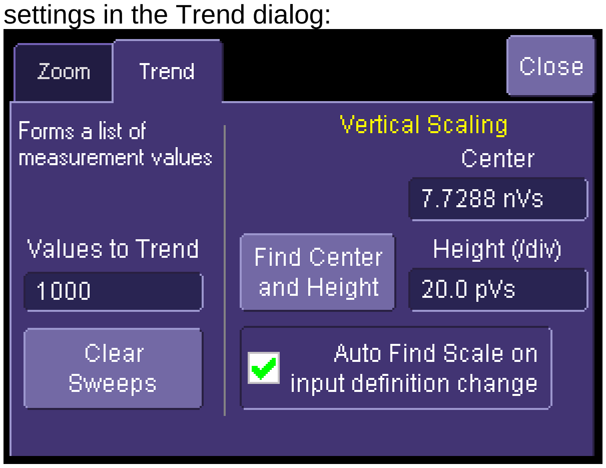
The restore point will be created and a confirmation message will be displayed.

TRACK VIEWS

Creating and Viewing a Trend

1. In the menu bar, touch **Measure**, then **Measure Setup** in the drop-down menu.
2. Touch one of parameter tabs **P1** through **Px**.
3. Touch inside the **Source1** data entry field and select an input waveform from the pop-up menu.
4. Touch inside the **Measure** data entry field and select a parameter from the pop-up menu.
5. Touch the **Trend** button 

at the bottom of the dialog; then, from the **Math selection for Trend** menu, select a math function location (F1 to Fx) to store the Trend display. The Trend will be displayed along with the trace label for the math function you selected.


6. Touch the newly displayed Trend math function trace label if you want to change any settings in the Trend dialog:


Creating a Track View

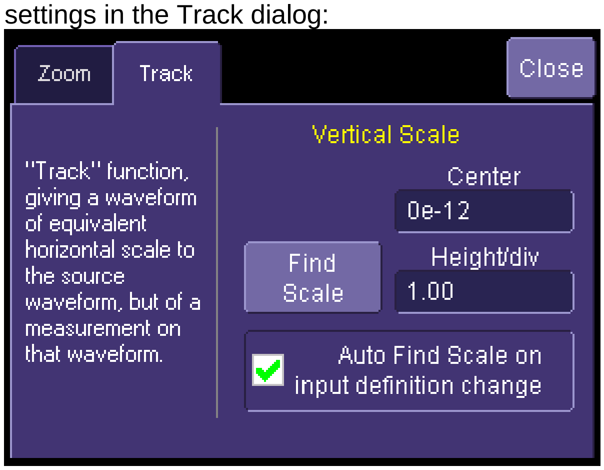
This feature is available in the XMAP option.

1. In the menu bar, touch **Measure**, then **Measure Setup** in the drop-down menu.
2. Touch one of parameter tabs **P1** through **Px**.
3. Touch inside the **Source1** data entry field and select an input waveform from the pop-up menu.
4. Touch inside the **Measure** data entry field and select a parameter from the pop-up menu.

5. Touch the **Track** button 

at the bottom of the dialog; then, from the **Math selection for Track** menu, select a math function location (F1 to Fx) to store the Track display. The Track will be displayed along with the trace label for the math function you selected.

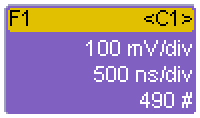
6. Touch the newly displayed Track math function trace label if you want to change any settings in the Track dialog:



HISTOGRAMS

Creating and Viewing a Histogram

Note: The number of sweeps comprising the histogram will be displayed in the bottom line of the trace descriptor label:

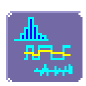


To Set Up a Single Parameter Histogram

From Measure Dialog

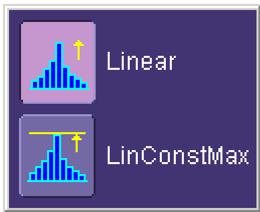
1. In the menu bar, touch **Measure**, then **Measure Setup**.
2. Touch the **My Measure** button.
3. Touch one of tabs **P1** through **Px**.
4. Touch inside the **Source1** field and select an input waveform from the pop-up menu.
5. Touch inside the **Measure** field and select a parameter from the pop-up menu.
6. Touch the **Histogram** button at the bottom of the dialog.
7. Touch a math trace in which to place the resulting histogram, then close the pop-up menu.
8. Touch the math trace label for the math trace you just created.
9. In the dialog to the right, touch the Histogram tab.
10. Under "Buffer," touch inside the #Values data entry field and enter a value.
11. Under "Scaling," touch inside the #Bins data entry field and enter a value from 20 to 2000.
12. Touch the Find Center and Width button to center the histogram. Or touch inside the Center, then the Width, data entry fields and enter a value using the pop-up numeric keypad.

From Math Dialog

1. In the menu bar, touch **Math**, then **Math Setup**.
2. Touch one of function tabs **F1** through **Fx**. (The number of math traces available depends on the software options loaded on your scope. See specifications.)
3. Touch the **Graph** button .
4. Touch inside the **Source1** field and select a source from the pop-up menu.
5. Touch inside the **Measurement** field and select a parameter from the pop-up menu.
6. Touch inside the **Graph with** field and select **Histogram** from the pop-up menu.
7. In the dialog to the right, touch the **Histogram** tab.
8. Under "Buffer," touch inside the **#Values** data entry field and enter a value from 20 to 1000.
9. Under "Scaling," touch inside the **#Bins** data entry field and enter a value from 20 to 2000.
10. Touch the **Find Center and Width** button to center the histogram. Or touch inside the

Center, then the **Width**, data entry fields and enter a value using the pop-up numeric keypad.

11. Touch inside the Vertical Scale field and select **Linear** or **Linear Constant Max** from the pop-up menu



.

To View Thumbnail Histograms

Histicons are miniature histograms of parameter measurements that appear below the grid. These thumbnail histograms let you see at a glance the statistical distribution of each parameter.

1. In the menu bar, touch **Measure**, then one of the Measure Mode buttons: **Std Vertical**, **Std Horizontal**, or **My Measure**.
2. Touch the **Histicons** checkbox to display thumbnail histograms below the selected parameters.

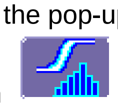
Note: For measurements set up in My Measure, you can quickly display an enlarged histogram of a thumbnail histogram by touching the Histicon you want to enlarge. The enlarged histogram will appear superimposed on the trace it describes. This does not apply to "Std Vertical" or "Std Horizontal" measurements.

Persistence Histogram

You can create a histogram of a persistence display also by cutting a horizontal or vertical slice through the waveform. You also decide the width of the slice and its horizontal or vertical placement on the waveform.

This math operation is different than the "Histogram" math operation and is not affected by **Center** and **Width** settings made there.

To Set Up Persistence Histograms

1. In the menu bar, touch **Math**, then **Math Setup**.
2. Touch one of function tabs **F1** through **Fx**. (The number of math traces available depends on the software options loaded on your scope. See specifications.)
3. Touch inside the **Source1** field and select a source from the pop-up menu.
4. Touch inside the **Operator1** field and select **Phistogram**  from the **Select Math Operator** menu.
5. Touch the "Phistogram" tab, then touch inside the Slice Direction field and select **Horizontal** or **Vertical** slice from the pop-up menu.
6. Touch inside the **Slice Center** field and enter a value, using the pop-up keypad.

7. Touch inside the **Slice Width** field and enter a value, using the pop-up keypad.

Note: You can use the front panel Adjust knobs to move the **Slice Center** line and the **Slice Width** boundary lines.

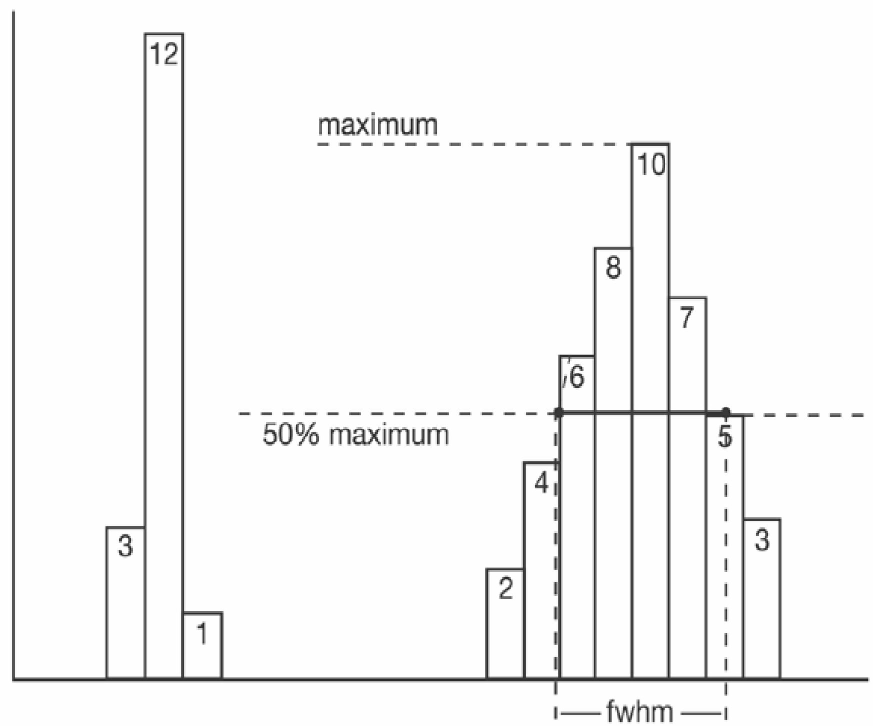
Persistence Trace Range

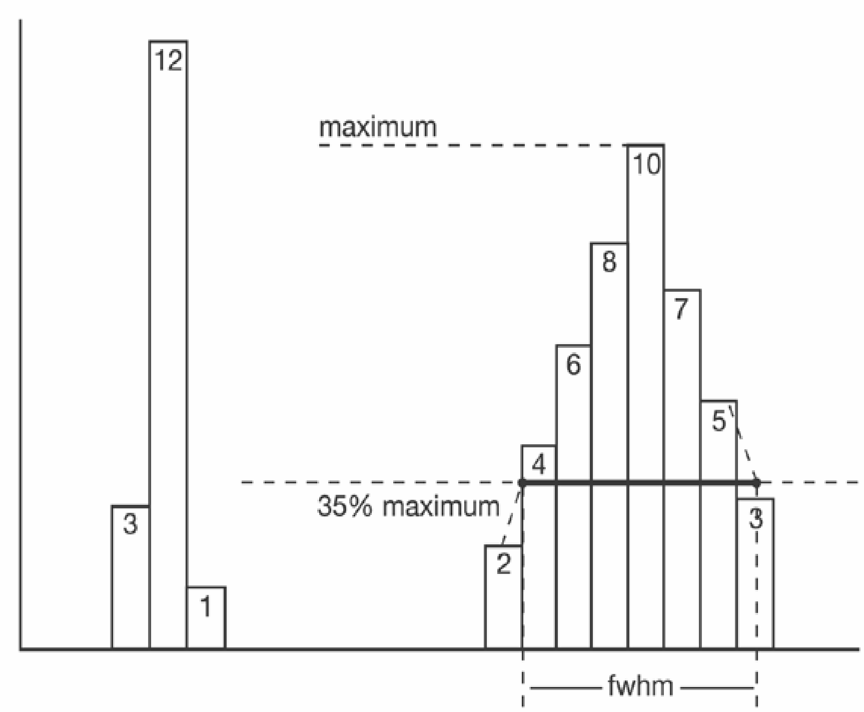
This math operation has a field where you can enter the percent of the persistence trace population to use in creating a new waveform.

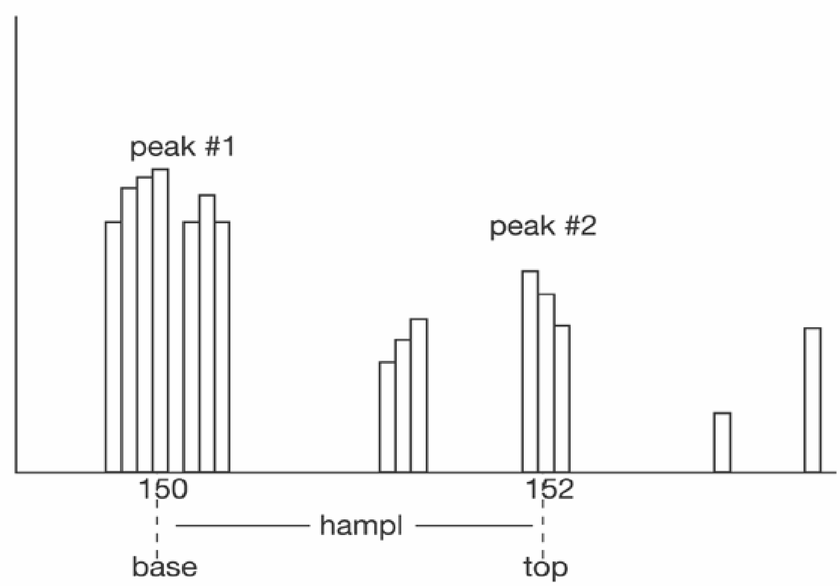
Persistence Sigma

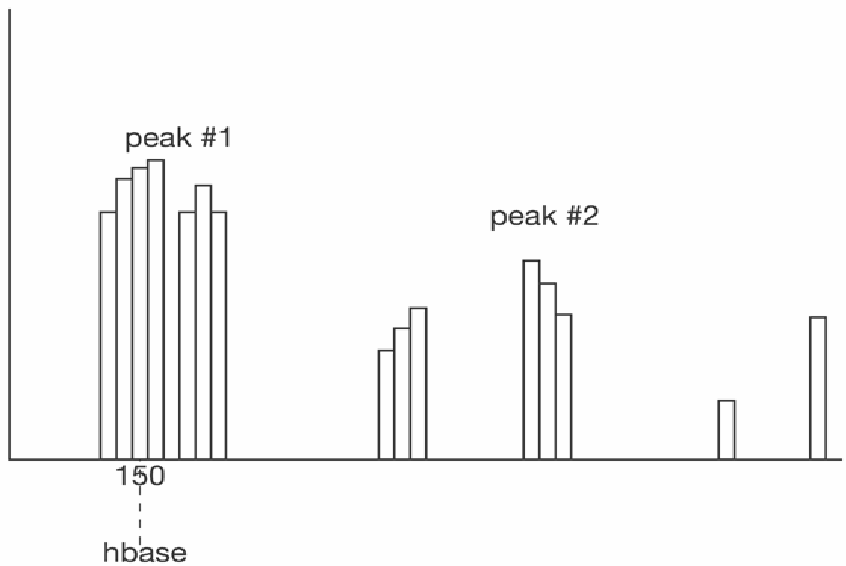
This math operation has a field where you can enter a scale, measured in standard deviations, by which to create a new waveform.

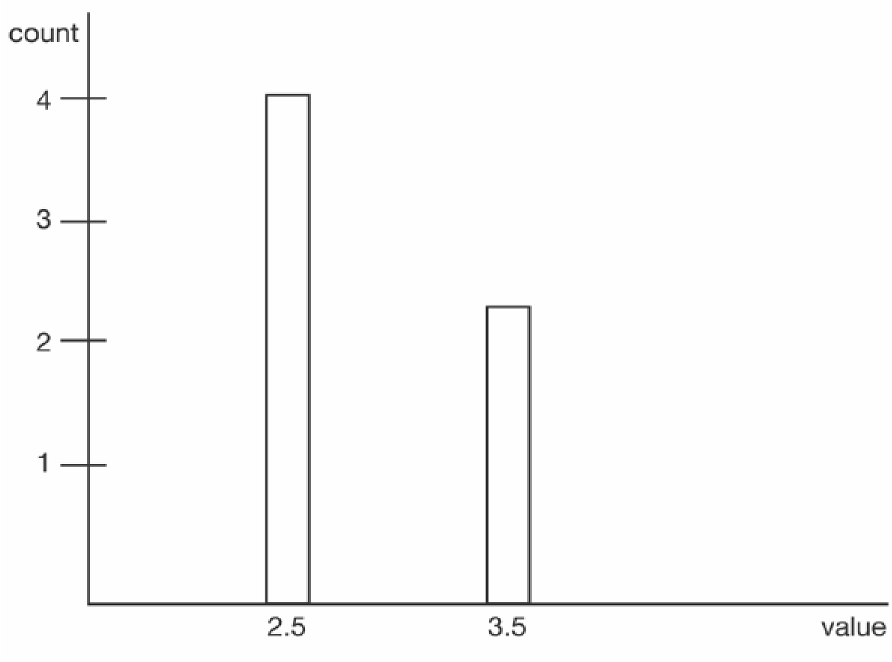
Histogram Parameters

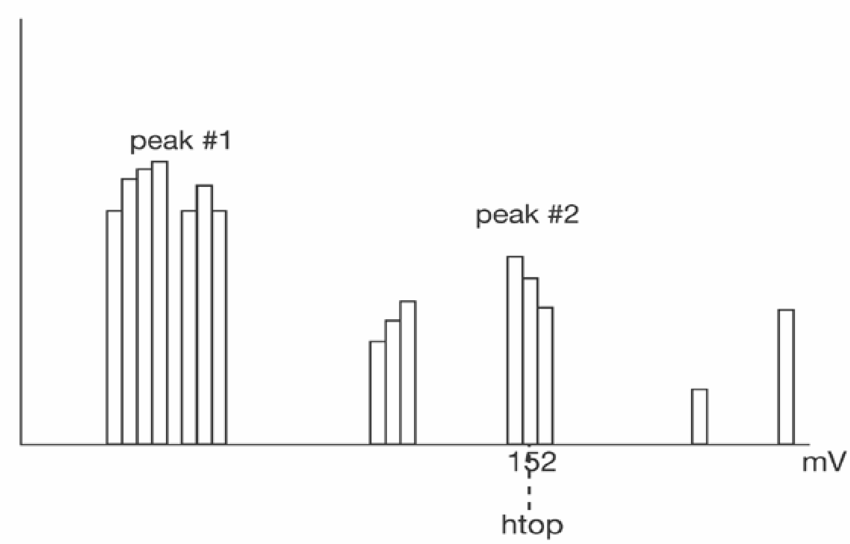
fwhm	Full Width at Half Maximum
Definition:	Determines the width of the largest area peak, measured between bins on either side of the highest bin in the peak that have a population of half the highest's population. If several peaks have an area equal to the maximum population, the leftmost peak is used in the computation.
Description:	First, the highest population peak is identified and the height of its highest bin (population) determined (for a discussion on how peaks are determined see the pks parameter Description:). Next, the populations of bins to the right and left are found, until a bin on each side is found to have a population of less than 50% of that of the highest bin's. A line is calculated on each side, from the center point of the first bin below the 50% population to that of the adjacent bin, towards the highest bin. The intersection points of these lines with the 50% height value is then determined. The length of a line connecting the intersection points is the value for fwhm.
Example:	

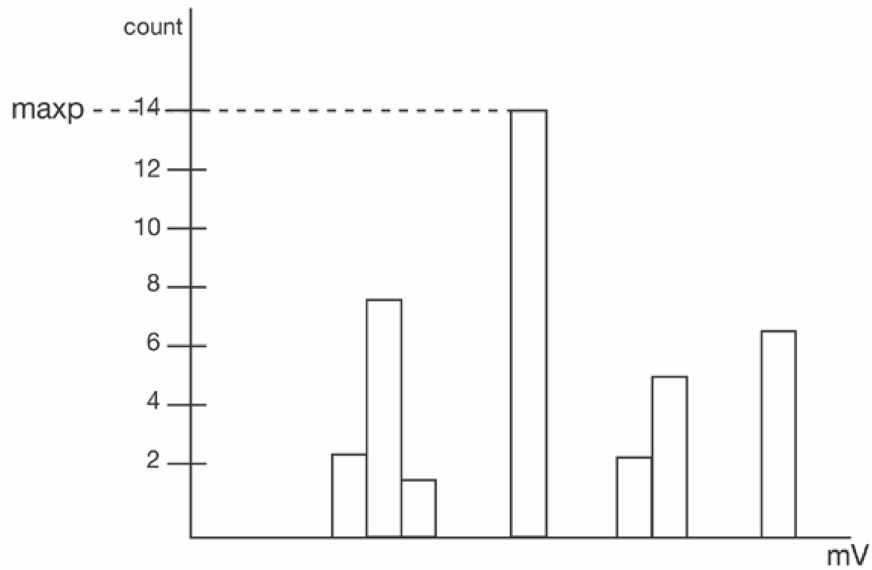
fwxx	**Full Width at xx% Maximum**
Definition:	Determines the width of the largest area peak, measured between bins on either side of the highest bin in the peak that have a population of xx% of the highest's population. If several peaks have an area equal to the maximum population, the leftmost peak is used in the computation.
Description:	First, the highest population peak is identified and the height of its highest bin (population) determined (see the pks description). Next, the bin populations to the right and left are found until a bin on each side is found to have a population of less than xx% of that of the highest bin. A line is calculated on each side, from the center point of the first bin below the 50% population to that of the adjacent bin, towards the highest bin. The intersection points of these lines with the xx% height value is then determined. The length of a line connecting the intersection points is the value for fwxx.
Example:	fwxx with threshold set to 35%:
	

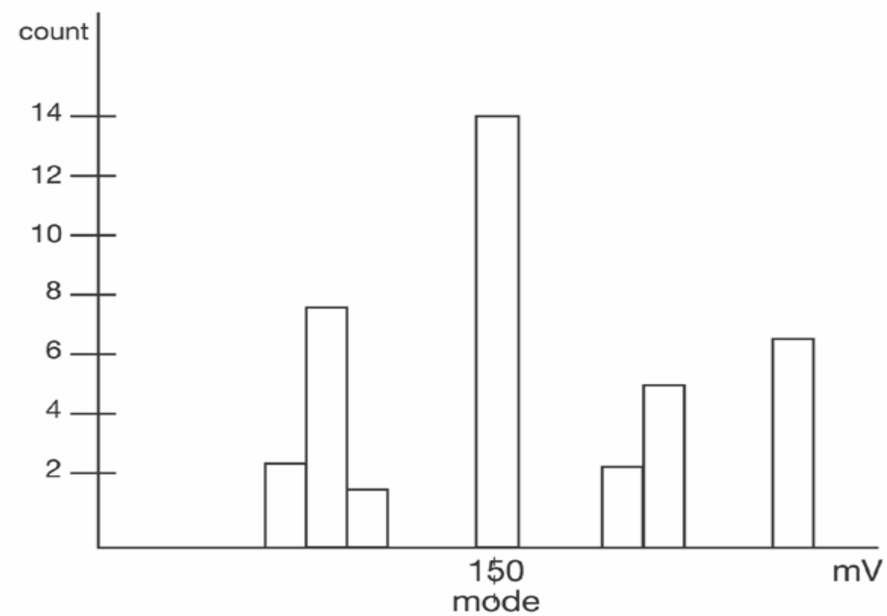
hist ampl	Histogram Amplitude
Definition:	The difference in value of the two most populated peaks in a histogram. This parameter is useful for waveforms with two primary parameter values, such as TTL voltages, where hampl would indicate the difference between the binary `1' and `0' voltage values.
Description:	The values at the center (line dividing the population of peak in half) of the two highest peaks are determined (see pks parameter description:). The value of the leftmost of the two peaks is the histogram base (see hbase). While that of the rightmost is the histogram top (see htop). The parameter is then calculated as:
	hampl = htop - hbase
Example:	
	In this histogram, hampl is 152 mV - 150 mV = 2 mV.

hbase	Histogram Base
Definition:	The value of the leftmost of the two most populated peaks in a histogram. This parameter is primarily useful for waveforms with two primary parameter values such as TTL voltages where hbase would indicate the binary `0' voltage value.
Description:	The two highest histogram peaks are determined. If several peaks are of equal height the leftmost peak among these is used (see pks). Then the leftmost of the two identified peaks is selected. This peak's center value (the line that divides the population of the peak in half) is the hbase.
Example:	

hist rms	**Histogram Root Mean Square**
Definition:	The rms value of the values in a histogram.
Description:	The center value of each populated bin is squared and multiplied by the population (height) of the bin. All results are summed and the total is divided by the population of all the bins. The square root of the result is returned as hrms.
Example:	Using the histogram shown here, the value for hrms is:
	hrms $= \sqrt{(3.5^2 * 2 + 2.5^2 * 4)/6} = 2.87$
	

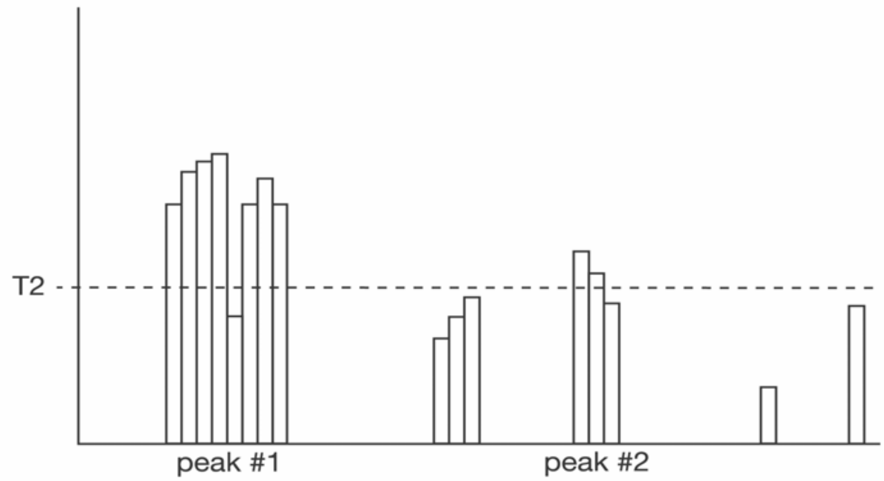
hist top	**Histogram Top**
Definition:	The value of the rightmost of the two most populated peaks in a histogram. This parameter is useful for waveforms with two primary parameter values, such as TTL voltages, where htop would indicate the binary `1' voltage value.
Description:	The two highest histogram peaks are determined. The rightmost of the two identified peaks is then selected. The center of that peak is htop (center is the horizontal point where the population to the left is equal to the area to the right).
Example:	

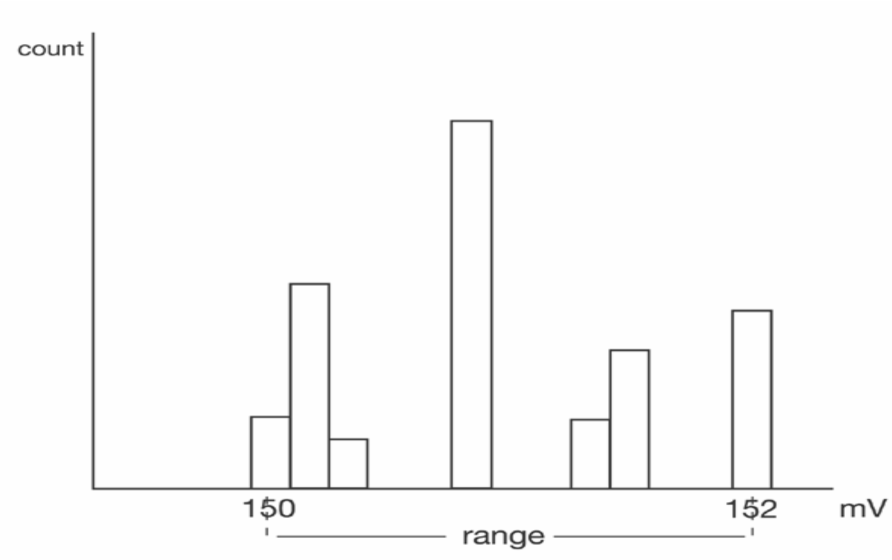
maxp	Maximum Population
Definition:	The count (vertical value) of the highest population bin in a histogram.
Description:	Each bin between the parameter cursors is examined for its count. The highest count is returned as maxp.
Example:	
	Here, maxp is 14.

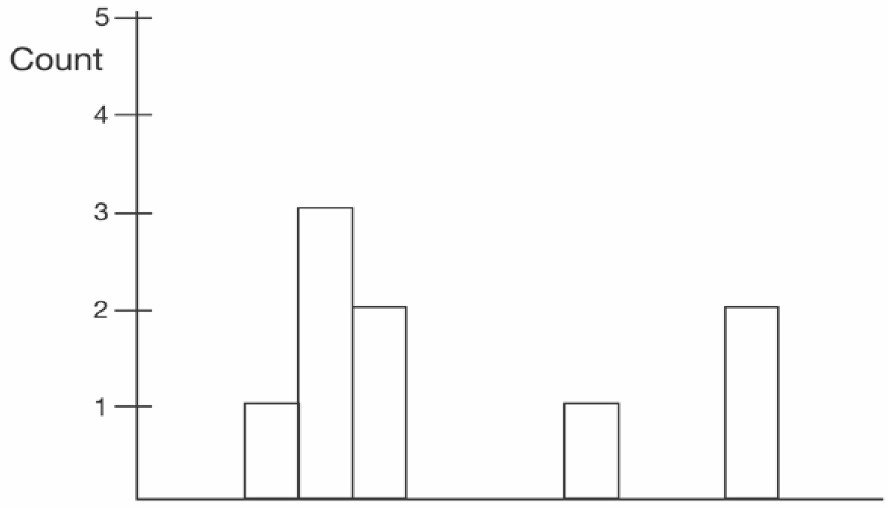
mode	**Mode**
Definition:	The value of the highest population bin in a histogram.
Description:	Each bin between the parameter cursors is examined for its population count. The leftmost bin with the highest count found is selected. Its center value is returned as mode.
Example:	
	Here, mode is 150 mV.

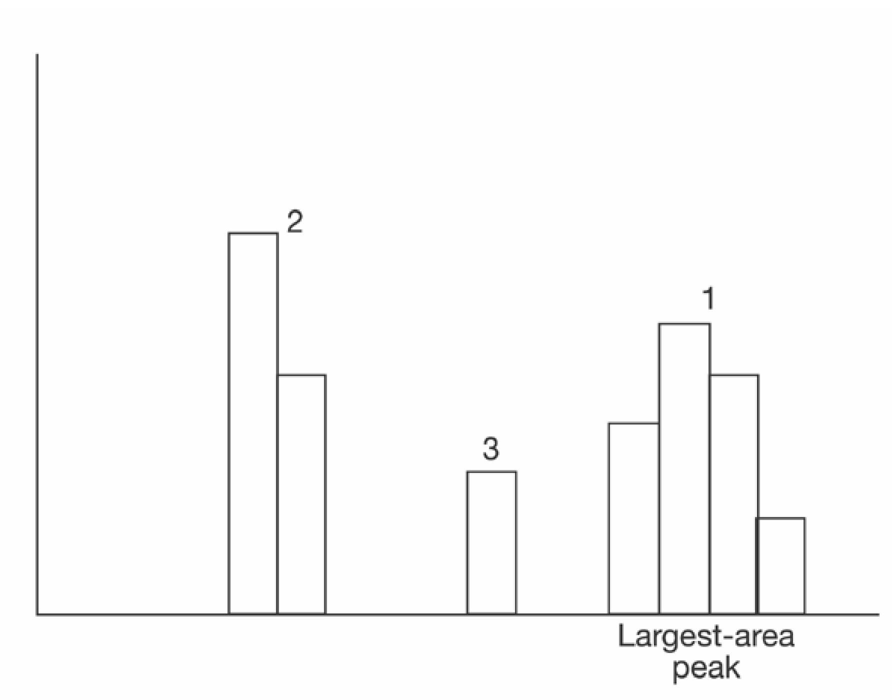
pctl	Percentile
Definition:	Computes the horizontal data value that separates the data in a histogram such that the population on the left is a specified percentage `xx' of the total population. When the threshold is set to 50%, pctl is the same as hmedian.
Description:	The total population of the histogram is determined. Scanning from left to right, the population of each bin is summed until a bin that causes the sum to equal or exceed `xx'% of the population value is encountered. A ratio of the number of counts needed for `xx'% population/total bin population is then determined for the bin. The horizontal value of the bin at that ratio point of its range is found, and returned as pctl.
Example:	The total population of a histogram is 100. The histogram range is divided into 20 bins and `xx' is set to 25%. The population sum at the sixth bin from the left is 22. The population of the seventh is 9 and its sub-range is 6.1 to 6.4 V. The ratio of counts needed for 25% population to total bin population is:
	3 counts needed / 9 counts = 1/3.
	The value for pctl is:
	6.1 volts + .33 * (6.4 - 6.1) volts = 6.2 volts.

pks	Peaks
Definition:	The number of peaks in a histogram.
Description:	The instrument analyzes histogram data to identify peaks from background noise and histogram binning artifacts such as small gaps.
	Peak identification is a 3-step process:
	1. The mean height of the histogram is calculated for all populated bins. A threshold (T1) is calculated from this mean, where: T1= mean + 2 sqrt (mean). 2. A second threshold is determined based on all populated bins under T1 in height, where: T2 = mean + 2 * sigma, and where sigma is the standard deviation of all populated bins under T1. 3. Once T2 is defined, the histogram distribution is scanned from left to right. Any bin that crosses above T2 signifies the existence of a peak. Scanning continues to the right until one bin or more crosses below T2. However, if the bins cross below T2 for less than a hundredth of the histogram range, they are ignored, and scanning continues in search of peaks that cross under T2 for more than a hundredth of the histogram range. Scanning goes on over the remainder of the range to identify additional peaks. Additional peaks within a fiftieth of the range of the populated part of a bin from a previous peak are ignored.
	NOTE: If the number of bins is set too high, a histogram may have many small gaps. This increases sigma and, thereby, T2. In extreme cases, it can prevent determination of a peak, even if one appears to be present to the eye.
Example:	Here the two peaks have been identified. The peak with the highest population is peak #1.

	T2 peak #1 peak #2

range	Range
Definition:	Computes the difference between the value of the rightmost and that of the leftmost populated bin.
Description:	The rightmost and leftmost populated bins are identified. The difference in value between the two is returned as the range.
Example:	
	In this example, range is 2 mV.

totp	Total Population
Definition:	Calculates the total population of a histogram between the parameter cursors.
Description:	The count for all populated bins between the parameter cursors is summed.
Example:	
	The total population of this histogram is 9.

xapk	X Coordinate of xx^{th} Peak
Definition:	Returns the value of the xx^{th} peak that is the largest by area in a histogram.
Description:	First the peaks in a histogram are determined and ranked in order of total area (for a discussion on how peaks are identified see the description for the pks parameter). The center of the n^{th} ranked peak (the point where the area to the left is equal to the area to the right), where n is selected by you, is then returned as xapk.
Example:	The rightmost peak is the largest, and is thus ranked first in area (1). The leftmost peak, although higher, is ranked second in area (2). The lowest peak is also the smallest in area (3).
	

Histogram Theory of Operation

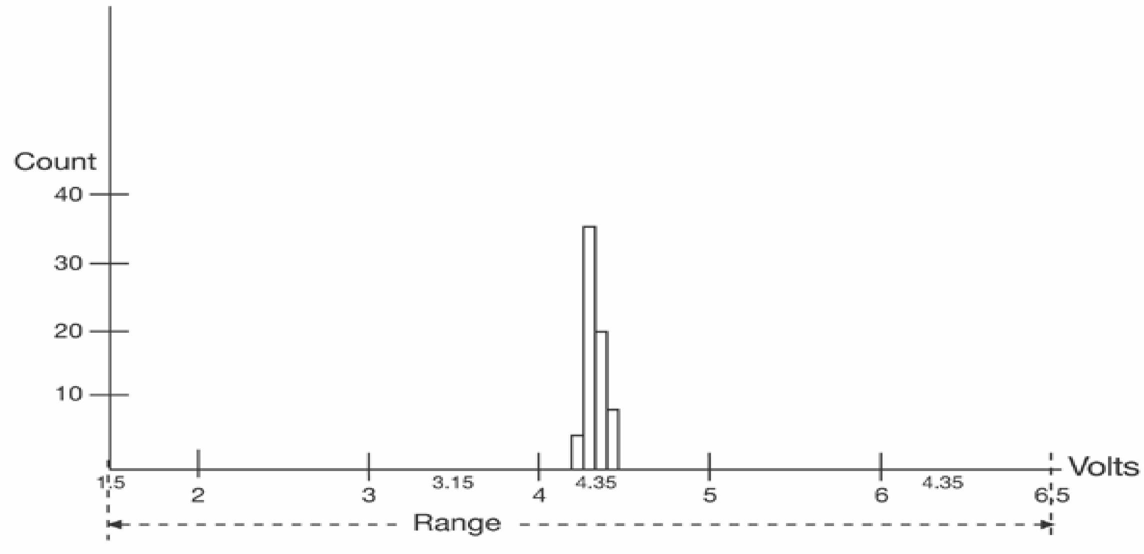
An understanding of statistical variations in parameter values is needed for many waveform parameter measurements. Knowledge of the average, minimum, maximum, and standard deviation of the parameter may often be enough, but in many cases you may need a more detailed understanding of the distribution of a parameter's values.

Histograms allow you to see how a parameter's values are distributed over many measurements. They do this by dividing a range of parameter values into sub-ranges called bins. A count of the number of parameter values (events) that fall within ranges of the bin itself is maintained for each bin.

While such a value range can be infinite, for practical purposes it need only be defined as large enough to include any realistically possible parameter value. For example, in measuring TTL high-voltage values a range of ±50 V is unnecessarily large, whereas one of 4 V ±2.5 V is more reasonable. It is the 5 V range that is then subdivided into bins. And if the number of bins used were 50, each would have a range of 5 V/50 bins or 0.1 V/bin. Events falling into the first bin would then be between 1.5 V and 1.6 V. While the next bin would capture all events between 1.6 V and 1.7 V, and so on.

After a process of several thousand events, the bar graph of the count for each bin (its histogram) provides a good understanding of the distribution of values. Histograms generally use the 'x' axis to show a bin's sub-range value, and the 'Y' axis for the count of parameter values within each bin. The leftmost bin with a non-zero count shows the lowest parameter value measurements. The vertically highest bin shows the greatest number of events falling within its sub-range.

The number of events in a bin, peak or a histogram is referred to as its population. The following figure shows a histogram's highest population bin as the one with a sub-range of 4.3 to 4.4 V (which is to be expected of a TTL signal).



The lowest-value bin with events is that with a sub-range of 3.0 to 3.1 V. As TTL high voltages need to be greater than 2.5 V, the lowest bin is within the allowable tolerance. However, because of its proximity to this tolerance and the degree of the bin's separation from all other values, additional investigation may be required.

DSO Process

The instrument generates histograms of the parameter values of input waveforms. But first, you must define the following:

- The parameter to be histogrammed
- The trace on which the histogram is to be displayed
- The maximum number of parameter measurement values to be used in creating the histogram
- The measurement range of the histogram
- The number of bins to be used

Some of these are pre-defined but can be changed. Once they are defined, the oscilloscope is ready to make the histogram. The sequence for acquiring histogram data is as follows:

1. Trigger
2. Waveform acquisition
3. Parameter calculations
4. Histogram update
5. Trigger re-arm

If you set the timebase for non-segmented mode, a single acquisition occurs prior to parameter calculations. However, in Sequence mode an acquisition for each segment occurs prior to parameter calculations. If the source of histogram data is a memory, saving new data to memory effectively acts as a trigger and acquisition. Because updating the screen can take much processing time, it occurs only once a second, minimizing trigger dead time. Under remote control the display can be turned off to maximize measurement speed.

Parameter Buffer

The oscilloscope maintains a circular parameter buffer of the last 20,000 measurements made, including values that fall outside the set histogram range. If the maximum number of events to be used for the histogram is a number `N' less than 20,000, the histogram will be continuously updated with the last `N' events as new acquisitions occur. If the maximum number is greater than 20,000, the histogram will be updated until the number of events is equal to `N.' Then, if the number of bins or the histogram range is modified, the scope will use the parameter buffer values to redraw the histogram with either the last `N' or 20,000 values acquired -- whichever is the lesser. The parameter buffer thereby allows histograms to be redisplayed, using an acquired set of values and settings that produce a distribution shape with the most useful information.

In many cases the optimal range is not readily apparent. So the scope has a powerful range finding function. If required it will examine the values in the parameter buffer to calculate an optimal range and redisplay the histogram using it. The instrument will also give a running count of the number of parameter values that fall within, below, or above the range. If any values fall below or above the range, the range finder can then recalculate to include these parameter values, as long as they are still within the buffer.

Capture of Parameter Events

The number of events captured per waveform acquisition or display sweep depends on the parameter type. Acquisitions are initiated by the occurrence of a trigger event. Sweeps are equivalent to the waveform captured and displayed on an input channel (1, 2, or 3 or 4). For non-segmented waveforms an acquisition is identical to a sweep. Whereas for segmented waveforms an acquisition occurs for each segment and a sweep is equivalent to acquisitions for all segments. Only the section of a waveform between the parameter cursors is used in the calculation of parameter values and corresponding histogram events.

The following table provides a summary of the number of histogram events captured per acquisition or sweep for each parameter, and for a waveform section between the parameter cursors.

Parameters	Number of Events Captured
duty, freq, period, width, time@lev, f@level, f80-20%, fall, r@level, r20-80%, rise	All events in the acquisition
ampl, area, base, cmean, cmedian, crms, csdev, cycles, delay, maximum, mean, minimum, nbph, nbpw, over+, over-, pkpk, npts, rms, sdev, dly	One event per acquisition

Histogram Parameters (XMAP and JTA2 Options)

Once a histogram is defined and generated, measurements can be performed on the histogram itself. Typical of these are the histogram's

- average value, standard deviation
- most common value (parameter value of highest count bin)
- leftmost bin position (representing the lowest measured waveform parameter value)
- rightmost bin (representing the highest measured waveform parameter value)

Histogram parameters are provided to enable these measurements. Available through selecting "Statistics" from the "Category" menu, they are calculated for the selected section between the parameter cursors:

fwhm full width (of largest peak) at half the maximum bin

fwxx full width (of largest peak) at xx% the maximum bin

hist ampl histogram amplitude between two largest peaks

hist base histogram base or leftmost of two largest peaks

hist max value of the highest (right-most) populated bin in a histogram

hist mean average or mean value of data in the histogram

hist median value of the x-axis of a histogram that divides the population into two equal halves

hist min value of the lowest (left-most) populated bin in a histogram

hist rms rms value of data in histogram

hist sdev standard deviation of values in a histogram

hist top histogram top or rightmost of two largest peaks

max populate population of most populated bin in histogram

mode data value of most populated bin in histogram

percentile data value in histogram for which specified `x'% of population is smaller

peaks number of peaks in histogram

pop @ x population of bin for specified horizontal coordinate

range difference between highest and lowest data values

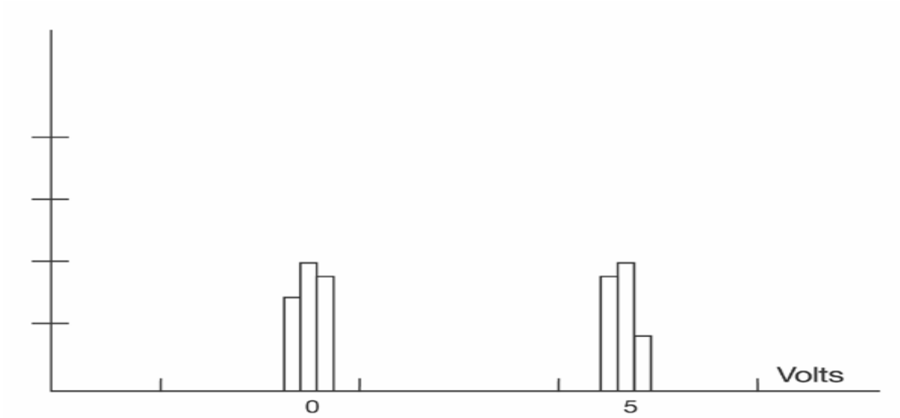
total pop total population in histogram

x at peak x-axis position of specified largest peak

Histogram Peaks

Because the shape of histogram distributions is particularly interesting, additional parameter measurements are available for analyzing these distributions. They are generally centered around one of several peak value bins, known, with its associated bins, as a histogram peak.

Example: In the following figure, a histogram of the voltage value of a five-volt amplitude square wave is centered around two peak value bins: 0 V and 5 V. The adjacent bins signify variation due to noise. The graph of the centered bins shows both as peaks.



Determining such peaks is very useful because they indicate dominant values of a signal.

However, signal noise and the use of a high number of bins relative to the number of parameter values acquired, can give a jagged and spiky histogram, making meaningful peaks hard to distinguish. The scope analyzes histogram data to identify peaks from background noise and histogram definition artifacts such as small gaps, which are due to very narrow bins.

Binning and Measurement Accuracy

Histogram bins represent a sub-range of waveform parameter values, or events. The events represented by a bin may have a value anywhere within its sub-range. However, parameter measurements of the histogram itself, such as average, assume that all events in a bin have a single value. The scope uses the center value of each bin's sub-range in all its calculations. The greater the number of bins used to subdivide a histogram's range, the less the potential deviation between actual event values and those values assumed in histogram parameter calculations.

Nevertheless, using more bins may require that you perform a greater number of waveform parameter measurements, in order to populate the bins sufficiently for the identification of a characteristic histogram distribution.

In addition, very fine grained binning will result in gaps between populated bins that may make it difficult to determine peaks.

The oscilloscope's 20,000-parameter buffer is very effective for determining the optimal number of bins to be used. An optimal bin number is one where the change in parameter values is insignificant, and the histogram distribution does not have a jagged appearance. With this buffer, a histogram can be dynamically redisplayed as the number of bins is modified by the user. In addition, depending on the number of bins selected, the change in waveform parameter values can be seen.

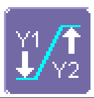
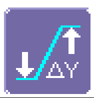
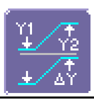
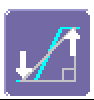
Measuring with Cursors

Cursors are important tools that aid you in measuring signal values. Cursors are markers — lines, cross-hairs, or arrows — that you can move around the grid or the waveform itself. Use cursors to make fast, accurate measurements and to eliminate guesswork. There are two basic types:

- **Horiz(ontal)** (generally Time or Frequency) cursors are markers that you move horizontally along the waveform. Place them at a desired location along the time axis to read the signal's amplitude at the selected time.
- **Vert(ical)** (Voltage) cursors are lines that you move vertically on the grid to measure the amplitude of a signal.

Cursor Measurement Icons

The **Readout** icons depict what is being measured for each measurement mode.

	Each cursor locates a point on the waveform. The cursor values can be read in the descriptor label for the trace. Use the **Position** data entry fields at the right side of the dialog to place the cursors precisely.
	This is the difference in Y values. The value can be read in the descriptor label for the trace.
	Displays absolute and delta cursors together.
	This gives the slope between cursors.

If there are non-time-domain waveforms displayed, there will also be a menu offering choices of x-axis units: **s** or **Hz**, for example.

Cursors Setup

Quick Display

At any time, you can change the display of cursor types (or turn them off) without invoking the "Cursors Setup" dialog as follows:

1. In the menu bar, touch **Cursors**, then **Off**, **Abs Horizontal**, **Rel Horizontal**, **Abs Vertical**, or **Rel Vertical**.
2. The cursors displayed will assume the positions previously set up. If you want to change their position or measurement mode, in the menu bar touch **Cursors**, then **Cursors Setup** in the drop-down menu.

Full Setup

1. In the menu bar, touch **Cursors**, then **Cursors Setup**. The "Standard Cursors" dialog opens.
2. In the dialog area, touch the **Cursors On** check box to display them.
3. Touch one of the **Horizontal** or **Vertical** mode buttons: **Relative** or **Absolute**.
4. If you chose a Relative mode, also touch a readout parameter button: Y position, delta Y, or slope.
5. If you chose a Relative mode, touch inside the **Position 1** and **Position 2** data entry fields and type in a value for each cursor. You can also use the Cursors knobs on the front panel to place the cursors. If you chose an Absolute mode, do the same for your single cursor.
6. If you chose a Relative mode and you would like both cursors to move in unison as you adjust the position, touch the **Track** check box to enable tracking.

Overview of Parameters

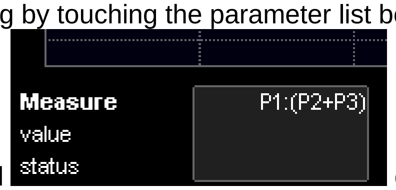
Parameters are measurement tools that determine a wide range of waveform properties. Use them to automatically calculate many attributes of your waveform, like rise-time, rms voltage, and peak-to-peak voltage, for example.

There are parameter modes for the amplitude and time domains, custom parameter groups, and parameters for pass and fail testing. You can make common measurements on one or more waveforms.

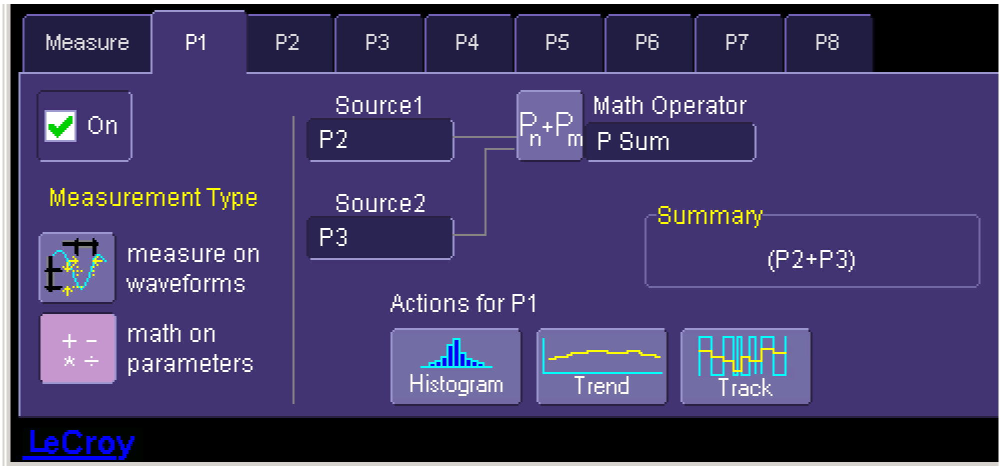
To Turn On Parameters

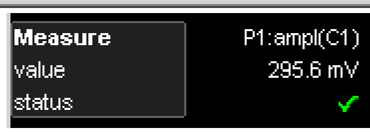
1. Touch **Measure** in the menu bar, then **Measure Setup...** in the drop-down menu.
2. Touch inside the **On** checkbox for each parameter you want to display.

Quick Access to Parameter Setup Dialogs

You can quickly gain access to a parameter setup dialog by touching the parameter list box below the grid. For example, touching within **P1** below the grid 

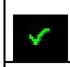
displays the setup dialog for P1:



Touching the row titles  displays the top Measure dialog.

Status Symbols

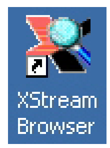
Below each parameter appears a symbol that indicates the status of the parameter, as follows:

	A warning symbol indicates that there is something wrong with the signal or the setup. Touch the parameter list box and read the explanation in the message line at the bottom of the screen.
	A green check mark means that the scope is returning a valid value.
	A crossed-out pulse means that the scope is unable to determine top and base; however, the measurement could still be valid.
	A downward pointing arrow indicates an underflow condition.
	An upward pointing arrow indicates an overflow condition.
	An upward-and-downward pointing arrow indicates an underflow and overflow condition.

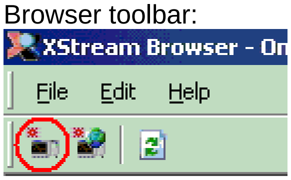
Using X-Stream Browser to Obtain Status Information

Example:

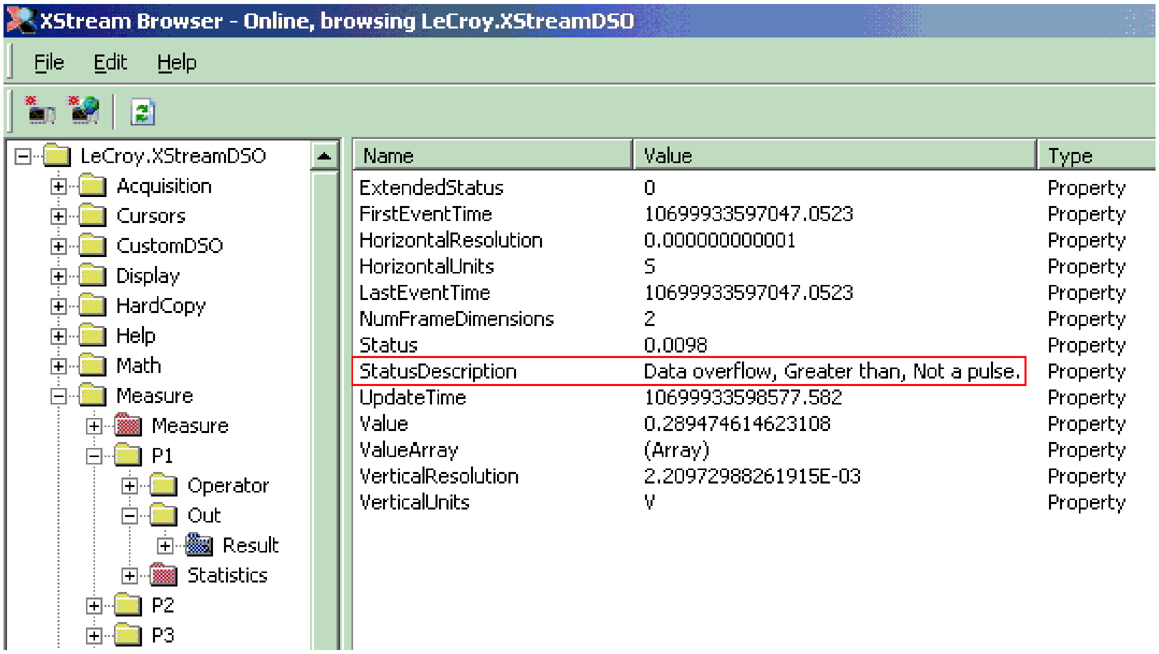
Here is a case of an overflow condition, in which the amplitude of the waveform cannot be determined:

1. Minimize the scope display by selecting File Minimize.
2. Touch the **X-Stream Browser** desktop icon 

to open the browser.
3. Touch the left scope icon ("Connect to a local X-Stream DSO device") in the X-Stream Browser toolbar:



4. Select **Measure Parameter in error (P1) Out Result:**



5. Read the status information in line **StatusDescription**.

Statistics

By touching the **Statistics On** checkbox in the "Measure" dialog, you can display statistics for standard vertical or horizontal parameters, or for custom parameters. The statistics that are displayed are as follows:

value (last)
mean
min.
max.
sdev
num

The values displayed in the **num** row is the number of measurements computed. For any parameter that computes on an entire waveform (like edge@level, mean, minimum, maximum, etc.) the value displayed represents the number of sweeps.

For any parameter that computes on every event, the value displayed is equal to the number of events per acquired waveform. If *x* waveforms were acquired, the value represents *x* times the number of cycles per waveform. Also, the "value" is equal to the measurement of the last cycle on the last acquisition.

To Apply a Measure Mode

1. In the menu bar, touch **Measure**, then **Measure Setup**.
2. Choose a **Measure Mode** from the dialog. The parameters are displayed below the grid.

Measure Modes

The selections for Measure Mode allow you to quickly apply parameters for standard vertical and standard horizontal setups, and custom setups.

Standard Vertical Parameters

These are the default Standard Vertical Parameters:

Vertical
mean
sdev
max.
min.
ampl
pkpk
top
base

Standard Horizontal Parameters

These are the default Standard Horizontal Parameters:

Horizontal
freq
period
width
rise
fall
delay

duty
npoints

My Measure

You can choose to customize up to eight parameters by touching **My Measure**.

Parameter Math (XMath or XMAP option required)

The instrument gives you the ability to perform arithmetic operations (addition, subtraction, multiplication, division) on the results of two parameter measurements. Alternatively, you can apply math to a single parameter (for example, invert). By customizing parameters in this way, you can effectively extend the range of parameter measurements based on your particular needs.

Logarithmic Parameters

The parameter math feature prevents multiplication and division of parameters that return logarithmic values. These parameters are as follows:

- narrow-band power (NBPW)
- top-to-base ratio when the units are in dB (TBR)

Excluded Parameters

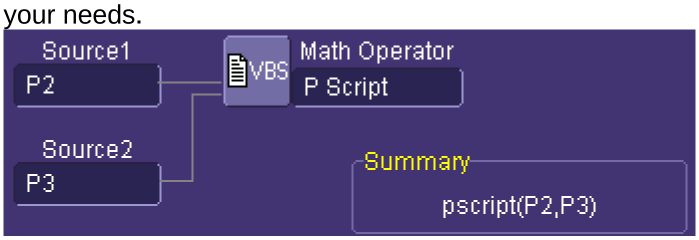
Parameters that are already the result of parameter math operations are excluded. If they are included in a remote control setup command, an error message is generated and the setup canceled.

Excluded parameters are as follows:

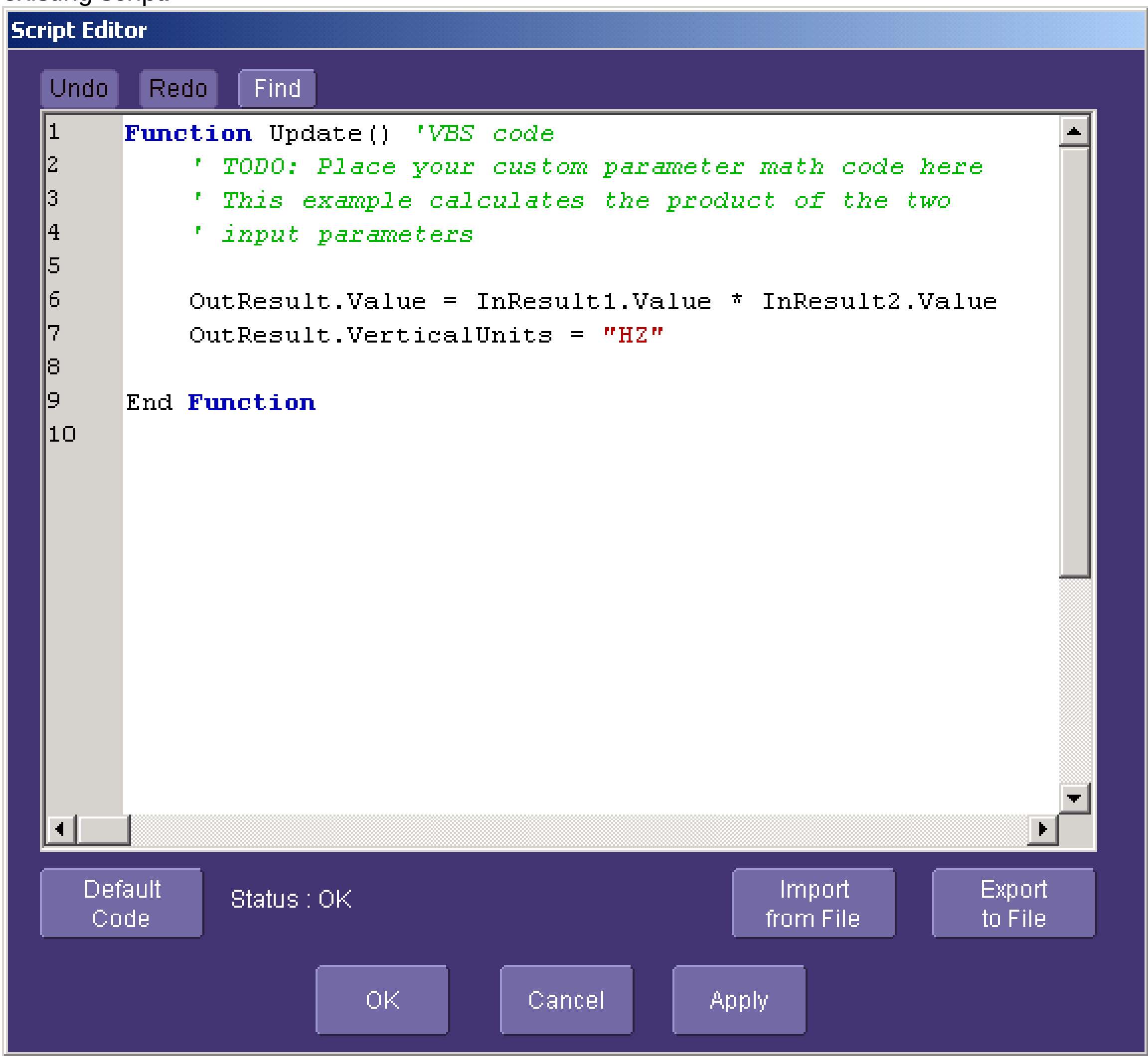
- delta delay (DDLY)
- delta time at level (DTLEV)
- phase (PHASE)

Parameter Script Parameter Math

In addition to the arithmetic operations, the Parameter Math feature allows you to use VBScript or JavaScript to write your own script for one or two measurements and produce a result that suits your needs.



Code entry is done in the Script Editor window directly on the instrument. You can also import an existing script.

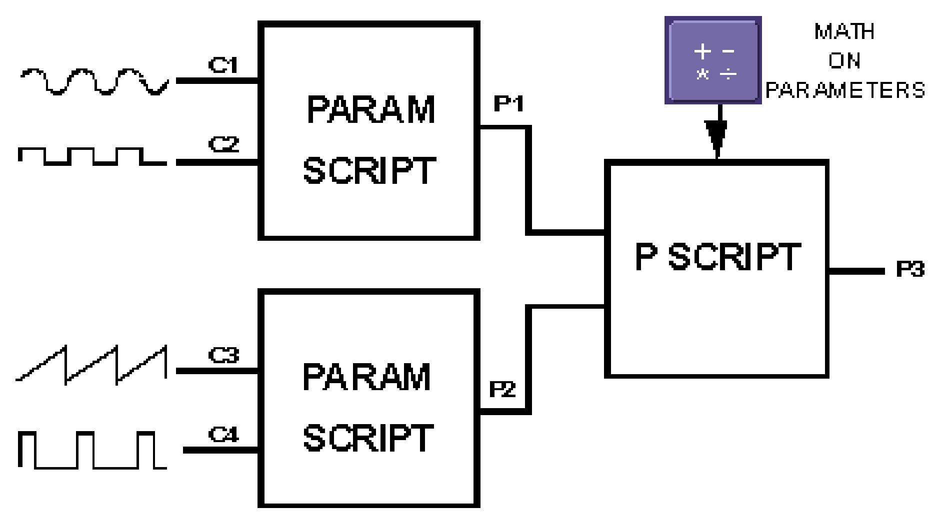


Param Script vs. P Script

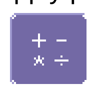
Param Script is a VBScript or JavaScript that operates on one or two *waveforms* and outputs a parameter measurement, as shown in the figure below. P Script, on the other hand, is another VBScript or JavaScript that takes as input one or two *parameters* and performs a math operation on them to produce another parameter output.

The inputs to Param Script can also be math (F1-Fx) or memory (M1-Mx) traces. The inputs to P

Script can be the results of any parameter measurement, not necessarily Param Script.



To Set Up Parameter Math

1. Touch **Measure** in the menu bar, then **Measure Setup...** in the drop-down menu.
2. Touch the **My Measure** button in the "Measure" dialog.
3. Touch the **Px** tab for the parameter to which you want to apply parameter math.
4. In the "Px" dialog, touch the **math on parameters** button . The **Source** field will expand to two fields.
5. Touch inside the **Source1** and **Source2** fields and select the parameters you want to apply math to (P1 to Px). If you are applying math to a single parameter (for example, invert), just touch inside the **Source1** field and select a parameter (P1 to Px).
6. Touch inside the **Math Operator** field and select a math operation from the **Select Measurement** menu. If you select an operation that requires two input parameters, the **Source** field will expand to two fields.

To Set Up Parameter Script Math

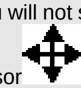
1. Touch **Measure** in the menu bar, then **Measure Setup...** in the drop-down menu.
2. Touch the **My Measure** button in the "Measure" dialog.
3. Touch the **Px** tab for the parameter to which you want to apply parameter math.
4. In the "Px" dialog, touch the **math on parameters** button . The **Source** field will expand to two fields.

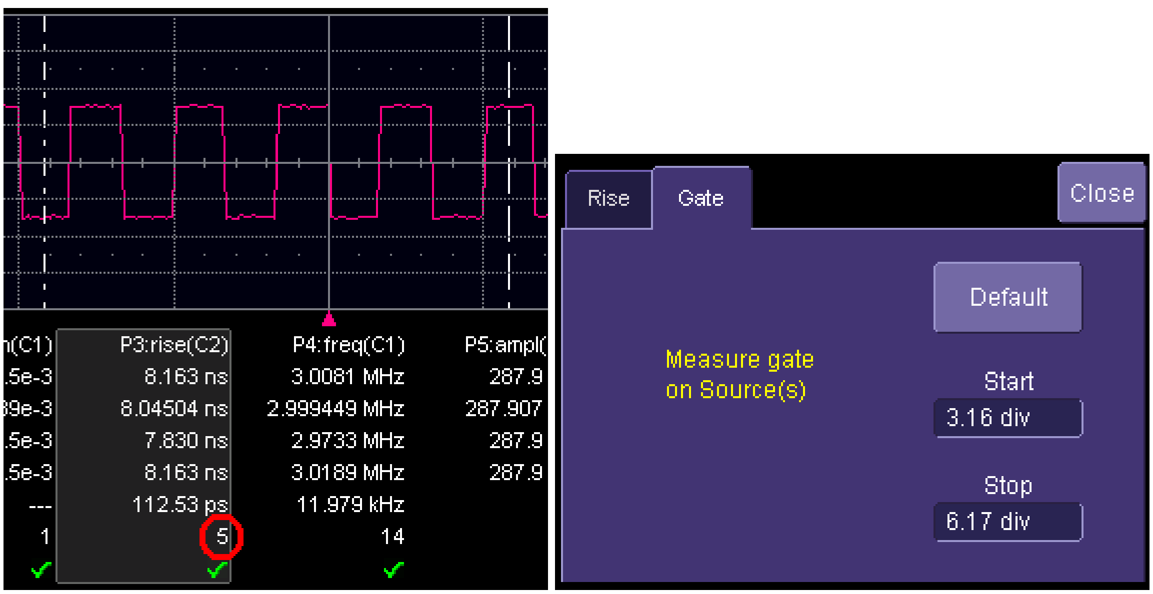
5. Touch inside the **Source1** and **Source2** fields and select the parameters you want to apply math to (P1 to Px). If you are applying math to a single parameter (for example, invert), just touch inside the **Source1** field and select a parameter (P1 to Px).
6. Touch inside the **Math Operator** field and select 

P Script from the **Select Measurement** menu.
7. In the "Script Math" dialog, touch inside the Script Language field and select either **VBScript** or **JScript** from the pop-up menu.
8. Touch the **Edit Code** button; the Script Editor window opens. You can enter code in this window or call up an existing script from a file storage location. If you create your script in this window, you can then export it and save it to file.

Measure Gate

Using Measure Gate, you can narrow the span of the waveform on which to perform parameter measurements, allowing you to focus on the area of greatest interest. You have the option of dragging the gate posts horizontally along the waveform, or specifying a position down to hundredths of a division. The default starting positions of the gate posts are 0 div and 10 div, which coincide with the left and right ends of the grid. The gate, therefore, initially encloses the entire waveform.

> **Note:** If you have Grid On Top enabled, you will not see the gate posts in their default position at each end of the grid. But if you touch either end of the grid, a drag cursor  will appear, indicating that you have control of the post and can now drag it.



In this example, you can see that the Measure Gate includes only five rising edges. Therefore, parameter calculations for rise time are performed only on the five pulses bounded by the gate posts. The position of the gate posts is shown in the **Start** and **Stop** fields in the accompanying dialog.

To Set Up Measure Gate

1. In the menu bar, touch **Measure Setup...**
2. Touch the **Px** tab for the parameter you want to gate. A mini-dialog to the right of the main setup dialog opens.

Note: If you already have the parameter of interest set up, you can simply touch the parameter directly below the grid.

3. Touch inside the **Start** data entry field and enter a value, using the pop-up numeric keypad. Or, you can simply touch the leftmost grid line and drag the gate post to the right.
4. Touch inside the **Stop** data entry field and enter a value, using the pop-up numeric keypad. Or, you can simply touch the rightmost grid line and drag the gate post to the left.

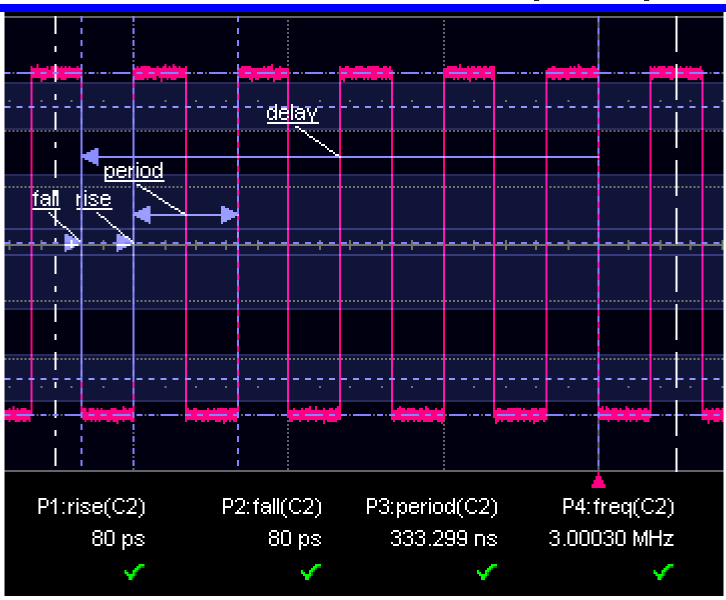
Help Markers

Help Markers clarify parameter measurements by displaying movable cursors and a visual representation of what is being measured. For the "at level" parameters, Help Markers make it easier to see where your waveform intersects the chosen level. This feature also displays the hysteresis band that you have set about that level.

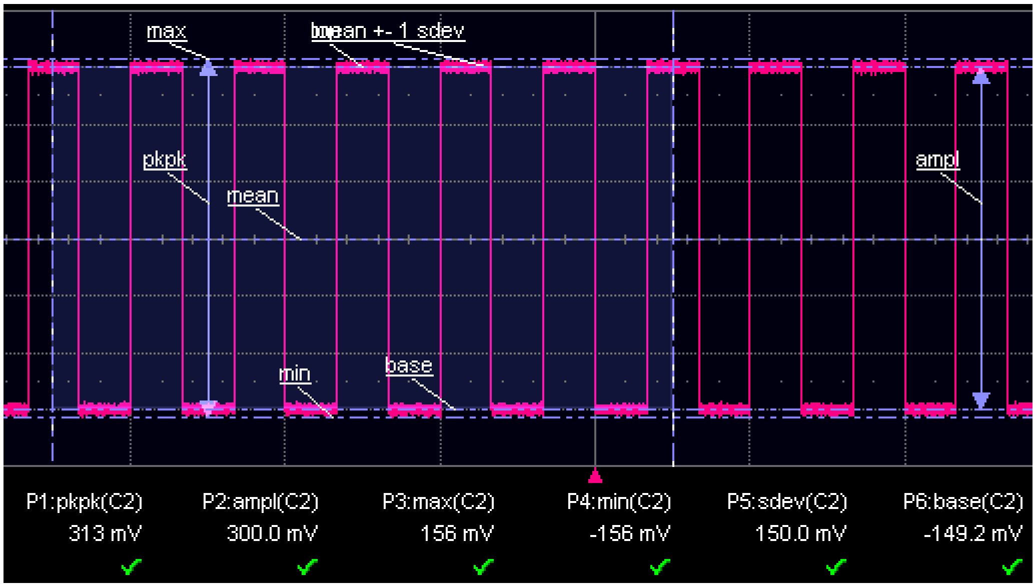
You also have the option, by means of an **Always On** checkbox, to leave the Help Markers displayed after you have closed the Help Markers setup dialog.

You have a choice of **Simple** or **Detailed** views of the markers:

- The **Simple** selection produces cursors and Measure Gate gate posts. The gate posts are independently placeable for each parameter.
- The **Detailed** selection produces cursors, Measure Gate gate posts, a label identifying the parameter being measured, and a level indicator and hysteresis band for "at level" parameters (not part of Standard Horizontal or Standard Vertical parameters).



Standard Horizontal Parameter Help Markers



Standard Vertical Parameter Help Markers

To Set Up Help Markers

1. In the menu bar, touch **Measure Setup...**

2. Select a Measure Mode: **Std Vertical**, **Std Horizontal**, or **My Measure**.
3. Touch the **Show All** button 

to display Help Markers for every parameter being measured on the displayed waveform (C2 in the examples above).
4. Touch a **P1** to **Px** tab 

for any parameter listed; it doesn't matter which. Touch inside the **Help Markers** field and select **Simple** The **Simple** selection produces cursors and Measure Gate gate posts. The gate posts are independently placeable for each parameter. or **Detailed** The **Detailed** selection produces cursors, Measure Gate gate posts, a label identifying the parameter being measured, and a level indicator and hysteresis band for "at level" parameters..

Note: The choice of Simple or Detailed is applied to all parameters at the same time. That is, if you choose Simple markers for one parameter, all parameters will be displayed in this mode.

5. Touch the **Always On** checkbox if you want to continuously display Help Markers for this parameter.

To Turn Off Help Markers

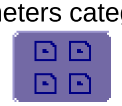
1. Touch the **Clear All** button 

to turn off Help Markers for all parameters.
2. To turn off Help Markers for individual parameters, touch the **Px** tab for the parameter in question. Then uncheck the **Always On** checkbox. When you close this dialog, the Help Markers for this parameter will no longer be displayed.

To Customize a Parameter

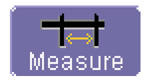
From the Measure Dialog

1. Touch the **My Measure** button in the "Measure" dialog. The dialog presents you with a panel of eight preset parameters.
2. For each parameter, touch the **On** check box to enable the parameter listed.
3. If you want to change the parameter listed, or a measurement characteristic, touch the parameter button 

(**P1** for example) alongside the check box. A pop-up menu of parameters categorized by type appears. To display parameter icons only, touch the icon button  at the bottom of the menu. To display the icons in list form, along with an explanation of each parameter, touch the list button . Use the Up/Down buttons to scroll through the list of icons.

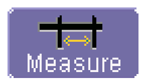
4. When you make a selection from the parameter icon menu, the setup dialogs for that parameter appear. You can then change the waveform source and other conditions of the parameter.
5. If you are setting up an "@level" parameter, make selections for **Level type** (percent or absolute), **Slope** (positive, negative, both), and **Hysteresis** level.
6. Touch the **Gate** tab, and set the position of the gate posts.

From a Vertical Setup Dialog

1. In the "Cx Vertical Adjust" dialog, touch the Measure button 

.
2. Select a parameter from the pop-up menu. (The **Actions for trace** source defaults to the channel or trace whose dialog is open. If a parameter, it goes into the next "available" parameter, or the last one if all are used.)
3. Select another parameter or touch **Close**.

From a Math Setup Dialog

1. In the "Fx" dialog, touch the Measure button 

.
2. Select a parameter from the pop-up menu. (The **Actions for trace** source defaults to the channel or trace whose dialog is open. If a parameter, it goes into the next "available" parameter, or the last one if all are used.)
3. Select another parameter or touch **Close**.

Parameter Calculations

Parameters and How They Work

Determining Top and Base Lines

Proper determination of the top and base reference lines is fundamental for ensuring correct parameter calculations. The analysis begins by computing a histogram of the waveform data over the time interval spanned by the left and right time cursors. For example, the histogram of a waveform transitioning in two states will contain two peaks (see Figure 1). The analysis will attempt to identify the two clusters that contain the largest data density. Then the most probable state (centroids) associated with these two clusters will be computed to determine the top and base reference levels: the top line corresponds to the top and the base line to the bottom centroid.

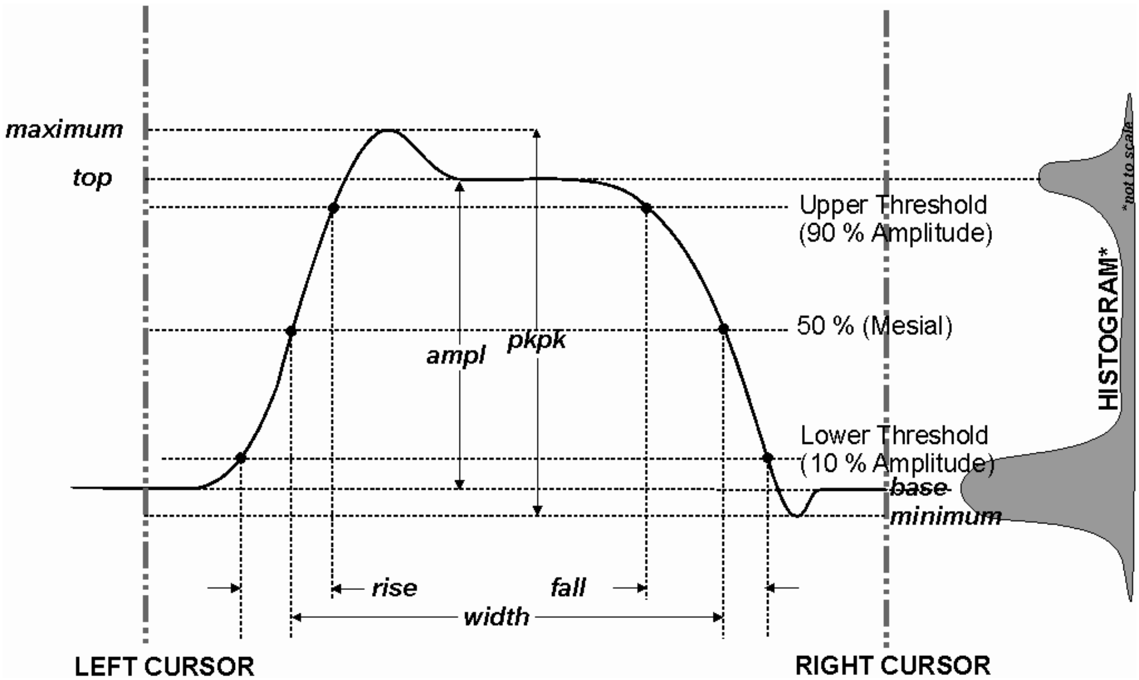


Figure 1

Determining Rise and Fall Times

Once top and base are estimated, calculation of the rise and fall times is easily done (see Figure 1). The 90% and 10% threshold levels are automatically determined by the instrument, using the amplitude (ampl) parameter.

Threshold levels for rise or fall time can also be selected using absolute or relative settings (r@level, f@level). If absolute settings are chosen, the rise or fall time is measured as the time interval separating the two crossing points on a rising or falling edge. But when relative settings are chosen, the vertical interval spanned between the base and top lines is subdivided into a percentile scale (base = 0 %, top = 100 %) to determine the vertical position of the crossing points.

The time interval separating the points on the rising or falling edges is then estimated to yield the rise or fall time. These results are averaged over the number of transition edges that occur within the observation window.

Rising Edge Duration	$\frac{1}{Mr}\sum_{i=1}^{Mr}(Tr_i^{90} - Tr_i^{10})$
Falling Edge Duration	$\frac{1}{Mf}\sum_{i=1}^{Mf}(Tf_i^{10} - Tf_i^{90})$
Where *Mr* is the number of leading edges found, *Mf* the number of trailing edges found, Tr_i^x the time when rising edge *i* crosses the x% level, Tf_i^x and the time when falling edge *i* crosses the x% level.	

Determining Time Parameters

Time parameter measurements such as width, period and delay are carried out with respect to the mesial reference level (see Figure 2), located halfway (50%) between the top and base reference lines.

Time-parameter estimation depends on the number of cycles included within the observation window. If the number of cycles is not an integer, parameter measurements such as rms or mean will be biased. However, only the last value is actually displayed, the mean being available when statistics are enabled. To avoid these bias effects, the instrument uses cyclic parameters, including crms and cmean, that restrict the calculation to an integer number of cycles.

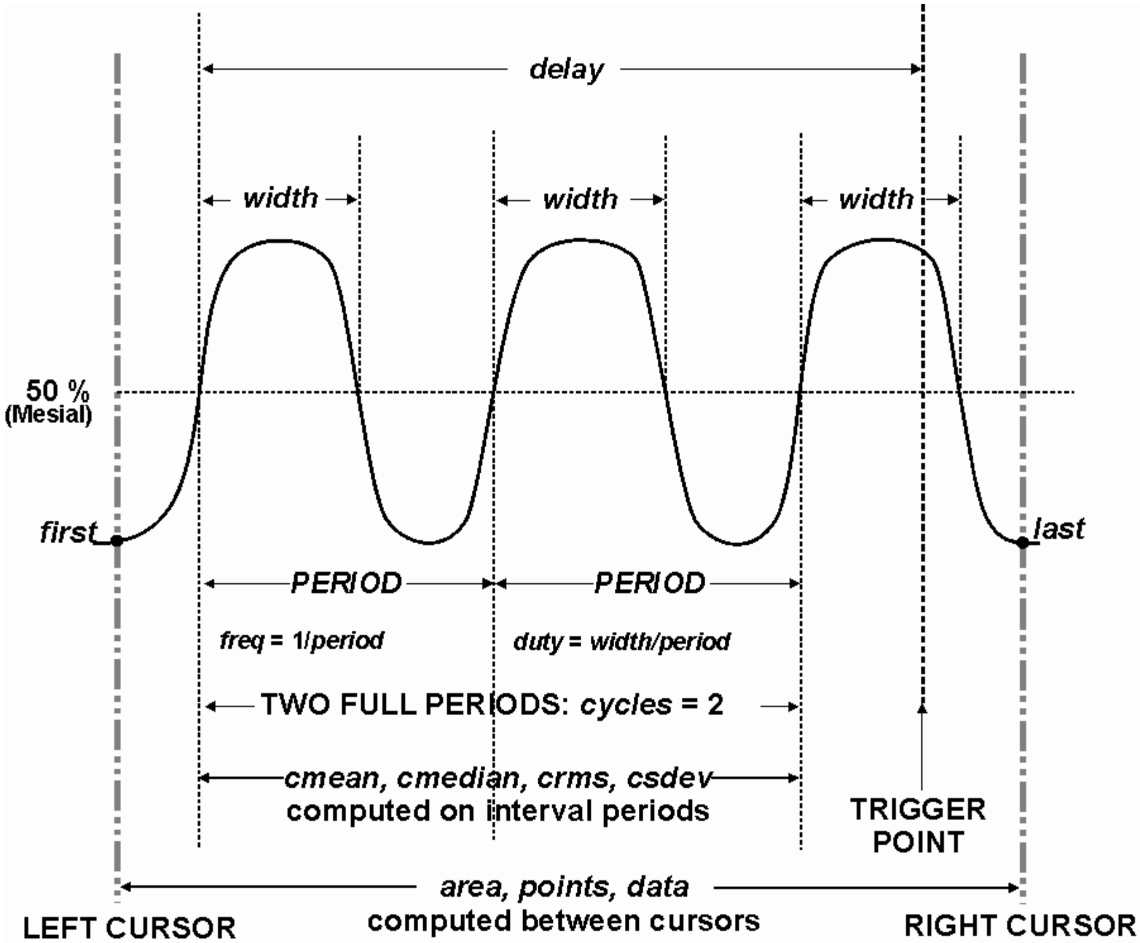


Figure 2

Determining Differential Time Measurements

The instrument enables accurate differential time measurements between two traces: for example, propagation, setup and hold delays (see Figure 3).

Parameters such as Delta c2d± require the transition polarity of the clock and data signals to be specified.

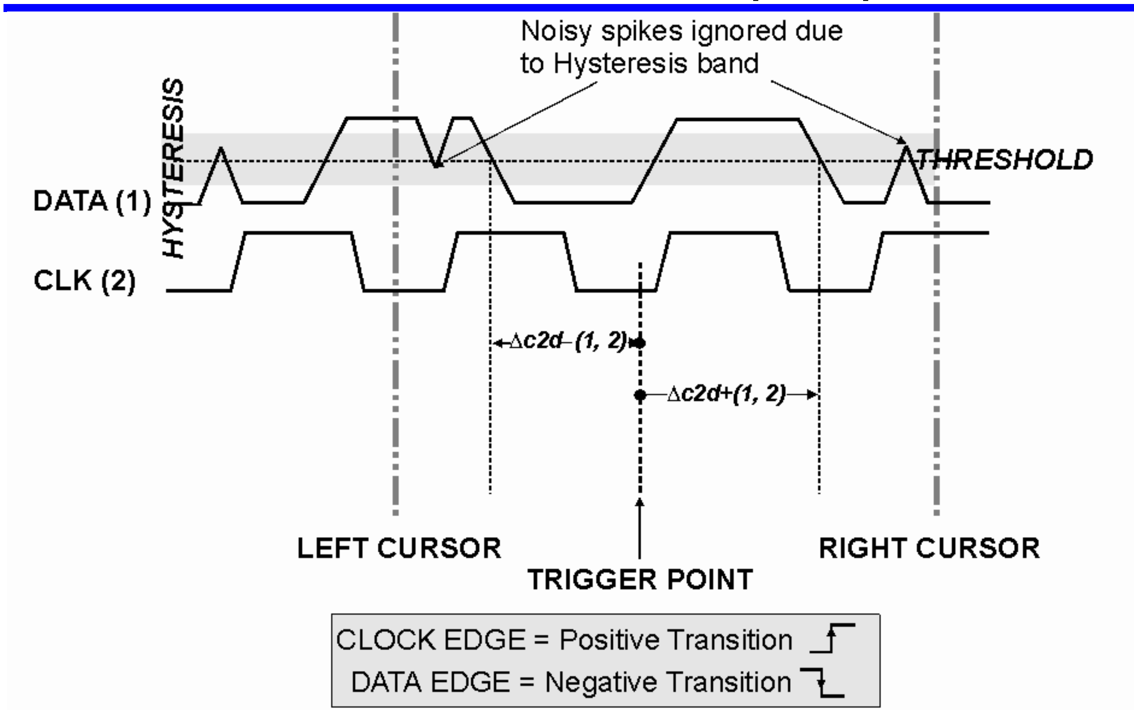


Figure 3

Moreover, a hysteresis range may be specified to ignore any spurious transition that does not exceed the boundaries of the hysteresis interval. In Figure 3, Delta c2d- (1, 2) measures the time interval separating the rising edge of the clock (trigger) from the first negative transition of the data signal. Similarly, Delta c2d+ (1, 2) measures the time interval between the trigger and the next transition of the data signal.

Level and Slope

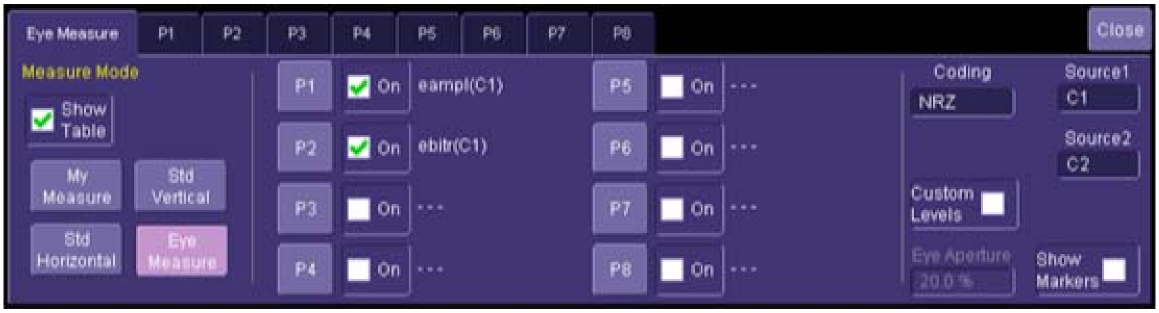
For several time based measurements, you can choose positive, negative, or both slopes to begin parameter measurements. For two-input parameters, such as Dtime@level, you can specify the slope for each input, as well as the level and type (percent or absolute).

Eye Parameters

A set of parameters called Eye Parameters is available, which measures waveform characteristics based on the statistics on the 3-dimensional persistence map of the eye pattern. This map displays voltage vs. time vs. frequency of occurrence (on the Z-axis). All of the eye parameters begin with the designation EyeXXXX. For example, the rise time based on a measurement of the eye pattern is referred to as Eye Rise Time.

There are two classes of digital signal whose eye patterns can be measured: Return to zero (RZ) and non-return to zero (NRZ). There are separate measurements for each of these signal types.

The type of signal is selected in the parameter menu using the **Coding** control. The "Eye Measure" menu also has controls to enter two sources. The **Source1** control is the primary input for all of the selected eye parameters, while **Source2** is used by eye parameters measured between two eye patterns. Sources for eye measurements can be any channel when the instrument is in Eye mode, and the eye pattern measured in Serial Data Analysis mode (Eye) on NRO9000 models with the SDA option or SDA100G models.



The amplitude related parameters associated with NRZ signals are based on the one and zero levels, as measured within the eye aperture, which is defined as a percentage of the unit interval centered in the eye. The default percentage is 20%. The one and zero levels are measured as the mean of histograms around the high and low levels within the eye aperture. The level at 50% between the high and low levels at 10% and 90% are used by parameters such as rise time. NRZ measurements operate on the histogram mean of the persistence map.

Parameter	Description	Definition	Notes
Ext Ratio	Extinction ratio	$ER = \frac{(P_1 - P_d)}{(P_0 - P_d)}$	Dark calibration required to determine Pd. Result can be displayed in %, dB (10log(ER)).
Eye AC RMS	RMS value of the signal within 1 unit interval UI		
Eye Amplitude	Difference between the 0 and 1 levels	$Eampl = P_1 - P_0$	P1 and P0 are the mean of the amplitude.
Eye BER	Estimated BER based on eye amplitude and noise level	$BER = \frac{e^{-Q^2/2}}{Q\sqrt{2\pi}}$	Q is the Q factor. BER assumes optimum sampling position within the eye.
Eye Bit Rate	Bit rate based on the inverse of the measured unit interval	$BR = \frac{1}{UI}$	

Eye Bit Time	Duration of one unit interval		
Eye Crossing	Crossing level of the eye diagram		
Eye Cyc Area	The area under the mean persistence trace over the first UI on the display		
Eye delay	Time to the first crossing of an eye relative to the trigger or eye reference		
Eye delta delay	Delay of crossing times between two eyes		
Eye fall time	Fall time of the mean of the persistence map		90% to 10% default; user selectable levels
Eye frequency	Bit rate expressed as a frequency	$F = \frac{1}{2 * UI}$	
Eye height	Vertical eye opening	$EH = P_1 - P_0 - 3\sigma_1 - 3\sigma_0$	
Eye mean	Mean level of the eye	$\frac{1}{M}\sum_0^{UI} y_i$	y_i are the samples between the eye crossing times. M is the number of samples in the region within the eye crossing points.
Eye Open Fac	Size of vertical opening of the eye	Vertical eye closure penalty $VECP = 10\log\left(\frac{OMA}{A_0}\right)$	
Eye over N	Suppression ratio (RZ only)	$SR = \frac{(P_1 - P_0)}{(P_{0-1/2} - P_0)}$	$P_{0-1/2}$ is the zero level 1/2 of a UI from the peak (one level) of the RZ signal
Eye over P	Positive overshoot of eye diagram		

List of Parameters

The following table describes the instrument parameters. Availability of some parameters depends on the options installed. See the comments in the "Notes" column of the table.

Parameter	Description	Definition	Notes
Amplitude	Measures the difference between upper and lower levels in two-level signals. Differs from pkpk in that noise, overshoot, undershoot, and ringing do not affect the measurement.	top base	On signals not having two major levels (such as triangle or saw-tooth waves), returns same value as pkpk. Standard parameter.
Ampl asym	Amplitude asymmetry between taa+ and taa-	1 - |(taa+ - taa-)|/(taa+ taa-)	Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
ACSN	Auto-correlation Signal-to-Noise provides a signal-to-noise ratio for periodic waveforms.		Available with DDM2 option. Standard in DDA-5005A.
Area	Integral of data: Computes area of waveform between cursors relative to zero level. Values greater than zero contribute positively to the area; values less than zero negatively.	Sum from first to last of data multiplied by horizontal time between points	Standard parameter..
Base	Lower of two most probable states (higher is top). Measures lower level in two-level signals. Differs from min in that noise, overshoot, undershoot, and ringing do not affect measurement.	Value of most probable lower state	On signals not having two major levels (triangle or saw-tooth waves, for example), returns same value as min. Standard parameter.

Cycles	Determines number of cycles of a periodic waveform lying between cursors. First cycle begins at first transition after the left cursor. Transition may be positive- or negative-going.	Number of cycles of periodic waveform	Standard parameter.
cyclic Mean	Cyclic mean: Computes the average of waveform data. Contrary to mean, computes average over an integral number of cycles, eliminating bias caused by fractional intervals.	Average of data values of an integral number of periods	Choose this parameter by selecting **Mean** from the parameter table, then touching the **Cyclic** checkbox. Standard parameter.
cyclic Median	Cyclic median: Computes average of base and top values over an integral number of cycles, contrary to median, eliminating bias caused by fractional intervals.	Data value for which 50% of values are above and 50% below	Choose this parameter by selecting **Median** from the parameter table, then touching the **Cyclic** checkbox. Standard parameter.
cyclic RMS	Cyclic root mean square: Computes square root of sum of squares of data values divided by number of points. Contrary to rms, calculation is performed over an integral number of cycles, eliminating bias caused by fractional intervals.	$\sqrt{\frac{1}{N} \sum_{i=1}^{N} (v_i)^2}$	Where: v_i denotes measured sample values, and N = number of data points within the periods found. Choose this parameter by selecting **RMS** from the parameter table, then touching the **Cyclic** checkbox. Standard parameter.
cyclic Std dev	Cyclic standard deviation: Standard deviation of data values from mean value over integral number of periods. Contrary to sdev, calculation is performed over an integral number of cycles, eliminating bias caused by fractional intervals.	$\sqrt{\frac{1}{N} \sum_{i=1}^{N} (v_i - mean)^2}$	Where: v_i denotes measured sample values, and N = number of data points within the periods found. Choose this parameter by selecting **Std dev** from the parameter

			table, then touching the **Cyclic** checkbox. Standard parameter.
Delay	Time from trigger to transition: Measures time between trigger and first 50% crossing after left cursor. Can measure propagation delay between two signals by triggering on one and determining delay of other.	Time between trigger and first 50% crossing after left cursor	Standard parameter.
Delta delay	delay: Computes time between 50% level of two sources.	Time between midpoint transition of two sources	Standard parameter.
Dperiod@level	Adjacent cycle deviation (cycle-to-cycle jitter) of each cycle in a waveform		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2 and XMAP options.
Dtime@level	t at level: Computes transition between selected levels or sources.	Time between transition levels of two sources, or from trigger to transition level of a single source This measurement gives the time of the source 2 edge minus the time of the source 1 edge.	Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Standard parameter.
Duration	For single sweep waveforms, dur is 0; for sequence waveforms: time from first to last segment's trigger; for single segments of sequence waveforms: time from previous segment's to current segment's trigger; for	Time from first to last acquisition: for average, histogram or sequence waveforms	Standard parameter.

	waveforms produced by a history function: time from first to last accumulated waveform's trigger.		
Duty@level	Percent of period for which data are above or below a specified level.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2 and XMAP options.
Duty cycle	Duty cycle: Width as percentage of period.	width/period	Standard parameter.
Dwidth@level	Difference of adjacent width above or below a specified level.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2 and XMAP options.
Edge@level	Number of edges in waveform.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2 and XMAP options.
Excel	Performs measurements in Excel by transferring one or two waveforms and reading the resulting parameter value.		Available with XMAP option. Excel must be loaded on the instrument.

<table>
<tr><td>Fall time</td><td>Fall time: Duration of falling edge from 90-10%.
<table><tr><th>Thresh.</th><th>Remote</th><th>Lower Limit</th><th>Upper Limit</th><th>Default</th></tr><tr><td>Lower</td><td>Low</td><td>1 %</td><td>45 %</td><td>10 %</td></tr><tr><td>Upper</td><td>High</td><td>55 %</td><td>99 %</td><td>90 %</td></tr></table>Threshold arguments specify two vertical values on each edge used to compute fall time. Formulas for upper and lower values:

lower = lower thresh. x amp/100 + base

upper = upper thresh. x amp/100 + base</td><td>Time at upper threshold minus

Time at lower threshold

averaged over each falling edge</td><td>On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results.

Standard parameter.</td></tr>
<tr><td>Fall 80-20%</td><td>Fall 80-20%: Duration of pulse waveform's falling transition from 80% to 20%, averaged for all falling transitions between the cursors.</td><td>Average duration of falling
80-20% transition</td><td>On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results.

Standard parameter.</td></tr>
<tr><td>Fall@level</td><td>Fall at level: Duration of pulse waveform's falling edges between user-specified transition levels. See also Rise@level.</td><td>Duration of falling edge between transition levels</td><td>On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results.

Standard parameter.</td></tr>
<tr><td>First</td><td>Indicates value of horizontal axis at left cursor.</td><td>Horizontal axis value at left cursor</td><td>Indicates location of left cursor. Cursors are interchangeable: for example, the left cursor</td></tr>
</table>

			may be moved to the right of the right cursor and first will give the location of the cursor formerly on the right, now on left. Standard parameter.
Frequency	Frequency: Period of cyclic signal measured as time between every other pair of 50% crossings. Starting with first transition after left cursor, the period is measured for each transition pair. Values then averaged and reciprocal used to give frequency.	1/period	Standard parameter.
Freq@level	Frequency at a specific level and slope for every cycle in waveform.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2 and XMAP options.
FWHM	Measures the width of the largest area histogram peak at half of the population of the highest peak.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
FWxx	Measures the width of the largest area histogram peak at xx% of the population of the highest peak.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Half period	Half period of a waveform.		Reference levels and edge-transition polarity can be selected. Hysteresis argument

			used to discriminate levels from noise in data. Available with JTA2 and XMAP options.
Hist ampl	Difference in value between the two most populated peaks in a histogram.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Hist base	Value of the left-most of the two most populated histogram peaks.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Hist maximum	Value of the highest (right-most) populated bin in a histogram.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Hist mean	Average or mean value of data in the histogram.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Hist median	Value of the "X" axis of a histogram that divides the population into two equal halves.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Hist minimum	Value of the lowest (left-most) populated bin in a histogram.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.

Hist rms	Root mean square of the values in a histogram.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Hist sdev	Standard deviation of values in a histogram.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Hist top	Value of the right-most of the two most populated histogram peaks.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Hold time	Time from the clock edge to the data edge. You can set levels, slope, and hysteresis independently for Hold Clock and Hold Data. See also Setup parameter.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2 and XMAP options.
Last	Time from trigger to last (rightmost) cursor.	Time from trigger to last cursor	Indicates location of right cursor. Cursors are interchangeable: for example, the right cursor may be moved to the left of the left cursor and first will give the location of the cursor formerly on the left, now on right. Standard parameter.
Level@X	Gives the vertical value at the specified *x* position. If the *x* position is between two points, it gives the		Standard parameter.

	interpolated value. When the **Nearest point** checkbox is checked, it gives the vertical value of the nearest data point.		
Local base	Value of the baseline for a local feature.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local bsep	Local baseline separation, between rising and falling slopes.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local max	Maximum value of a local feature.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local min	Minimum value of a local feature.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.

Local number	Number of local features (peak/trough pairs).		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local pkpk	Vertical difference between the peak and trough of a local feature (lmax lmin).		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tbe	Time between events (between local peak and next trough or local trough and next peak).		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tbp	Time between a local feature peak and the next local peak.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tbt	Time between a local feature trough and the next local trough.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2

			option. Standard in DDA-5005A.
Local tmax	Time of the maximum value of a local feature.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tmin	Time of the minimum value of a local feature.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tot	Time a local feature spends over a user specified percentage of its peak-to-trough amplitude.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tpt	Time between local feature peak and trough.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.

Local ttp	Time between local feature trough and the next local peak.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Local tut	Time a local feature spends under a user specified percentage of its peak-to-trough amplitude.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
Mathcad	Produces a parameter using a user-specified Mathcad function.		Available with XMAP option. Mathcad 2001i or later must be loaded on the instrument.
MATLAB	Produces a parameter using a user-specified MATLAB function.		Available with XMAP option. MATLAB must be loaded on the instrument.
Maximum	Measures highest point in waveform. Unlike top, does not assume waveform has two levels.	Highest value in waveform between cursors	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition. Computes horizontal axis location of rightmost non-zero bin of histogram -- not to be

			confused with maxp. Standard parameter.
Max populate	Peak with maximum population in a histogram.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Mean	Average of data for time domain waveform. Computed as centroid of distribution for a histogram.	Average of data	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition. Standard parameter.
Median	The average of base and top values.	Average of Base and Top.	Standard parameter.
Minimum	Measures the lowest point in a waveform. Unlike base, does not assume waveform has two levels.	Lowest value in waveform between cursors	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition. Standard parameter.
Mode	Position of the highest histogram peak.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Nb phase	Provides a measurement of the phase at a specific frequency of a waveform (narrow band).		Available with DDM2 and XMAP options. Standard in DDA-5005A.

Nb Power	Provides a measurement of the power at a specific frequency of a waveform (narrow band).		Available with DDM2 and XMAP options. Standard in DDA-5005A.
N-cycle jitter	Peak-to-peak jitter between edges spaced *n* UI apart.	Compares the expected time to the actual time of leading edges n bits apart.	Available in SDA analyzers.
NLTS	Provides a measurement of the nonlinear transition shift for a prml signal.		Available with DDM2 option. Standard in DDA-5005AA.
Npts	Number of points in the waveform between the cursors.		Standard parameter.
Overshoot-	Overshoot negative: Amount of overshoot following a falling edge, as percentage of amplitude.	(base - min.)/ampl x 100	Waveform must contain at least one falling edge. On signals not having two major levels (triangle or saw-tooth waves, for example), may not give predictable results. Standard parameter.
Overshoot+	Overshoot positive: Amount of overshoot following a rising edge specified as percentage of amplitude.	(max. - top)/ampl x 100	Waveform must contain at least one rising edge. On signals not having two major levels (triangle or saw-tooth waves, for example), may not give predictable results. Standard parameter.
Overwrite	Ratio of residual-to-original power of a low frequency waveform overwritten by a higher frequency.		Available with DDM2 option. Standard in DDA-5005A.

Param Script	Visual Basic or Java script that produces a measurement from one or two input waveforms.		Available with XMAP option. Standard in DDA-5005A.
Peaks	Number of peaks in a histogram.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Peak to peak	Peak-to-peak: Difference between highest and lowest points in waveform. Unlike ampl, does not assume the waveform has two levels.	maximum - minimum	Gives a similar result when applied to time domain waveform or histogram of data of the same waveform. But with histograms, result may include contributions from more than one acquisition. Standard parameter.
Percentile	Horizontal data value that divides a histogram so the population to the left is xx% of the total.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Period	Period of a cyclic signal measured as time between every other pair of 50% crossings. Starting with first transition after left cursor, period is measured for each transition pair, with values averaged to give final result.	$\frac{1}{Mr}\sum_{i=1}^{Mr}(Tr_i^{50} - Tr_i^{50})$	Where: Mr is the number of leading edges found, Mf the number of trailing edges found, Tr_i^x the time when rising edge i crosses the x% level, and Tf_i^x the time when falling edge i crosses the x% level. Standard parameter.

Period@level	Period at a specified level and slope for every cycle in waveform.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2 and XMAP options.
Phase	Phase difference between signal analyzed and signal used as reference. You can set the output type to percent, degrees, or radians. After setting up the reference, touch the **More** tab for signal setups.	Phase difference between signal and reference	Standard parameter.
Pop@X	Population at bin for specified horizontal coordinate. You can place the cursor at any bin and use either Absolute, Reference, or Difference cursor shape.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
PW50	Average pulse width at the 50% point between the local baseline and the local peak or trough.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
PW50-	Average pulse width at the 50% point between the local baseline and the local trough.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.

<table>
<tr><td>PW50+</td><td>Average pulse width at the 50% point between the local baseline and the local peak.</td><td></td><td>Hysteresis argument used to discriminate levels from noise in data.
Available with DDM2 option.
Standard in DDA-5005A.</td></tr>
<tr><td>Range</td><td>Calculates range (max min) of a histogram.</td><td></td><td>Available with DDM2, JTA2, and XMAP options.
Standard in DDA-5005A.</td></tr>
<tr><td>Resolution</td><td>Ratio of taa for a high and low frequency waveform</td><td>taa (HF)/mean taa (LF)*100</td><td>Hysteresis argument used to discriminate levels from noise in data.
Standard parameter.</td></tr>
<tr><td>Rise</td><td>Rise time: Duration of rising edge from 10-90%.
<table>
<tr><th>Thresh.</th><th>Remote</th><th>Lower Limit</th><th>Upper Limit</th><th>Default</th></tr>
<tr><td>Lower</td><td>Low</td><td>1 %</td><td>45 %</td><td>10 %</td></tr>
<tr><td>Upper</td><td>High</td><td>55 %</td><td>99 %</td><td>90 %</td></tr>
</table>
Threshold arguments specify two vertical values on each edge used to compute rise time.
Formulas for upper and lower values:
lower = lower thresh. x amp/100 + base
upper = upper thresh. x amp/100 + base</td><td>Time at lower threshold minus
Time at upper threshold
averaged over each rising edge</td><td>On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results.
Standard parameter.</td></tr>
</table>

Rise 20-80%	Rise 20% to 80%: Duration of pulse waveform's rising transition from 20% to 80%, averaged for all rising transitions between the cursors.	Average duration of rising 20-80% transition	On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results. Standard parameter.
Rise@level	Rise at level: Duration of pulse waveform's rising edges between transition levels.	Duration of rising edges between transition levels	On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving, however, less predictable results. Standard parameter.
RMS	Root Mean Square of data between the cursors -- about same as sdev for a zero-mean waveform.	$\sqrt{\frac{1}{N} \sum_{i=1}^{N} (v_i)^2}$	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition. Where: v_i denotes measured sample values, and N = number of data points within the periods found up to maximum of 100 periods. Standard parameter.

Setup	Time from the data edge to the clock edge.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2 and XMAP options.
Skew	Time of clock1 edge minus time of nearest clock2 edge.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Hysteresis on a measurement (if set to 500 mdiv) requires that the signal must transition one way 1/2 division (total swing) across the threshold for the measurement to be valid. Available with JTA2 and XMAP options.
Std dev	Standard deviation of the data between the cursors -- about the same as rms for a zero-mean waveform.	$\sqrt{\frac{1}{N}\sum_{i=1}^{N}(v_i - mean)^2}$	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition. Where: v_i denotes measured sample values, and N = number of data points within the periods found up to maximum of 100

			periods. Standard parameter.
TAA	Average peak-to-trough amplitude for all local features.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
TAA-	Average local baseline-to-trough amplitude for all local features.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
TAA+	Average local baseline-to-peak amplitude for all local features.		Hysteresis argument used to discriminate levels from noise in data. Available with DDM2 option. Standard in DDA-5005A.
TIE@level	Difference between the measured times of crossing a given slope and level and the ideal expected time. For Slope you can choose positive, negative, or both. For output units you can choose time or unit interval (UI). A unit interval equals one clock period. The Virtual Clock setup gives you a choice of Standard	Cutoff Freq = (1/1.667e3) x Clock Freq	Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with JTA2 and XMAP options.

	(1.544 MHz) or Custom reference clocks. You can also use a mathematically derived Golden PLL to filter low frequency jitter. The cutoff frequency is user selectable.		
Time@level	Time at level: Time from trigger (t=0) to crossing at a specified level.	Time from trigger to crossing level	Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Standard parameter.
Top	Higher of two most probable states, the lower being base; it is characteristic of rectangular waveforms and represents the higher most probable state determined from the statistical distribution of data point values in the waveform.	Value of most probable higher state	Gives similar result when applied to time domain waveform or histogram of data of same waveform. But with histograms, result may include contributions from more than one acquisition. Standard parameter.
Total Pop	Total population of a histogram.		Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
Width	Width of cyclic signal determined by examining 50% crossings in data input. If first transition after left cursor is a rising edge, waveform is considered to consist of positive pulses and width the time between adjacent rising and falling edges. Conversely, if falling	Width of first positive or negative pulse averaged for all similar pulses	Similar to fwhm, though, unlike width, that parameter applies only to histograms. Standard parameter.

	edge, pulses are considered negative and width the time between adjacent falling and rising edges. For both cases, widths of all waveform pulses are averaged for the final result.		
Width@level	Width measured at a specific level.		Reference levels and edge-transition polarity can be selected. Hysteresis argument used to discriminate levels from noise in data. Available with DDM2, JTA2, and XMAP options. Standard in DDA-5005A.
WidthN	Width measured at the 50% level and negative slope.		Standard parameter.
X at max	Determines the horizontal axis location of the maximum value between the cursors.		Restricted to time and frequency waveforms only.
X at min	Determines the horizontal axis location of the minimum value between the cursors.		Restricted to time and frequency waveforms only.
X at peak	The value of the nth highest histogram peak.		Applies only to histograms. Available with JTA2 and XMAP options. Standard in DDA-5005A.

Qualified Parameters

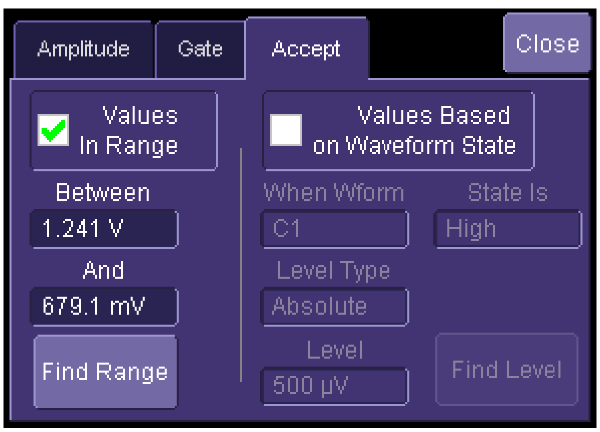
Some LeCroy instruments and software packages give you the ability to constrain parameter measurements to a vertically or horizontally limited range, or to occurrences gated by a second waveform. Furthermore, both constraints can operate together. This capability enables you to exclude unwanted characteristics from your measurements. It is much more restrictive than Measure Gate, which is used only to narrow the span of the waveform for analysis, along the horizontal axis.

Note: Since this feature operates on only a subset of the data, possible alerts or status indicators concerning the measurement (such as "Data range too low") are not displayed.

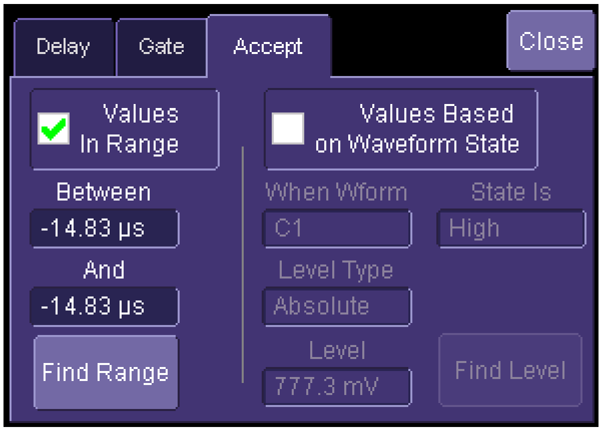
Range Limited Parameters

To Set Up Range Qualifiers

1. From the menu bar, select **Measure**, then **Measure Setup...** from the drop-down menu.
2. Touch a **Px** tab to open the setup dialog.
3. Touch inside the **Source** field and select a source from the pop-up menu.
4. Touch inside the **Measure** field and select a parameter from the pop-up menu.
5. Touch the **Accept** tab of the right-hand dialog, then touch the **Values In Range** checkbox. Depending on whether you select a vertical or horizontal parameter, the correct units will be automatically displayed (V, s, Hz, dB) in the **Between** and **And** fields. Or, if you select a simple ratio parameter (such as power factor) that yields a dimensionless number, no units will be displayed.
6. Touch the **Find Range** button to quickly display the most recent value of the parameter measurement. From there it is a simple matter to set the desired range.



Amplitude Parameter (range set manually)

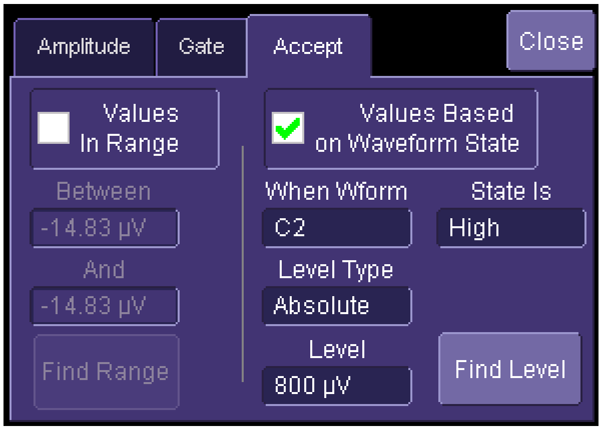


Delay Parameter (Find Range selected)

Waveform Gated Parameters

To Set Up Waveform Qualifiers

1. From the menu bar, select **Measure**, then **Measure Setup...** from the drop-down menu.
2. Touch a **Px** tab to open the setup dialog.
3. Touch inside the **Source** field and select a source from the pop-up menu.
4. Touch inside the **Measure** field and select a parameter from the pop-up menu.
5. Touch the **Accept** tab of the right-hand dialog, then touch the **Values Based on Waveform State** checkbox.
6. Touch inside the **When Wform** field and select the gating source.
7. Touch inside the **State Is** field and select **High** or **Low** from the pop-up menu. Parameter measurements on the subject waveform will only be taken when the gating waveform is in the selected state.
8. Touch inside the **Level Type** field and select **Absolute** or **Percent** from the pop-up menu.
9. Touch inside the **Level** field and enter the crossing level value at which you want measurements to begin. Alternatively, touch the **Find Level** button to automatically select the 50% level of your gating waveform.


Amplitude
Gate
Accept
Close
Values In Range
Values Based on Waveform State
Between
-14.83 µV
And
-14.83 µV
Find Range
When Wform
C2
State Is
High
Level Type
Absolute
Level
800 µV
Find Level

WAVEFORM MATH

Introduction to Math Traces and Functions

With the instrument's math tools you can perform mathematical functions on a waveform displayed on any channel, or recalled from any of the four reference memories M1 to M4. You can also set up traces F1 to Fx to do math on parameter measurements P1 to Px.

For example: you could set up Trace F1 as the difference between Channels 1 and 2, Trace F2 as the average of F1, and Trace F3 as the integral of F2. You could then display the integral of the averaged difference between Channels 1 and 2. Any trace and function can be chained to another trace and function. For example, you could make Trace F1 an average of Channel 1, Trace F2 an FFT of F1, and Trace F3 a zoom of F2.

Note: Math traces F5-F8 are available only if you have loaded software option package XMATH or XMAP on WaveMaster or WavePro scopes, but are standard on Disk Drive Analyzers and Serial Data Analyzers.

Math Made Easy

With the instrument's math tools you can perform mathematical functions on a waveform displayed on any channel C1 to C4, or recalled from any of the four reference memories M1 to M4. To do computations in sequence, you can also use math functions F1 to Fx as a source input waveform. Or you can use Parameters P1 through Px

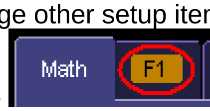
For example: you could set up F1 as the difference between Channels 1 and 2, F2 as the average of F1, and F3 as the integral of F2. You could then display the integral of the averaged difference between Channels 1 and 2. Any trace and function can be chained to another trace and function. For example, you could make F1 an average of Channel 1, F2 an FFT of F1, and F3 a zoom of F2.

Refer to the Specifications to find out which math tools are available in each optional package.

To Set Up a Math Function

This setup mode allows you to quickly apply frequently used math functions.

1. In the menu bar, touch **Math**, then **Math setup...**
2. If there are math functions already assigned to **F1** through **Fx**, touch the checkbox for the function you want to enable.
3. To assign a new math function to a trace, touch the Fx button for that trace, for example 

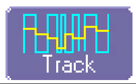
. The math function menu appears.
4. Touch a menu selection; your new function is automatically assigned, with the same setups as were in place for the last function in that Fx position.
5. If you want to change other setup items, like the source waveform, touch the appropriate Fx tab, for example 

. The setup dialog for that Fx position appears.

6. Touch the **Single** function button 

if you want to perform just one math function on the trace, or touch the **Dual** function button 

to perform math on math.
7. Touch the **Graph** button, then touch inside the **Graph with** field to select a graph mode. The Graph modes are as follows:

	Histogram of the values of a parameter
	Track of the values of a parameter
	Trend of the values of a parameter

Resampling To Deskew

Deskew whenever you need to compensate for different lengths of cables, probes, or anything else that might cause timing mismatches between signals. Resample a signal on one channel and adjust it in time relative to a signal on another channel.

To Resample

1. In the menu bar, touch **Math**, then **Math Setup...** in the drop-down menu.
2. Touch a math function trace tab **F1** through **Fx** The number of math traces available depends on the software options loaded on your scope. See Specifications..
3. Touch the **single** function button.
4. Touch inside the **Source1** field and select a source: channel, math trace, memory location.
5. Touch inside the **Operator1** field and select **Deskew** from the **Functions** category.
6. In the dialog on the right, touch the **Deskew** tab.
7. Touch inside the **Delay by** data entry field and type in a time value, using the pop-up keypad.

Rescaling and Assigning Units

This feature allows you to apply a multiplication factor (*a*) and additive constant (*b*) to your waveform: *a*X + *b*. You can do it in the unit of your choice, depending on the type of application.`

Allowable unit abbreviations are as follows:

(blank)	no units
A	Ampere
C	Coulomb
CYCLE	cycles
DB	Decibel
DBC	Decibel referred to carrier
DBM	Decibel Milliwatt
DBV	Decibel Volts
DBUZ	Decibel Microamp
DEC	Decade
DIV	Divisions
Event	Events
F	Farad
G	Gram
H	Henry
HZ	Hertz
J	Joule
K	Degree Kelvin
CEL	Degree Celsius
FAR	Degree Fahrenheit
L	Liter
M	Meter
FT	Foot
IN	Inch

YARD	yard
MILE	mile
N	Newton
OHM	Ohm
PAL	Pascal
PCT	Percent
POISE	Poise
PPM	parts per million
RAD	Radian
DEG	Degree (of arc)
MNT	Minute (of arc)
SAMPLE	sample
SWEEP	sweeps
SEC	Second (of arc)
S	Second
SIE	Siemens
T	Tesla
UI	Unit interval
V	Volt
VA	Volt amps
W	Watt
WB	Weber
MIN	min
HOUR	Hour
DAY	Day
WEEK	Week

You can also enter combinations of the above units following the SI rules:

- for the quotient of two units, the character **/** should be used
- for the product of two units, the character **.** should be used
- exponents can be represented by a digit appended to the unit without a space

For example,

- acceleration can be entered as M/S2 for meters per second squared
- volts seconds can be entered as V.S

In some cases, the units entered may be converted to simple units. For example entering V.A will display W (watts)

To Set Up Rescaling

1. In the menu bar, touch **Math**, then **Math Setup...** in the drop-down menu.
2. Touch a math function trace tab **F1** through **Fx** The number of math traces available depends on the software options loaded on your scope. See Specifications..
3. Touch the **single** function button.
4. Touch inside the **Source1** data entry field and select a source: channel, math trace, memory location.
5. Touch inside the **Operator1** data entry field and select **Rescale** from the **Functions** category.
6. In the dialog on the right, touch the **Rescale** tab.
7. Touch inside the **First multiply by** checkbox and enter a value for *a*, the multiplication factor.
8. Touch inside the **then add:** data entry field and enter a value for *b*, the additive constant.
9. Touch inside the **Override units** checkbox to disregard the source waveform's units, using the pop-up keyboard.

Averaging Waveforms

Summed vs. Continuous Averaging

For Summed averaging, you specify the number of acquisitions to be averaged. The averaged data is updated at regular intervals and presented on the screen.

On the other hand, Continuous averaging (the system default) helps to eliminate the effects of noise by continuously acquiring new data and adding the new waveforms into the averaging buffer. You determine the importance of new data vs. old data by assigning a weighting factor. Continuous averaging allows you to make adjustments to a system under test and to see the results immediately.

Note: Continuous Averaging is accessible from the channel "Vertical Adjust" dialog under "Pre-Processing," and from the math function menu.

Summed Averaging

Summed Averaging is the repeated addition, with equal weight, of successive source waveform records. If a stable trigger is available, the resulting average has a random noise component lower than that of a single-shot record. Whenever the maximum number of sweeps is reached, the averaging process stops.

An even larger number of records can be accumulated simply by changing the number in the dialog. However, the other parameters must be left unchanged or a new averaging calculation will be started. You can pause the averaging by changing the trigger mode from NORM/AUTO to STOP. The instrument resumes averaging when you change the trigger mode back to NORM/AUTO.

You can reset the accumulated average by pushing the CLEAR SWEEPS button or by changing an acquisition parameter such as input gain, offset, coupling, trigger condition, timebase, or bandwidth limit. The number of current averaged waveforms of the function, or its zoom, is shown in the acquisition status dialog. When summed averaging is performed, the display is updated at a reduced rate to increase the averaging speed (points and events per second).

Continuous Averaging

Continuous Averaging, the default setting, is the repeated addition, with unequal weight, of successive source waveforms. It is particularly useful for reducing noise on signals that drift very slowly in time or amplitude. The most recently acquired waveform has more weight than all the previously acquired ones: the continuous average is dominated by the statistical fluctuations of the most recently acquired waveform. The weight of ‘old’ waveforms in the continuous average gradually tends to zero (following an exponential rule) at a rate that decreases as the weight increases.

The formula for continuous averaging is

new average = (new data + weight * old average)/(weight + 1)

This is also the formula used to compute summed averaging. But by setting a "sweeps" value, you establish a fixed weight that is assigned to the old average once the number of "sweeps" is reached.

For example, for a sweeps (weight) value of **4**:

1st sweep (no old average yet): new average = (new data +0 * old average)/(0 + 1) = new data only

2nd sweep: new average = (new data + 1*old average)/(1 + 1) = 1/2 new data +1/2 old average

3rd sweep: new average = (new data + 2 * old average)/(2 + 1) = 1/3 new data + 2/3 old average

4th sweep: new average = (new data + 3 * old average)/(3 + 1) = 1/4 new data + 3/4 old average

5th sweep: new average = (new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

6th sweep: new average = (new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

7th sweep: new average = (new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

In this way, for sweeps > 4 the importance of the old average begins to decrease exponentially.

Note: The number of sweeps used to compute the average will be displayed in the bottom line of the trace descriptor label:

To Set Up Continuous Averaging

1. In the menu bar, touch **Math**, then **Math Setup...** in the drop-down menu.
2. Select a function tab from **F1** through **Fx**. [The number of math traces available depends on the software options loaded on your scope. See Specifications.]
3. Touch inside the **Source1** field and select a source waveform from the pop-up menu.
4. Touch inside the **Operator1** field and select **Average** from the **Select Math Operator** menu.
5. Touch the **Average** tab in the dialog to the right of the "Fx" dialog, touch the **Continuous** button.
6. Touch inside the **Sweeps** data entry field and enter a value using the pop-up keypad. The valid range is 1 to 1,000,000 sweeps.

To Set Up Summed Averaging

1. In the menu bar, touch **Math**, then **Math Setup...** in the drop-down menu.
2. Select a function tab from **F1** through **Fx**. [The number of math traces available depends on the software options loaded on your scope. See Specifications.]
3. Touch inside the **Source1** field and select a source waveform from the pop-up menu.
4. Touch inside the **Operator1** field and select **Average** from the **Select Math Operator**

menu.

5. Touch the **Average** tab in the dialog to the right of the "Fx" dialog, then touch the **Summed** button.
6. Touch inside the **Sweeps** data entry field and type in a value using the pop-up keypad. The valid range is 1 to 1,000,000 sweeps.

Enhanced Resolution

ERES (Enhanced Resolution) filtering increases vertical resolution, allowing you to distinguish closely spaced voltage levels. The functioning of the instrument's ERES is similar to smoothing the signal with a simple, moving-average filter. However, it is more efficient concerning bandwidth and pass-band filtering. Use ERES on single-shot waveforms, or where the data record is slowly repetitive (when you cannot use averaging). Use it to reduce noise when your signal is noticeably noisy, but you do not need to perform noise measurements. Also use it when you perform high-precision voltage measurements: zooming with high vertical gain, for example.

How the Instrument Enhances Resolution

The instrument's enhanced resolution feature improves vertical resolution by a fixed amount for each filter. This real increase in resolution occurs whether or not the signal is noisy, or your signal is single-shot or repetitive. The signal-to-noise ratio (SNR) improvement you gain is dependent on the form of the noise in the original signal. The enhanced resolution filtering decreases the bandwidth of the signal, filtering out some of the noise.

The instrument's constant phase FIR (Finite Impulse Response) filters provide fast computation, excellent step response in 0.5 bit steps, and minimum bandwidth reduction for resolution improvements of between 0.5 and 3 bits. Each step corresponds to a bandwidth reduction factor of two, allowing easy control of the bandwidth resolution trade-off. The parameters of the six filters are given in the following table.

Resolution increased by	-3 dB Bandwidth (× Nyquist)	Filter Length (Samples)
0.5	0.5	2
1.0	0.241	5
1.5	0.121	10
2.0	0.058	24
2.5	0.029	51
3.0	0.016	117

With low-pass filters, the actual SNR increase obtained in any particular situation depends on the power spectral density of the noise on the signal.

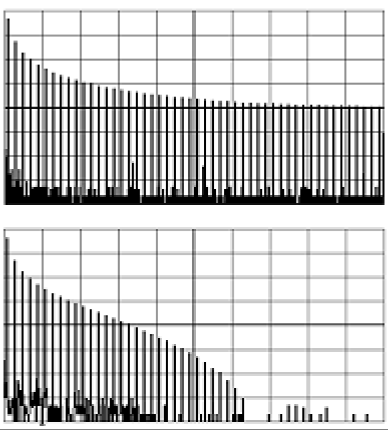
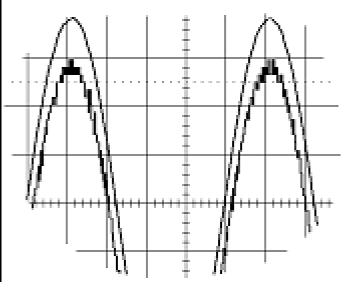
The improvement in SNR corresponds to the improvement in resolution if the noise in the signal is white -- evenly distributed across the frequency spectrum.

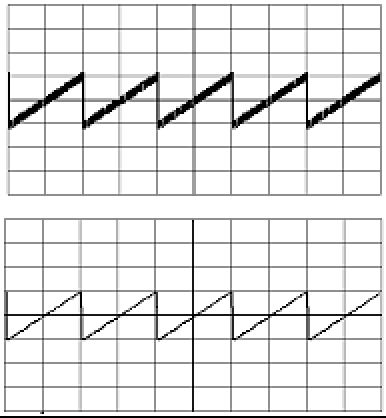
If the noise power is biased towards high frequencies, the SNR improvement will be better than the resolution improvement.

The opposite may be true if the noise is mostly at lower frequencies. SNR improvement due to the removal of coherent noise signals -- feed-through of clock signals, for example -- is determined by the fall of the dominant frequency components of the signal in the passband. This is easily ascertained using spectral analysis. The filters have a precisely constant zero-phase response. This has two benefits. First, the filters do not distort the relative position of different events in the waveform, even if the events' frequency content is different. Second, because the waveforms are stored, the delay normally associated with filtering (between the input and output waveforms) can be exactly compensated during the computation of the filtered waveform.

The filters have been given exact unity gain at low frequency. Enhanced resolution should therefore not cause overflow if the source data is not overflowed. If part of the source trace were to overflow, filtering would be allowed, but the results in the vicinity of the overflowed data -- the filter impulse response length -- would be incorrect. This is because in some circumstances an overflow may be a spike of only one or two samples, and the energy in this spike may not be enough to significantly affect the results. It would then be undesirable to disallow the whole trace.

The following examples illustrate how you might use the instrument's enhanced resolution function.

	**In low-pass filtering:** The spectrum of a square signal before (left top) and after (left bottom) enhanced resolution processing. The result clearly illustrates how the filter rejects high-frequency components from the signal. The higher the bit enhancement, the lower the resulting bandwidth.
	**To increase vertical resolution:** In the example at left, the lower ("inner") trace has been significantly enhanced by a three-bit enhanced resolution function.



To reduce noise: The example at left shows enhanced resolution of a noisy signal. The original trace (left top) has been processed by a 2-bit enhanced resolution filter. The result (left bottom) shows a "smooth" trace, where most of the noise has been eliminated.

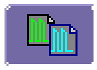
Note: Enhanced resolution can only improve the resolution of a trace; it cannot improve the accuracy or linearity of the original quantization. The pass-band will cause signal attenuation for signals near the cut-off frequency. The highest frequencies passed may be slightly attenuated. Perform the filtering on finite record lengths. Data will be lost at the start and end of the waveform: the trace will be slightly shorter after filtering. The number of samples lost is exactly equal to the length of the impulse response of the filter used: between 2 and 117 samples. Normally this loss (just 0.2 % of a 50,000 point trace) is not noticed. However, you might filter a record so short there would be no data output. In that case, however, the instrument would not allow you to use the ERES feature.

To Set Up Enhanced Resolution (ERES)

1. In the menu bar, touch **Math**, then **Math Setup...** in the drop-down menu.
2. Touch a function tab **F1** through **Fx** [The number of math traces available depends on the software options loaded on your scope. See Specifications.].
3. Touch inside the **Operator1** data entry field.
4. Select **ERES** from the **All Functions** or **Filter** group of Math functions.
5. Touch the **Trace On** checkbox.
6. Touch the "ERES" tab in the right-hand dialog, then touch inside the **bits** field and make an "Enhance by" selection from the pop-up menu:

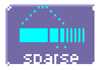
.

Waveform Copy

The **Copy** math function  makes a copy of your present waveform in its unprocessed state. While processing may continue on the original waveform, the copy enables faster throughput in some cases by preserving the original data. That is, no calculations need to be undone on the copy before additional math can be calculated.

This benefit of faster throughput, however, comes at the expense of memory usage.

Waveform Sparser

The Sparse math function 

allows you to thin out an incoming waveform by skipping points at regular intervals, and by starting acquisition at a particular "offset" (point). The **Sparsing factor** specifies the number of sample points to reduce the input waveform by. A sparsing factor of 4, for example, tells the scope to retain only one out of every 4 samples. A **Sparsing offset** of 3, on the other hand, tells the scope to begin on the third sample, then skip the number of samples specified by the sparsing factor (4). In this way, the sample rate is effectively reduced.

For the sparsing factor (interval), you can set a value from 1 to 1,000,000 points. For the sparsing offset you can set a value from 0 to 999,999.

Note: The maximum sparsing offset that can be entered for any sparsing factor equals Sparsing Factor 1.

To Set Up Waveform Sparser

1. In the menu bar, touch **Math**, then **Math setup...** in the drop-down menu.
2. Touch the tab for the function (F1 to Fx) you want to assign the Sparse operation to.
3. Touch inside the **Source1** field and select an input waveform.
4. Touch inside the **Operator1** field and select **Sparse** from the **Select Math Operator** menu.
5. Touch inside the **Sparsing factor** field and enter a value, using the pop-up keypad.
6. Touch inside the **Sparsing offset** field and enter a value, using the pop-up keypad.

Interpolation

Linear interpolation, which inserts a straight line between sample points, is best used to reconstruct straight-edged signals such as square waves. (Sinx)/x interpolation, on the other hand, is suitable for reconstructing curved or irregular waveshapes, especially when the sampling rate is 3 to 5 times the system bandwidth. The instrument also gives you a choice of Cubic interpolation.

For each method, you can select a factor from 2 to 50 points by which to interpolate (upsample).

To Set Up Interpolation

1. In the menu bar, touch **Math**, then **Math setup...** in the drop-down menu.
2. Touch the tab for the function (F1 to Fx you want to assign the Interpolate operation to.
3. Touch inside the **Source1** field and select an input waveform.
4. Touch inside the **Operator1** field, then touch the **Filter** button in the **Select Math Operator** menu.

5. Select **Interpolate** from the Filter submenu.
6. Touch the "Interpolate" tab in the mini setup dialog to the right of the main dialog.
7. Touch inside the **Algorithm** field and select an interpolation type.
8. Touch inside the **Upsample by** [Upsampling is the factor by which sampling is increased.] field and enter a value, using the pop-up numeric keypad, if you want to enter a specific value. Otherwise, use the **Up/Down** buttons to increment the displayed value in a 1-2-5 sequence.

Fast Wave Port

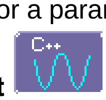
FastWavePort is a processing function for the LeCroy X-Stream family of digital oscilloscopes that enables you to insert your own custom processing algorithm, written in the C/C++ language, into the DSO's processing stream. FastWavePort maximizes data throughput from the acquisition system to your processing function. It also makes it simple to create these custom processing functions.

The technology that makes this system possible is the ability of two processes in a Windows system to share a region of memory. This enables the transfer of data at high-speed between the acquisition software and the custom processing function, which runs in a separate process from the DSO application. A major benefit of FastWavePort is that your application may be implemented and, more importantly, debugged independently of the main application.

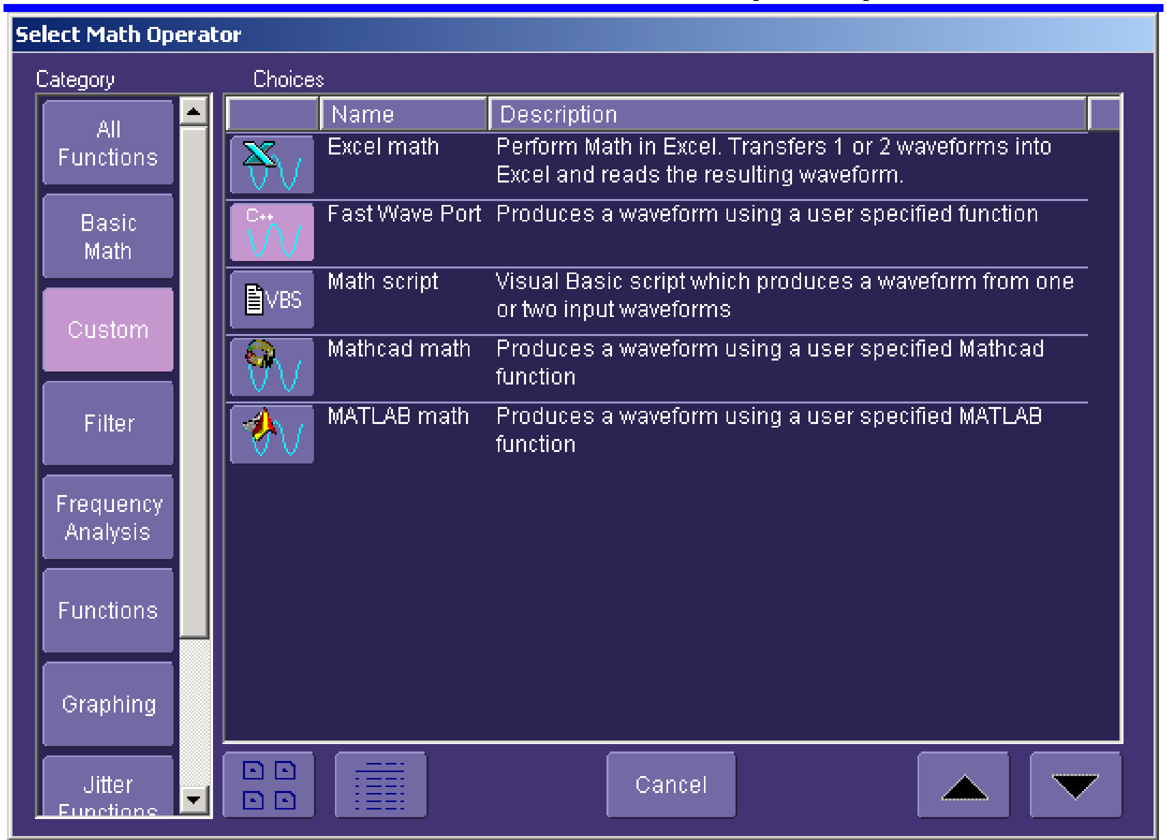
It is important to note that the transfer of the results of your processing function back into the X-Stream processing stream is optional. If performance is the primary goal, and display or further processing of the results within the DSO software is not required, then this may be skipped.

Fast Wave Port Setup -- Initial

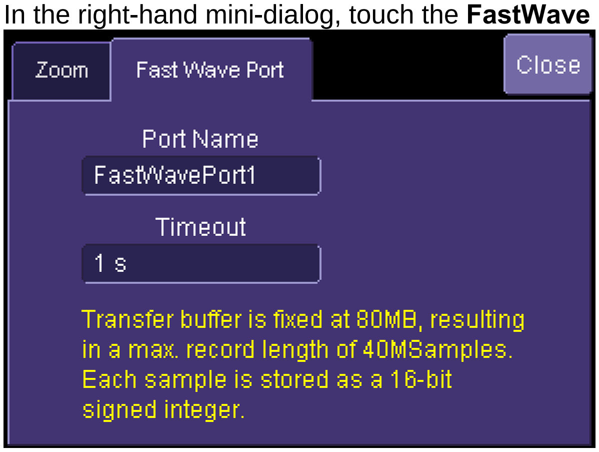
1. In the menu bar, touch **Math**, then **Math Setup...** in the drop-down menu.
2. Touch one of the Math function tabs, 

for example.
3. Touch inside the "Source1" field, and select a signal source from the pop-up menu. The source can be a channel waveform, math or memory trace, or a parameter.
4. Touch inside the "Operator"1 field and select **FastWavePort** 

from the **Custom** menu:



5. In the right-hand mini-dialog, touch the **FastWave Port** tab:



6. Touch inside the "Timeout" field and enter a suitable value.

Setup -- Case 1

This scenario assumes that you have developed your application on a PC.

1. Compile your application on your PC
2. Copy the compiled file onto the scope, using a memory stick or network drive.
3. Open the Command Prompt window (Start --> Programs --> Accessories --> Command Prompt) and run your application.

Setup -- Case 2

This scenario assumes that you have Visual C++ loaded on the scope.

- Use the Visual C++ editor to develop and run your application.

Setup -- Case 3

This scenario assumes that you are using a compiler other than Visual C++ (such as GNU's MinGW). It should be noted that the optimizer in the GNU C Compiler (GCC) is less efficient than that in Visual C++ and will result in lower performance.

1. Save your application in a text file, and copy it onto memory stick or network drive.
2. Using Windows Explorer, copy the text file to the scope.
3. Download and install the compiler onto the scope.
4. Configure Environmental Variables as follows:

   A. Open Start --> Settings --> Control Panel --> System.

   B. Click the "Advanced" tab, then the **Environmental Variables** button.

   C. In the "System variables" window, click **Path**, then the **Edit** button.

   D. At the end of the "Variable Value" string, append **;C:\MinGW\bin** for the case of the GNU C Compiler (GCC) for example.

   E. Click OK.
5. Open the Command Prompt window (Start --> Programs --> Accessories --> Command Prompt) and compile your application.
6. Run your application.

Operational Notes

Once FastWavePort is selected, the right-hand dialog shows the current settings. The first of these is critical, and indicates the base name of the memory window and the two events, which are global within the Windows O/S. This should be left at its default value and only changed if multiple FastWavePort functions are used in parallel. Note that this name must match the base name used in the client application.

The full names of these global objects are:

Memory Mapped File	"FastWavePort1File"
Data Available Event	"FastWavePort1MutexDataAvailable"
Processing Complete Event	"FastWavePort1MutexProcessingComplete"

The "Timeout" field specifies the amount of time that the DSO will wait for the custom processing function to complete. This prevents the DSO from waiting indefinitely for a potentially unforthcoming custom processing function. Be careful to set this value to something reasonable, which means a time that is longer by a reasonable margin than the custom processing is ever expected to take.

Data Length Limitations

The size of the memory window is fixed at 80 Mbytes, which equates to 40 MSamples.

Performance

This is by far the fastest way to process data using a user-defined algorithm on an X-Stream DSO.

Choice of Programming Language

The system was designed for use with the C/C++ programming language. However, it is theoretically possible for the processing to be implemented in any language that supports Windows named events (Mutex) and can open a named memory-mapped file. No guarantee can be given, however, as to the behavior of the system using anything but C/C++.

Example Application

This simple C++ application may be used as a starting point for a custom processing function. It demonstrates the following:

- How to create handles to the global objects (the memory window and the two events)
- How to read data from the memory window when the DSO flags that it's available
- How to scale the data into units of volts using data in the header that's stored at the beginning of the memory window.
- How to perform a simple processing function (in this case the absolute value)
- How to define the physical units of the output of the processing function (in this case 'Amps')
- How to flag to the DSO application that processing is complete

```
//-----------------------------------------------------------------------------------------
// FastWavePortClient.cpp :
//
// Prototype C++ client application for "Fast Wave Port' Math Processor
```

```
//
// Compatibility:
//      Microsoft Visual C++ 6.0, 7.1
//      MinGW 'gcc' based compiler (free download from http://www.mingw.org/)
//         Compile with: mingw32-c++ -o fastWavePortClient.exe fastwaveportclient.cpp
//-----------------------------------------------------------------------------------------
#include "windows.h"
#include <stdio.h>
//-----------------------------------------------------------------------------------------
// FastWavePort header, describes various properties of the waveform passed to the
user-processing
// function. Also used to carry the properties of the processed waveform back to the DSO.
//
#define FLAGS_OUTPUT_VALID      0x01
typedef unsigned __int64 lecTimeStamp;
lecTimeStamp lecTimeStampOneSecond = 1000000000;        // 1 ns units in a second
#pragma pack(push, 4)           // pack on 4-byte boundaries (Important!)
struct CDescHeader
{
   int descVersion;          // header version number
   int flags;                // misc. flags indicating the status of input, and how to treat the output
   int headerSize;           // size of the header, data starts immediately after the hdr.
   int windowSize;           // total size of the window (header + data)
   int numSamples;           // total number of samples in the input waveform
   int segmentIndex;         // index of this segment, usually zero when input waveform is not a
sequence
   int numSweeps;
   int _dummy1;              // not used
   double verGain;           // scale factor that relates integer sample data values to the vertical units
of the waveform.
```

```
   double verOffset;          // vertical offset (in vertical units, e.g. Volts) of the waveform
   double verResolution;      // vertical resolution of the measurement system (also in vertical units)
   double horInterval;       // scale factor that relates integer sample indices to the horizontal units of the waveform.
   double horOffset;          // horizontal offset (in horizontal units, e.g. seconds) of the waveform
   double horResolution;      // horizontal resolution of the measurement system (also in horizontal units)
   lecTimeStamp trigTime;     // trigger time, units of 1ns since 00:00:00 on Jan 1st 2000, 64-bit unsigned integer
   char verUnit[48];          // vertical units of the waveform ("V" for example)
   char horUnit[48];          // horizontal units of the waveform ("s" for example)
};
#pragma pack(pop)             // restore packing


//--------------------------------------------------------------------------------------
// The buffer size is 80MB (40,000,000 samples, stored as short integers) plus 0x1000 bytes for the header.
const unsigned long HEADER_SIZE = 0x1000
const unsigned long MEM_MAP_FILE_SIZE = 80000000 + HEADER_SIZE;    // = 40MSamples, or 80MBytes
int main(int argc, char* argv[])
{
   // names based on 'FastWavePort1' name defined in Processor setup.
   char szMapFileName[]                = "FastWavePort1File";
   char szMutexDataAvailableName[]         = "FastWavePort1MutexDataAvailable";
   char szMutexProcessingCompleteName[]    = "FastWavePort1MutexProcessingComplete";
   // Associate shared memory file handle value.
      HANDLE m_hMMFile = CreateFileMapping ((HANDLE)0xffffffff, NULL, PAGE_READWRITE, 0,  MEM_MAP_FILE_SIZE, szMapFileName);
      if(m_hMMFile == 0)
      {
         printf("Unable to create file mapping\n");
```

```
        return 0;
    }

  // Map a view of this file for writing.
    short *m_lpMMFile = (short *)MapViewOfFile (m_hMMFile, FILE_MAP_ALL_ACCESS, 0, 0, 0);
    if(m_lpMMFile == 0)
    {
        printf("Unable to map view of file\n");
        return 0;
    }
  // create/open events used for synchronization
  // if the client app. was run before the scope then these events will be created, if the scope was run first then these events
  // will just be opened
    HANDLE m_hDataAvailable = CreateEvent(NULL, FALSE, FALSE /* initial state */, szMutexDataAvailableName);
    HANDLE m_hProcessingComplete = CreateEvent(NULL, FALSE, FALSE /* initial state */, szMutexProcessingCompleteName);
    if(m_hDataAvailable == 0 || m_hProcessingComplete == 0)
    {
        printf("Unable to open events\n");
        return 0;
    }
  // main loop
    while(1)
    {
        int i = 0;
        printf("Waiting for new data...\n");
        // wait an infinite amount of time for data to be available
            DWORD waitSuccess = WaitForSingleObject(m_hDataAvailable, INFINITE);
```

```
        // print the first few bytes of the input waveform
          CDescHeader *descHeader = (CDescHeader *) &m_lpMMFile[0];
          short *m_lpWaveform = &m_lpMMFile[descHeader->headerSize / sizeof(short)];
          for(i = 0; i < 4; ++i)
            printf("%f ", (m_lpWaveform[i] * descHeader->verGain) + descHeader->verOffset);
        // compute the mean of all data values, while computing the abs value of the waveform
in-place
          double sum = 0.0;
          for(i = 0; i < descHeader->numSamples; ++i)
          {
            sum += (m_lpWaveform[i] * descHeader->verGain) + descHeader->verOffset;
            m_lpWaveform[i] = abs(m_lpWaveform[i]);
          }
          sum /= descHeader->numSamples;
        // modify the output units, set to Amps
          strcpy(descHeader->verUnit, "A");
        // print the mean, numer of samples, trigger time in seconds, and the segment index
          printf("    (%f) %d %d %d\n", sum, descHeader->numSamples, (int)
(descHeader->trigTime / lecTimeStampOneSecond), descHeader->segmentIndex);
        // use to flag that the output is not valid, increasing performance when
        // it is not necessary to read data back into the DSO
          //descHeader->flags &= ~FLAGS_OUTPUT_VALID;
        // flag that processing is complete
          SetEvent(m_hProcessingComplete);
      }
  return 0;
}
```

Header Description

int descVersion; // header version number

int flags; // misc. flags indicating the status of input, and how to treat the output int headerSize; // size of the header, data starts immediately after the hdr.

int windowSize; // total size of the window (header + data)

int numSamples; // total number of samples in the input waveform

int segmentIndex; // index of this segment, usually zero when input waveform is not a sequence

int numSweeps;

int _dummy1; // not used

double verGain; // scale factor that relates integer sample data values to the vertical units of the waveform.

double verOffset; // vertical offset (in vertical units, e.g. Volts) of the waveform

double verResolution; // vertical resolution of the measurement system (also in vertical units)

double horInterval; // scale factor that relates integer sample indices to the horizontal units of the waveform.

double horOffset; // horizontal offset (in horizontal units, e.g. seconds) of the waveform

double horResolution; // horizontal resolution of the measurement system (also in horizontal units)

lecTimeStamp trigTime; // trigger time, units of 1ns since 00:00:00 on Jan 1st 2000, 64-bit unsigned integer

char verUnit[48]; // vertical units of the waveform ("V" for example)

char horUnit[48]; // horizontal units of the waveform ("s" for example)

Data Length Limitations

The size of the memory window is fixed at 80 Mbytes, which equates to 40M samples.

Performance

Under optimal conditions, on a DSO with a 1.7 GHz Celeron processor, rates of up to 75 MS/s have been observed. Due to the differences between the acquisition and processing hardware in each of the X-Stream DSOs, this value may vary significantly and therefore cannot be guaranteed. However, this is by far the fastest way to process data using a user-defined algorithm on an X-Stream DSO.

Choice of Programming Language

The system was designed for use with the C/C++ programming language, and all furnished examples use this language. It is theoretically possible, however, for the processing to be implemented in any language that supports Windows named events (Mutex) and can open a named memory-mapped file. Nevertheless, no guarantee can be given as to the behavior of the system using anything but C/C++.

FFT

Why Use FFT?

For a large class of signals, you can gain greater insight by looking at spectral representation rather than time description. Signals encountered in the frequency response of amplifiers, oscillator phase noise and those in mechanical vibration analysis, for example, are easier to observe in the frequency domain.

If sampling is done at a rate fast enough to faithfully approximate the original waveform (usually five times the highest frequency component in the signal), the resulting discrete data series will uniquely describe the analog signal. This is of particular value when dealing with transient signals because, unlike FFT, conventional swept spectrum analyzers cannot handle them.

Spectral analysis theory assumes that the signal for transformation is of infinite duration. Since no physical signal can meet this condition, a useful assumption for reconciling theory and practice is to view the signal as consisting of an infinite series of replicas of itself. These replicas are multiplied by a rectangular window (the display grid) that is zero outside of the observation grid.

An FFT operation on an N-point time domain signal can be compared to passing the signal through a comb filter consisting of a bank of N/2 filters. All the filters have the same shape and width and are centered at N/2 discrete frequencies. Each filter collects the signal energy that falls into the immediate neighborhood of its center frequency. Thus it can be said that there are N/2 frequency bins. The distance in Hz between the center frequencies of two neighboring bins is always the same: Delta f.

Power (Density) Spectrum

Because of the linear scale used to show magnitudes, lower amplitude components are often hidden by larger components. In addition to the functions offering magnitude and phase representations, the FFT option offers power density and power spectrum density functions. These latter functions are even better suited for characterizing spectra. The power spectrum (V^2) is the square of the magnitude spectrum (0 dBm corresponds to voltage equivalent to 1 mW into 50 ohms.) This is the representation of choice for signals containing isolated peaks — periodic signals, for instance.

The power density spectrum (V^2/Hz) is the power spectrum divided by the equivalent noise bandwidth of the filter associated with the FFT calculation. This is best employed for characterizing broadband signals such as noise.

Memory for FFT

The amount of acquisition memory available will determine the maximum range (Nyquist frequency) over which signal components can be observed. Consider the problem of determining the length of the observation window and the size of the acquisition buffer if a Nyquist rate of 500 MHz and a resolution of 10 kHz are required. To obtain a resolution of 10 kHz, the acquisition time must be at least:

T = 1/Delta f = 1/10 kHz = 100 ms

For a digital oscilloscope with a memory of 100 kB, the highest frequency that can be analyzed is:

Delta f x N/2 = 10 kHz x 100 kB/2 = 500 MHz

FFT Pitfalls to Avoid

Take care to ensure that signals are correctly acquired: improper waveform positioning within the observation window produces a distorted spectrum. The most common distortions can be traced to insufficient sampling, edge discontinuities, windowing or the "picket fence" effect.

Because the FFT acts like a bank of band-pass filters centered at multiples of the frequency resolution, components that are not exact multiples of that frequency will fall within two consecutive filters. This results in an attenuation of the true amplitude of these components.

Picket Fence and Scallop

The highest point in the spectrum can be 3.92 dB lower when the source frequency is halfway between two discrete frequencies. This variation in spectrum magnitude is the picket fence effect. The corresponding attenuation loss is referred to as scallop loss. LeCroy scopes automatically correct for the scallop effect, ensuring that the magnitude of the spectra lines correspond to their true values in the time domain.

If a signal contains a frequency component above Nyquist, the spectrum will be aliased, meaning that the frequencies will be folded back and spurious. Spotting aliased frequencies is often difficult, as the aliases may ride on top of real harmonics. A simple way of checking is to modify the sample rate and observe whether the frequency distribution changes.

Leakage

FFT assumes that the signal contained within the time grid is replicated endlessly outside the observation window. Therefore if the signal contains discontinuities at its edges, pseudo-frequencies will appear in the spectral domain, distorting the real spectrum. When the start and end phase of the signal differ, the signal frequency falls within two frequency cells, broadening the spectrum.

The broadening of the base, stretching out in many neighboring bins, is termed leakage. Cures for this are to ensure that an integral number of periods is contained within the display grid or that no discontinuities appear at the edges. Another is to use a window function to smooth the edges of the signal.

Choosing a Window

The choice of a spectral window is dictated by the signal's characteristics. Weighting functions control the filter response shape, and affect noise bandwidth as well as side lobe levels. Ideally, the main lobe should be as narrow and flat as possible to effectively discriminate all spectral components, while all side lobes should be infinitely attenuated. The window type defines the bandwidth and shape of the equivalent filter to be used in the FFT processing.

In the same way as one would choose a particular camera lens for taking a picture, some experimenting is generally necessary to determine which window is most suitable. However, the following general guidelines should help.

Rectangular windows provide the highest frequency resolution and are thus useful for estimating the type of harmonics present in the signal. Because the rectangular window decays as a (sinx)/x function in the spectral domain, slight attenuation will be induced. Alternative functions with less attenuation (Flat Top and Blackman-Harris) provide maximum amplitude at the expense of frequency resolution. Whereas, Hamming and Von Hann are good for general purpose use with continuous waveforms.

Window Type	Applications and Limitations
Rectangular	These are normally used when the signal is transient (completely contained in the time-domain window) or known to have a fundamental frequency component that is an integer multiple of the fundamental frequency of the window. Signals other than these types will show varying amounts of spectral leakage and scallop loss, which can be corrected by selecting another type of window.
Hanning (Von Hann)	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Hamming	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Flat Top	This window provides excellent amplitude accuracy with moderate reduction of leakage, but with reduced frequency resolution.
Blackman–Harris	It reduces the leakage to a minimum, but with reduced frequency resolution.

FFT Window Filter Parameters				
Window Type	**Highest Side Lobe (dB)**	**Scallop Loss (dB)**	**ENBW (bins)**	**Coherent Gain (dB)**
Rectangular	-13	3.92	1.0	0.0
von Hann	-32	1.42	1.5	-6.02
Hamming	-43	1.78	1.37	-5.35

Flat Top	-44	0.01	2.96	-11.05
Blackman-Harris	-67	1.13	1.71	-7.53

Improving Dynamic Range

Enhanced resolution uses a low-pass filtering technique that can potentially provide for three additional bits (18 dB) if the signal noise is uniformly distributed (white). Low-pass filtering should be considered when high frequency components are irrelevant. A distinct advantage of this technique is that it works for both repetitive and transient signals. The SNR increase is conditioned by the cut-off frequency of the ERES low-pass filter and the noise shape (frequency distribution).

LeCroy digital oscilloscopes employ FIR digital filters so that a constant phase shift is maintained. The phase information is therefore not distorted by the filtering action.

Record Length

Because of its versatility, FFT analysis has become a popular analysis tool. However, some care must be taken with it. In most instances, incorrect positioning of the signal within the display grid will significantly alter the spectrum. Effects such as leakage and aliasing that distort the spectrum must be understood if meaningful conclusions are to be arrived at when using FFT.

An effective way to reduce these effects is to maximize the acquisition record length. Record length directly conditions the effective sampling rate of the scope and therefore determines the frequency resolution and span at which spectral analysis can be carried out.

FFT Algorithms

A summary of the algorithms used in the oscilloscope's FFT computation is given here in a few steps:

1. The data are multiplied by the selected window function.
2. FFT is computed, using a fast implementation of the DFT (Discrete Fourier Transform):

$$X_n = \frac{1}{N}\sum_{k=0}^{k=N-1} x_k \times W^{nk}$$

   where: x_k is a complex array whose real part is the modified source time domain waveform, and whose imaginary part is 0; X_n is the resulting complex frequency-domain waveform; $W = e^{-2\pi i/N}$; and N is the number of points in x_k and X_n.

   The generalized FFT algorithm, as implemented here, works on N, which need *not* be a power of 2.

3. The resulting complex vector X_n is divided by the coherent gain of the window function, in order to compensate for the loss of the signal energy due to windowing. This compensation provides accurate amplitude values for isolated spectrum peaks.
4. The real part of X_n is symmetric around the Nyquist frequency, that is

$$R_n = R_{N-n}$$

while the imaginary part is asymmetric, that is

$I_n = -I_{N-n}$

The energy of the signal at a frequency *n* is distributed equally between the first and the second halves of the spectrum; the energy at frequency 0 is completely contained in the 0 term.

The first half of the spectrum (Re, Im), from 0 to the Nyquist frequency is kept for further processing and doubled in amplitude:

$R'_n = 2 \times R_n\_0\ \ n < N/2$

$I'_n = 2 \times I_n\_\_0\ \ n < N/2$

5. The resultant waveform is computed for the spectrum type selected.

   If "Magnitude" is selected, the magnitude of the complex vector is computed as:

$$M_n = \sqrt{R'^2_n + I'^2_n}$$

Steps 1–5 lead to the following result:

An AC sine wave of amplitude 1.0 V with an integral number of periods N_p in the time window, transformed with the rectangular window, results in a fundamental peak of 1.0 V magnitude in the spectrum at frequency N_p x Delta f. However, a DC component of 1.0 V, transformed with the rectangular window, results in a peak of 2.0 V magnitude at 0 Hz.

The waveforms for the other available spectrum types are computed as follows:

Phase: angle = arctan $(I_n/R_n)\_M_n > M_{min}$

angle = 0 $M_n\ \ M_{min}$

Where M_{min} is the minimum magnitude, fixed at about 0.001 of the full scale at any gain setting, below which the angle is not well defined.

The dBm Power Spectrum:

$$dBm\ PS = 10 \times \log_{10}\left(\frac{M_n^2}{M_{ref}^2}\right) = 20 \times \log_{10}\left(\frac{M_n}{M_{ref}}\right)$$

where M_{ref} = 0.316 V (that is, 0 dBm is defined as a sine wave of 0.316 V peak or 0.224 V rms, giving 1.0 mW into 50 ohms).

The dBm Power Spectrum is the same as dBm Magnitude, as suggested in the above formula.

dBm Power Density:

$$dBm\,PD = dBm\,PS - 10 \times \log_{10}\left(ENBW \times \Delta f\right)$$

where *ENBW* is the equivalent noise bandwidth of the filter corresponding to the selected window, and *Delta f* is the current frequency resolution (bin width).

6. The FFT Power Average takes the complex frequency-domain data R'_n and I'_n for each spectrum generated in Step 5, and computes the square of the magnitude:

$$M_n^2 = R'^2_n + I'^2_n,$$

then sums M_n2 and counts the accumulated spectra. The total is normalized by the number of spectra and converted to the selected result type using the same formulas as are used for the Fourier Transform.

Glossary

This section defines the terms frequently used in FFT spectrum analysis and relates them to the oscilloscope.

Aliasing If the input signal to a sampling acquisition system contains components whose frequency is greater than the Nyquist frequency (half the sampling frequency), there will be less than two samples per signal period. The result is that the contribution of these components to the sampled waveform is indistinguishable from that of components below the Nyquist frequency. This is **aliasing**.

The timebase and transform size should be selected so that the resulting Nyquist frequency is higher than the highest significant component in the time-domain record.

Coherent Gain The normalized coherent gain of a filter corresponding to each window function is 1.0 (0 dB) for a rectangular window and less than 1.0 for other windows. It defines the loss of signal energy due to the multiplication by the window function. This loss is compensated for in the oscilloscope. The following table lists the values for the implemented windows.

Window Frequency Domain Parameters				
Window Type	Highest Side Lobe (dB)	Scallop Loss (dB)	ENBW (bins)	Coherent Gain (dB)
Rectangular	-13	3.92	1.0	0.0
Hanning (Von Hann)	-32	1.42	1.5	-6.02
Hamming	-43	1.78	1.37	-5.35
Flattop	-44	0.01	2.96	-11.05
Blackman–Harris	-67	1.13	1.71	-7.53

ENBW Equivalent Noise BandWidth (ENBW) is the bandwidth of a rectangular filter (same gain at the center frequency), equivalent to a filter associated with each frequency bin, which would collect the same power from a white noise signal. In the table on the previous page, the ENBW is listed for each window function implemented, given in bins.

Filters Computing an N-point FFT is equivalent to passing the time-domain input signal through N/2 filters and plotting their outputs against the frequency. The spacing of filters is Delta f = 1/T, while the bandwidth depends on the window function used (see Frequency Bins).

Frequency Bins The FFT algorithm takes a discrete source waveform, defined over N points, and computes N complex Fourier coefficients, which are interpreted as harmonic components of the input signal.

For a real source waveform (imaginary part equals 0), there are only N/2 independent harmonic components.

An FFT corresponds to analyzing the input signal with a bank of N/2 filters, all having the same shape and width, and centered at N/2 discrete frequencies. Each filter collects the signal energy that falls into the immediate neighborhood of its center frequency. Thus it can be said that there are N/2 "frequency bins."

The distance in hertz between the center frequencies of two neighboring bins is always:

Delta f = 1/T

where T is the duration of the time-domain record in seconds.

The width of the main lobe of the filter centered at each bin depends on the window function used. The rectangular window has a nominal width at 1.0 bin. Other windows have wider main lobes (see table).

Frequency Range The range of frequencies computed and displayed is 0 Hz (displayed at the left-hand edge of the screen) to the Nyquist frequency (at the rightmost edge of the trace).

Frequency Resolution In a simple sense, the frequency resolution is equal to the bin width Delta f. That is, if the input signal changes its frequency by Delta f, the corresponding spectrum peak will be displaced by Df. For smaller changes of frequency, only the shape of the peak will change.

However, the effective frequency resolution (that is, the ability to resolve two signals whose frequencies are almost the same) is further limited by the use of window functions. The ENBW value of all windows other than the rectangular is greater than Delta f and the bin width. The table of Window Frequency-Domain Parameters lists the ENBW values for the implemented windows.

Leakage In the power spectrum of a sine wave with an integral number of periods in the (rectangular) time window (that is, the source frequency equals one of the bin frequencies), the spectrum contains a sharp component whose value accurately reflects the source waveform's amplitude. For intermediate input frequencies this spectral component has a lower and broader peak.

The broadening of the base of the peak, stretching out into many neighboring bins, is termed *leakage*. It is due to the relatively high side lobes of the filter associated with each frequency bin.

The filter side lobes and the resulting leakage are reduced when one of the available window functions is applied. The best reduction is provided by the Blackman–Harris and Flattop windows. However, this reduction is offset by a broadening of the main lobe of the filter.

Number of Points The FFT is computed over the number of points (Transform Size) whose upper bounds are the source number of points, and by the maximum number of points selected in the menu. The FFT generates spectra of N/2 output points.

Nyquist Frequency The Nyquist frequency is equal to one half of the effective sampling frequency (after the decimation): Delta f x N/2.

Picket Fence Effect If a sine wave has a whole number of periods in the time domain record, the power spectrum obtained with a rectangular window will have a sharp peak, corresponding exactly to the frequency and amplitude of the sine wave. Otherwise the spectrum peak with a rectangular window will be lower and broader.

The highest point in the power spectrum can be 3.92 dB lower (1.57 times) when the source frequency is halfway between two discrete bin frequencies. This variation of the spectrum magnitude is called the *picket fence effect* (the loss is called the scallop loss).

All window functions compensate for this loss to some extent, but the best compensation is obtained with the Flattop window.

Power Spectrum The power spectrum (V^2) is the square of the magnitude spectrum.

The power spectrum is displayed on the dBm scale, with 0 dBm corresponding to:

$$V_{ref}^2 = (0.316\ V_{peak})^2,$$

where V_{ref} is the peak value of the sinusoidal voltage, which is equivalent to 1 mW into 50 ohms.

Power Density Spectrum The power density spectrum (V^2/Hz) is the power spectrum divided by the equivalent noise bandwidth of the filter, in hertz. The power density spectrum is displayed on the dBm scale, with 0 dBm corresponding to (V_{ref}^2/Hz).

Sampling Frequency The time-domain records are acquired at sampling frequencies dependent on the selected time base. Before the FFT computation, the time-domain record may be decimated. If the selected maximum number of points is lower than the source number of points, the effective sampling frequency is reduced. The effective sampling frequency equals twice the Nyquist frequency.

Scallop Loss This is loss associated with the picket fence effect.

Window Functions All available window functions belong to the sum of cosines family with one to three non-zero cosine terms:

$$W_k = \sum_{m=0}^{m=M-1} a_m \cos\left(\frac{2 p k}{N} m\right) \qquad 0 \le k < N$$

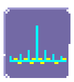
where: $M = 3$ is the maximum number of terms, a_m are the coefficients of the terms, N is the number of points of the decimated source waveform, and k is the time index.

The table of Coefficients of Window Functions lists the coefficients a_m. The window functions seen in the time domain are symmetric around the point k = N/2.

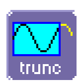
Coefficients of Window Functions			
Window Type	a_0	a_1	a_2
Rectangular	1.0	0.0	0.0
Hanning (Von Hann)	0.5	-0.5	0.0
Hamming	0.54	-0.46	0.0
Flattop	0.281	-0.521	0.198
Blackman–Harris	0.423	-0.497	0.079

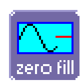
FFT Setup

To Set Up an FFT

1. In the menu bar touch **Math**, then **Math Setup...** in the drop-down menu.
2. Touch a Math function trace button: **F1** through **Fx**; a pop-up menu appears. Select **FFT**  from the menu.
3. Touch the **Single** 

or **Dual** (function of a function) 

button if the FFT is to be of the result of another math operation.
4. Touch inside the **Source1** field and select a channel, memory, or math trace on which to perform the FFT.
5. Touch inside the **Operator1** field: Select **FFT** from the pop-up menu if you selected **Single** function. Select another math function if you selected **Dual** function. Then touch inside the **Operator2** field and select **FFT** from the pop-up menu.
6. In the right-hand dialog, touch the **FFT** tab.
7. Choose whether to **Truncate**[1] 

or **Zero-fill**[2] 

the trace display.

---

[1] When the FFT transform size does not match the record length, you can truncate the record and perform an FFT on the shorter record. This will increase the resolution bandwidth.

[2] Zero-fill is useful when the source data for the FFT comes from a math operation that shortens the record. This is commonly encountered in filtering operations like enhanced resolution. The missing data points are replaced by data values, whose amplitudes are interpolated to fit between the last data point and th first data point in the record. This guarantees that there is not a first-order discontinuity in the filled data. Since the data at the end of the record is "filled" data, it is advisable to select a weighting window other than rectangular to minimize the effect of the fill on the resulting spectrum.

8. Touch the **Suppress DC** checkbox if you want to make the DC bin go to zero. Otherwise, leave it unchecked.
9. Touch inside the **Output** type field, and make a selection from the pop-up menu.
10. Touch inside the **Window** field, select a window type.
11. Touch inside the **Algorithm** field and select either **Least Prime**[3] or **Power of 2**[4] from the pop-up menu.

[3] The default algorithm is a least primes algorithm that computes FFTs on transform sizes having lengths that can be expressed as factors of 2^N*5^K. This is very compatible with the record lengths encountered in the oscilloscope, which are often multiples of 1, 2, 4, 5, or 10.

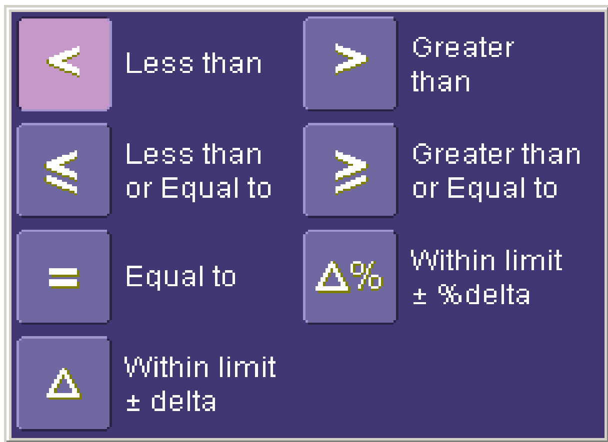
[4] The other choice is a power of two algorithm where the record lengths are in the form of 2^N. The power of 2 algorithm generally runs faster than the least primes algorithm. The price that is paid is a record length that is not the same as the acquired signal. The power-of-two FFT uses the first 2^N points of the record. For example, if you acquire 500 points in your trace, the power-of-two FFT would only use the first 256 points.

Pass/Fail Testing

Comparing Parameters

Each Pass/Fail input (**Qx**) can compare a different parameter result to a user-defined limit (or statistical range) under a different condition.

The conditions are represented by these comparison operators:



At the touch of a button, test results can also be compared to these standard statistical limits:

- current mean
- mean + 1 SD
- mean + 3 SD

In Dual Parameter Compare mode, your X-Stream scope gives you the option to compare to each other parameter results measured on two different waveforms. You can set your test to be true if Any waveform or All waveforms fit the criterion stipulated by the comparison condition. Your setup is conveniently shown in the Summary box of the **Qx** dialog. For example:

Mask Tests

You have the choice to do mask testing by using an existing mask, or by using a mask created from your actual waveform, with vertical and horizontal tolerances that you define. Existing masks can be loaded from a floppy disk or from a network.

You can set your mask test to be True for waveforms All In, All Out, Any In, or Any Out. For example, if you select All In, the test will be False if even a single waveform falls outside the mask.

Masks that you create from your waveform can be confined to just a portion of the trace by use of a measure gate. (See Measure Gate for an explanation of how this feature works.)

Actions

Setting Up Pass/Fail Actions

By touching the **Stop Test** checkbox 

in the "Actions" dialog, you can set up the test to end after a predetermined number of sweeps that you decide.

You can also decide the actions to occur upon your waveforms' passing or failing, by selecting one or all of the following:

- stop
- audible alarm
- print image of display
- emit pulse
- save waveform

The selection **Pulse** 

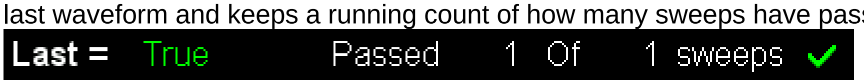
causes a pulse to be output through the Aux Out connector at the front of the scope. This pulse can be used to trigger another scope. You can set the amplitude and width of the pulse as described in Auxiliary Output Signals.

Depending on your scope model, you can configure up to 8 pass/fail conditions. The Boolean conditions to determine if your waveform passes are as follows:

All True	All False
Any True	Any False
All Q1 to Q4 **Or** All Q5 to Q8	Any Q1 to Q4 **And** Any Q5 to Q8

Setting Up Pass/Fail Testing

Initial Setup

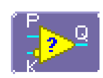
1. Touch **Analysis** in the menu bar, then **Pass/Fail Setup...** in the drop-down menu.
2. Touch the **Actions** tab.
3. Touch the **Enable Actions** checkbox. This will cause the actions that you will select to occur upon your waveform's passing or failing a test.
4. Touch the **Summary View** to enable a line of text that shows concisely the status of your last waveform and keeps a running count of how many sweeps have passed.


5. Touch inside the **Pass If** field, and select a Boolean condition from the pop-up menu.
6. If you want to set up the test to end after a finite number of sweeps, touch the **Stop Test** checkbox. Then touch inside the **After** data entry field and enter a value, using the pop-up numeric keypad.
7. Under "If", touch either the **Pass** 

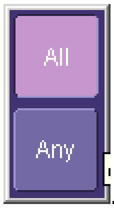
or **Fail** 

button to set the actions to occur upon your waveform's passing or failing the test.
8. Under "Then", touch the actions you want to occur: stop test, sound alarm, print result, emit pulse, or save the waveform. If you want to have the results printed and your scope is not equipped with a printer, be sure that the it is connected to a local or network printer. See Printing.
9. If you want to save your waveform automatically, touch the **Save Setup**. This will take you out of the current dialog and will open the "Save Waveform" dialog. See Saving and Recalling Waveforms.
10. Test your Pass/Fail conditions by touching the **Force Actions Once** button. Press the **Clear All** button to quickly uncheck all checkboxes if you want to change your selections.

Comparing a Single Parameter

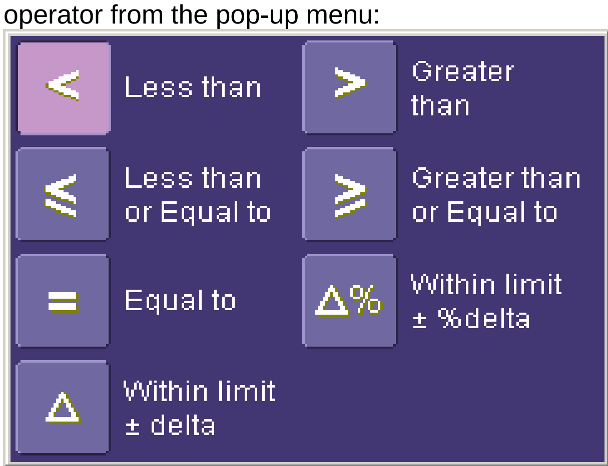
1. Touch **Analysis** in the menu bar, then **Pass/Fail Setup...** in the drop-down menu.
2. Touch a **Qx** tab; a setup dialog for that position will open.
3. Touch inside the **Source1** field and select a source from the pop-up menu.
4. Touch inside the **Condition** field in the main dialog and select **ParamCompare** 

.
5. Touch inside the **Compare Values** field and select **All** or **Any** from the pop-up menu



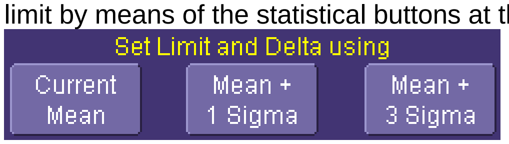
. By selecting **All**, the test will be true only if every waveform falls within the limit that you will set. By selecting **Any**, the test will be true if just one waveform falls within the limit.

6. Touch inside the **Condition** field in the "ParamCompare" mini-dialog and select a math operator from the pop-up menu:



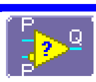
.

7. Touch inside the **Limit** field and enter a value, using the pop-up numeric keypad. This value takes the dimensions of the parameter that you are testing. For example, if you are testing a time parameter, the unit is seconds. If you chose either **WithinDeltaPct**  or **WithinDeltaAbs**  from the **Condition** menu, you also have the choice of setting the limit by means of the statistical buttons at the bottom of the "ParamCompare" dialog:

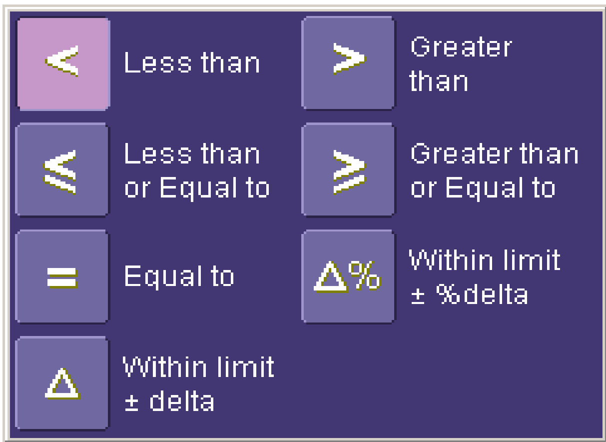


Comparing Dual Parameters

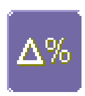
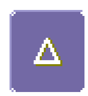
1. Touch **Analysis** in the menu bar, then **Pass/Fail Setup...** in the drop-down menu.
2. Touch a **Qx** tab; a setup dialog for that position will open.
3. Touch inside the **Condition** field in the main dialog and select **DualParamCompare**


.

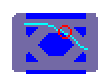
4. Touch inside the **Source1** and **Source2** fields and select a source from the pop-up menu.
5. Touch inside the "ParamCompare" mini-dialog field and select a source from the pop-up menu.
6. Touch inside the **Compare Values** field and select **All** or **Any** from the pop-up menu. By selecting **All**, the test will be true only if every waveform falls within the limit that you will set. By selecting **Any**, the test will be true if just one waveform falls within the limit.
7. Touch inside the **Condition** field in the "ParamCompare" mini-dialog and select a math operator from the pop-up menu:

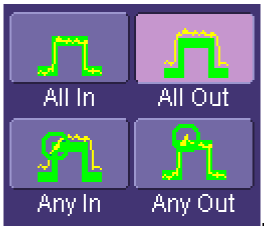


.

8. Touch inside the **Limit** field and enter a value, using the pop-up numeric keypad. This value takes the dimension of the parameter that you are testing. For example, if you are testing a time parameter, the unit is seconds.
9. If you chose either **WithinDeltaPct**  or **WithinDeltaAbs**  from the **Condition** menu, touch inside the **Delta** field and enter a value.

Mask Testing

1. Touch **Analysis** in the menu bar, then **Pass/Fail Setup...** in the drop-down menu.
2. Touch a **Qx** tab; a setup dialog for that position will open.
3. Touch inside the **Source1** field and select a source from the pop-up menu.
4. Touch inside the **Condition** field in the main dialog and select **Mask Test**  .
5. From the "Test" mini-dialog, make a selection in the **Test is True when** group of buttons



.

This selection means, for example, that if you select **All In** the test will be False if even a single waveform falls outside the mask.

6. From **Show Markers**, choose whether or not to have mask violations displayed.
7. If you are loading a pre-existing mask, touch the **Load Mask** tab, then the **File** button. You can then enter the file name or browse to its location.
8. If you want to make a mask from your waveform, touch the **Make Mask** tab.
9. Touch inside the **Ver Delta** and **Hor Delta** fields and enter boundary values, using the pop-up numeric keypad.
10. Touch the **Browse** button to create a file name and location for the mask if you want to save it.
11. Touch the **Gate** tab, then enter values in the **Start** and **Stop** fields to constrain the mask to a portion of the waveform. Or, you can simply touch and drag the Gate posts, which initially are placed at the extreme left and right ends of the grid.

Status

The status read-only dialog displays system information including serial number, firmware version, and installed software and hardware options.

To Access Status Dialog

1. In the menu bar, touch **Utilities**.
2. Touch the **Status** tab.

Remote communication

The Remote dialog is where you can select a network communication protocol, establish network connections, and configure the Remote Control Assistant log. The choice of communication protocols is limited to TCPIP and GPIB.

Note: GPIB is an option and requires a GPIB card to be installed in a card slot at the rear of the scope.

Note: The instrument uses Dynamic Host Configuration Protocol (DHCP) as its addressing protocol. Therefore, it is not necessary to set up an IP address if your network supports DHCP. If it does not, you can assign a static address in the standard Windows 2000 network setup menu.

The Remote Control Assistant monitors communication between your PC and scope when you are operating the instrument remotely. You can log all events, or errors only. This log can be invaluable when you are creating and debugging remote control applications.

To Set Up Remote Communication.

If you are connecting the scope to a network, first contact your Information Systems administrator. If you are connecting the scope directly to your PC, connect a GPIB or Ethernet cable between them.

1. In the menu bar touch **Utilities**, then **Utilities Setup...** in the drop-down menu.
2. Touch the **Remote** tab.
3. Make a **Port** selection:**TCPIP** (transmission control protocol/Internet protocol) or **GPIB** (general purpose interface bus). If you do not have a GPIB card installed, the GPIB selection will not be accessible.
4. If you are using GPIB, set a GPIB address by touching inside the **GPIB Address** data entry field and enter an address.
5. Press the **Net Connections** button; the Windows **Network and Dial-up Connections** window appears.
6. Touch **Make New Connection** and use the Windows Network Connection Wizard to make a new connection; or, touch Local Area Connection to reconfigure the scope's connection if it is already connected to the network.

To Configure the Remote Control Assistant Event Log

1. In the menu bar touch **Utilities**, then **Utilities Setup...** in the drop-down menu.
2. Touch the **Remote** tab.

3. Touch inside the **Log Mode** data entry field.
4. Select **Off**, **Errors Only**, or **Full Dialog** from the pop-up menu.
5. To export the contents of the event log to an ASCII text file, touch the **Show Remote Control Log** button: the "Event Logs" pop-up window appears. Touch inside the **DestFilename** data entry field and enter a file name, using the pop-up keyboard. Then touch the **Export to Text File** button.

Hardcopy

Printing

For print setup, refer to Printing.

Clipboard

This selection prints to the clipboard so you can paste a file into another application (like MS Word, for example).

To Print from the Clipboard

1. In the menu bar touch **Utilities**, then **Utilities Setup...** in the drop-down menu.
2. Touch the **Hardcopy** tab.
3. Under **Colors**, touch the **Use Print Colors** checkbox if you want the traces printed on a white background. A white background saves printer toner.
4. Touch the **Grid Area Only** checkbox if you do not need to print the dialog area and you only want to show the waveforms and grids.
5. Touch the **Print Now** button.

File

Choose **File** if you want to output the screen image to storage media such as floppy drive or hard drive. When outputting to floppy disk, be sure to use a preformatted disk.

To Print to File

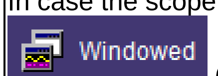
1. In the menu bar touch **Utilities**, then **Utilities Setup...** in the drop-down menu.
2. Touch the **Hardcopy** tab, then the **File** icon.
3. Touch inside the **File Format** data entry field and select a graphic file format from the pop-up menu.
4. Under **Colors**, touch the **Use Print Colors** checkbox if you want the traces printed on a white background. A white background saves printer toner.
5. Touch inside the **Directory** data entry field and type the path to the folder you want to print to, using the pop-up keyboard. Or touch the **Browse** button and navigate to the folder.
6. Touch inside the **File Name** data entry field and enter a name for the display image, using the pop-up keyboard.
7. Touch the **Grid Area Only** checkbox if you do not need to print the dialog area and you only want to show the waveforms and grids.
8. Touch the **Print Now** button.

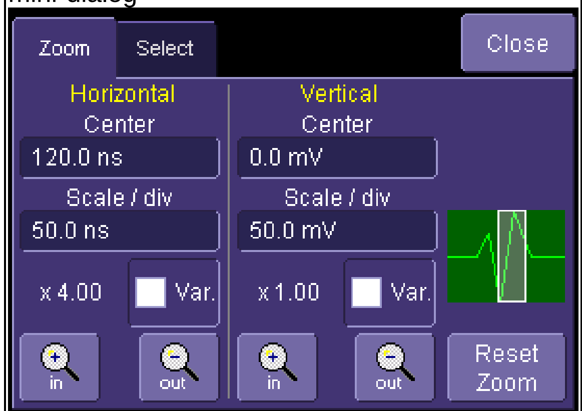
E-Mail

The instrument also gives you the option to e-mail your screen images, using either the MAPI or SMTP protocols. Before you output to e-mail from the Utilities dialog, you first have to set up the e-mail server and recipient address in **Preference Setup**.

To Send E-mail

1. In the menu bar touch **Utilities**, then **Utilities Setup...** in the drop-down menu.
2. Touch the **Hardcopy** tab, then the **E-mail** button.
3. Touch inside the **File Format** data entry field and select a graphic file format from the pop-up menu.
4. Under **Colors**, touch the **Use Print Colors** checkbox if you want the traces printed on a white background. A white background saves printer toner.
5. Touch the **Prompt for message to send with mail** checkbox if you want to include remarks with the image.
6. Touch inside the **Hardcopy Area** field and make a selection from the pop-up menu:

Grid Area Only	Sends only the waveforms and whatever annotation is contained in the grid.
DSO Window	Sends the grid area plus open dialogs.
Full Screen	In case the scope display is windowed  (reduced), sends the scope UI and revealed desktop.
Menu Only	Sends only the menu bar at the top of the scope.
Message Line Only	Sends only the status line at the bottom of the screen.

Docked Dialog Only	Sends only the dialog area of the screen, if a dialog is open (docked).
RHS Dialog Only	For Math setup displays, sends only the right-hand side mini-dialog  Example RHS dialog.

7. Touch the **Print Now** button  .

Date & Time

The instrument gives you the choice of manually setting the time and date or getting it from the Internet. If you elect to get the time and date from the Internet, you need to have the scope connected to the Internet through the LAN connector on the rear panel. You can also set time zones and daylight savings time.

To Set Time and Date Manually

1. In the menu bar touch **Utilities**, then **Utilities Setup...** in the drop-down menu.
2. Touch the **Date/Time** tab.
3. Touch inside each of the **Hour**, **Minute**, **Second**, **Day**, **Month**, and **Year** data entry fields and enter a value, using the pop-up numeric keypad.
4. Touch the **Validate Changes** button.

To Set Time and Date from the Internet

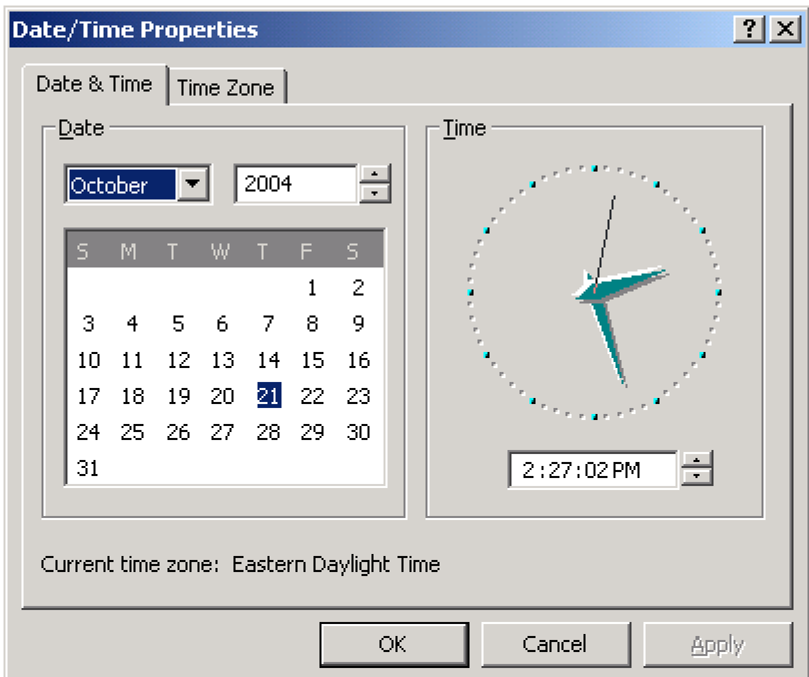
The Simple Network Time Protocol (SNTP) is used.

1. Ensure that the scope is connected to the Internet through the LAN connector at the rear of the scope.
2. In the menu bar touch **Utilities**, then **Utilities Setup...** in the drop-down menu.

3. Touch the **Date/Time** tab.
4. Touch the **Set from Internet** button.

To Set Time and Date from Windows

1. In the menu bar touch **Utilities**, then **Utilities Setup...** in the drop-down menu.
2. Touch the **Date/Time** tab.
3. Touch the **Windows Date/Time** button  .
4. Use the **Time & Date Properties** window to configure the time, including time zone.



Options

Use this dialog to add or remove software options. For information about software options, contact your local LeCroy Sales and Service office, or visit our Web site at http://www.lecroy.com/options.

Options that you purchase, such as JTA2, add performance to you instrument. This added performance is seen in the new math functions or parameters that you can choose from when doing Measure or Math setups.

Preferences

Audible Feedback

You can elect to have audible confirmation each time you touch a screen or front panel control.

1. In the menu bar touch **Utilities**; then touch **Preferences** in the drop-down menu.
2. Touch the "Audible Feedback" **Enable** checkbox so that the scope emits a beep with each touch of the screen or front panel control.

Auto-calibration

You can choose to have your instrument automatically recalibrate itself whenever there is a significant change in ambient temperature. If you do not enable this option, the scope will only recalibrate at startup and whenever you make a change to certain operating conditions.

1. In the menu bar touch **Utilities**; then touch **Preferences** in the drop-down menu.
2. Touch the "Automatic Calibration" **Enable** checkbox.

UI Language Selection

The "Language" field allows you to select a language, other than English, for the UI at any time. To change the current selection, touch inside the "Language" field and select a language from the pop-up menu.

Performance Optimization

You can set up the scope to optimize either calculating speed or display speed. If the display update rate is of primary concern to you, optimize for Display. If acquisition and analysis are more important, optimize for analysis. Optimizing for analysis can be useful when persistence or averaging is used, giving higher priority to waveform acquisition at the expense of display update rate.

The choices are presented as a spectrum with highest values at the extremes:

1. In the menu bar touch **Utilities**; then touch **Preferences** in the drop-down menu.
2. Touch one of the optimization icons.

Offset Control

As you change the gain, this control allows you to either keep the vertical offset level indicator stationary (when **Div** is selected) or to have it move with the actual voltage level (when **Volts** is selected). The advantage of selecting **Div** is that the waveform will remain on the grid as you increase the gain; whereas, if **Volts** is selected, the waveform could move off the grid.

> **Note:** Regardless of whether you select **Volts** or **Div**, the "Offset" shown in the channel setup dialog always indicates volts. However, when **Div** is selected for the Offset Control, the offset in volts is scaled proportional to the change in gain, thereby keeping the division on the grid constant.

1. In the menu bar touch **Utilities**; then touch **Preferences** in the drop-down menu.

2. Touch the **Acquisition** tab.
3. Under **Offset Setting constant in:**, touch either the **Div** or **Volts** button.

Delay Control

As you change the timebase, this control allows you to either keep the horizontal offset indicator stationary (when **Div** is selected) or to have it move with the trigger point (when **Time** is selected). The advantage of selecting **Div** is that the trigger point will remain on the grid as you increase the timebase; whereas, if **Time** is selected, the trigger point could move off the grid.

Note: Regardless of whether you select **Time** or **Div**, the "Delay" shown in the timebase setup dialog always indicates time. However, when **Div** is selected for Delay In, the delay in time is scaled proportional to the change in timebase, thereby keeping the division on the grid constant.

1. In the menu bar touch **Utilities**; then touch **Preferences** in the drop-down menu.
2. Touch the **Acquisition** tab.
3. Under **Delay Setting constant in:**, touch either the **Div** or **Volts** button.

E-mail

Before you can send e-mail from the scope, it must first be configured.

1. In the menu bar touch **Utilities**, then **Preference Setup...** in the drop-down menu.
2. Touch the **E-mail** tab.
3. Choose an e-mail server protocol: **MAPI** (Messaging Application Programming Interface) is the Microsoft interface specification that allows different messaging and workgroup applications (including e-mail, voice mail, and fax) to work through a single client, such as the Exchange client included with Windows 95 and Windows NT. MAPI uses the default Windows e-mail application (usually Outlook Express). **SMTP** (Simple Mail Transfer Protocol) is a TCP/IP protocol for sending messages from one computer to another through a network. This protocol is used on the Internet to route e-mail. In many cases no account is needed.
4. If you chose MAPI, touch inside the **Originator Address (From:)** data entry field and use the pop-up keyboard to type in the instrument's e-mail address. Then touch inside the **Default Recipient Address (To:)** data entry field and use the pop-up keyboard to enter the recipient's e-mail address.
5. If you chose SMTP, touch inside the **SMTP Server** data entry field and use the pop-up keyboard to enter the name of your server. Touch inside the **Originator Address (From:)** data entry field and use the pop-up keyboard to type in the instrument's e-mail address. Then touch inside the **Default Recipient Address (To:)** data entry field and use the pop-up keyboard to enter the recipient's e-mail address.
6. You can send a test e-mail text message by touching the **Send Test Mail** button. The test message reads "Test mail from [name of scope's email address]."

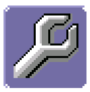
Acquisition Status

For each general category of scope operation, you can view a summary of your setups. These dialogs are not accessible through the Utilities menu, but are instead accessed from the menu bar drop-down menus. The categories are as follows:

- Vertical -- select **Channels Status . . .** from drop-down menu
- Timebase -- select **Acquisition Status . . .** from drop-down menu
- Trigger -- select **Acquisition Status . . .** from drop-down menu
- Math -- select **Math Status . . .** from drop-down menu

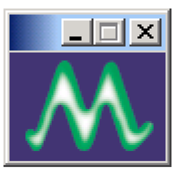
In addition to these dialogs, summaries are also provided for XY setups, memory (M1-M4) setups, and time stamps for sequence mode sampling.

Service

This button provides access to service dialogs, which are for the sole use of LeCroy service personnel. A security code is required to gain access.

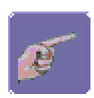
Show Windows Desktop

Touching the **Show Windows Desktop** button  in the main "Utilities" dialog minimizes the instrument application to reveal the underlying desktop. To maximize the application, touch the shortcut icon:



.

Touch Screen Calibration

Touching the **Touch-Screen Calibration** button  starts the calibration procedure. During the procedure, you will be prompted to touch the center of a small cross in 5 key locations on the touch screen. Because sufficient accuracy cannot be achieved using your finger, use a stylus instead for this procedure. The calibration has a ten-second timeout in case no cross is touched.

To avoid parallax errors, be sure to place your line of sight directly in front of each cross before touching it.

CUSTOMIZATION

Customizing Your Instrument

The instrument provides powerful capability to add your own parameters, functions, display algorithms, or other routines to the scope user interface without having to leave the instrument application environment. You can customize the instrument to your needs by using the power of programs such as Excel™, Mathcad™, and MATLAB™, or by scripting in VBS. Whichever method you use, the results appear on the instrument's display together with the signals that you started with. This ability offers tremendous advantages in solving unique problems for a large range of applications, with comparatively little effort from you.

Caution

Accessing the scope's automation interface from within an XDEV custom processor (VBScript, Mathcad, MATLAB, Excel, etc.) is NOT recommended.

Cases where the scope's behavior cannot be guaranteed, or worse, cases which can cause the scope's software to crash include the following:

1. Changing "Upstream" Controls

'Upstream' controls are considered to be any control that, if changed, could provoke an infinite loop. An example would be a VBScript processor, in F1, which uses C1 as a source. If this processor changes the offset or vertical scale of C1 as a result of examining its input (C1) data, an infinite loop could occur. This could eventually cause the scope software to crash. This is not limited only to changing upstream channel controls, but includes any upstream processing also.

2. Accessing Other Results

Access to results (waveform, measurement, etc.), other than the scripting processor's own inputs, may cause incorrect measurements. The reason for this is simple: the scope contains a complex algorithm to determine in which order results are computed. This algorithm ensures that all inputs required by a processor are computed before the processor itself. If a VBScript processor decides to access, via automation, results other than those supplied to its inputs, the scope's dependency algorithm cannot be used. Therefore, the results accessed may not be coherent (they may be from a previous acquisition, or worse, could cause an infinite computation loop).

3. Reconfiguring Math or Measurements

Reconfiguring (adding or removing) Math and/or Measurements from within a custom processing function is not recommended, especially when the reconfiguration would cause the custom processor to remove itself.

Introduction

Instrument customization provides these important capabilities:

- You can export data to programs, without leaving the instrument environment.
- You can get results back from those programs, and display them on the instrument, without leaving the instrument application environment.
- Once the result is returned, you can perform additional scope operations, such as measuring with cursors, applying parameters, or performing additional functions on the waveform, in exactly the same way as for a normal waveform.
- You can program the scope yourself.

The instrument does not just provide connectivity with data downloads to other programs. It provides true customizable interaction with these other programs, and allows you to truly customize the scope to do the exact job you want it to do. The advantages to this are many:

- You can use the standard processing power of the instrument to do most of your calculations
- You only need to write the function, parameter, display algorithm, etc. that specifically applies to your need and that the instrument doesn't contain.
- You can view the final result on the instrument display, and use all of the instrument's tools to understand the result.
- You can do additional processing on the result by applying either standard instrument parameters, functions, etc. to the returned result, or even more powerfully, adding chained customized functions. For example, you can do an Excel calculation on a result with a MATLAB function applied to it.

Solutions

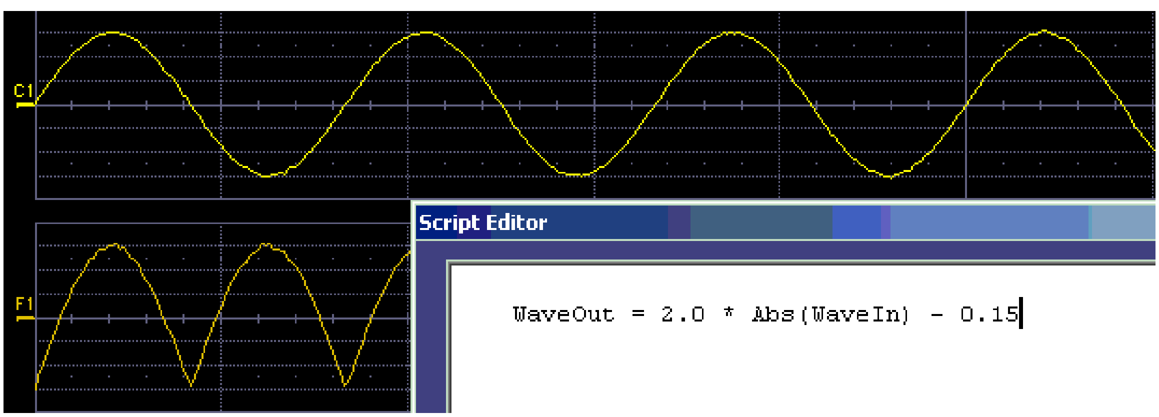
Engineers do not buy equipment; they buy solutions. But what solutions can be reached from a set of instrument waveform data? In principle, anything that can be logically derived from those data, given the limitations of signal-to-noise ratio and processing time. Here are some examples of what can be done with a customized instrument:

- Changing the units of a grid to joules, newtons, amps, etc.
- Creating a new waveform by manipulating the data of one or two input waveforms
- Creating a new waveform without using any of the input data
- Creating a new parameter by manipulating the data of one or two input waveforms
- Changing a vertical scale or a horizontal scale from linear to non-linear

You don't have to use all the data from the input waveforms: you can select data from one or more segments, which need not be aligned in the two-input waveforms.

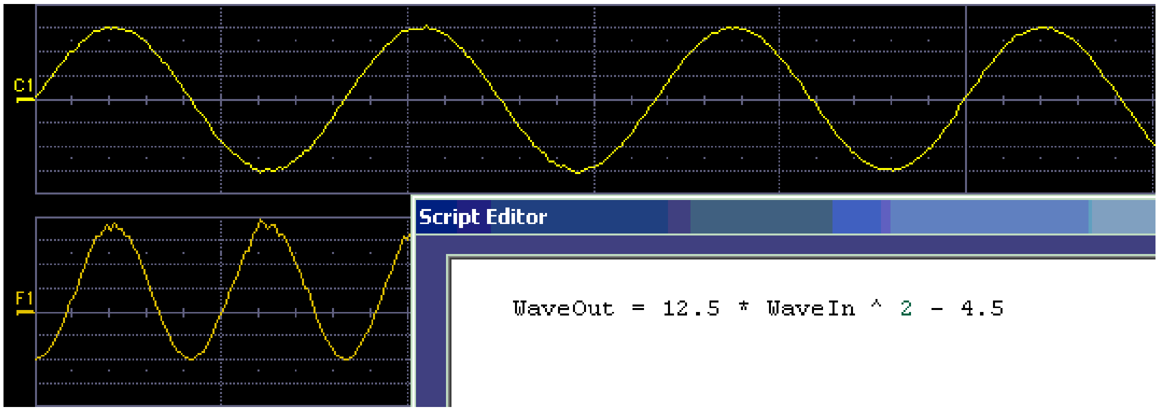
Examples

Example 1: Simple math functions using VBScript



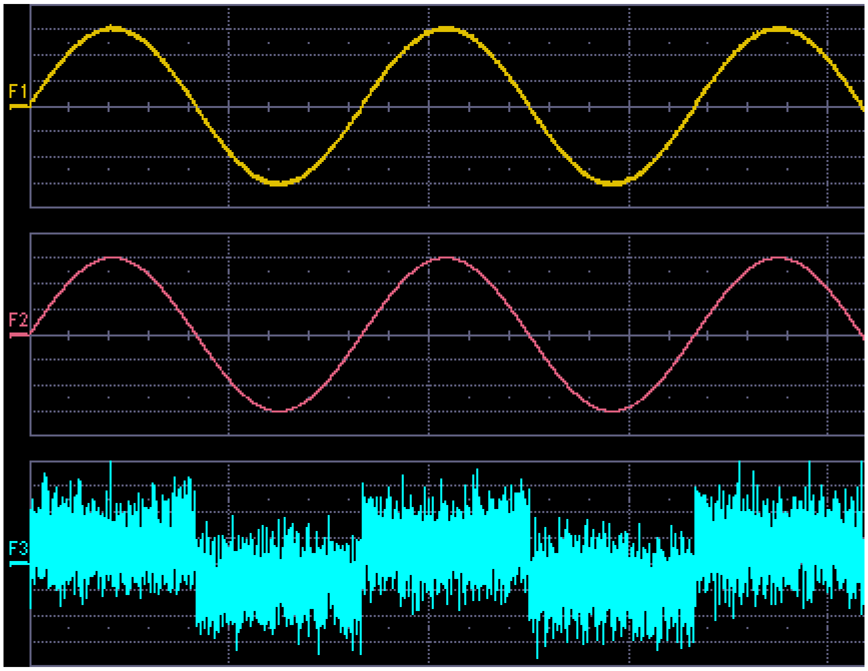
WaveOut is the waveform being returned to the instrument (F1 in this case). WaveIn is the input waveform (C1 in this case) You can see that the F1 result is displayed on the scope, and can be processed further.

Example 2: Another simple math functions using VBScript



Example 3 below doesn't use the input data at all. The middle waveform (F2) is a "golden waveform", in this case a perfect sine (subject to 16-bit resolution), that was created using a VBScript. The lower trace (F3) is a subtraction of the acquired waveform (upper trace) and the golden waveform. The subtraction (of course) contains all the noise, but it also shows the presence of a very small square wave signal.

Example 3



Here is the VBScript that produced the "golden sine" (F2 above):

```
Frequency = 3000000.0   ' Frequency of real data
SampleTime = InResult.HorizontalPerStep
Omega = 2.0 * 3.1416 * Frequency * SampleTime
Amplitude = 0.15        ' Amplitude of real data
    For K = 0 To LastPoint
    newDataArray(K) = Amplitude * Sin(Omega * K)
    Next
OutResult.DataArray(True) = newDataArray ' Data in volts
```

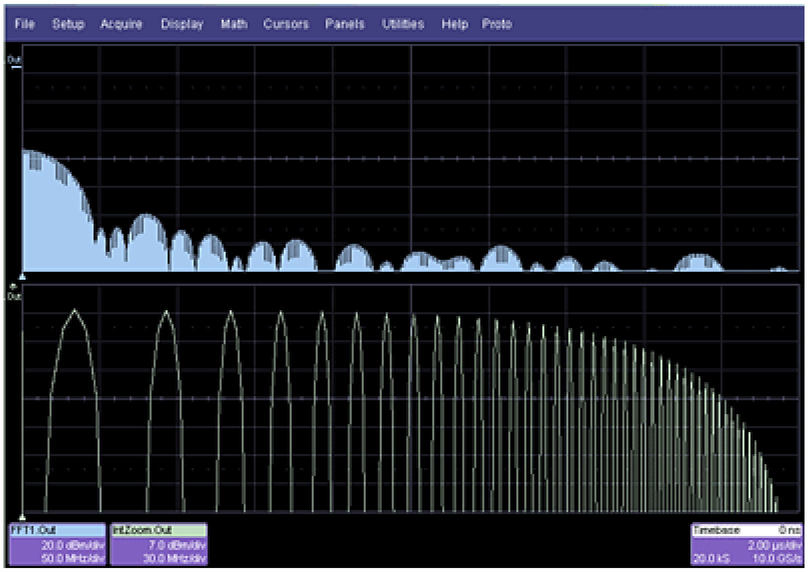
OutResult.DataArray is the waveform returned to the scope and displayed on the scope as the F2 waveform.

Example 4

Example 4 is a measurement of DVI (Digital Video Interface) Data-Clock skew jitter measurement, using a VBScript to emulate the PLL.

In this example, a customer was not able to probe the desired clock signal. The only probing point available was the output differential clock signal (C2). However, that clock was a factor of 10 slower than the clock embedded in the data signal (C3). By using a VBScript to create a clock waveform of the appropriate frequency (waveform F1), the customer was able to display and measure data-clock skew using a LeCroy instrument function and parameter.

Example 5

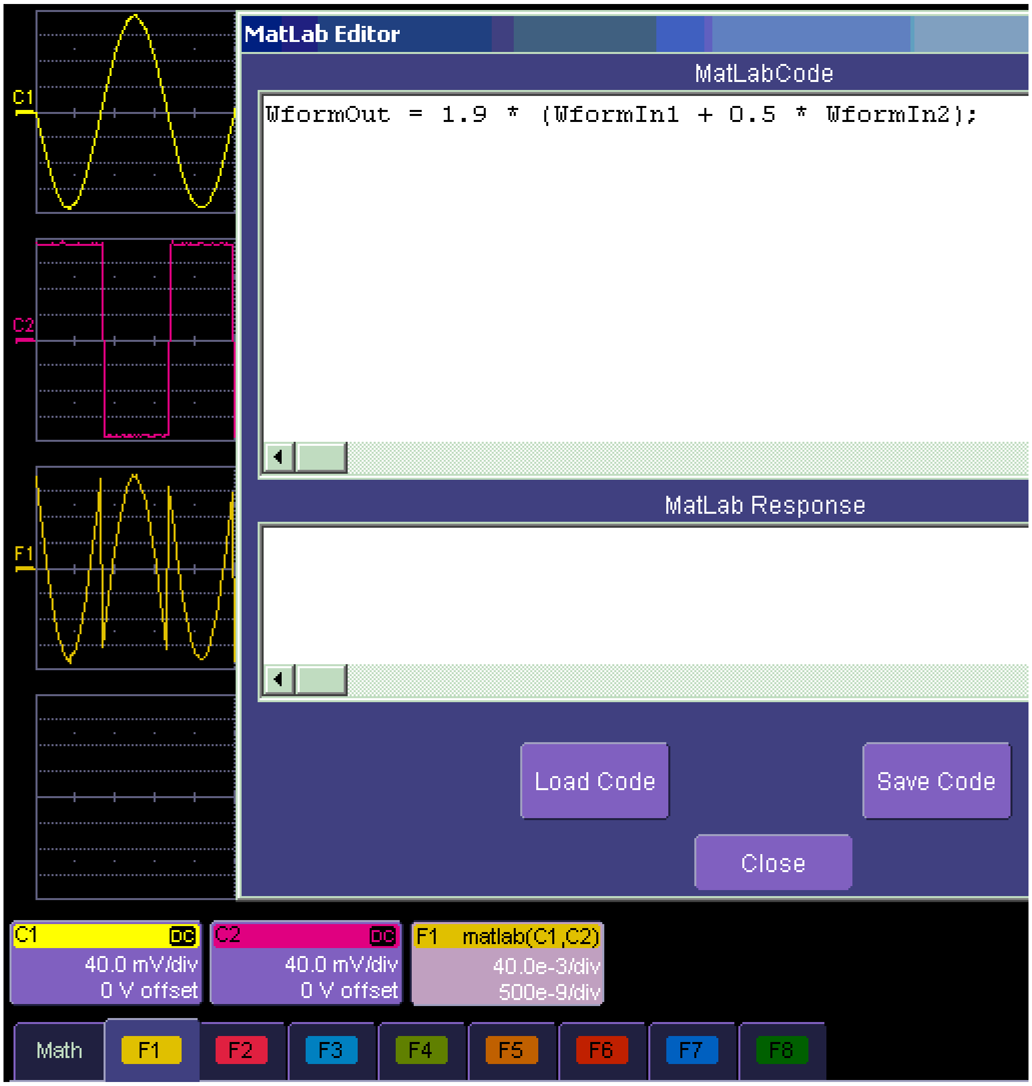


Next, a logarithmic vertical scale, for which the script can be found here. (Most scripts would be far simpler than this one.)

Frequency response curves are frequently drawn on a logarithmic scale. The upper trace is a frequency spectrum of a square wave after enhanced resolution has been applied. It was created using instrument functions. The lower trace is the first lobe of the FFT display. But with a logarithmic frequency scale. Click here for the VBScript.

In addition to VBScripting, MATLAB, Mathcad, or Excel can also be used to generate a result. The F1 trace (shown below in Example 6) was calculated in MATLAB (F1=WformOut) from C1 (WformIn1) and C2 (WformIn2). The same calculation could also be done in Excel by using a simple formula in a spreadsheet cell.

Example 6



Summary

The examples above illustrate only the capability to use VBScript and MATLAB. The instrument with the LeCroy XMAP software option allows you to use Excel, Mathcad, MATLAB, and VBScript in this manner. Of course, you will need to load Excel, Mathcad, or MATLAB in the scope (VBScript does not require any additional software) to take advantage of the capability. You can think of these functions as "subroutines" of the instrument's main software, which take in waveform data and other variables like vertical scale and offset, and horizontal scale and offset. These functions then return a waveform or a parameter as required. In addition, you can view the calculated data directly in Excel, MATLAB, or Mathcad, if you desire.

What is Excel?

Excel is a program within Microsoft Office. With it you can place data in the cells of a spreadsheet, calculate other values from them, prepare charts of many kinds, use mathematical and statistical functions, and communicate with other programs in Office. From the instrument you can send data to Excel (where processing can take place) and return the results to the instrument.

What is Mathcad?

Mathcad is a software package from MathSoft. It provides an integrated environment for performing numerical calculations and solving equations, and communicating with other programs. Results can be presented in tabular or graphical form.

What is MATLAB?

MATLAB is a software package from MathWorks that provides an environment for work in computation and mathematics. An interactive language and graphics are provided.

What is VBS?

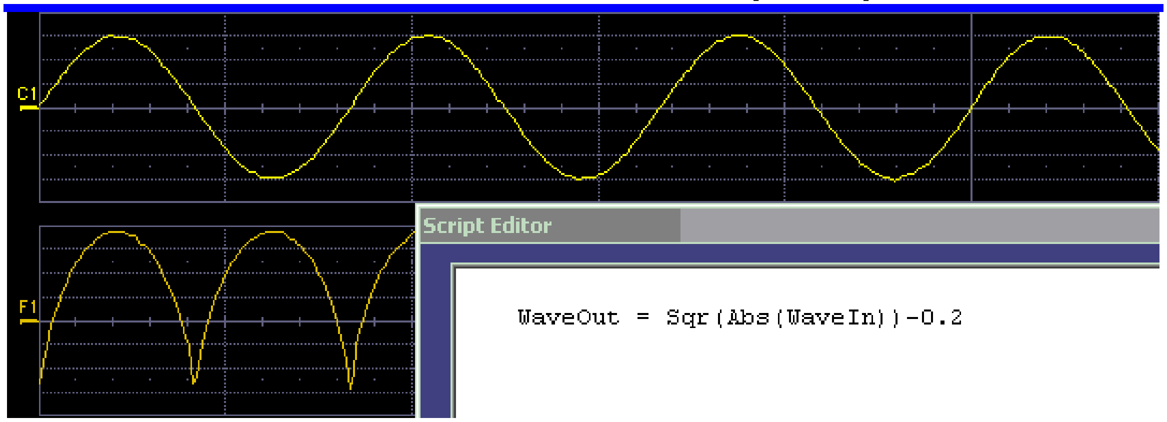
VBS is a programming language, but you don't write it in a special environment such as C++ or Visual Basic; you write it within your own application. In the instrument, a few clicks or button pushes will get you into an editing panel where you can write what you want. You cannot crash the scope, or in any other way interfere with its workings, because the system is completely protected.

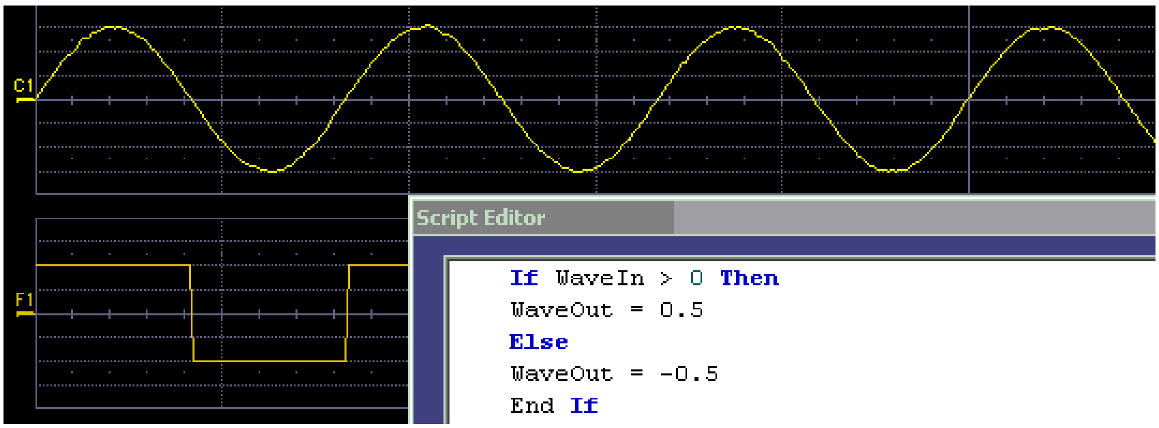
A product of Microsoft and a subset of Visual Basic, VBS can be learned very quickly if you have some experience in any programming language. The VBS processing function can collect a number of useful variables from the scope, including waveform data and useful variables such as volts per division and time per division. The output from a script can be a waveform or a parameter, and you can choose your own values for variables such as volts per division.

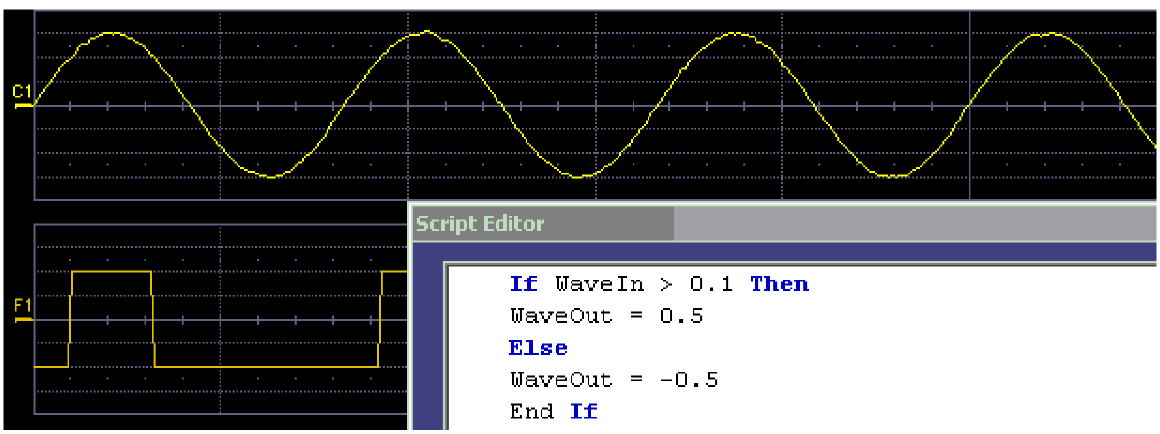
The idea of a VBS function is that you start with an input waveform, operate on some or all of the values with a script, and show the result on a scope grid, like any other waveform.

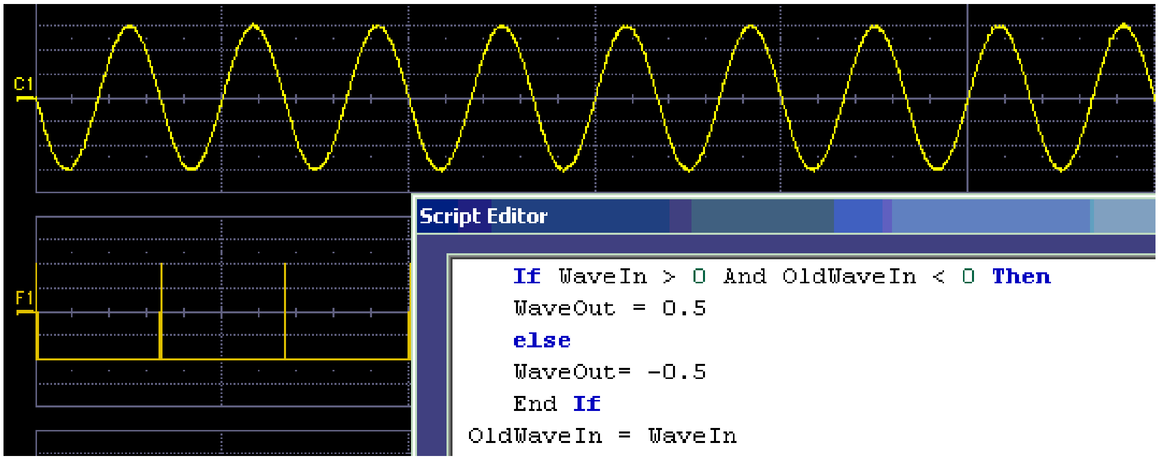
VBScript customization is built into the instrument, so no additional programs need to be loaded to take advantage of this capability.

The following diagrams were made by changing a small part, in some cases just one line, of a standard VBScript. VBS is a well-known standard language, with excellent support documentation, and it is easy to use in several different environments.


C1
F1
Script Editor
WaveOut = Sqr(Abs(WaveIn))-0.2


C1
F1
Script Editor
If WaveIn > 0 Then
WaveOut = 0.5
Else
WaveOut = -0.5
End If


C1
F1
Script Editor
If WaveIn > 0.1 Then
WaveOut = 0.5
Else
WaveOut = -0.5
End If



These examples are purely illustrative, but you can easily imagine that with a VBScript you can add value to the scope in a very short time. This gives you an instrument that does exactly what you want, time after time, by using your stored setups and scripts.

What can you do with a customized instrument?

If you require a result that can be derived logically from the input waveform, you can do it. Many calculations can be done with remarkably small scripts, but if you have no time for scripting, you can use one of the proprietary packages, such as Excel, MATLAB, or Mathcad, which offer immense processing power.

Scaling and Display

Scripting and programming allow a large variety of opportunities. You may, for example, be using transducers. If so, you can change the units of your waveforms, and write N (newtons), J (joules) and so on, and you can introduce scaling factors. If the transducers are non-linear, you can correct for that, too. You can also transform horizontal scales and vertical scales by manipulating the data. Logarithmic scales in amplitude and frequency are often required. Squaring and taking square roots are needed in certain applications. Here is a picture showing some graphs related to white noise, showing ways of detecting small deviations from the true distribution. The lower two graphs were generated and placed in one trace using a VBScript.

In the next example, four graphs are placed in one trace.

Golden Waveforms

This is a rich field for VBS. An example was given earlier. The only limits to the shapes that can be generated are the vertical resolution and the number of samples.

A practical example – DVI Data-Clock skew

The next example is a measurement of DVI Data-Clock skew jitter measurement, using a VBScript to emulate the PLL. A solution to a practical measurement problem was shown earlier.

These are just a few of the many solutions that can be created.

Number of Samples

The various math packages can process samples as follows:

Excel	65,535 samples
Mathcad	5 MS

The number of samples that MATLAB can process is determined by memory option, as follows:

Memory Length	System DRAM	Buffer Length
STD, S, M	256 MB	40 MB
L, VL	512 MB	200 MB
XL	1 GB	400 MB
XXL	2 GB	400 MB

Calling Excel From Your Instrument

Calling Excel Directly from the Instrument

Excel can be directly called from the instrument in two ways:

Using a function	F1 through Fx [The number of math traces available depends on the software options loaded on your scope. See Specifications.]	Excel returns a waveform
Using a parameter	P1 through Px [The number of parameters available depends on the software options loaded on your scope. See Specifications.]	Excel returns a parameter

In both cases, one call to Excel can use two separate waveforms as input.

Notes:
Excel has a calculation algorithm of 64,000 points (32,000 if you have created a chart in Excel). Therefore, make sure that your acquisition has less than this number of points if you are going to use an Excel calculation.

To use this capability, you must have the LeCroy XMAP software option and Excel loaded in your instrument. Select **Minimize** from the instrument's **File** menu to access the Excel program directly.

How to Select a Math Function Call

The Excel math function is selected from the **Math Operator** menu, where it appears in the

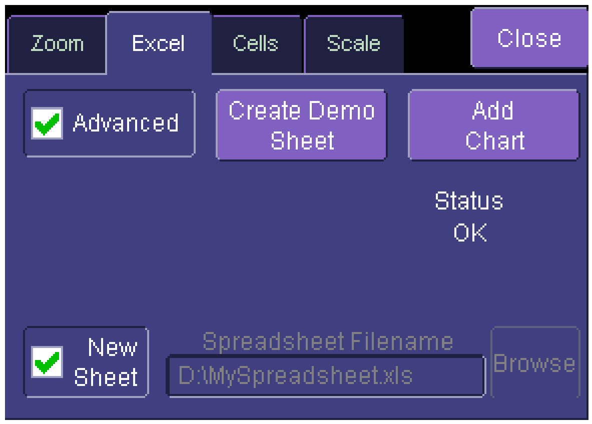
Custom group.

How to Select a Parameter Function Call

The Excel Parameter function is selected from the **Select Measurement** menu, where it appears in the **Custom** group.

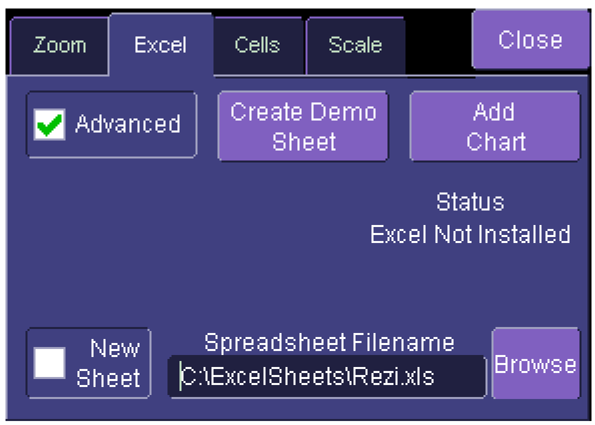
The Excel Control Dialog

Once you have invoked an Excel call, you will see a dialog at the right of the screen, allowing you to control the zoom, Excel properties, linking cells, and scale of the output trace from Excel:



Entering a File Name

If you uncheck the **New Sheet** checkbox, you can enter the file name of an existing file.

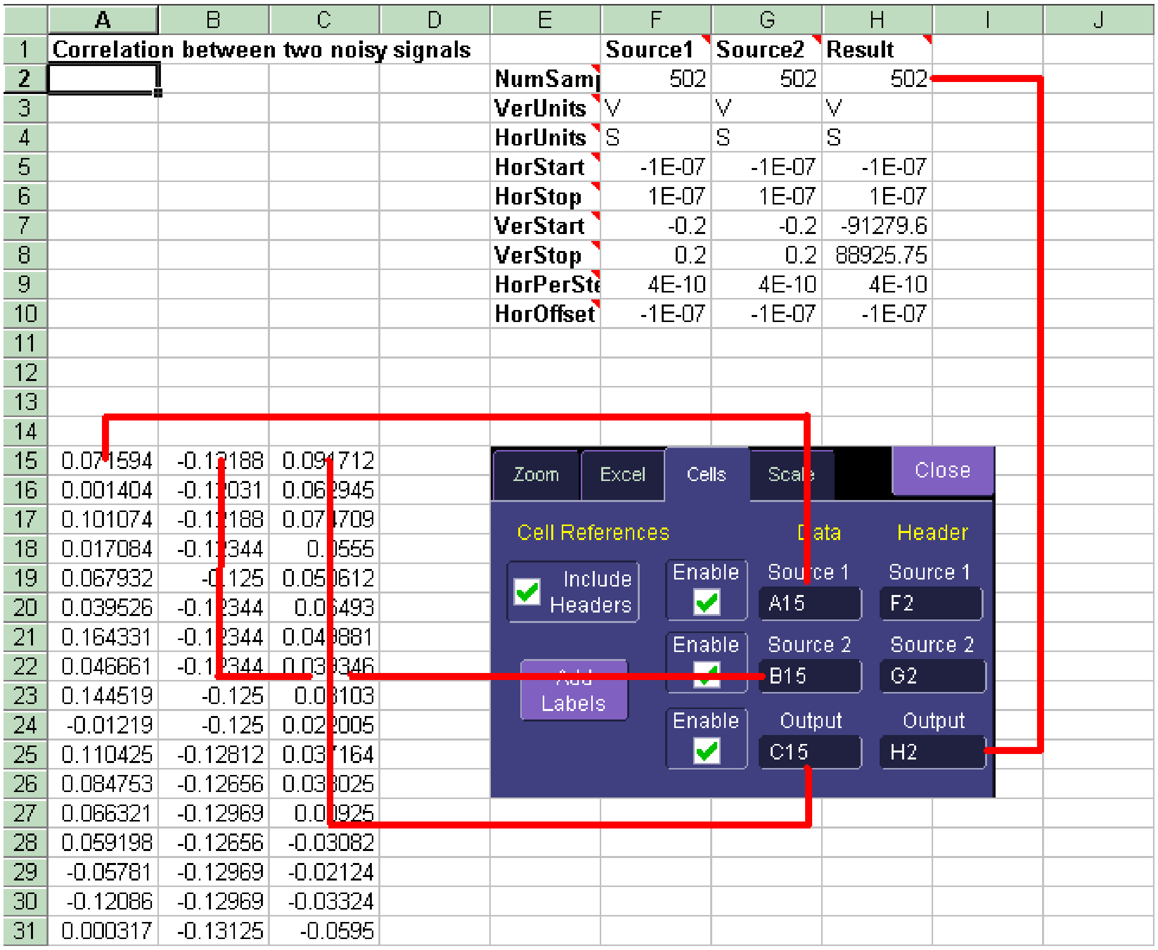


Create Demo Sheet Calls up a default Excel spreadsheet.

Add Chart Adds charts of your waveforms to Excel. You can go into Excel and create as many charts as you want.

Organizing Excel Sheets

The **Cells** tab allows you to organize your Excel chart. When placing the components in the sheet, be careful to avoid over-writing needed information, especially when you are using multiple input waveforms. As depicted here, the instrument panel has been pasted over the Excel sheet.



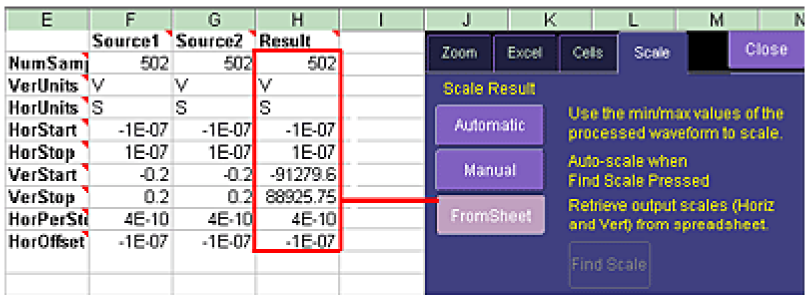
	A	B	C	D	E	F	G	H	I	J
1	Correlation between two noisy signals					Source1	Source2	Result		
2					NumSamj	502	502	502		
3					VerUnits	V	V	V		
4					HorUnits	S	S	S		
5					HorStart	-1E-07	-1E-07	-1E-07		
6					HorStop	1E-07	1E-07	1E-07		
7					VerStart	-0.2	-0.2	-91279.6		
8					VerStop	0.2	0.2	88925.75		
9					HorPerSte	4E-10	4E-10	4E-10		
10					HorOffset	-1E-07	-1E-07	-1E-07		
11										
12										
13										
14										
15	0.071594	-0.12188	0.091712							
16	0.001404	-0.12031	0.062945							
17	0.101074	-0.12188	0.071709							
18	0.017084	-0.12344	0.0555							
19	0.067932	-0.125	0.050612							
20	0.039526	-0.12344	0.06493							
21	0.164331	-0.12344	0.048881							
22	0.046661	-0.12344	0.039346							
23	0.144519	-0.125	0.03103							
24	-0.01219	-0.125	0.022005							
25	0.110425	-0.12812	0.037164							
26	0.084753	-0.12656	0.038025							
27	0.066321	-0.12969	0.00925							
28	0.059198	-0.12656	-0.03082							
29	-0.05781	-0.12969	-0.02124							
30	-0.12086	-0.12969	-0.03324							
31	0.000317	-0.13125	-0.0595							

There are three arrays of data for the three waveforms: up to two inputs and one output. There are corresponding small arrays of information about each trace.

Scale Setting the Vertical Scale

The vertical scale of the output waveform from Excel may be set in three ways:

Automatic	For each acquisition, the instrument fits the waveform into the grid.
Manual	For one acquisition, click **Find Scale**; the instrument fits the current waveform into the grid. All subsequent acquisitions will use this scale until you make a change.
From Sheet	The scale is taken from the specified cells in the Excel sheet, H2 through H10 in the example above, where cell H2 was specified as the top of the data set, as depicted below.



Trace Descriptors

The next figure explains the meanings of the descriptors for each trace.

C	D	E	F	G	H
n two noisy signals			Source1	Source2	Result
Samples in trace		NumSam	502	502	502
Vertical units		VerUnits	V	V	V
Horizontal units		HorUnits	S	S	S
Left edge of grid		HorStart	-5E-08	-5E-08	-5E-08
Right edge of grid		HorStop	1.5E-07	1.5E-07	1.5E-07
Bottom of grid		VerStart	-0.2	-0.2	-0.32585
Top of grid		VerStop	0.2	0.2	0.266995
Time per sample		HorPerSte	4E-10	4E-10	4E-10
Trigger position		HorOffset	-5E-08	-5E-08	-5E-08

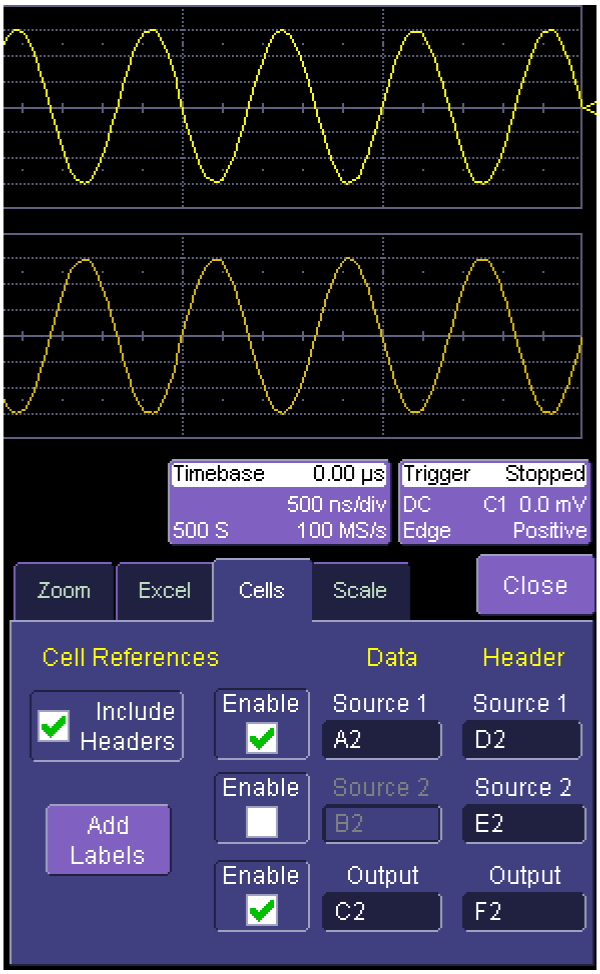
Multiple Inputs and Outputs

If you invoke two or more instrument parameter functions or waveform functions that call Excel, you will find that they all refer to the same spreadsheet by default. Thus, your spreadsheet can use the data from several waveforms, and you can derive many different combinations of output parameters and waveforms, including some of each, from your spreadsheet. You only have to be careful about the positioning of your cell ranges within the sheet so that no conflicts occur.

Because filling cells in the spreadsheet is a relatively slow process, all unwanted sources (inputs) should be left disabled (unchecked). For example, if you want one waveform and two parameters derived from the data of three waveforms, you can have one function with both sources enabled, one with one source enabled, and one with no sources enabled. The alternative is to use one input in each function.

Simple Excel Example 1

In this example we use Excel to invert or negate a waveform. The first figure shows a part of the screen. The upper trace is the original signal. The lower is the result from Excel.

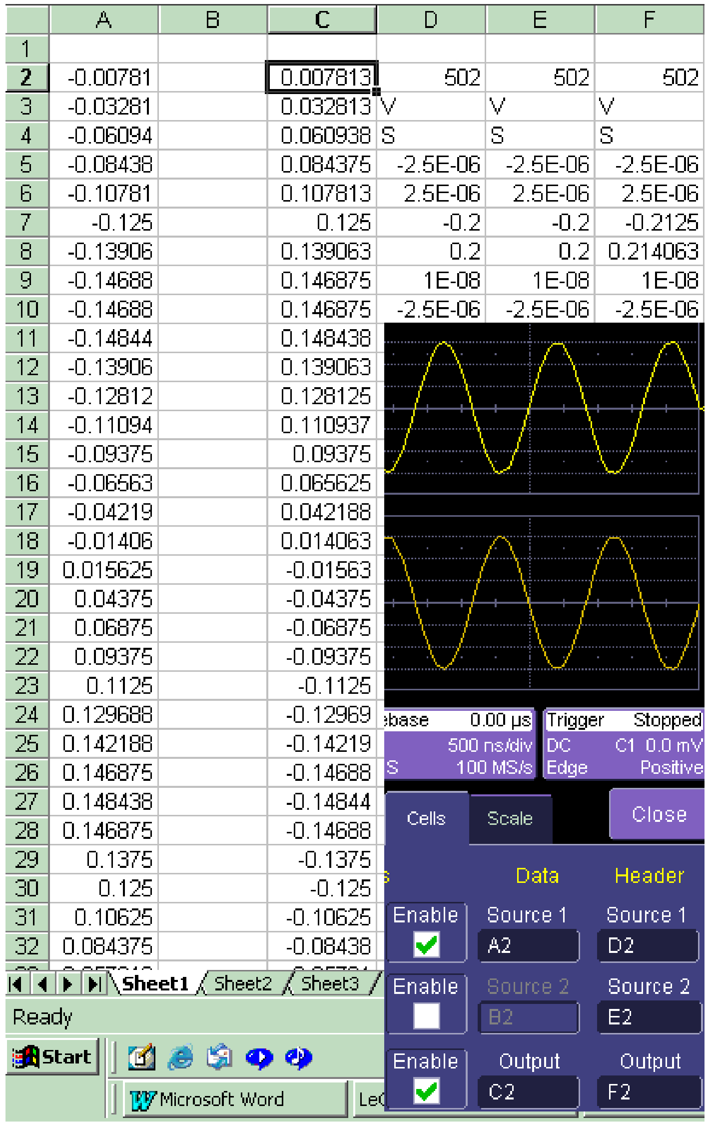


The dialog is the one that controls the location of the data in the Excel worksheet.

The input data are in columns A and B (though, only the first is used) and the output is in column C. All have been set to start at row 2, allowing space for a title in row 1.

Columns D, E and F contain the headers for the three waveforms. These are the set of numbers that provide the description of the scope settings, such as vertical scale and offset, and number of samples.

In this figure, the panel has been pasted onto the Excel sheet for comparison:



	A	B	C	D	E	F
1						
2	-0.00781		0.007813	502	502	502
3	-0.03281		0.032813	V	V	V
4	-0.06094		0.060938	S	S	S
5	-0.08438		0.084375	-2.5E-06	-2.5E-06	-2.5E-06
6	-0.10781		0.107813	2.5E-06	2.5E-06	2.5E-06
7	-0.125		0.125	-0.2	-0.2	-0.2125
8	-0.13906		0.139063	0.2	0.2	0.214063
9	-0.14688		0.146875	1E-08	1E-08	1E-08
10	-0.14688		0.146875	-2.5E-06	-2.5E-06	-2.5E-06
11	-0.14844		0.148438			
12	-0.13906		0.139063			
13	-0.12812		0.128125			
14	-0.11094		0.110937			
15	-0.09375		0.09375			
16	-0.06563		0.065625			
17	-0.04219		0.042188			
18	-0.01406		0.014063			
19	0.015625		-0.01563			
20	0.04375		-0.04375			
21	0.06875		-0.06875			
22	0.09375		-0.09375			
23	0.1125		-0.1125			
24	0.129688		-0.12969			
25	0.142188		-0.14219			
26	0.146875		-0.14688			
27	0.148438		-0.14844			
28	0.146875		-0.14688			
29	0.1375		-0.1375			
30	0.125		-0.125			
31	0.10625		-0.10625			
32	0.084375		-0.08438			

To get the output values in column C, we set C2 = - A2 and copy this formula down the column. This is the only action needed in Excel, and can be seen in the next figure:

C2 = =-A2

	Name Box	B	C	D	E	F
1						
2	-0.00781		0.007813	502	502	502
3	-0.03281		0.032813	V	V	V
4	-0.06094		0.060938	S	S	S
5	-0.08438		0.084375	-2.5E-06	-2.5E-06	-2.5E-06
6	-0.10781		0.107813	2.5E-06	2.5E-06	2.5E-06
7	-0.125		0.125	-0.2	-0.2	-0.2125
8	-0.13906		0.139063	0.2	0.2	0.214063
9	-0.14688		0.146875	1E-08	1E-08	1E-08
10	-0.14688		0.146875	-2.5E-06	-2.5E-06	-2.5E-06
11	-0.14844		0.148438			
12	-0.13906		0.139063			
13	-0.12812		0.128125			
14	-0.11094		0.110937			
15	-0.09375		0.09375			
16	-0.06563		0.065625			
17	-0.04219		0.042188			
18	-0.01406		0.014063			
19	0.015625		-0.01563			
20	0.04375		-0.04375			

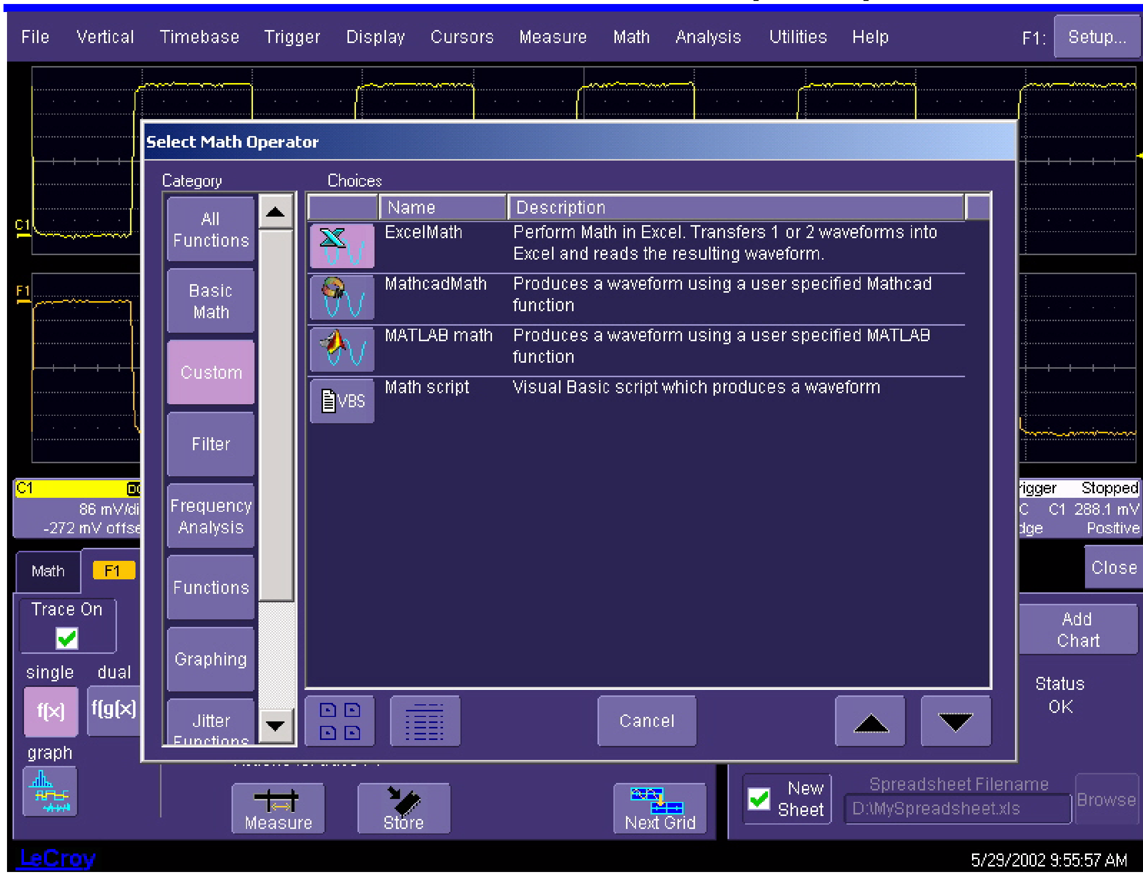
Simple Excel Example 2

In this example we use Excel to invert or negate a waveform. The first figure shows a part of the instrument screen. The upper trace (C1) is the original signal. The lower trace (F1) is the result calculated in Excel and displayed on the screen.

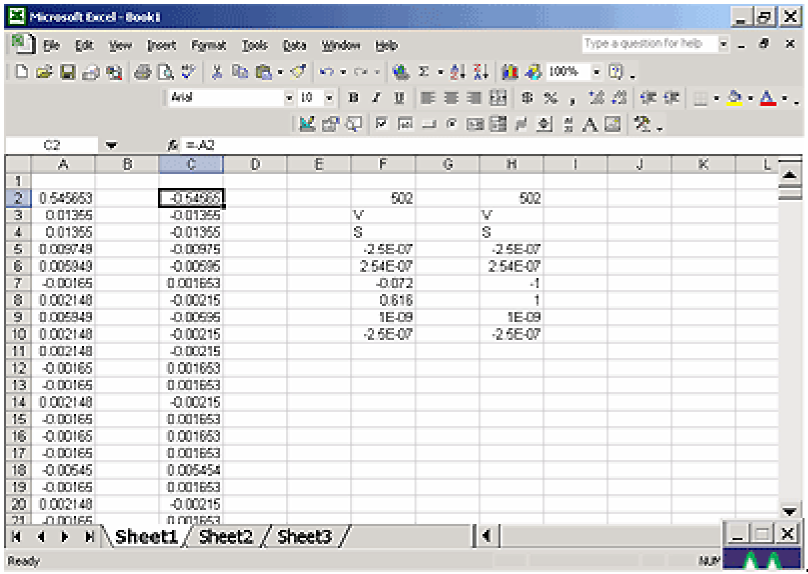
The input data is in columns A and B (though by default, only a single input/column is used), and the output is in column C. All have been set to start at row 2 (which allows for a header in row 1).

To create this waveform, you would simply do the following:

1. Ensure that your acquisition has no more than 64 kpts (the Excel calculation limit)
2. Choose a function, and select ExcelMath as **Operator1** for the function. Excel will open automatically in the background.



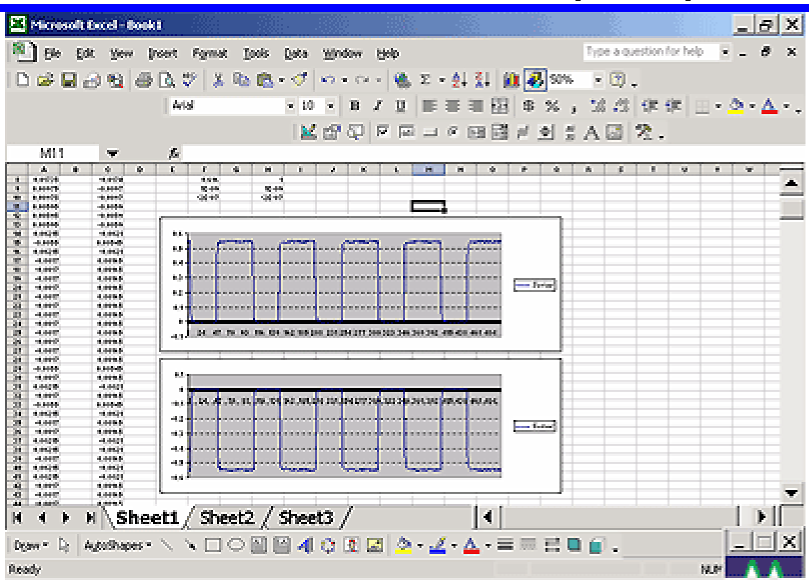
3. Choose File, Minimize from the menu bar to minimize the instrument display and open the Excel program.
4. Create your formula for each data point in column A (in this case, our formula for cell C2 is **-A2**, copied for the entire column), as shown here



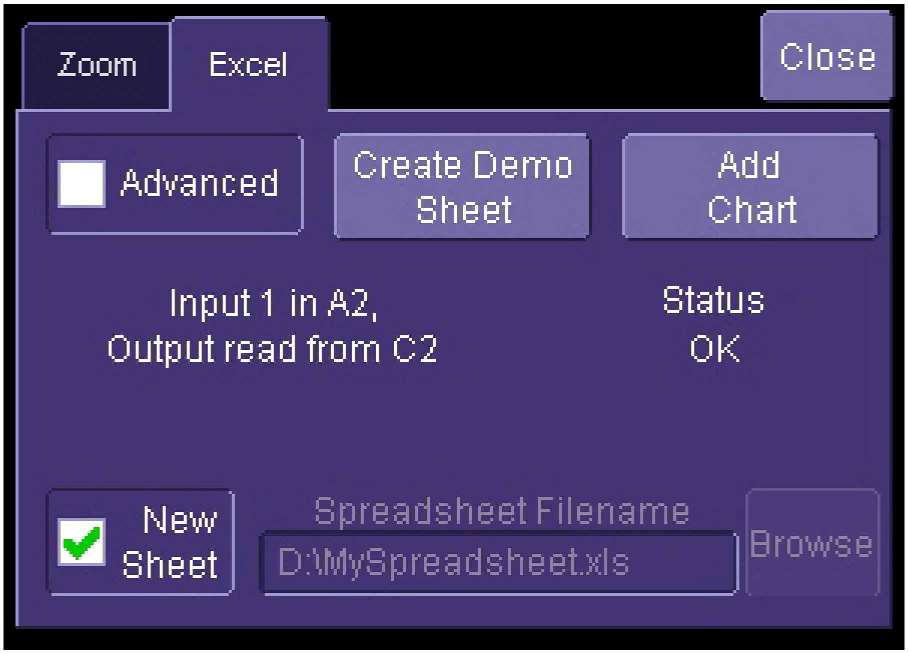
5. Retrigger the scope (if it is not currently triggering)
6. Return to the program

Note that the only action that was needed in Excel was to create the formula in column C for each data point in column A. The instrument automatically opens Excel, puts the waveform data in the correct columns, and returns the calculated data back to the display as the chosen F trace. This Excel-calculated trace can have further measurements or math calculations performed on it, if desired.

You can also create a chart of the data in Excel automatically and view the data there.



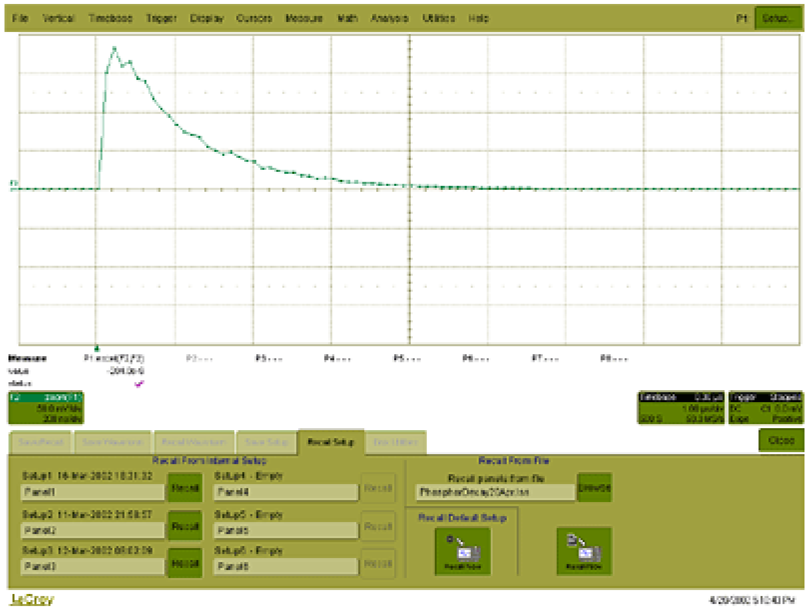
Simply press the **Add Chart** button in the instrument's Excel dialog and a chart of the input (top chart) and Excel calculated output (bottom chart) will be automatically created in the spreadsheet. The chart will be updated automatically as the scope is triggered.



Exponential Decay Time Constant Excel Parameter (Excel Example 1)

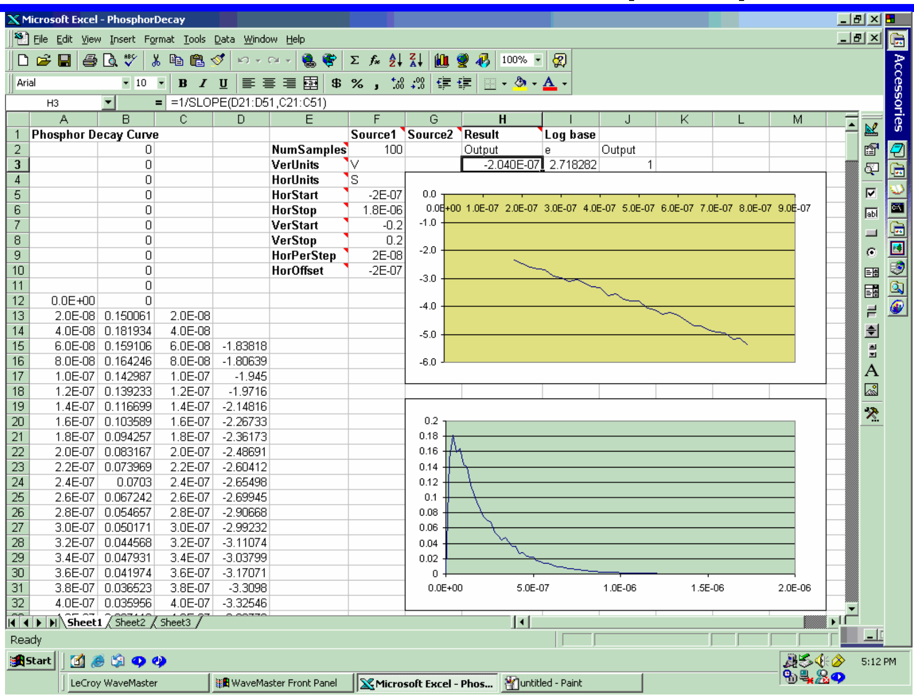
This example calculates the time constant of an exponentially falling pulse, such as the light output of a phosphor.

The first figure shows a typical pulse, including pseudo-random noise, generated by a VBScript.



The pulse was generated by a formula of the form $e^{(1 - t/TC1)} * e^{-t/TC2}$, where TC1 and TC2 are time constants, The requirement is to measure the time constant TC2, using the portion of the trace where TC1 has negligible effect. This was done using Function F1, which is not a part of the measurement process.

For the actual measurement, Parameter P1 was set up as an Excel call. In Excel, the selected portion of the trace was converted to logarithms, and the Excel function SLOPE was used, as shown here:



Here we see the input data in column B (with a time scale in A) created using the contents of cell F9, Horizontal Per Step. The logarithmic data are in column D, with the time scale repeated in C. The output appears in cell H3, using the formula **=1/SLOPE(D21:D51,C21:C51)**.

Required files:

Setup: PhosphorDecay20Apr.lss

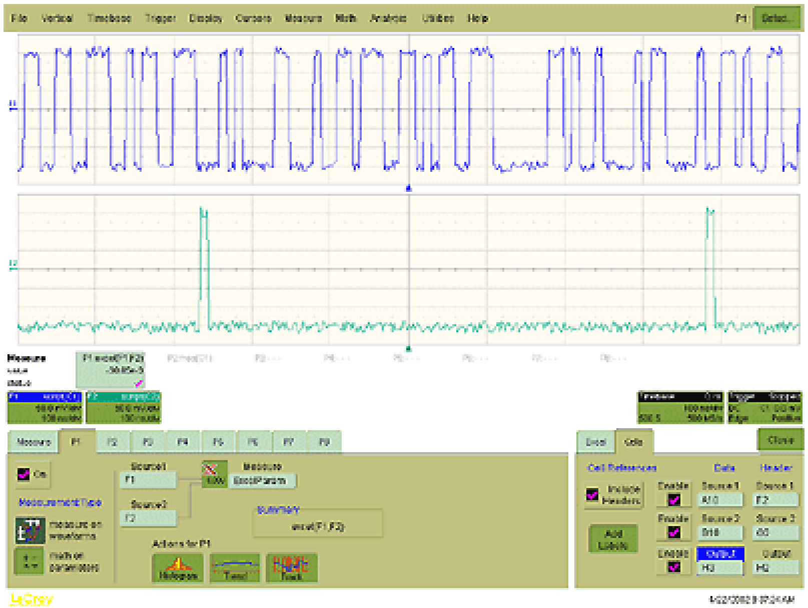
F1 Generator: PhosphorPulseGen.txt

P1 Excel: PhosphorDecay.xls

Gated Parameter Using Excel (Excel Example 2)

This example calculates a parameter of a waveform, in a region of interest defined by the leading edges of two pulses in a separate waveform.

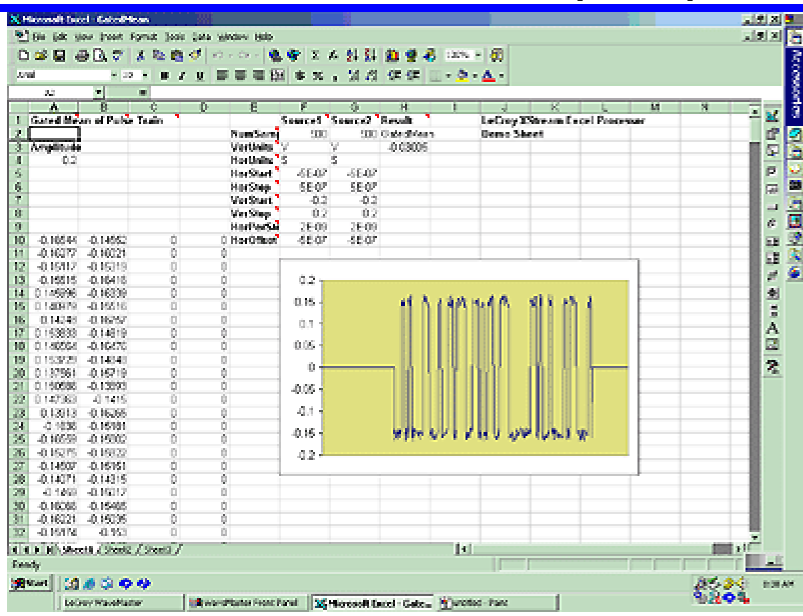
This figure shows the instrument screen:



The traces were made using VBS scripts in functions F1 and F2, based on pseudo-random numbers to provide noise and varying pulse widths. Randomize Timer: Randomize Timer was used in both scripts to ensure that successive acquisitions produced different data. Script F1 generates pulses with widths that are multiples of a set clock period. F2 generates one pulse in the first half of the time window, and one pulse in the second half. Both pulses are constrained to coincide with the clock pulses of F1. F1 and F2 are used here only as simulations and are not part of the measurement process, which only uses P1.

The call to Excel is made through Parameter P1.

The next figure shows a part of the Excel workbook.



Here we see the gated waveform that has been created in Excel. The Mean parameter during the region of interest (ROI) is placed in cell H3.

How Does this Work?

The amplitude of the signal is about 0.3 volts, and the screen height is 0.4 volts, as derived from cells F7 and Fx. A threshold value for amplitude was calculated by placing 0.5 * (Fy – Fx) in cell A4.

Remember that in the instrument the sources were defined to be A10 and B10. This means that the first point on the waveform will be read into A10, and, since the waveform has 500 points, the last point will be read into A510. The same holds true for F2 and column B, since F2 is assigned as Source2, and data is defined to write into column B starting with cell B10.

To create the gating function in column C, the cell C10 was given the following formula:

IF ((B10 – B9) > A4, 1 – C9, C9). This was copied down the column. Column D, the output column, is simply A * C.

The output was defined as cell H3.

The required mean in cell H3 is given by SUM (D10 : D509) / SUM (C10 : C509), for a 500 point waveform.

Requires files:

Setup: GatedParameterExcel.lss

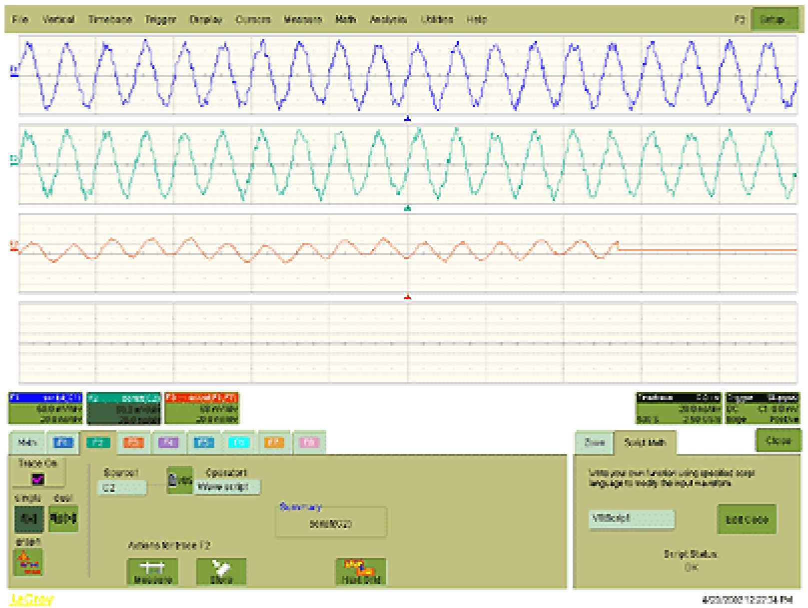
Function F1: RandomPulses22Apr.txt

Function F2: RandomGate22Apr.txt

Parameter P1: GatedMean.xls

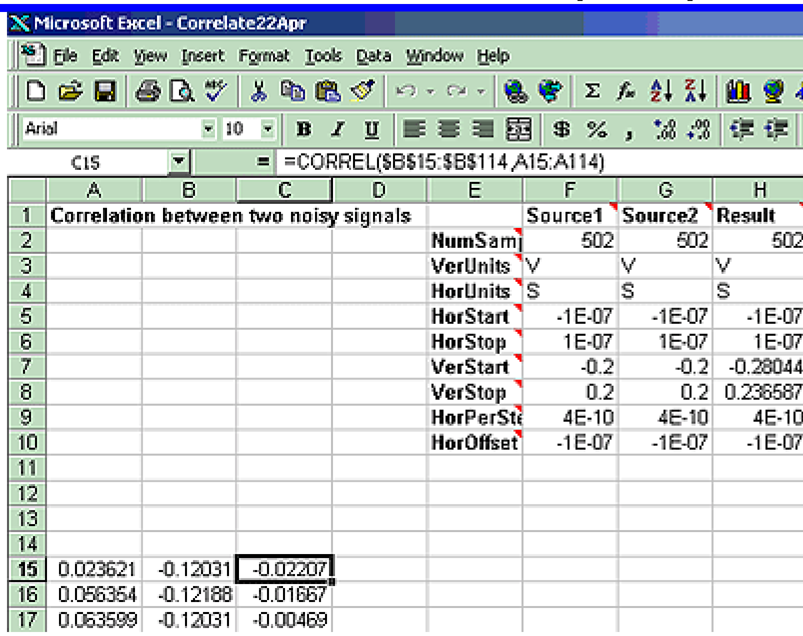
Correlation Excel Waveform Function (Excel Example 3)

This example uses an Excel waveform function to examine the cross-correlation between two signals, which are both noisy sinusoidal segments. The correlation trace is, of necessity, shorter than the input traces.



The noise was generated using pseudo-random numbers. Randomize Timer was included in the VBScript to ensure that the two traces differed, and that subsequent acquisitions differed. Functions F1 and F2 are included only to simulate signals, and are not part of the measurement process, which is performed by F3.

This example used the CORREL (Array1, Array2) function of Excel, as depicted below:



	A	B	C	D	E	F	G	H
1	Correlation between two noisy signals					Source1	Source2	Result
2					NumSamp	502	502	502
3					VerUnits	V	V	V
4					HorUnits	S	S	S
5					HorStart	-1E-07	-1E-07	-1E-07
6					HorStop	1E-07	1E-07	1E-07
7					VerStart	-0.2	-0.2	-0.28044
8					VerStop	0.2	0.2	0.236587
9					HorPerSte	4E-10	4E-10	4E-10
10					HorOffset	-1E-07	-1E-07	-1E-07
11								
12								
13								
14								
15	0.023621	-0.12031	-0.02207					
16	0.056354	-0.12188	-0.01667					
17	0.063599	-0.12031	-0.00469					

Required files:

Setup: CorrelateExcel22Apr.lss

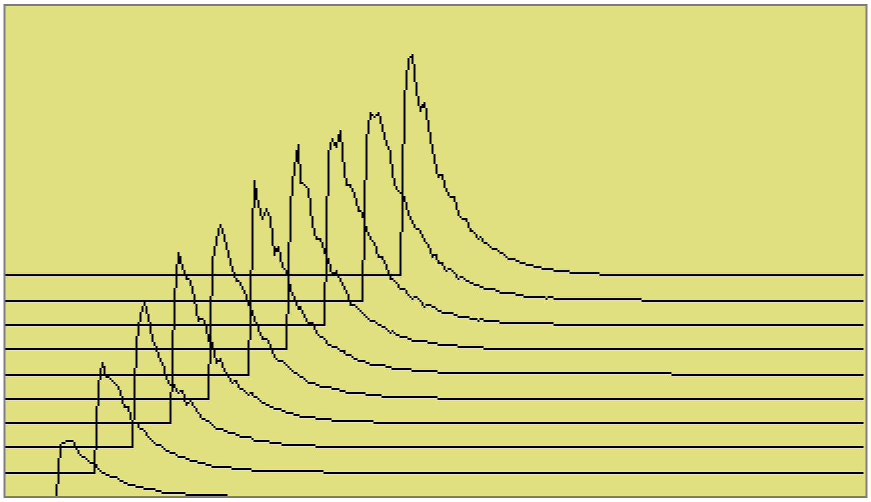
Function F1: NoisySine22Apr.txt

Function F2: NoisySine22Apr.txt

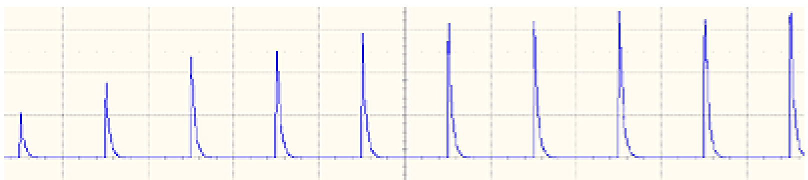
Function F3: Correlate22Apr.xls

Multiple Traces on One Grid (Excel Example 4)

This example shows how you can place multiple traces in one picture, with only two operations in an Excel sheet. Depicted below is an example from an Excel spreadsheet.

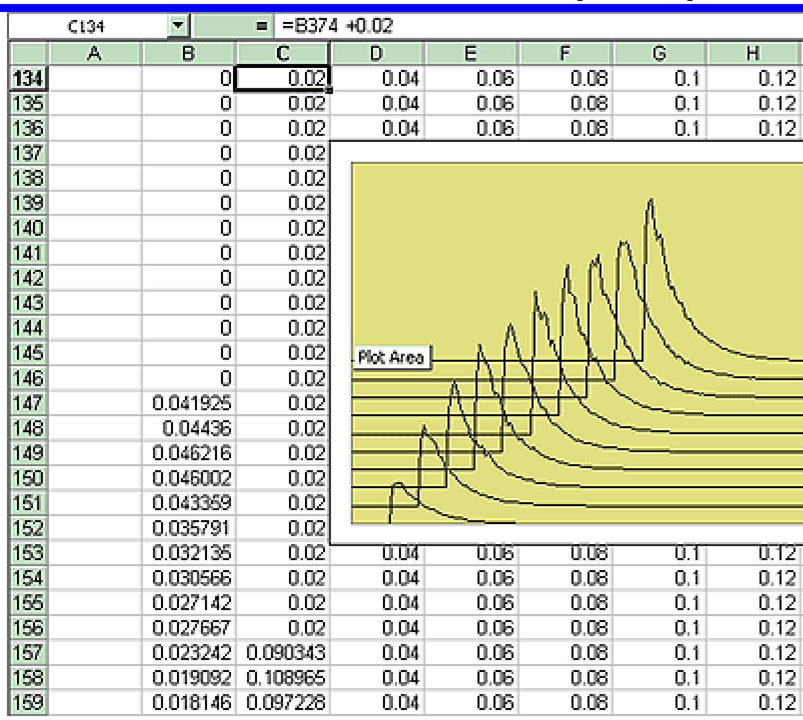


Here is an original instrument trace



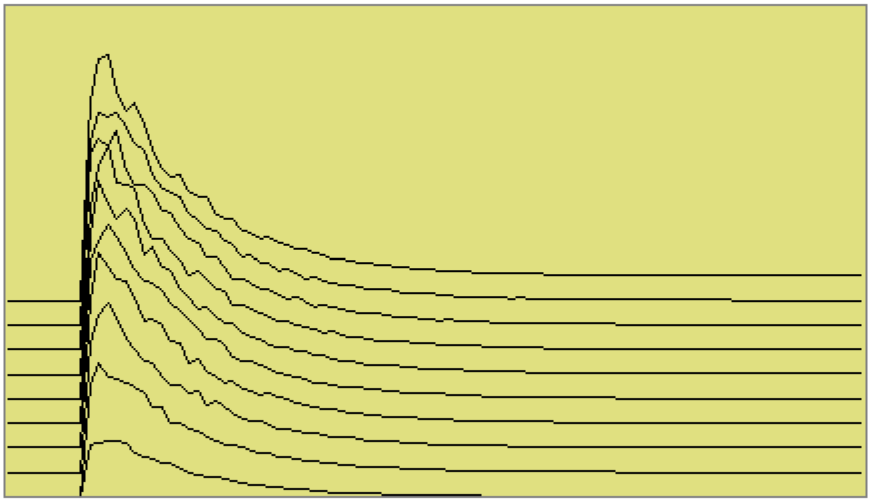
The method is very simple. First, the waveform is transferred to an Excel spreadsheet by means of an instrument Excel call. Second, two operations are needed in Excel: placing a simple formula in one cell, and copying that formula into a range of cells.

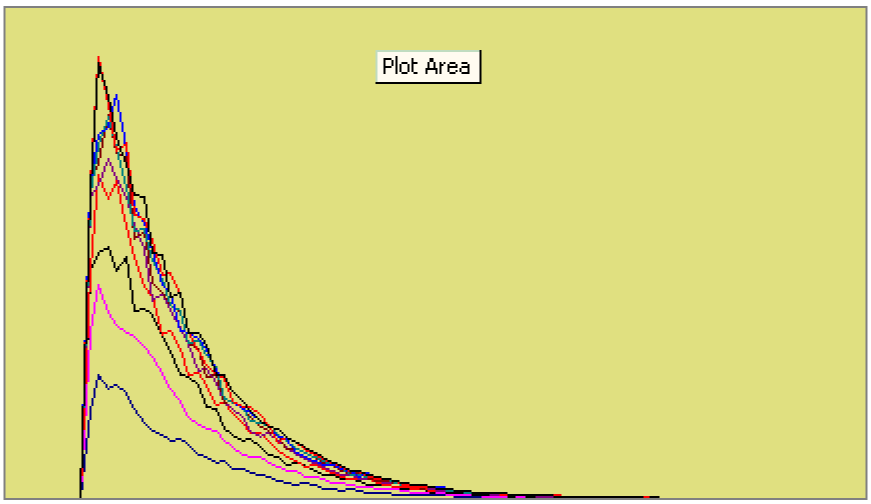
Depicted below is the required Excel formula.



	A	B	C	D	E	F	G	H
134		0	0.02	0.04	0.06	0.08	0.1	0.12
135		0	0.02	0.04	0.06	0.08	0.1	0.12
136		0	0.02	0.04	0.06	0.08	0.1	0.12
137		0	0.02					
138		0	0.02					
139		0	0.02					
140		0	0.02					
141		0	0.02					
142		0	0.02					
143		0	0.02					
144		0	0.02					
145		0	0.02					
146		0	0.02					
147		0.041925	0.02					
148		0.04436	0.02					
149		0.046216	0.02					
150		0.046002	0.02					
151		0.043359	0.02					
152		0.035791	0.02					
153		0.032135	0.02	0.04	0.06	0.08	0.1	0.12
154		0.030566	0.02	0.04	0.06	0.08	0.1	0.12
155		0.027142	0.02	0.04	0.06	0.08	0.1	0.12
156		0.027667	0.02	0.04	0.06	0.08	0.1	0.12
157		0.023242	0.090343	0.04	0.06	0.08	0.1	0.12
158		0.019092	0.108965	0.04	0.06	0.08	0.1	0.12
159		0.018146	0.097228	0.04	0.06	0.08	0.1	0.12

In fact, the simple expression B374 + 0.02 comprises several components. The original instrument trace is in column B, and the plot is required to start at cell B134. The traces repeat at intervals of 250 cells. Let us call this interval R. If we require a horizontal displacement D, then in cell CN we write B(N + R - D). In this example D is 10. Finally we may want a vertical displacement V, and we write B(N + R - D) + V. In this example, V is 0.02. D and V can be zero if required, as depicted below. All that remains is to copy the formula to the required range of cells.





Required files:

F1 is needed only as a simulator of signals.

Instrument setup: LaserStartup25Apr.lss

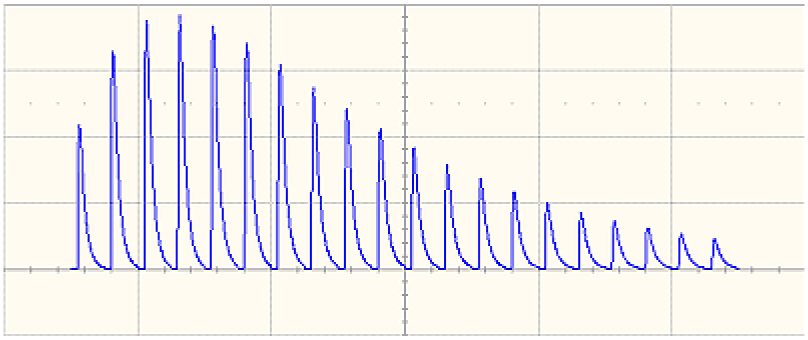
Function F1: LaserStartupApr25.txt

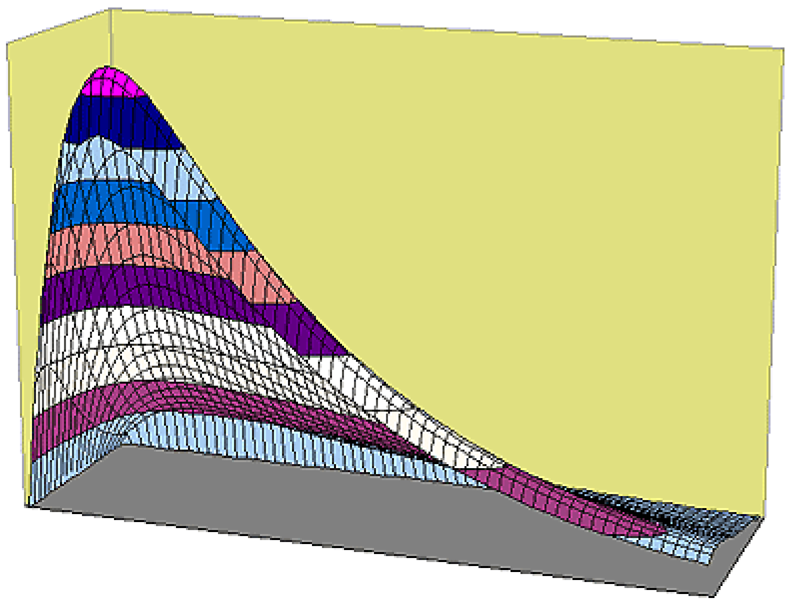
Function F2: LaserStartupPulses.xls No offset

LaserStartupPulses2.xls Vertical offset

LaserStartupPulses3.xls Vertical and horizontal offset

Using a Surface Plot (Excel Example 5)





Required files:

Setup: LaserSurface1May.lss

Function F1 Generator: LaserSurface2May.txt

Function F2 Excel: LaserSurface2May.xls

WRITING VB SCRIPTS

VBScripting is one of the custom features of your instrument. Others include the ability to work with programs such as Excel, Mathcad and MATLAB.

Types of Scripts in VBS

The instrument's VBS provides two types of script.

- The Waveform Function script allows you to take the data from one or two traces and make a new trace whose values may depend on the values of the input trace.
- The Parameter Function script also takes in the data from one or two traces, but it only has one output. This output is the zeroth element in the output array. It appears as a parameter value on the instrument's screen. The remainder of the array is currently not used, and is not accessible.

Within both types of script, you can call Excel.

Loading and Saving VBScripts

From the editing panel you can save your script and you can load a previous one. Should you forget to save a script, please note that when you save your setup, it has your current scripts embedded in it. Therefore it is a good idea to save your setup frequently. It is worth saving the script separately as well, because it is saved in a suitable format for printing or off-line editing with Notepad. Note that in both these examples the input data are referred to as InResult.DataArray. You can also write InResult1.DataArray and InResult2.DataArray, which refer to the two input traces. InResult.DataArray always refers to input trace 1. These remarks hold for any script that you write.

Example Waveform Function Script: Square of a waveform

```
' Example script to produce a waveform
This example calculates the square of
the input waveform.
OutResult.Samples = InResult.Samples ' Visible trace length + 1
' Note that a trace of nominal length 1000 comprises data numbered from
' 0 to 1001. The 1001st point is not visible, so you
' normally use points 0 to 1000,
' giving 1001 points and 1000 intervals between points.
startData = 0
endData = OutResult.Samples
LastPoint = endData - 1 ' because the last point is invisible.
```

```
ReDim newArray(OutResult.Samples) ' to store the results
unscaledData = InResult.DataArray(False)
' InResult.DataArray(False) provides
' integer data from -32768 to 32767.
' InResult.DataArray(True) provides real data
' in the same physical unit as the vertical scale of the input trace.
ScaleFactor = 1.0 / 32768 ' to make the trace fill the screen.
       For i = 0 To LastPoint
       newArray(i) = ScaleFactor * (unscaledData(i)) ^ 2
       Next
OutResult.DataArray(False) = newArray ' signed long integer data output
```

Example Parameter Function Script: RMS of a waveform

```
' Example script to produce a parameter.
' This script calculates the root mean square
' of the input waveform.
' Note that a trace of nominal length 1000 has data from
' 0 to 1001. The 1001st point is not visible, so you
' normally use points 0 to 1000,
' giving 1001 points and 1000 intervals between points.
startData = 0
endData = InResult.Samples
LastPoint = endData - 1 ' because the last point is invisible.
ReDim newArray(InResult.Samples) ' to store the results
unscaledData = InResult.DataArray(True)
' InResult.DataArray(False) provides
' integer data from -32768 to 32767.
' InResult.DataArray(True) provides real data
' in the same unit as the vertical scale of the trace.
Total = 0
        For i = 0 To LastPoint
```

```
        Total = Total + (unscaledData(i)) ^ 2
        Next
NewArray(0) = Sqr (Total / (LastPoint + 1)) Place the result in the zeroth element.
OutResult.ValueArray(True) = newArray ' integer data output
```

The default waveform function script: explanatory notes

InResult.Samples is the number of points in the incoming waveform.

InResult.DataArray(Boolean) (or InResult1.DataArray or InResult2.DataArray) is the array of input data. If the Boolean is True you get scaled real data in the units of the trace. If the Boolean is false you get unscaled integer data in the range -32768 to + 32767.

The value of InResult.Samples is the total number of data in a trace. It is two more than the nominal value given on the screen. The first point DataArray(0), coincides with the left edge of the screen, apart from the wobble caused by the trigger-to-sample clock difference. If the trace length is nominally 500, the right edge of the screen coincides with DataArray(500), which is the 501st point. The last point, number 502, is just off the right of the screen, and is never seen. That is why the loop in the script runs only to endData - 1.

OutResult.Samples is the number of data in the output trace, and is set to be the same as the number of data in the input trace. If you set the output length less than the input length, you get a shorter trace, the remainder being made of zeroes. If you try to set the output values to something illegal, you may find that a part of the trace retains the values from a previous acquisition.

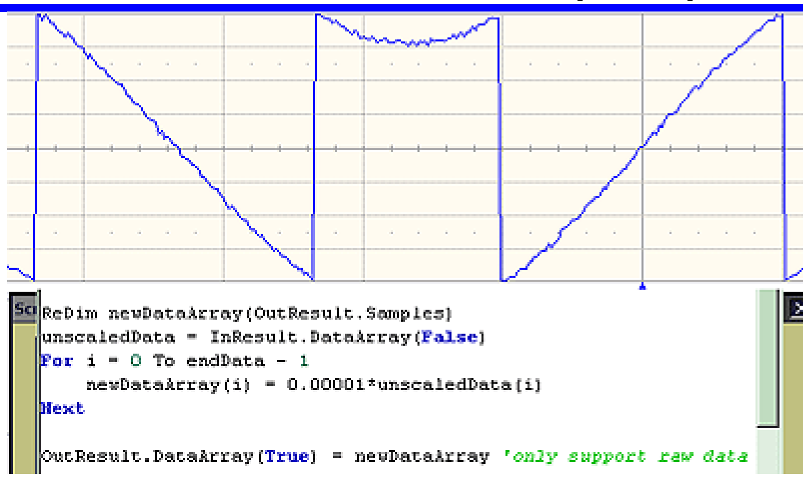
If you try to set something outside the bounds of an array, or you make some other error, or something overflows, or you ask for something impossible, such as log(-13), the instrument tells you the line number, and the nature of the problem. Other types of error may not be given the correct line number, for example, if "Next" or "End If" is omitted, because VBS does not know where it should have been.

UnscaledData is simply a copy of the input data set.

ReDim newDataArray(OutResult.Samples) defines an array of data for use as a scratch pad. Dim is short for Dimension, which is used in Visual Basic to declare a variable (even if it only has one element, in which case you omit the size of the array).

InResult.DataArray(False) means that the data are signed integers in the range -32768 to 32767. False is a Boolean value applying to the property Scaled. Scaled data are specified in the units of the vertical scale, such as volts. You get these by putting "True" instead of "False". If you want to make a section of the output trace invisible, you simply set the data values to full scale or bigger, top or bottom.

You can start with the unscaled data (False) as input, and then set the output data to scaled data (True), and you can go from scaled to unscaled. Using scaled data, an overflow will make a picture like this:



You can also start with True and convert to False, but in this case overflows will cause an error message.

Anything after a single quotation mark on a line will not be used by the instrument. This feature is intended for comments, for example

```
' This is a comment.
A = Amp * Sin(Omega * T) Calculate the output.
```

InResult.DataArray and OutResult.DataArray are only to be used as shown in the default scripts and in the example scripts: you cannot refer directly to individual elements of these arrays. You have to use your own arrays, in this example, unscaledData and newDataArray. You are not allowed to write statements like the following:

```
Y = InResult.DataArray (17)
OutResult.DataArray (257) = Z
```

Some parts of the default script must not be changed because they are a part of the interface. These are **highlighted** in the following script .

' TODO add your custom code here accessing OutResult and InResult objects

' Here's a small example that just inverts the waveform.

OutResult.Samples = InResult.Samples

startData = 0

endData = **OutResult.Samples**

newNumPoints = endData - startData

ReDim newDataArray (**OutResult.Samples**)

unscaledData = **InResult.DataArray** (False)

```
        For i = 0 To endData - 1
        newDataArray (i) = - unscaledData (i)
     Next
OutResult.DataArray (False) = newDataArray _' only support raw data
```

The four highlighted quantities are parts of the interface. The names must be retained. Furthermore, InResult.Samples and InResult.DataArray are inputs, and their values cannot be changed. OutResult.Samples and OutResult.DataArray are outputs, and can be changed, but not directly through their individual elements.

The default parameter function script: explanatory notes

The default parameter script is similar to the default waveform script, but there are subtle differences.

First, the size of the data array is the same as the nominal value: you cannot use or see the extra two points. So "500 points" means just that: 500 points.

Second, the output looks like an array, but only element zero is currently used. You must copy your parameter result into newValueArray(0). As with the arrays of the Waveform Script, you cannot refer directly to elements of the input and output arrays. You may not write something like

```
     OutResult.ValueArray (0) = P.
```

Note that the unit of the parameter is displayed as the same as the vertical unit of the trace, even if you have squared the data, for example, unless you change the unit yourself.

To find out how to edit a parameter script, click here.

The default parameter script is shown below.

```
' TODO add your custom code here accessing OutResult and InResult objects
' Here's a small example that just inverts the waveform
numParam = InResult.Samples
ReDim newValueArray(numParam)
scaledData = InResult.DataArray
     For i = 0 To numParam-1
     newValueArray(i) = -scaledData(i)_' Change this to do something useful.
     Next
OutResult.ValueArray = newValueArray 'only support raw data
```

Your parameter script should include something like this:

A. Do calculation to obtain your parameter value from the input data array.

B. newValueDataArray (0) = ParameterValue

C. OutResult.ValueArray = newValueArray

You can test this script using setup MeanDemoScriptApr2.lss.

You can edit scripts using Notepad, but you will not get any notification of errors.

You are **not** allowed to write OutResult.ValueArray(0) = MeanParameter.

InResult.DataArray and OutResult.DataArray are only to be used as shown in the default scripts and in the example scripts. You cannot refer to, or modify, any individual element in these arrays.

Scripting with VBScript

Separators

The two separators in VBS are the colon **:** and the single quotation mark .

Using the colon, you can place two or more statements on a line, for example:

```
XMin = 0.0 : XMax = 800.0 : YMin = 0.0 : YMax = 600.0
```

There is also an implied separator whenever a new line is begun.

Using the quotation mark you can signify that the remainder of the line is a comment: non-executable material that is usually used to clarify the workings of the script. For example:

```
RMSMax = 32767 / Sqr (2)    ' RMS of the largest sinusoid that can
 be
                            ' fitted into the screen in unscaled mode.
```

To continue a comment on to another line, another quotation mark is required on the new line.

Variable Types

VBS supports the following variable types:

Integer	signed 16 bit value in the range -32768 to 32767
Long	signed 32 bit value in the range -2^{31} to $+2^{31} - 1$
Single	real number or floating point number
Double	real number or floating point number
Boolean	Boolean or logical value
String	string of characters

When making comparisons using real numbers, beware of testing for equality, because of rounding errors. It may be better to apply a tolerance band. For Boolean, integers and strings, equality is valid.

You can use variables in VBS without declaring the type. The context may force an implicit type assignment. For example, if the result of a calculation is of a different type from the defined type, the type may be changed. Always set out calculations in such a way that type changes will not affect the final result in an undesirable or unpredictable way. If you want to change the type of a variable or a result, use a conversion function that will show others what you intend to happen. The conversion functions are CDbl, CInt, CLng, CSng, CStr.

Variable Names

Upper and lower case have no significance in VBS, either in variable names or in keywords (the names reserved by the system), but it is a good idea to be consistent about the spelling of a variable name to avoid confusion. At least 36 characters may be used in a variable name. These can include any combination of alphabetic and numeric characters, and the underscore character. No other punctuation character may be used in a variable name.

Do not use any of the following characters in a variable name:

`! @ & $ # ? , * . { } ( ) [ ] = + - ^ % / ~ < > : ;`

Just use alphanumerics and underscore, for example: Example_Name

If you have to introduce constants, give them sensible names, just like variables. For example, do not write:

```
_If RMS < 23169 Then OutputY = Y
```

Its meaning may not be obvious to someone else.

It is better to write something like this:

```
     FullScale = 32767
     RootTwo = Sqr (2.0)
     MaxRMS = FullScale / RootTwo
      . . . . .
          If RMS < MaxRMS Then . . . . .
```

But to keep your scripts fast, leave definitions like this outside your loops.

General usage

Note that white space has no effect, so you can introduce spaces for clarity, except of course within variable names, function names and other keywords. Indenting control statements can be a great help in understanding a program. For example:

```
     For K = Kstart To Kstop
     X = K * Sqr (3)
          For N = NStart To Nstop
          Y = N * N
               If Y < FullScale Then
```

```
              . . . . . .
              . . . . . .
               End If   ' End of main calculation
          Next   ' End of N loop
     Next   ' End of K loop
```

If a section becomes very long, you could provide the end with a comment, to show where it comes from.

Arithmetic Operators

As with most other languages, the arithmetic operators are used as follows:

^	Exponentiation	A ^ B = A^B = A raised to the power B
/	Division	A / B = A divided by B
\	Integer division	A \ B = A divided by B, truncated to next integer below
*	Multiplication	A * B = A multiplied by B
+	Addition	A + B = B added to A
-	Subtraction	A B = B subtracted from A

Notes:
If there is any possibility that you will be taking the exponent of a negative number, make sure to trap any possible errors arising from such operations as trying to take the square root of a negative number. Logs of negative numbers are forbidden also.

If there is any possibility that you will be dividing by zero, make sure to trap this.

There are two ways of dealing with these types of problem. One is to prevent it happening by making suitable tests before the calculation is performed. The other is to let it happen, and use an error handling routine. This will be discussed later.

Normally in VBScript you will know the range of the data, since all the incoming data are, by definition, integer (unscaled data) or real (scaled data), and they must fit into the screen of the instrument.

Results of Calculations

Sometimes you may see a statement like this:

```
A = A * A * (Cos (A) + Sin (A) )
```

The program takes the quantity represented by A and performs all of the following operations, using that original value:

1. Multiply A by itself.
2. Calculate the cosine of A.
3. Calculate the sine of A.
4. Add the cosine and the sine together.

5. Multiply that result by the square of A.

At this point, the quantity represented by A has not been changed. Only at the end of the calculation is the final value placed in the memory location labeled A.

Note that you can write more than one statement on a line, separated by colons, like this

```
A = B * Cos (34 * Theta) * Sin (55 * Theta) : B = A * A + Z * Z
```

Order of Calculations

Operations are performed in the following order:

1. Contents of brackets
2. Exponentiation
3. Division and multiplication
4. Addition and subtraction

If there is any doubt as to how the calculation will be done, use brackets. These will also make the order of the calculations clear to any reader of the program, which is desirable if you are to give it to a customer, who will want to know what was intended.

Here are some examples of the uses of brackets:

Brackets are worked out before any other operations are performed.

Use brackets to force the result you want, and also to clarify a calculation.

A	1 1 1 1 1 1 1 1	255	0 1 0 1 1 0 1 0	90	(B OR C) AND (D OR E)
B	1 1 1 1 0 0 0 0	240	0 0 0 0 0 0 0 0	0	B OR (C AND D) OR E
C	1 0 1 0 1 0 1 0	130	1 1 1 1 1 0 1 0	250	B OR (C AND (D OR E))
D	0 1 0 1 0 1 0 1	85	0 1 0 1 1 1 1 1	95	((B OR C) AND D) OR E)
E	0 0 0 0 1 1 1 1	15			
F	0 0 0 0 0 0 0 0	0			

A	7	315	A * B * (C / D) * E * F
B	6	8.75	A * B * C / (D * E * F)
C	5	35	A * B * (C / (D * E)) * F
D	4		
E	3		
F	2		

Check these results to see whether any errors, deliberate or otherwise, have been introduced.

These results are from file Brackets.Xls. You can make a copy of that file in order to experiment with different combinations of brackets.

VBS Controls

```
Do                  Do                  Do
. . . .             . . . .             . . . .
Loop         Loop Until . . . .         Loop While
Do Until     Do While                   Exit Do
. . . .      . . . .
Loop         Loop
For . . . Next                          Exit For
```

GoTo__**This is not allowed in instrument VBS.**

```
If . . . . Then . . . . _' On one line__
If . . . Then
ElseIf . . . Then
End If
If . . . Then . . . End If__
If . . . Then . . . Else . . . End If
Select Case
. . . .
End Select
While
. . . .
Wend
```

Choose the construction that best satisfies the requirements of speed and clarity.

The construction GoTo LabelledStatement is available in many languages, including VBA, but not in VBS. ***GOTO*** is not allowed in VBS.

IF . . . Then . . . Else . . . End If

A very simple example:

```
If A >= 0 Then B = Sqr (A) 'Take the square root of A if A is not negative.
```

```
    If A + B < C + D Then E = F : G = H_ 'No End Is needed if all on one
line.
```

If you need to perform a longer procedure, make this construction:

```
If A >= 0 Then
B = Sqr (A)
C = 32766 * Sin ( TwoPi * B / PeriodOfSinusoid)
End If      ' End If is needed to terminate the construction.
```

The If statement is very often used with the following Boolean expressions:

A > B	A is greater than B
A >= B	A is greater than B or equal to B
A = B	A is equal to B
A < B	A is less than B
A <= B	A is less than B or equal to B
A <> B	A is not equal to B

These statements are not like the usual program statements, such as A = B. These statements are Boolean (logic) statements, which can take the values True or False. You may even see things like "If A Then B", which means that if A is True, B gets done.

In the first example, if A is negative, we might want to write something like this:

```
If A >= 0 Then
B = Sqr (A)
Else
B = 0
End If
```

and in fact you can make some very complex constructions using If, as in the examples below:

```
If A < 0 Then
    If A < - 1 Then
    Z = 17
    Else_
    Z = 31
    End If
```

```
Else_
    If A > 3 Then
    Z = 63
    Else
    Z = 127
    End If
End If
If A > 0 Then
    If B > 0 Then
    Z = Y
    End If
End If
```

This is equivalent to:

```
If ( (A > 0) And (B > 0) ) Then
Z = Y
End If
```

Summary of If . . . . Then . . . . Else

```
If Boolean Then AnyVBScriptingOnOneLine
If Boolean Then
AnyVBScriping
End If
If Boolean Then
AnyVBScripting
Else
AnyOtherVBScripting
End If
```

If you write a list like this, all the Booleans will be evaluated, whether you want that or not:

```
If A > 9 Then VBScripting1
If A > 7 Then VBScripting2
If A > 6 Then VBScripting3
```

```
If A > 4 Then VBScripting4
If A > 3 Then VBScripting5
If A > 1 Then VBScripting6
```

Be very careful when testing for equality. There will be no trouble with Integers, Long Integers, and Strings, but Real numbers are different. Because they have so many significant digits, values that should be equal, may differ minutely after a computation. It is safer with Real numbers to test using a tolerance band.

File for this example: IfThenElse.xls

If you find that you are building up a rather complicated set of Ifs, you might want to consider the Select Case construction.

Select Case

This is a very powerful construction, which is also easy to understand when written out. It is best for Integers and Strings, where exact values are always obtained. Here is a simple example:

```
Select Case K
     Case 7 : Y = 6 : Z = 3
     Case 7 : Y = Sqr (Sin (A) ) : Z = Sqr (Cos (A) )
     Case N : Z = Y + X
     Case Else :
End Select
```

Case N assumes that the value of N has already been set. Case Else is included to cover other cases, whether foreseen or not. It should always be included.

You can also provide lists of values.

```
Select Case K
     Case 1, 2, 3, 5, 8, 13 : Y = 55 : Z = 89
     Case 4, 9, 16, 25, 36 : Y = Sqr (Sin (A) ) : Z = Sqr (Cos (A) )
     Case 7, 15, 31, 63, 127 : Z = Y + X
     Case Else : Z = 3
End Select
```

Case N assumes that the value of N has already been set. Case Else is included to cover other cases, whether foreseen or not. It should always be included.

This is very much neater than a string of Ifs and Elses, but remember: you cannot use Select Case unless you are sure of exact equality, which allows you to compare integers and strings only. You cannot put Case > 5, for example. File for this example: SelectCase.Xls

Summary of Select Case . . . . End Select

```
     SelectCase VariableName
          Case Alist : VBScriptingA
          Case Blist : VBScriptingB
          . . . .
          Case Else : VBScriptingElse_ VBScriptingElse can be empty.
     End Select
```

Do . . . Loop

This construction is useful when you do not know at programming time how many times the loop will be executed. Here are some examples:

```
Do
AnyVBSCalculation
Loop Until D > Pi
Do Until Z < Y
AnyVBSCalculation
Loop
Do
AnyVBSCalculation
Loop While D <= Pi
Do While Y >=Z
AnyVBSCalculation
Loop
```

These constructions enable you to make the test before or after the calculation. If before, the calculation might not be done even one time, if the condition for terminating were already true. With the condition at the end, the calculation is done at least one time.

Sometimes you might want to exit the loop from somewhere inside: for example, if some kind of problem is looming, such as the logarithm of a negative number.

For this case, you can use **If . . . . Then Exit Do**.

To make a pause of 10 seconds you can write:

```
     NewTime = Timer + 10.0
```

```
Do Loop Until Timer >= NewTime
```

where Timer is a clock function in the PC, which has a resolution of one second.

Example file for these constructions: DoLoops.Xls

While . . . Wend

This is similar to Do While . . . Loop. You can write things like:

```
While ( (A > 2) And (C < 92677663) )
AnyVBCalculation
Wend
```

For . . . Next

Sometimes you know, or you think you know, the number of times that you want to do a job. For this case a For loop is ideal, especially when you have an array of numbers to work with.

Examples:

```
For K = 0 To Total
HistogramBin (K) = 0
Next
```

```
Omega = TwoPi / Period
For N = 0 To Period
Y (N) = A * Sin (Omega * N)
Next
```

Be careful about changing the counting variable in any loop. You can do this to terminate the loop early (but **Exit For** is better), but you could also prevent it from terminating at all.

For emergency exit, you can use **Exit For**. For example:

```
For K = 0 To Total
     If HistogramBin(K) = 0 Then Exit For
AnyVBScripting
Next
```

It is possible to make a For loop with steps greater than 1, as in the following example in which K takes the values 3, 7, 11, 15, ... 83.

```
For K = 3 To 82 Step 4
AnyVBScripting
```

```
Next K
```

You may place loops inside one another (nested loops), but they must all use different control variables. Example:

```
For K = 0 To N
VBScriptingK
    For L = - 7 To 17
    VBScriptingL
        For M = S To T
        VBScriptingM
        Next
    Next
Next
```

VBS keywords and functions

The ones in *italics* do not apply to the instrument.

+	Add two values or concatenate two strings.
-	Subtract two values.
*****	Multiply two values.
/	Divide two values.
****	Divide two values to obtain an integer result
Abs	Make absolute value.
Asc	Make ASCII value of a character.
Atn	Make $\tan^{-1}$ of a value. Result in range from $-\pi/2$ to $+\pi/2$ radians.
Cdbl	Convert a value to double precision floating point.
Chr	Create a character from an integer in range 0 to 255.
Cint	Convert a value to nearest integer in the range -32768 to +32767
Clng	Convert a value to nearest long integer in the range -2^{31} to $+2^{31} - 1$.

Close	*Close a file.*
Cos	Make the cosine of an angle expressed in radians.
Csng	Convert a number to single precision floating point.
Cstr	Convert a variable to a string.
Exp	Raise e to the power of the input.
Get	*Get a value from a file.*
Input	*Get some ASCII data from a file.*
Instr	Find the position of a string in a longer string.
Int	Convert to nearest integer below the input value.
Left	Take some characters at the left end of a string.
Log	Take the natural logarithm of a positive value.
Ltrim	Remove spaces at the left end of a string.
Mid	Take or insert some characters in a string.
Mod	Take the modulus of a value in terms of another value.
On Error	Take some action if an error occurs.
Open	*Open a file.*
Print	*Send some ASCII data to a file.*
Put	*Send some data to a file.*
Randomize	Randomize Timer re-seeds the pseudo-random number generator.
Read	*Read from a file.*
Right	Take some characters at the right end of a string.
Rnd	Make a random real number in the range from 0.0 to 1.0
Rtrim	Remove spaces from right hand end of a string.
Sin	Make the sine of an angle expressed in radians.
Sqr	Make the square root of a positive number.

Str	Make a string from a numerical value.
Timer	Time since midnight in seconds, with a resolution of one second.
Trim	Remove leading and trailing spaces from a string.
Val	*Get the ASCII value of a string beginning with numerical characters.*

Other VBS Words

Const	Define a constant value.
Dim	Dimension a variable.
Redim	Dimension a variable again.
Boolean	Boolean variable
Double	Double precision real variable.
Integer	Integer in the range -32768 to + 32767
Long	Long integer in the range -2^{31} to $+ 2^{31} - 1$
Single	Single precision real variable
String	String variable
And	Logical AND
Or	Logical OR

To make a bit-by-bit comparison, logical constructions can be used with variables, as in A and B, or with tests such as If A > B Then . . .

Functions

These are mainly of the form C = F (A), where A is the argument, or input to the function.

Abs	Abs (A) calculates the absolute value of an integer or a real number, so the result is always positive or zero. A can be any number in the range of the VB system.
Atn	Atn (A) calculates the angle of which A is the tangent. Because infinitely many angles can have the same tangent, the output of Atn always lies in the range minus π / 2 to plus π / 2. The input can be any positive or negative value in the

	range of the VB system.
CDbl	CDbl (A) calculates a double precision real variable, equal to A.
CInt	Cint (A) calculates the integer value nearest to A, which can be any acceptable VBS number. Cint (-7.4) = -7. Integers are signed 16-bit values in the range -32767 to + 32767.
CLng	CLng (A) calculates the nearest long integer to the value A. Long integers are signed 32-bit values in the approximate range -21.5 M to + 21.5 M.
Cos	Cos (A) calculates the cosine of any integer or real number, giving an output that is never greater than plus one or less than minus one.
CSng	CSng (A) calculates a single precision real variable equal to A.
Exp	Exp (A) calculates the value of e^A.
Int	Cint (A) calculates the integer value next below A, which can be any acceptable VBS number. Int (-7.4) = -8.
Log	Log (A) calculates the natural logarithm (to base e), of any acceptable VBS number greater than zero. A negative number or zero will create an error. To calculate $Log_{10}(A)$, use $Log_{10}(A) = Log(A) / Log(10)$
Mod	A Mod (B) calculates the modulus of A, which is the remainder after A has been divided by B. 34 Mod 8 = 2. 34 Mod 55 = 0. -34 Mod 13 = -8. 21 Mod -8 = 5.
Randomize	Calculates a new seed for the pseudo-random number generator. Randomize Timer uses the real-time clock for this purpose.
Sin	Sin (A) calculates the sine of any integer or real number, giving an output that is never greater

	than +1 or less than -1.
Sqr	Sqr (A) calculates the square root of any integer or a real number that is not negative. If A is negative, an error will occur.
Timer	Time since the previous midnight in whole seconds.

Hints and Tips for VBScripting

Set the trigger to Single or Stopped if you need to do a lot of editing: it is faster.

Before starting a script, remove any existing scripts that you do not need. This is because errors in an existing script will give you error messages, even if your current script is perfect. And an existing good script may develop a fault if you change the setup. For example, you might change the vertical scale or the memory length and get an overflow if you did not guard against it in the script.

When starting a script, make sure that you have chosen the right kind: function or parameter. You can get some very frustrating problems if you are in the wrong mode. You can cut and paste the VBS statements if you discover this error.

If your calculation requires a long memory, development might be quicker if you test the principles on a shorter trace at first.

Note that the pseudo-random number generator is reset at the start of a script. If you want a different set of pseudo-randoms every time, put Randomize Timer in the program, to be run once, before any pseudo-randoms are generated. You can use this instruction to re-seed the generator at any time during execution.

Do not put the final statement in a loop, hoping that you can see a progressive result as some parameter changes. No output will be seen on the screen of the instrument until the script has been completely run and quitted, so only the final result will appear. If the loop runs many times, you will think that the scope has hung up.

If you want a For loop, end it with "Next" and not "Next X".

If you make a script that takes a long time to run, go back to the default setup before quitting or powering down, or you will have a long wait next time you power up.

Always use a recursive calculation when this will speed things up.

Keep everything outside a loop that does not have to be inside, to speed things up.

Make your scripts clear, not only by indenting and commenting, but by structuring neatly as well.

Sometimes it might be easier to develop your script in Excel VBA (remembering that VBA is not identical to VBS), so that you can display intermediate results. If you do this, note that you can read from a cell or write to it using statements like these:

```
A = Worksheets("Sheet1").Cells(Row, Column).Value
Worksheets("Sheet1").Cells(Row, Column).Value = B
```

Note that in VBS, after you have corrected an error and clicked on "Apply," the error message may go on flashing for a few seconds, or a few acquisitions, before being erased. Look for the "Script OK" message. Be patient before assuming that you still have a bug.

If your calculation requires data to be used at some other horizontal positions than their original ones, make sure that your algorithm does not try to send data to non-existent array positions, that is, beyond the edges of the screen. You may have to truncate your output trace, as happens with the instrument's Enhanced Resolution and Boxcar functions.

No output will emerge from a script until you press Apply.

No output will emerge from a script until it has received an input. This includes the case where the input data are not used in calculating the output data. So you must have had at least one acquisition before you see anything.

Because you can introduce undeclared variables at any point in a calculation, VBS does not check your spelling.

You can make a portion of a trace disappear if you set the values to 32767 or -32768.

You can highlight a section of a trace by making the points alternately too high and too low by a suitable amount. Providing the memory length is not too short, the compaction algorithm will give the effect of a thicker trace.

The lengths of the output trace and the input trace need not be the same. You can even make the output trace longer than the input trace, but you will need to unzoom it to see it all. This feature can be used to avoid compaction problems with non-linear horizontal scales. It can also be used to show several versions of a function at the same time, without having to set up a separate script for each one.

If your program structure is complicated, consider typing all the IFs, ELSEIFs, ENDIFs, FORs, NEXTs, etc and then clicking Apply. You wont get any output, but the system will tell you if the structure is acceptable. Then you can insert the actual program statements.

Always try to make the script as independent as possible of variables such as V/Div, T/Div, and memory length, unless that would make it harder to understand. If so, give some values as examples, and explain how the script would have to change if the variables changed.

ERRORS

The instrument VBS tries hard to help you when errors occur.

Errors may be of two main types:

- The script may not be usable because the interpreter cannot construct a logical structure from it.
- The script may be usable, but may fail while running because an incomputable function has been requested.

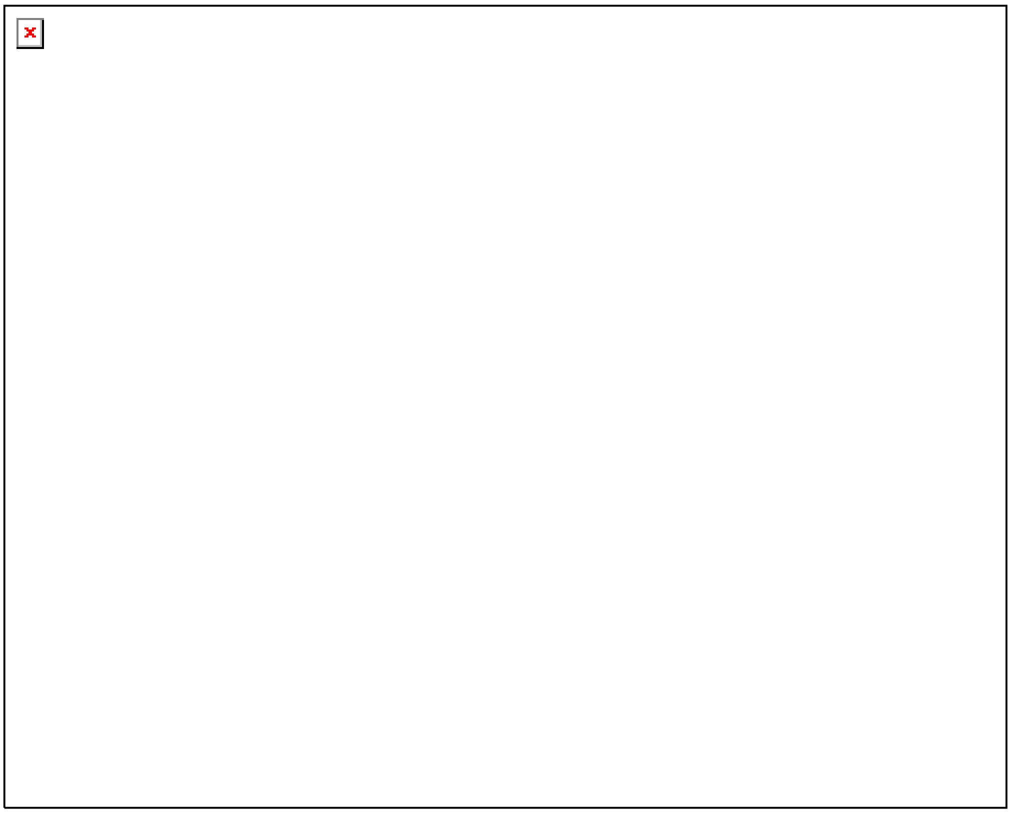
Sometimes the line number given for an error is wrong. This can happen when the error is of this general type:

```
Missing "Next" Missing "End If"

Extra "Next" Missing "Until" etc.
```

This happens because VBS cannot know where you should have put the statement.

If at some point during the calculation of an output array, a value goes outside the allowed range, the calculation will stop, and you will see the new values up to the point of the stoppage. To the right of that point, the trace will display the previous values. In fact, if you deliberately recalculate only a part of a trace, you can have a mixture of new and old values.

In the figure below is a type of error message that you may see if one of your calculations has tried to set a value outside the range -32768 to +32767. It takes extra time to guard against this, but unless you are sure that it will not happen, you need some kind of check. In the example on the next page, the red trace has gone outside the allowed range at the beginning, resulting in the message at the bottom of the instrument screen: This array is fixed or temporarily locked: `OutResult.DataArray`.



Error Handling

Note that the construction OnError GoTo Label: is not allowed in VBS. In fact no GoTos or labels are allowed. Therefore there is no way for you to provide handlers to deal with errors and exceptions. You must be aware of all possibilities at all points in your program, and you must either be certain that errors will not occur, or you must take action to ensure that they do not.

Examples:

Sqr	You cannot take the square root of a negative number.
Log	You cannot take the log of zero or of a negative number.
A / B	You cannot divide by zero.
Array	You cannot use an index outside the bounds of an array.
Size	Unscaled data cannot go outside the range -32768 to 32767.

If there is any possibility that any of these might occur, take steps to deal with this before it can happen.

For example, you may write some kind of generator of pseudo-random statistical values. If these belong to a distribution that in principle has an infinite range, or a finite range which is wider than the signed 16-bits allowed, check each value. If a value falls outside the range, you could set it to the maximum or generate another example.

You can, however, use one of the following:

```
On Error Resume Next
```

followed by some code that may make some attempt to deal with the problem, or at least to allow execution to continue.

```
On Error GoTo 0
```

This cancels On Error Resume Next_

Speed of Execution

To maximize the speed of execution of a script, the most important thing you can do is to minimize the number of operations that are performed inside loops. Anything done once only is unlikely to be an important source of delay. Please note that VBS is much slower than the internal computations of the instrument, so do everything you can to save time, unless time is irrelevant to the application.

Using an array element takes longer than using a single variable. Here is an example:

```
For K = 1 to Total
```

```
        If X (K) > X (K - 1) Then
        Y = Cos (X (K) ) * Sin (X (K) ) * Sqr (X (K) )
        End If
    Next
```

To do the same thing we could also write this, using the index only once:

```
OldXK = X (0)
    For K = 1 To Total
    XK = X (K)
        If XK > OldXK Then
        Y = Cos (XK) * Sin (XK) * Sqr (XK)
        OldXK = XK
        End If
    Next
```

VBS runs slower than the "internal" calculations, because the scripts are interpreted. This could be serious for calculations where many operations are needed on each sample, such as convolution, correlation, and long digital filters.

Scripting Ideas

What can we do in a VBS script that we cannot do with the normal instrument functions? Here are some possibilities.

- Create a new function that acts on waveform values.
- Create a new parameter.
- Create a new form of non-linear vertical scale.
- Create a new form of non-linear horizontal scale.
- Move some or all data horizontally, including reflections.
- Combine data to form digital filters.
- Show several function results side by side.
- Show several function results interleaved.

You can even create output data that are not related to the input. The output data need not even be in the same domain as the input data, because the system treats them as pure numbers. So you can create your own transforms into the frequency domain, for example.

Debugging Scripts

Until we have integrated a more comprehensive debugger for VBScript there is a workaround.

1. Download the Windows Scripting Debugger for Windows 2000 from here:

http://download.microsoft.com/download/winscript56/Install/1.0a/NT45XP/EN-US/scd10en.exe

2. Enable JIT (Just In Time) debugging by setting the following registry key

HKCU\Software\Microsoft\Windows Script\Settings\JITDebug = to 1 (DWORD value)

3. Place a **Stop** statement in your script.

Now, when the Stop statement is executed the debugger will open and allow single-stepping, variable examination, etc.

Using VBA or Visual Basic to debug VBScripts is not recommended since the language syntax for these three variants of basic is slightly different.

Horizontal Control Variables

InResult.HorizontalOffset	Double precision	Time shift of input waveform on grid in units of horizontal scale
OutResult.HorizontalOffset	Double precision	Time shift of output waveform on grid in units of horizontal scale
InResult.HorizontalPerStep	Double precision	Time between successive samples in the input waveform
OutResult.HorizontalPerStep	Double precision	Time between successive samples in the output waveform
InResult.HorizontalUnits	String	Horizontal units of input waveform
OutResult.HorizontalUnits	String	Horizontal units of output waveform
InResult.Samples	Integer	Number of samples in input waveform

Vertical Control Variables

InResult.VerticalOffset	Double precision	Vertical shift of input waveform on grid
OutResult.VerticalOffset	Double precision	Vertical shift of output waveform on grid
InResult.VerticalPerStep	Double precision	Difference between successive possible levels in the input waveform memory
OutResultVerticalPerStep	Double precision	Difference between successive

		possible levels in the output waveform memory 1 / 65536 of vertical full scale
InResult.VerticalResolution	Double precision	Difference between successive possible physical levels in the input waveform
OutResultVerticalResolution	Double precision	Difference between successive possible physical levels in the output waveform 1 / 256 of vertical full scale for channel waveforms 1 / 65536 of vertical full scale for math waveforms
InResult.VerticalUnits	String	Vertical units of input waveform
OutResult.VerticalUnits	String	Vertical units of output waveform

List of Variables Available to Scripts

FirstEventTime([out, retval] VARIANT * pVal); FirstEventTime([in] VARIANT newVal);

LastEventTime([out, retval] VARIANT * pVal); LastEventTime([in] VARIANT newVal);

UpdateTime([out, retval] VARIANT * pVal); UpdateTime([in] VARIANT newVal);

Details([in] BSTR strDetailsIID, [out, retval] VARIANT * pVal);

Status([out, retval] VARIANT * pVal); Status([in] VARIANT newVal);

ExtendedStatus([out, retval] VARIANT * pVal); ExtendedStatus([in] VARIANT newVal);

StatusDescription([out, retval] BSTR * pVal); StatusDescription([in] BSTR newVal);

```
DataArray([in, defaultvalue(TRUE)] BOOL arrayValuesScaled,
        [in, defaultvalue(LEC_ALL_DATA)] int numSamples,
        [in, defaultvalue(0)] int startIndex,
        [in, defaultvalue(1)] int sparsingFactor,
        [out, retval] VARIANT *pArray);
DataArray([in, defaultvalue(TRUE)] BOOL arrayValuesScaled,
        [in, defaultvalue(LEC_ALL_DATA)] int numSamples,
        [in, defaultvalue(0)] int startIndex,
```

[in, defaultvalue(1)] int sparsingFactor,

[in] VARIANT array);

HorizontalUnits([out, retval] BSTR *pVal); HorizontalUnits([in] BSTR newVal);

Samples([out, retval] int *pVal); Samples([in] int newVal);

HorizontalResolution([out, retval] double *pVal); HorizontalResolution([in] double newVal);

HorizontalPerStep([out, retval] double *pVal); HorizontalPerStep([in] double newVal);

HorizontalOffset([out, retval] double *pVal); HorizontalOffset([in] double newVal);

Sweeps([out, retval] int *pVal); Sweeps([in] int newVal);

HorizontalVariances([out, retval] int *pVal); HorizontalVariances([in] int newVal);

HorizontalVarianceArray([out, retval] VARIANT * pArray);

HorizontalVarianceArray([in] VARIANT array);

HorizontalFrameStart([out, retval] double *pVal); HorizontalFrameStart([in] double newVal);

HorizontalFrameStop([out, retval] double *pVal); HorizontalFrameStop([in] double newVal);

VerticalFrameStart([out, retval] double *pVal); VerticalFrameStart([in] double newVal);

VerticalFrameStop([out, retval] double *pVal); VerticalFrameStop([in] double newVal);

VerticalResolution([out, retval] double *pVal); VerticalResolution([in] double newVal);

VerticalPerStep([out, retval] double *pVal); VerticalPerStep([in] double newVal);

VerticalOffset([out, retval] double *pVal); VerticalOffset([in] double newVal);

VerticalMinPossible([out, retval] double *pVal); VerticalMinPossible([in] double newVal);

VerticalMaxPossible([out, retval] double *pVal); VerticalMaxPossible([in] double newVal);

VerticalUnits([out, retval] BSTR *pVal); VerticalUnits([in] BSTR newVal);

Communicating with Other Programs from a VBScript

The ability of The instrument to communicate with other programs opens up immense possibilities, both for calculation and for graphics, making the assembly of reports relatively simple.

Communicating with Excel from a VBScript

Although there are direct instrument calls to Excel and other programs, you may wish to do this from a VBScript. Here is an example:

```
OutResult.Samples = InResult.Samples

startData = 0

endData = OutResult.Samples
```

```
ReDim newData(OutResult.Samples)

USD = InResult.DataArray(False)

LastPoint = endData - 1

Set ExcelApp = GetObject(,"Excel.Application")

ExcelApp.Visible = True

ExcelColumnA = 2      'Column where the data will appear in Excel

ExcelRow = 10         'Row where the data will start

ExcelColumnB = 3     ' Column where the output data will appear in Excel

     For K = 0 To LastPoint

     ExcelApp.ActiveSheet.Cells("ExcelRow + K, ExcelColumnA ") = -USD(K)

     Next
```

Once the data are in Excel, any Excel functions can be applied to the data. The results can be returned to the VB script.

```
     For K = 0 To LastPoint

     NDA(K) = ExcelApp.ActiveSheet.Cells("ExcelRow + K, ExcelColumnB")

     Next
```

Transferring data cell by cell is very slow, so it is better to do a block transfer.

Calling MATLAB from The Instrument

Note: Load MATLAB version 6.5 just as you would on any PC. Once it is loaded, open MATLAB from the desktop, then close it again, before you attempt to open it from the instrument application. This is to update the registry.

MATLAB can be directly called from the instrument in two ways:

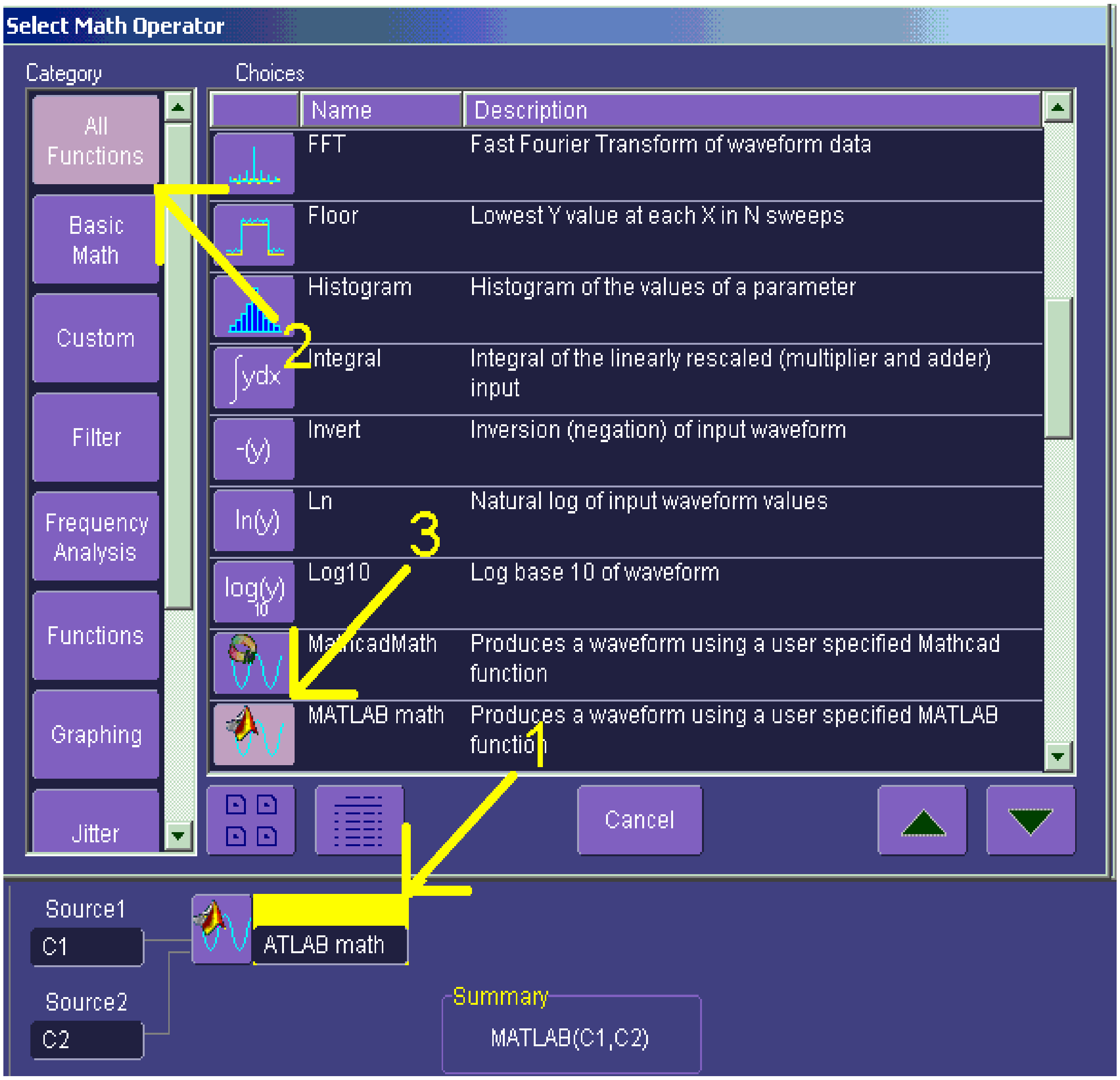
Using a function	F1 through Fx [The number of math traces available depends on the software options loaded on your scope. See Specifications.]	MATLAB returns a waveform
Using a parameter	P1 through Px	MATLAB returns a parameter

In both cases, one call to MATLAB can use two separate waveforms as input, providing much greater computing power than is available by calling MATLAB from a VBScript.

Note: If you do not place a semicolon ";" at the end of a line, MATLAB will show the calculated value in the result window, significantly slowing down the processing rate. This feature is best kept for diagnostics.

How to Select a Waveform Function Call

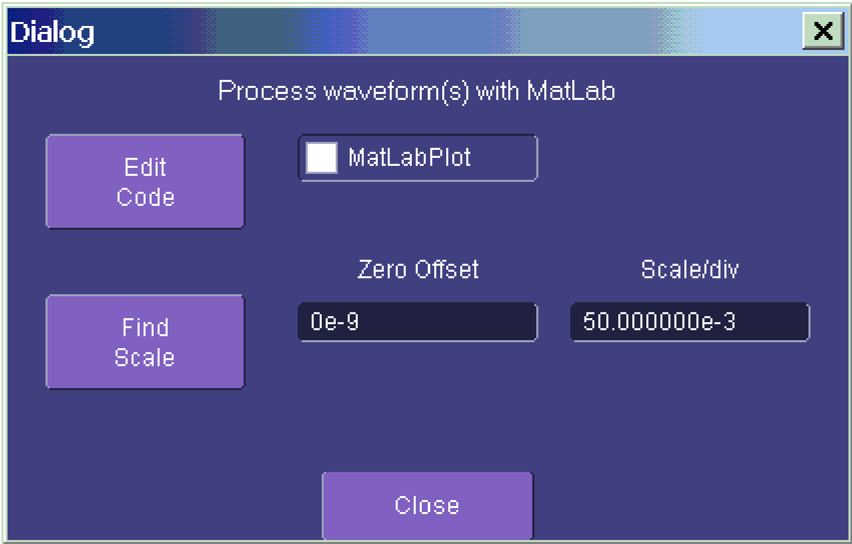
The MATLAB Waveform functions are selected from the **Select Math Operator** menu. Please note that once you have clicked on "MATLAB Wave" there will be a slight pause before MATLAB starts.



Source 1 and Source 2 are the waveforms that MATLAB will use.

The MATLAB Waveform Control Panel

Once you have invoked a MATLAB waveform call, you will see the zoom dialog at the right of the screen. Touch the MATLAB tab to see a panel like this:

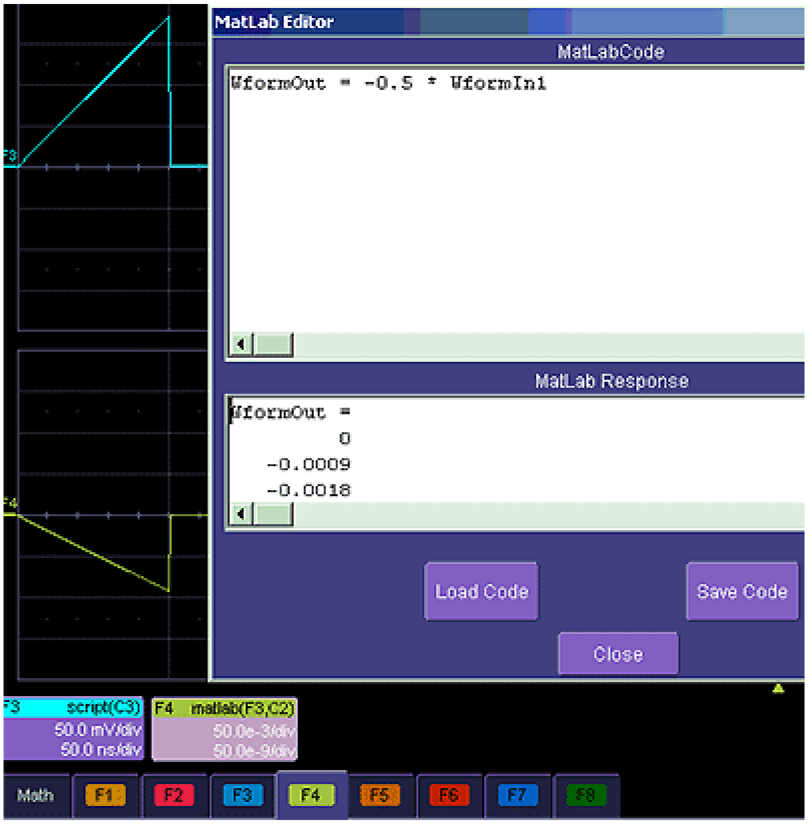


.

Touch **Find Scale** to make your output fit the grid, or use the text boxes to choose a scale.

MATLAB Waveform Function Editor -- Example

By touching **Edit Code**, you can reach the MATLAB Editor where you will see the default waveform function. If you are familiar with MATLAB, you might prefer to launch MATLAB and create a MATLAB function that performs your task. Your program in the instrument could then be a one-line call of your MATLAB function.

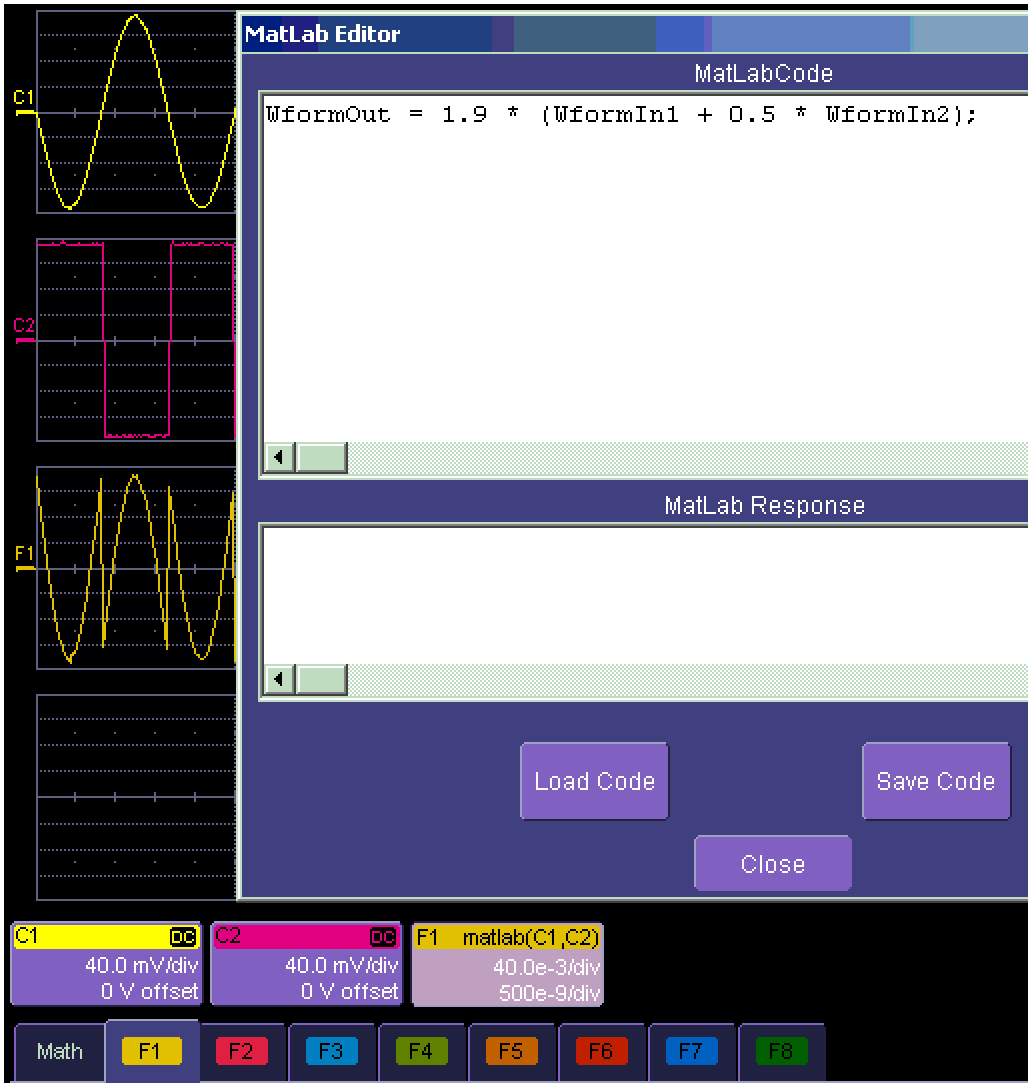


This is the default waveform function, with one important change – the semi-colon (;) has been removed from the end of the line. If the semicolon is present, your function will run much faster, because the output values will not be shown in MATLAB Response. With a long waveform, the time needed to display it could be quite long. The response values can be useful during development and debugging. Any line without a semicolon will produce a visible MATLAB Response.

From this panel you can save your code, load a previous code, and edit your function. A powerful feature of MATLAB is that you can refer to an entire waveform as a vector. The two input waveforms are WformIn1 and WformIn2, while the output is WformOut. You can also refer to individual samples, such as WformIn1(34), and sequences of samples, such as WformIn(55:89)

You can write statements such as these:

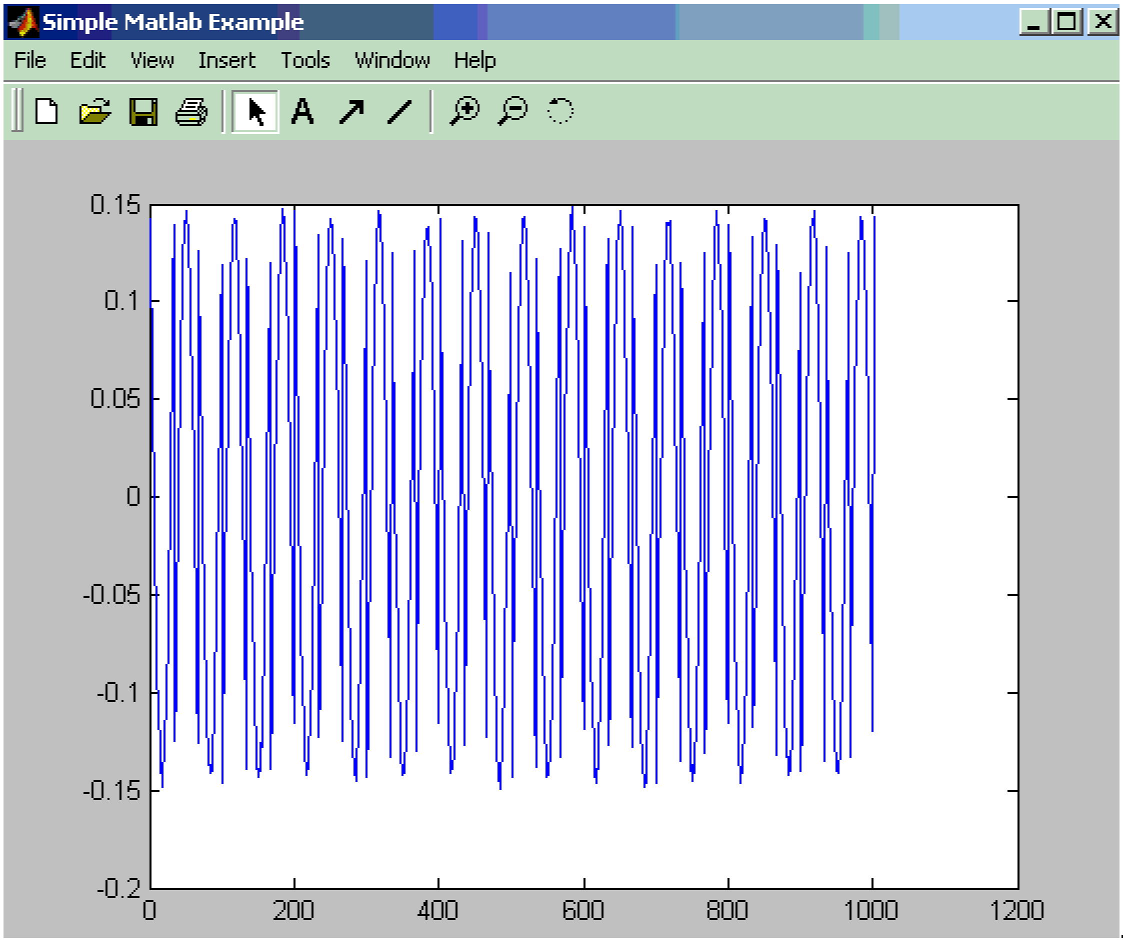
```
WformOut(5) = WformIn(5)

WformOut(89) = WformIn(144)

WformOut(34:55) = WformIn(34:55)

WformOut(233:377) = WformIn(100:244)
```



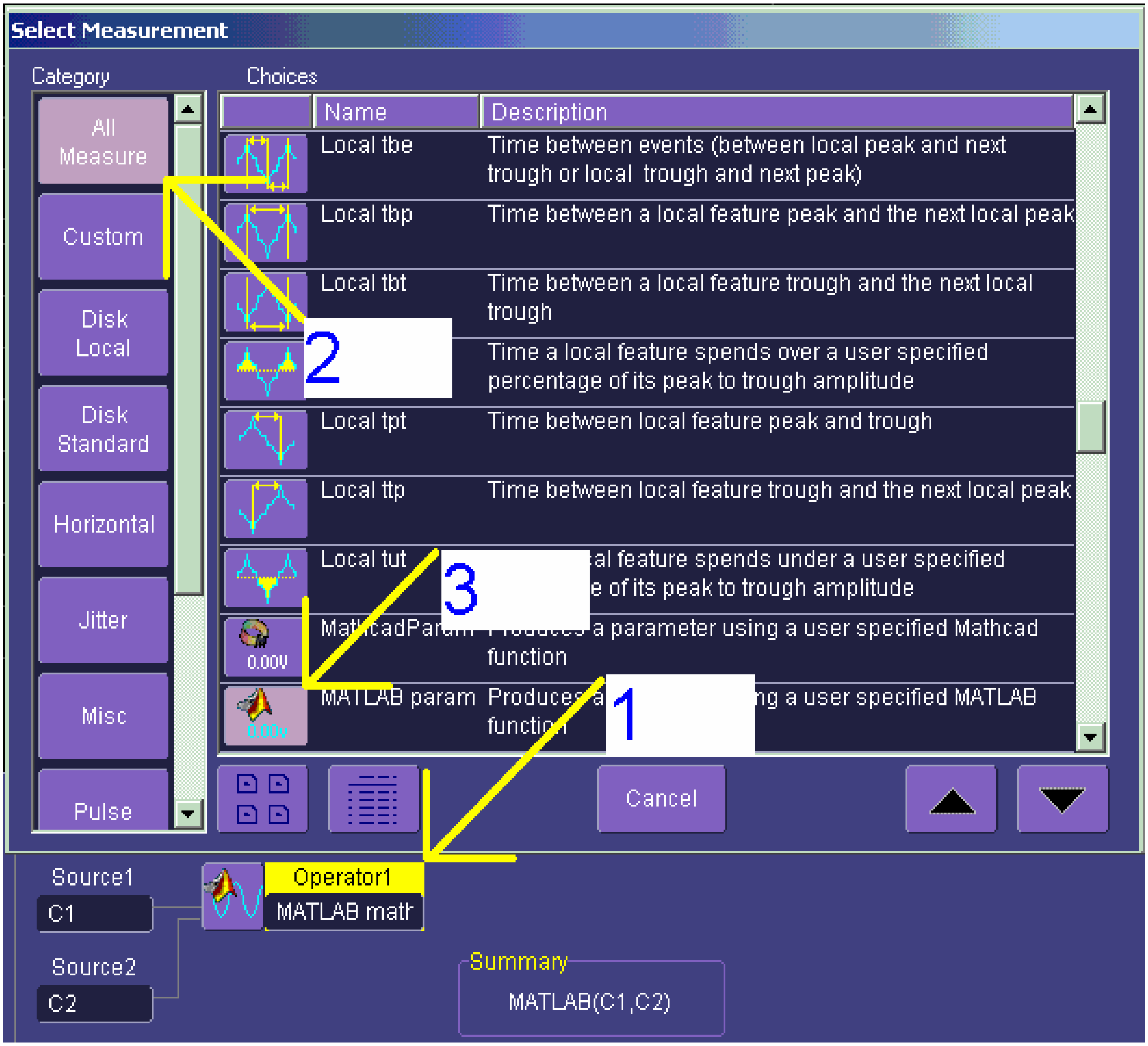
This very simple example adds a rescaled copy of Channel 2 to a copy of Channel 1, and then rescales the result.

MATLAB Example Waveform Plot

If you touch the **MATLAB Plot** checkbox you will see a MATLAB plot like this one:



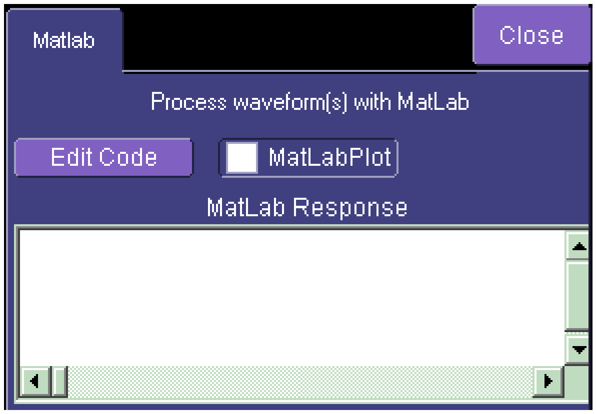
How to Select a MATLAB Parameter Call



Menu position for MATLAB parameter call in **Select Measurement** menu.

The MATLAB Parameter Control Panel

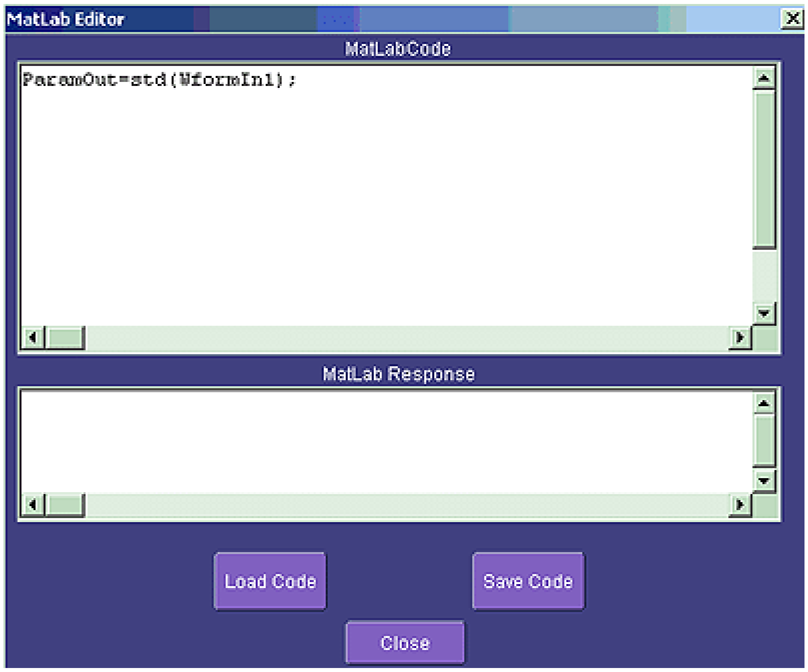
Once you have invoked a MATLAB parameter call, a mini-dialog to the right of the main dialog will appear:



You can touch the **MATLAB Plot** checkbox if you want to see a plot in MATLAB as well as getting a result in the instrument.

The MATLAB Parameter Editor

By touching **Edit Code**, you can reach the MATLAB Editor:

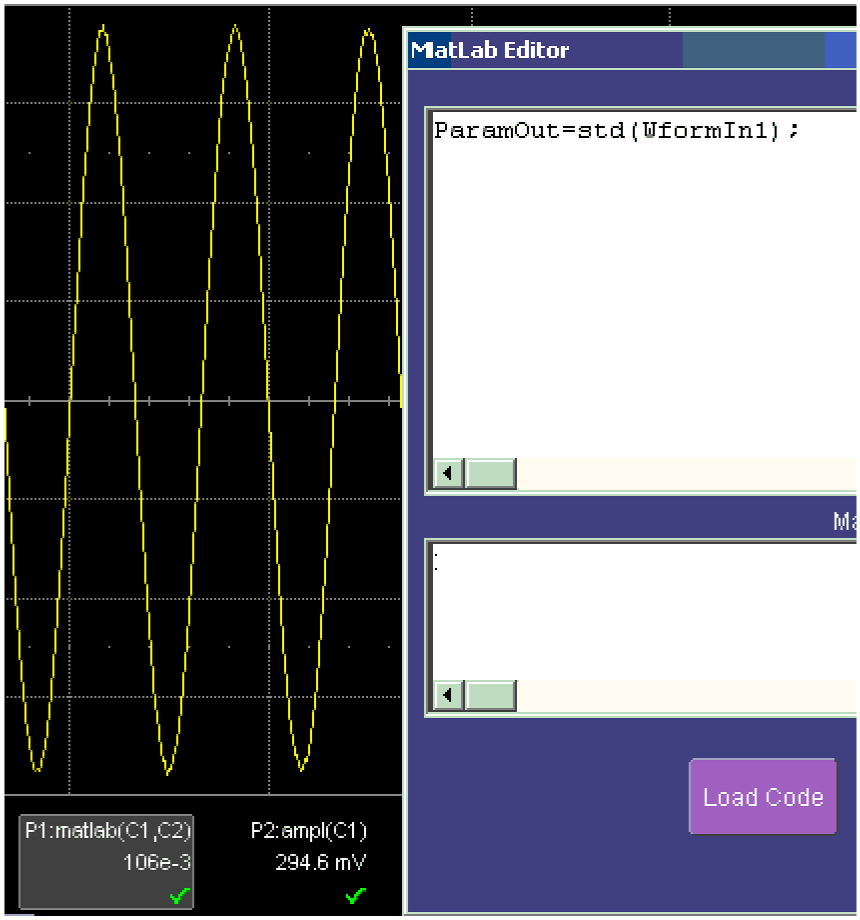


This simple example shows the MATLAB function Standard Deviation acting on input channel 1, and the result would be shown in the **MATLAB Response** pane for an amplitude of 0.15 volt.

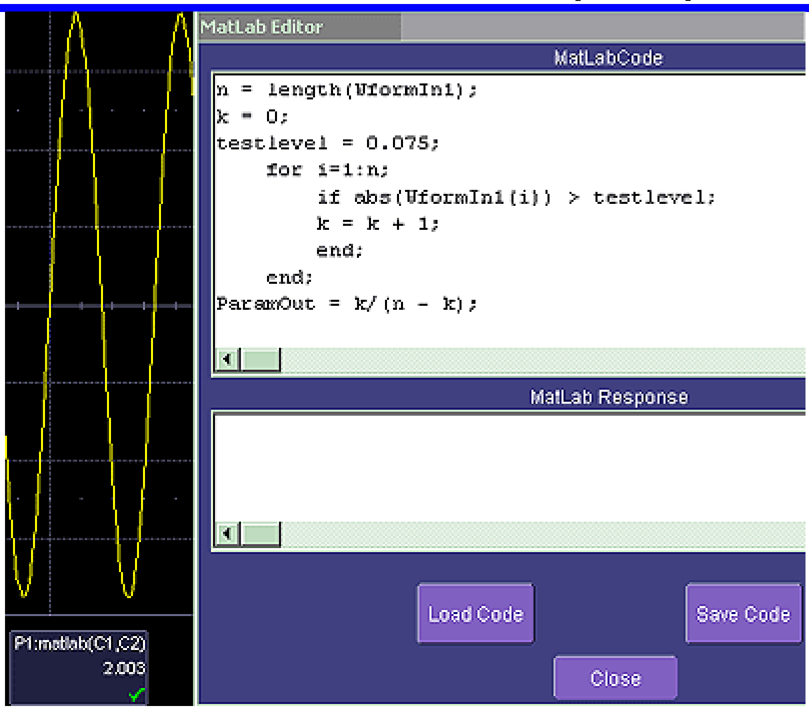
You can load an existing MATLAB program, using the **Load Code** button, and you can save the current program, using the **Save Code** button.

If you are familiar with MATLAB you might prefer to launch MATLAB and create a MATLAB function that performs your task. Your program in the instrument could then be a one-line call of your MATLAB function.

MATLAB Example Parameter Panel

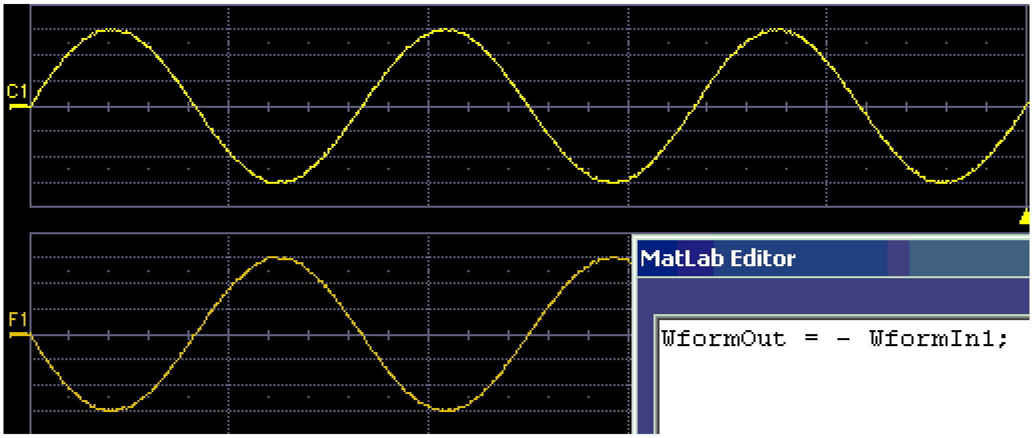


The next example calculates the ratio of the number of data points that are above a given level to the number of points below the level, in this case one half of the amplitude.

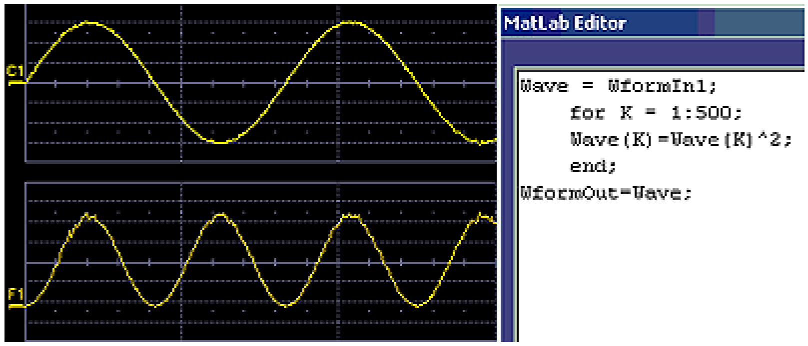


Further Examples of MATLAB Waveform Functions

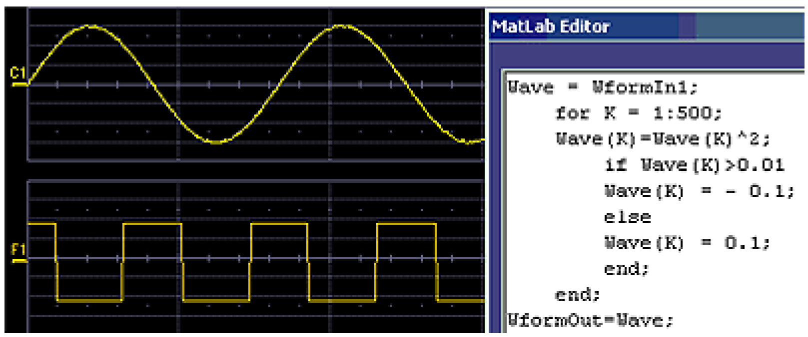
Negate the input signal:



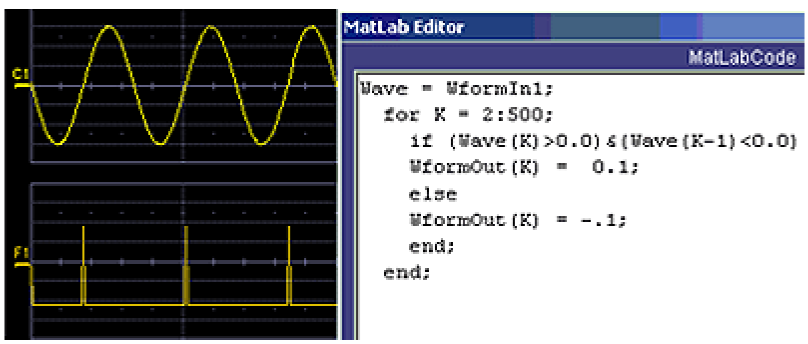
Square the input signal:



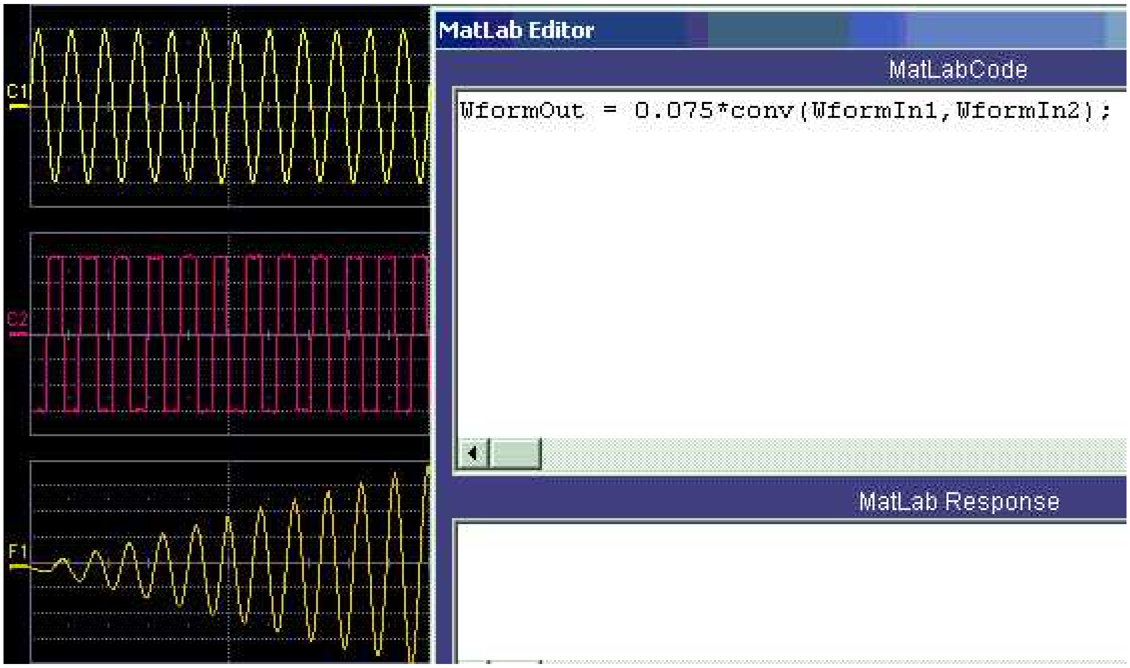
Create pulses from a sinusoid.



Create pulses at the zero crossings of the signal.



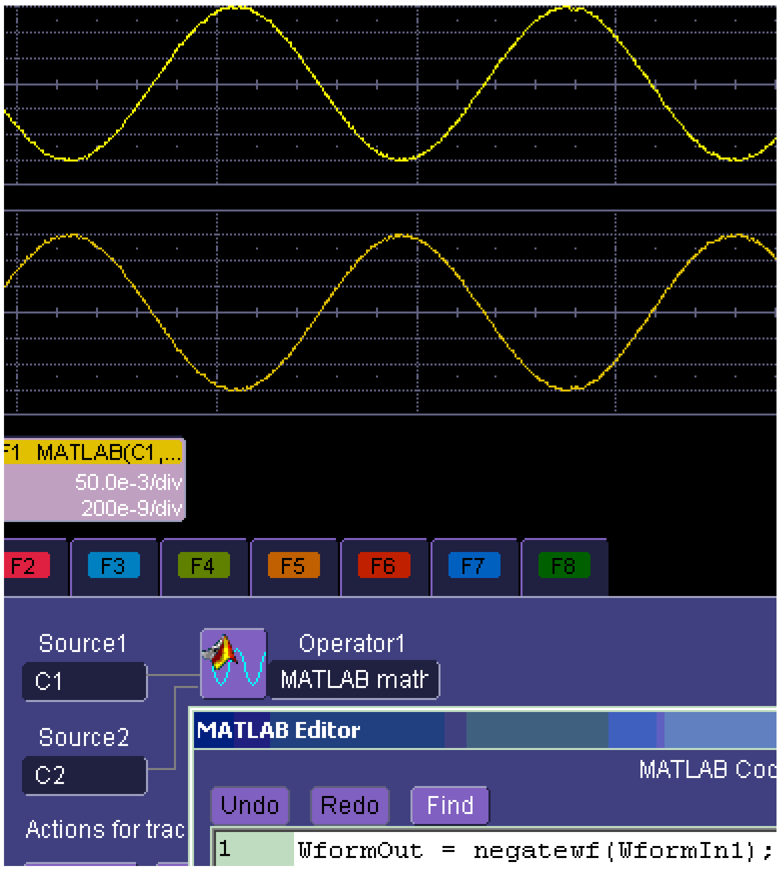
Convolve two signals.



Creating Your Own MATLAB Function

The procedure is simple. Create a MATLAB function using any text editor, and save it as a MATLAB m-file by giving it a name of the form Filename.m. Call the function using the MATLAB math editor or the MATLAB parameter editor as appropriate. A simple example is shown below.

```
function out = negatewf(wf1)
      % NEGATEWF changes the sign of all the data.
out = -wf1;
```



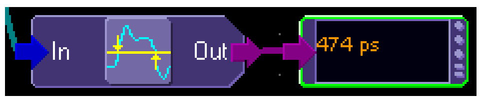
PROCESSING WEB OPTION

This feature is available with the XWEB software option.

The Processing Web provides a graphical way to quickly and easily set up math functions and parameter measurements. Using the Processing Web, you can chain together many more math-on-math functions than you can using the Math Setup dialog, where you are limited to two functions. In addition, you can insert a parameter measurement for any math output waveform anywhere in the web.

The "web" analogy derives from the nodes and connecting lines used to construct the web. Nodes are math functions selected from the **Add Math Processor** menu, parameters selected from the **Add Measure Processor** menu, or parameter math functions from the **Add Parameter Math Processor** menu.

Another key feature of the Processing Web is that you can preview your waveform at any math or parameter node in the web. Math previews are thumbnail images of the waveform. For parameters, the statistic displayed is the value of the last acquisition:



Statistic for Width Parameter

Once you have created a Processing Web setup, you can save and recall it for future use, the same as for any panel setup.

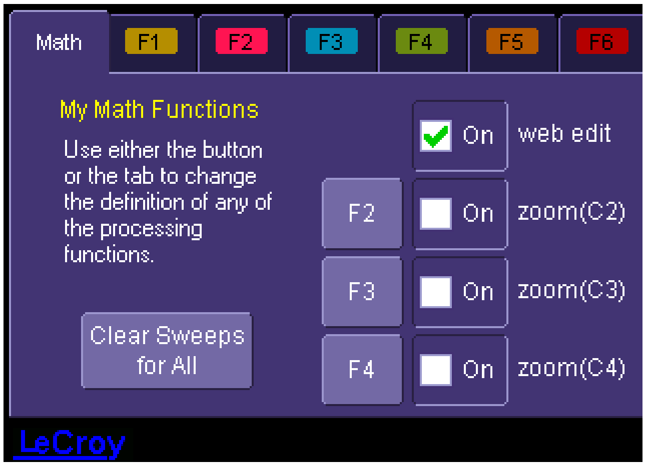
To Use the Web Editor

1. In the menu bar, touch **Display**, then **Web Editor** in the drop-down menu.

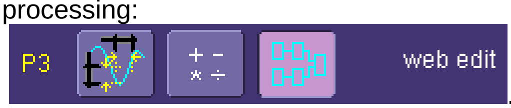
2. Touch the **Math** tab and select a math location (F1 to Fx) for the new math function that you are about to create by touching the **Web Edit** button:

.

Once you select a math location for web editing, it cannot be used for another math function, and will appear as unavailable in the Math Setup dialog:

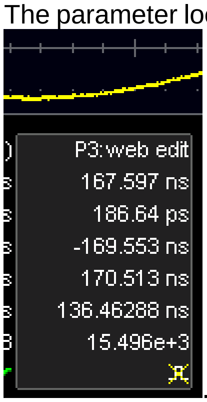


However, you can cancel web processing within the "Math Setup" dialog by touching the single function, double function, or graph button. Touch the **Measure** tab, then touch the **Web Edit** button, if you want to dedicate a parameter location (P1 to Px) for web processing:



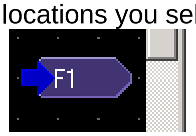
.

The parameter location you choose will display "Web Edit" under the waveform display grid



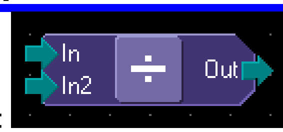
.

3. Touch the **Web Editor** tab to return to the web setup dialog. The math and parameter locations you selected appear as outputs at the far right:

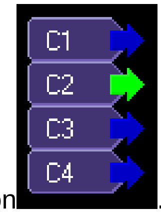


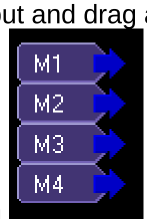
You may have to scroll up or down to see it.

4. Touch the **Add Math** button and select a math function from the **Add Math Processor**

menu. The math function icon will appear on the web setup field: 

. Touch and drag the icon to the desired location.

5. If you are using channel inputs, touch the arrow of a channel input icon 

. Then drag a line from the channel to the input of the math function icon. If your math function is a dual input function (such as ratio), select a second input and drag another line to the second math input. If you are using a memory location 

(M1 to M4) as an input, drag a line to the math function in the same way as for channel inputs.

Note: You can use a combination of channel input and memory input to your math function.

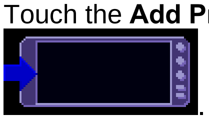
6. Touch the output arrow of the math function icon and drag a line to the Fx output on the right-hand side of the setup field. Your math function is complete.

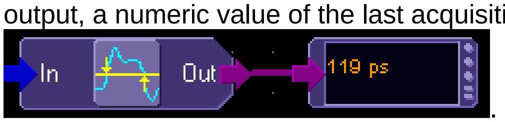
Adding Parameters

Add parameter measurements in the same way as for math functions. Parameters can be connected to any math function in the web.

Adding Previews

1. Touch the **Add Preview** button: 

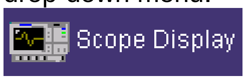
. A scope-like icon will appear: 
.

2. Touch the output arrow on the math function or parameter icon and drag a line to the input arrow of the preview icon. A thumbnail view of your signal will appear if the preview icon is connected to a channel output or math function output. If it is connected to a parameter output, a numeric value of the last acquisition will be displayed: 

.

Exiting the Web Editor

To exit, touch the **Close** tab; or, in the menu bar, touch **Display** then **Scope Display** in the drop-down menu.



The scope display will return to the normal waveform display grid.

Viewing the Output

1. Touch **Math** in the menu bar, then **Math Setup...** in the drop-down menu.
2. Touch the **On** checkbox for the function you want to view.

LABNOTEBOOK

Introduction to LabNotebook

LeCroy's LabNotebook feature extends the documentation capabilities of your scope. It allows you to create an annotated notebook entry containing all displayed waveforms, the setup of the DSO, and user-supplied annotation. The notebook entry can then be converted to hardcopy format -- pdf, rtf, or html -- and printed or e-mailed. You can also use the default report layout or configure your own, and even substitute your own company logo in the header.

Notebook entries are stored in an internal database and are available for recall at any time. Besides storing the waveform data, LabNotebook also stores your panel setups and parameter measurements. You have the capability to back up the database to external media.

The Flashback feature allows you to recall the state of the DSO at a later date, including the saved waveforms and the DSO setup, so that you can make additional measurements. A keyword filter makes it easy to find the correct notebook entry to recall.

You can choose which notebook to use for your entries, and label the notebook by project or user. If the scope is shared among several users, for example, or used for different projects, the data can be kept separately. Similarly, hardcopy reports can be stored in different folders.

Preferences

You should set your preferences before creating notebook entries.

Miscellaneous Settings

You can elect to name notebook entries with the default date and time by leaving the top box unchecked. Check the box if you want the opportunity to rename the notebook entry as soon as it is created.

Check the middle box if you want to be able to annotate a notebook entry as soon as it is created.

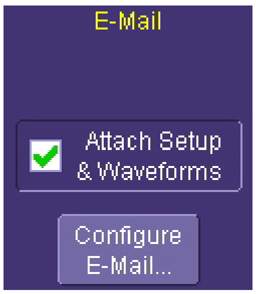
Check the last box if you want to generate a notebook entry by simply touching the Hardcopy (Print) front panel button. By checking this box, you override any other configuration for this button; for example, send e-mail or output to printer.

Hardcopy Setup

Check the **Use Print Colors** checkbox to place your waveforms on a white background in the notebook entry. This will save printer ink later when you print the hardcopy report.

Touch inside **Hardcopy Area** to determine how much of the screen image to include in the report: grid area only, grid area plus dialog, whole screen.

E-mail Setup

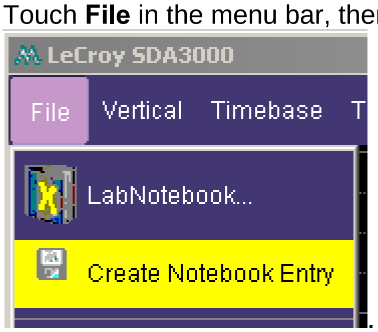


You can e-mail just the pdf or html report; or, you can include additional files: trace data (.trc) for each waveform in the report, a screen dump, a scope setup file, and an xml report record. Touch the checkbox to enable the extra report segments.

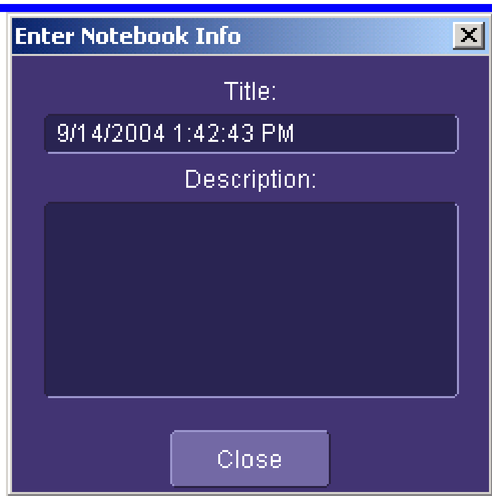
Touch the **Configure E-Mail** button to set the recipient address and server information.

Creating a Notebook Entry

1. Touch **File** in the menu bar, then **Create Notebook Entry** in the drop-down menu

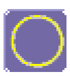


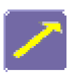
A dialog box is displayed in which to enter a title and comments for the entry. By default, the entry is titled with the current date and time.

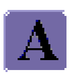


2. Touch inside the **Title** field and enter a title, using the pop-up keyboard. Then touch inside the **Description** field and enter a description, if desired, and touch **Close**.
3. The notebook entry will display your waveforms in "print colors," that is, on a white background to save printer ink, if you selected that option in notebook Preferences. Otherwise, the waveforms will appear on a black background. A drawing toolbar appears at top:


The pen tool enables you to write or draw in freehand. You can use a mouse, or a stylus to do this using the touch screen. Once you click off, you can drag your note anywhere on your waveform.


The circle tool enables you to create a circle around a waveform feature that you want to point out. Once you click off, the circle is drawn and you can drag it anywhere on the screen.

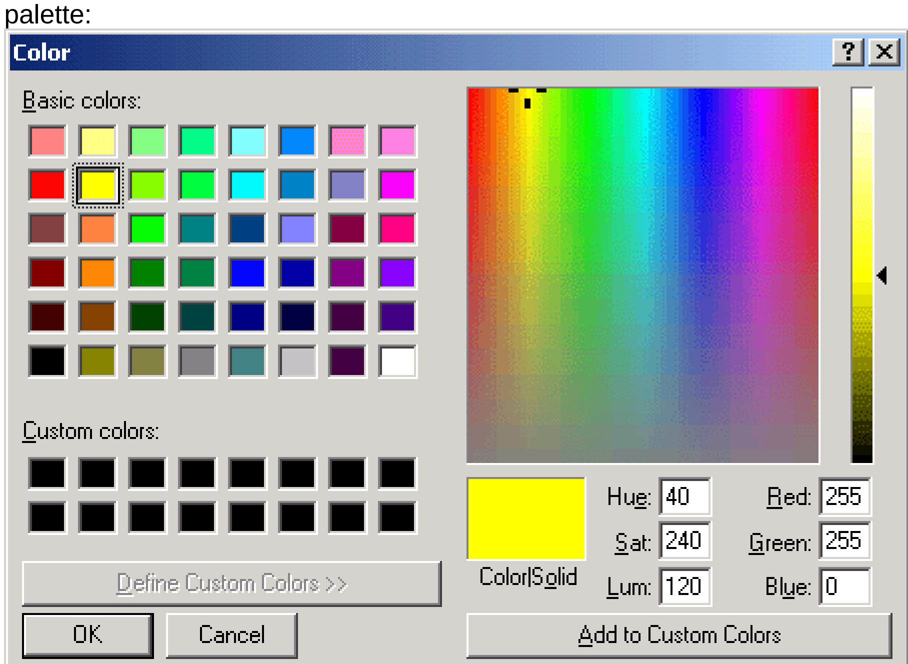

The arrow tool enables you to draw lines with arrowheads for callouts. You can rotate these lines through 360 degrees and drag them to any location on the screen.


The text tool enables you to enter text callouts on your report. When you touch this tool, a dialog box opens in which to enter text by means of a pop-up keyboard.

After you touch **Close**, your text will appear on the display as a draggable object.

These are the three default colors that you can select for shapes, lines, and text. To use additional colors, touch **More**.

When you touch **More**, a Custom box opens with the default color yellow displayed. Touch the yellow button to open the full color palette:



When you have chosen a custom color, touch **Add to Custom Colors**; the color will appear in the **Custom Colors** palette:

Then touch the color to enable it, and touch **OK**. The next object that you create will be in that color.

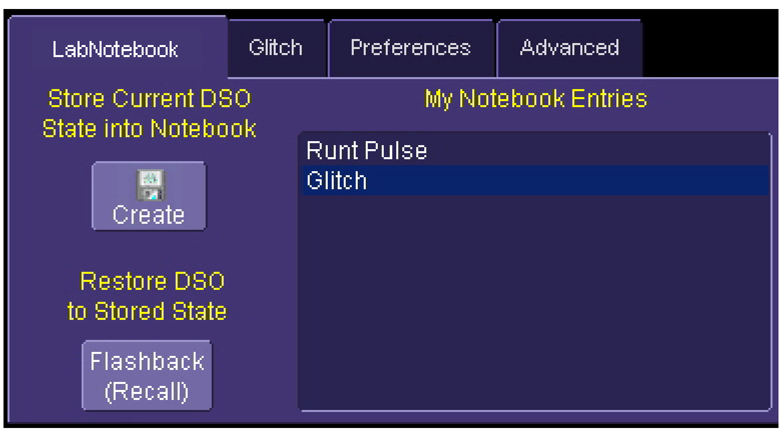
If you want to erase a drawing object, touch it to select it, then touch **Erase Selected**.

Touch **Erase All** to erase all drawn objects and text.

Touch **Undo** to discard the last object drawn.

The **Move Toolbar** button enables you to place the toolbar anywhere on the screen. Touch the button a second time to return it to its original fixed location.

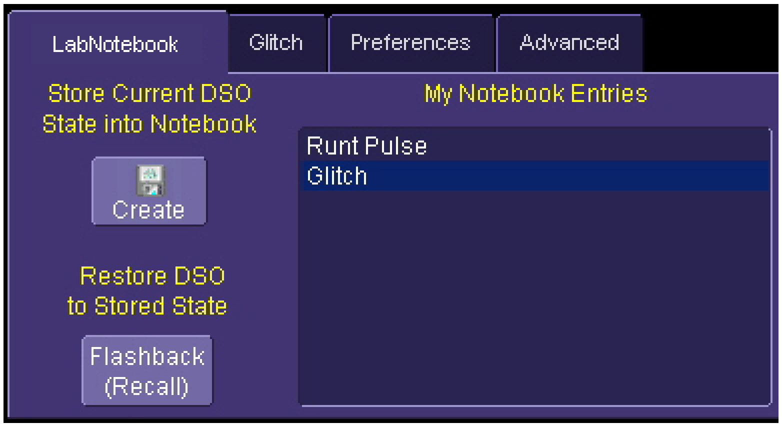
Touch **Done** when you are finished annotating the notebook entry. The name of the entry will appear in the list box in the "LabNotebook" dialog.



You can now create a hardcopy report of it, and email or print it out.

Recalling Notebook Entries

After a notebook entry is made, you can recall it at any time. The recall includes waveforms and scope settings.



1. Select the notebook entry from the list box.
2. Touch **Flashback**.
3. To exit Flashback, touch the **Undo Flashback** button in the top-right corner of the screen,

   or press the **Auto** trigger button.

Note: The flashback feature currently recalls the DSO Setup, and all displayed waveforms. Some forms of 'result data' are not recalled, including:

a. Persistence data. This will be saved in the hardcopy, and will be printed in the report, but will not be recalled during Flashback.

b. Histogram data. Histograms internally have a 32-bit resolution, but when stored into a trace file and recalled during flashback they are clipped to 16-bits.

c. Floating point waveforms. Certain math operations result in the creation of floating point waveforms with much higher resolution than can be stored in a 16-bit waveform file. This extra resolution will not be preserved when traces are recalled using flashback.

d. Cumulative Measurements. Any measurements that are on when the Lab Notebook entry is created are not saved individually in the database, other than being embedded in the hardcopy image. This means that when flashback is used, the measurements will be recomputed using the waveform data that was recalled. Normally this will not pose a problem, but if cumulative measurements were on, which accumulated data from multiple acquired waveforms, they will loose their history and show instead only the results from the stored waveforms.

Creating a Report

Once the notebook entry is created, you can easily generate a hardcopy report for e-mailing or printing.

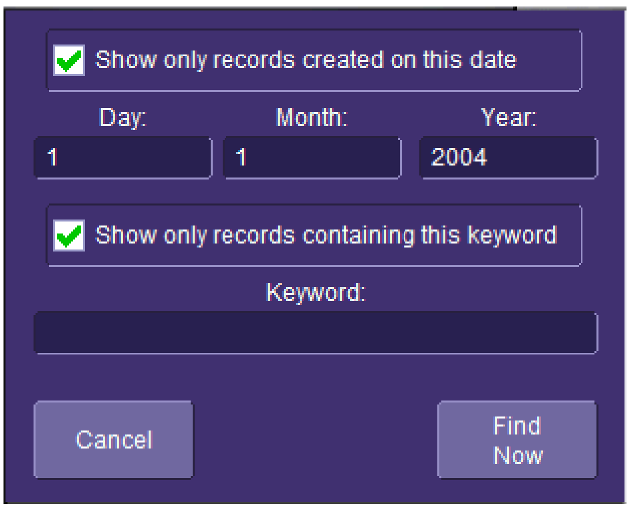
Previewing a Report

Before creating a report, you can preview it by simply touching the **View** button 

. To exit the preview, touch the **Close** button at the right of the dialog.

Locating a Notebook Entry

A search filter is provided to help you locate the notebook entry you want to make a report of. You can search by date or keyword.

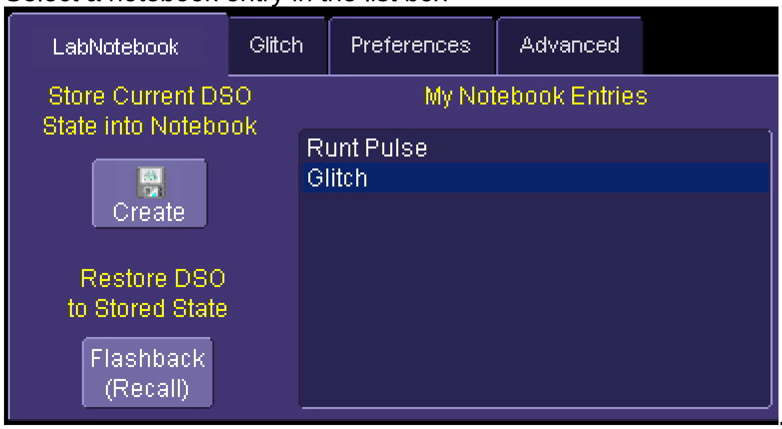


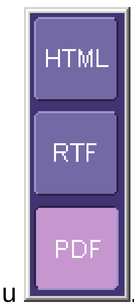
1. Touch the **Filter** button 

. A search dialog box opens.
2. Touch inside the **Day**, **Month**, and **Year** fields and enter a date. Or touch inside the **Keyword** field and enter a keyword or phrase.
3. Touch **Find Now**. Only the entries fitting the date or keyword criteria will now appear in the list box.

Creating the Report

1. Select a notebook entry in the list box



2. Touch inside the **Format** field and select a report format from the pop-up menu .

3. Touch the **Create Report** button.
4. A dialog box opens in which to name the report and select a folder to contain the report. Touch inside the **File name** field and enter a name using the pop-up keyboard.
5. If you want to e-mail or print the data to a network printer, touch **More Actions**, then the **Print** or **E-Mail** button. If you select **Print**, a Windows dialog box will open for you to select a printer and set options. If you select **E-Mail**, the report will be sent immediately to the e-mail address configured in Utilities Preferences.

Formatting the Report

LeCroy provides a default report format (template); however, you can use your own format, including company logo.

1. Touch the **Advanced** tab.
2. Touch inside the **Directory** field and navigate to a folder to contain the reports.

3. Touch the **Browse** button next to Template to navigate to an existing report format that you want to use. Or touch inside the **Template** field and enter the name and path to the template, using the pop-up keyboard. Otherwise, touch the **Use Default** checkbox to use LeCroy's format.
4. To use a logo other that the one provided, which indicates the scope that produced the report, browse to the bit map file or touch inside the **Logo** field and enter the name and path to the file, using the pop-up keyboard. Otherwise, touch the **Use Default** checkbox to use LeCroy's logo
.

Note: If you elect to use your own logo bit map, do not use a bit map larger than 180 pixels (height) x 100 pixels (width).

Managing Notebook Entry Data

Adding Annotations

You can add annotations to your notebook entry at any time.

1. Touch the "LabNotebook" tab.
2. Touch the notebook entry you want to annotate in the scroll list box. A new tab will appear bearing the name of the selected notebook entry.
3. Touch the new tab, then the **Scribble** button . The notebook entry will appear again with the drawing toolbar, described in Creating a Notebook Entry.

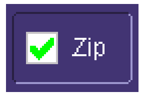
Deleting Notebook Entries

1. Touch the "LabNotebook" tab.
2. Touch the **Delete All** button to clear the database, or select a notebook entry in the list box, then touch the **Delete** button to discard just that one entry.

Saving Notebook Entries to a Folder

You can save notebook entries to a folder other than the default.

1. Touch the tab bearing the name of the notebook entry.
2. Touch the **Save Data to** button . A navigation window opens, which provides the opportunity also to open Windows Explorer to navigate to the folder.

3. Touch the **Zip** checkbox 

if you want to compress the data before archiving.

Managing the Database

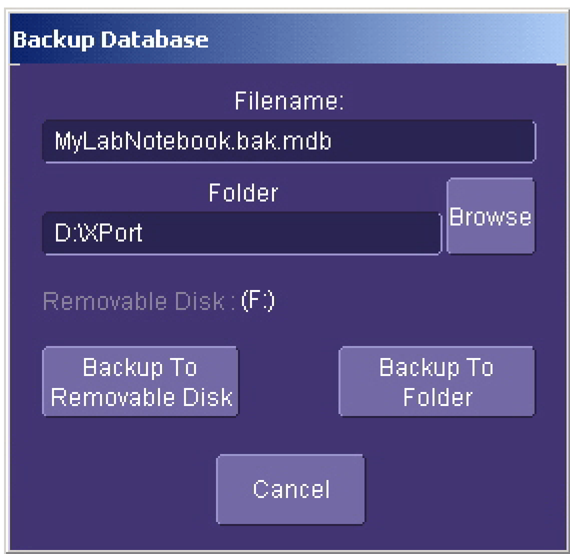
You can begin a new database for your notebook entries at any time, back up the current one, or compress the data.

To Select a Database for Backup or Compression

1. Touch the **Advanced** tab.
2. Touch the **Browse** button. A navigation window opens. Navigate to the database you want to work on.

Touch **Compact** to reduce the size of a database. This function "defragments" the notebook after a large amount of entries have been deleted.

Insert a memory stick into a USB port, then touch **Backup** to send the database to the external media



.

To Start a New Database

Touch the **Start New** button. The name of the notebook database will be incremented by 1:

§ § §