dScope Series III

Operation Manual

by Ian Dennis

This manual is also available as 'on-line help' from the dScope software. You can access the on-line help from the 'Help' menu. The on-line version is context-sensitive: by pressing F1, you can get immediate help for whichever menu or dialogue box you are currently using.

Table of Contents

Part 1

General information

1 General information

Manual revision history

Rev	Date	Author	Notes
1.00	8th January 2003	I.G.Dennis	To accompany software 1.00
1.01	30th March 2004	I.G.Dennis	To accompany software 1.01
1.10	1st May 2005	I.G.Dennis	To accompany software 1.10
1.11	15th August 2005	I.G.Dennis	To accompany software 1.11
1.20	8th January 2007	I.G.Dennis	To accompany software 1.20
1.21	28th March 2007	I.G.Dennis	To accompany software 1.21
1.30	1st July 2009	L.R.Elliott	To accompany software 1.30
1.40	9th September 2009	L.R.Elliott	To accompany software 1.40
1.40d	24th June 2011	L.R.Elliott	To accompany software 1.40d
1.42	17th August 2012	L.R.Elliott	To accompany software 1.42
1.44	1st March 2013	L.R.Elliott	To accompany software 1.44
1.45	6th November 2013	K.S.C.Stubbs	To accompany software 1.45

Support contacts

Prism Media Products Limited
The Old School
Stretham
Ely
Cambridgeshire CB6 3LD
UK

Telephone: +44 1353 648888
Fax: +44 1353 648867

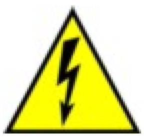
Prism Media Products Inc
21 Pine Street
Rockaway
NJ 07866
USA

Telephone: +1 973 983 9577
Fax: +1 973 983 9588

Email: tech.support@prismsound.com
Web: http://www.prismsound.com

Or contact your local Prism Sound distributor as detailed on the website.

WARNING!



TO PREVENT FIRE OR SHOCK HAZARD DO NOT EXPOSE THIS EQUIPMENT TO RAIN OR MOISTURE. DO NOT REMOVE THE COVER. NO USER-SERICEABLE PARTS INSIDE. REFER SERVICING TO QUALIFIED SERVICE PERSONNEL.

Statements of conformity

This equipment has been tested and found to comply with the limits for a Class B digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against interference in a residential area. This device generates and uses radio frequency energy and, if not installed and used in accordance with the instructions, may cause interference to radio or TV reception. If this unit does cause interference to radio or TV reception, please try to correct the interference by one or more of the following measures:
a) Reorient or relocate the receiving antenna.
b) Increase the separation between the equipment and the receiving antenna.
c) Plug the equipment into an outlet on a different circuit from the receiver.
d) If necessary, consult your dealer or an experienced radio or TV technician.
CAUTION: Changes or modifications to this equipment not expressly approved by the manufacturer could void the user's authority to operate this equipment.

THIS DIGITAL APPARATUS MEETS ALL CLASS B LIMITS FOR RADIO NOISE EMISSIONS AS LAID DOWN IN THE RADIO INTERFERENCE REGULATIONS OF THE CANADIAN DEPARTMENT OF COMMUNICATIONS.
CET APPAREIL NUMÉRIQUE RESPECTE TOUTES LES EXIGIENCES APPLICABLES AUX APPAREILS NUMÉRIQUES DE CLASSE B SUR LE BROUILLAGE RADIOELECTRIQUE EDICTE PAR LE MINISTERE DES COMMUNICATIONS DU CANADA.

Prism Media Products Ltd hereby declares that this equipment conforms to the following standards:
EN55103-1, environment category E4
EN55103-2, environment category E4

NOTE: The use of this equipment with non-shielded interface cabling is not recommended by the manufacturer and may result in non-compliance with one or more of the above directives. All coaxial connections should be made using a properly screened 75R cable with the screen connected to the outer of the connector at both ends. All XLR connections should use a screened twisted pair cable with the screen connected to pin 1 of the XLR connector at both ends. In the case of the digital XLR connections this cable should be of 110R impedance.

Trademark acknowledgements

Access, ActiveX, Excel, Microsoft, MS-DOS, Visual Basic, VB, VBA, VBScript, Visual C++ and Windows are trademarks of Microsoft Corporation.
Borland and Delphi are trademarks of Borland Software Corporation.
LabVIEW and LabWindows/CVI are trademarks of National Instruments Corporation.
Alesis and ADAT are trademarks of Alesis Corporation.
Dolby and the double-D symbol are trademarks of Dolby Laboratories.
DTS is a trademark of DTS, Inc.
Audio Precision is a trademark of Audio Precision Inc.

All trademarks acknowledged.

In accordance with our policy of continual development, features and specifications are subject to change without notice.

Part 2

Introduction to dScope

2 Introduction to dScope

The dScope Series III is a powerful audio test and measurement system.

dScope can generate and analyze a wide range of digital and analogue audio signals, and can also generate and analyze different parameters of the digital audio interface itself. dScope provides a unique blend of accuracy, functionality and ease of use, and its portability lets it operate in situations where other test instruments would not be practical.

For details of what's new in this software version, go to What's new...

For a quick tour of dScope operation go to Quick tour

For a no-nonsense guide to common operations go to How do I...

For an introduction to dScope's user interface go to User-interface basics

For a description of dScope architecture and capabilities go to Architecture

For an in-depth user-interface reference go to Operation reference

Whilst dScope is easy and intuitive in performing routine tests and measurements, its full power can be unleashed using scripting; this is a technique whereby many of the dScope's functions can be modified or enhanced to fulfill a user's special requirements. Scripting is especially useful in production testing, where sequences of tests and limits can be automated and the results passed to other Windows applications; also in development, where measurement methods can be mathematically tailored to produce customised results.

dScope is unique in its easy application of multi-tone testing techniques. Using multi-tones, it is possible to measure many different parameters of the equipmeny under test simultaneously, and to automatically check them against acceptable limits. Using the dScope's multi-tone features, most types of audio device can be thoroughly tested in a few seconds, without the need for sweeps or multiple spot-measurements. For more about multi-tone testing, see the Multi-tone Generation and Analysis section.

dScope can also measure transfer functions of transducers, rooms or electronic devices employing impulse response techniques using swept sine ('chirp') or noise sequence stimuli. For more information, see the Impulse Response Parameters section.

To facilitate testing of equipment with multiple inputs or outputs (analogue or digital), the dScope's high-quality I/O Switchers provide user-friendly channel selection. Multiple switchers can be combined into larger matrices with seamless user-controls. For direct connection to CODECs or other chip or board level audio devices, the Versatile Serial I/O (VSIO) can provide user-programmable serial interfacing, complete with serial control capability if required. For testing digital power amplifiers, a passive low-pass filter is available.

2.1 About this manual

The dScope Series III Operation Manual is provided in two different formats: as a conventional printed manual, and also as 'online help' which can be viewed whilst operating the dScope. The printed version is also provided in 'electronic' format, as a 'pdf' file, with the dScope software. These files can be viewed and printed using the Adobe Acrobat Reader, which can be downloaded free at www.adobe.com. Updates of the manuals are available from the Prism Sound website at www.prismsound.com.

When viewed on-line, the manual pages are accompanied by a navigation area to the left. Therein, a

"Contents" section shows a hierarchical map of the entire document from which desired pages can be selected. Next to "Contents", the "Index" section allows instant access to pages describing particular topics. The "Search" section lists all pages containing a particular word or phrase, and the "Favorites" section can be used to save page locations for future reference.

Entry into the on-line help from the dScope application is 'context-sensitive', so pressing the F1 key takes you directly to the help page for whichever dialogue box or panel you are using at that time.

When viewed as on-line help, each page is headed by a title block which shows the name of the page, plus some links on the right-hand side. The upper row of links refer to topics above the current page in the manual's hierarchy. Below, a "See Also" link often appears which accesses a pop-up box containing a list of related topics.

Within the body of each page, certain font and highlighting conventions are used:

Links to other parts of the manual are shown like this.

Buttons on the dScope dialogue boxes are designated, for example, [OK]

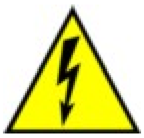
and Results are referred to, for example, as <amplitude>

```
Code samples are shown in this font...
```

NB: **Noteworthy items are indicated like this.**

Important warnings are designated like this.

.



This symbol designates warnings of risk of electric shock.

Part 3

Operation overview

3 Operation overview

For a summary of what's new in the latest major software revision, got to What's new...

For an introduction to dScope's user interface go to User-interface basics

For a quick tour of dScope operation go to Quick tour

For a no-nonsense guide to common operations go to How do I...

3.1 What's new in Version 1.44?

Major feature enhancements in Version 1.44 of the dScope software include:

- **ASIO Soundcard generation and analysis**

dScope can now generate and analyze audio on ASIO soundcards as well as WDM soundcards.

- **Continuous-Time analysis available to soundcard inputs**

dScope is no longer limited to FFT-based analysis when the Signal Analyzer source is set to 'Soundcard'; Continuous-Time analysis of soundcard inputs is now available.

3.2 What's new in previous versions

Version 1.40

Major feature enhancements in Version 1.40 of the dScope software include:

- **64-bit operation**

dScope's software will now install and run on both 32-bit and 64-bit Windows operating systems.

Version 1.30

Major feature enhancements in Version 1.30 of the dScope software include:

- **Model numbers**

dScope's software now has several optional features, defined by the which version of the hardware is purchased. For full details, see model numbers.

Version 1.20

Major feature enhancements in Version 1.20 of the dScope software include:

- **192kHz single-wire digital I/O**

dScope can now generate and analyze AES3 and S/PDIF data at 176.4kHz and 192kHz sampling rates. This includes full support of carrier degradation and analysis features (e.g. jitter), including Carrier Display.

- **Generation and analysis using Windows sound devices**

It is now possible to drive a Windows sound device from the dScope Generator, and to feed the FFT Analyzer from a sound device input. This is particularly useful for testing PC-hosted EUTs such as soundcards or bluetooth headsets. Routing matrices are included to facilitate the use of multi-channel sound devices. For more information, see the Soundcard Outputs dialogue box and Soundcard Inputs dialogue box sections.

- **Acoustic measurements of transducers and rooms**

It is now possible to generate an impulse response of a signal path (e.g. a loudspeaker or a room) driving it with a noise or swept-sine (log chirp) stimulus and correlating the output signal with either the generated data or the signal entering the other Analyzer channel (e.g. a 'direct' signal). The impulse response can be time-windowed (for example to simulate anechoic conditions) and then used to derive the frequency response of the signal path. Multiple contiguously-acquired buffers can be averaged prior to analysis to reduce the effects of background noise. For more information, see the Impulse Response Parameters dialogue box section.

dBSPL units are now available in both the Generator and the Analyzer, as well as the ability to apply measurement microphone calibration data (both gain and frequency response).

- **Dolby and DTS stream generation**

It is now possible to generate a wide variety of multi-channel Dolby Digital and DTS encoded streams directly from the dScope's digital outputs. For more information, see the dScope Applications Manual
.

- **Script debugger**

It is now possible to set breakpoints, single-step and to modify/examine variables all from within the Script Edit window.

- **Time-domain averaging**

The FFT Analyzer can now apply time-domain averaging as well as frequency-domain averaging. Whilst frequency-domain averaging is useful in flattening an FFT Trace to eliminate variations in uncorrelated noise, time-domain averaging can actually lower the noise floor to improve the dynamic range of the measurement. dScope can apply BOTH time- and frequency-domain averaging to the same measurement, providing an FFT with a flattened AND lowered noise floor.

- **New Generator functions: "Swept sine" and "Bin centres"**

The Swept sine function is primarily useful for acoustic testing. The user may specify start and end frequencies, length of sweep, log or linear progression and the duration of a padding silence period (which is especially useful to wait out room reverberance in acoustic testing).

The Bin centres function generates a multi-tone stimulus with a tone in every bin of the subsequent FFT analysis. Bin centres is a wideband noise-like stimulus, but is preferred over noise because it allows instant and solid plotting of EUT frequency response using a synchronous FFT technique without the random variations which occur with a white noise stimulus.

For more information, see the Signal Generator dialogue box section.

- **48kHz operation of analogue I/O**

In addition to the 96kHz and 192kHz analogue sampling options, dScope now provides an additional 48kHz option to allow improved frequency resolution per FFT point in applications where extreme HF capability is not required (e.g. acoustic testing).

- **Internal loopback of individual digital I/O channels**

This allows the two-channel Analyzer to monitor both the input and output of an EUT with digital I/O. It is now possible to measure delay and input-to-output phase of a digital device in the same way as for an analogue device. For more information, see the Digital Inputs dialogue box section.

- **More versatile graph exporting**

Trace window images can now be exported in BMP, JPG, GIF, TIF and PNG formats as well as Windows Enhanced Metafile (EMF) format.

- **Importing and exporting of sample buffers**

Sample buffers can now be saved/loaded as part of dScope configurations. WAV files can also be imported.

- **Concurrent Scripting**

Multiple VBScripts can now be run simultaneously.

- **Sweep-step-triggered Scriptlets**

The user can now script actions to be executed between each step of a Sweep. This can simplify custom-Trace generation Scripts by removing the need to set up User traces.

- **New Trace transform "Relative to other Trace"**

Allows easy checking of EUT Trace variations from a 'golden unit'.

- **Clip flags on Monitor Outputs**

The addition of clip indicators in the Monitor Outputs dialogue box aids manual gain setting of the Monitors.

- **Overhauled warning system**

The new non-intrusive warning system gives the user clearer and more comprehensive information about potential set-up problems.

- **User-interface enhancements**

Including improved Trace window drawing and a range of new Toolbar icons. NB: users upgrading to Version 1.20 from a previous dScope version will retain their previous Toolbar configuration; to take advantage of the new icons, it is necessary to reset the Toolbar configuration (see Customize Toolbar dialogue box).

3.3 User interface basics

The user interface of the dScope comprises a number of basic elements:

Menu bar
Main Toolbar
User bar
Pages
Status bar

The Main Toolbar, User bar and Status bar may be individually turned on or off from the View menu.

Menu bar

The Menu bar is situated at the top of the dScope window. It provides access to all the functions of the dScope, although it is usual for the more commonly used functions to be included as one-click 'icons' on the Main Toolbar.

All of the dScope's menus are detailed in the Operation reference chapter.

Main Toolbar

Below the Menu bar is the Main Toolbar, which contains icons used as shortcuts to the most commonly used functions. The selection of icons displayed on the Main Toolbar can be customized by the user. The functions of all available Main Toolbar icons are described in the Main Toolbar icons section of the Icons and Hotkeys reference.

User bar

Below the Toolbar is the User bar, which contains buttons allocated by the user to give instant access to saved scripts or Configurations (setups). Details of how to do this are in the Customize User bar dialogue box section. After first installation, a 'default User bar' is installed containing shortcuts to a variety of common tasks. Details of the default User bar are available by clicking the [About User bar] button on the left of the bar.

Pages

The main part of the dScope window contains the currently open dialogue boxes, Readings, Trace window etc. This area is notionally arranged as five different 'Pages', one of which is selected for viewing using the Page tabs in the lower right-hand corner of the dScope window.

This facility allows different objects to be arranged on different Pages to alleviate the limits of the screen size. In general, any dialogue box, Reading etc. can be opened on more than one Page if desired.

Page tabs are designated in a bold font to show that the Page has some content; empty Pages are designated in a lighter font.

Status bar

The bottom line of the dScope window, to the left of the Page tabs is the Status bar. This shows important indications of the current state of the dScope, including warning messages. This is described in detail in the Status bar section of the Operation reference.

Dialogue boxes and panels.

Most of the dScope's controls are arranged within 'dialogue boxes' containing all controls for a particular function, for example the Signal Generator. Many of the dialogue boxes are subdivided into 'panels' (shown within an indented box) which may be used stand-alone by dragging them off their parent dialogue boxes. This is useful in making the best use of the available space on each Page.

Each dialogue box, and each of its panels is described separately in the Operation reference chapter of this manual.

Results and Readings

'Results' measured by the dScope are shown as blue-green text within a black window on a dialogue box or panel. Important Results can be turned into 'Readings' by dragging them off their parent dialogue box or panel whilst holding down the right mouse button. Readings have more flexibility than simple Results; for example they can be re-sized, re-coloured, or have bar graphs or limit checking added to them as described in the Reading window section.

Dockable Toolbars

The Main Toolbar and the User bar are 'dockable', as are some other Toolbars within the dScope. Any of the dockable Toolbars can be undocked from its default position by either double-clicking on it (away from any of the actual Toolbar icons) or by holding down the left mouse button and dragging it away. Once undocked, it can be replaced by either double-clicking on its blue title bar, or by holding the left mouse button down over the title bar and dragging it back close to its docking position.

Whilst undocked, the Toolbar can be 'shaped' by dragging its edges with the left mouse button held down; however, it will always remain large enough to contain all the icons assigned to it. From the undocked state, a dockable Toolbar can be hidden completely by clicking the right mouse-button over its blue title bar, and then selecting the 'Hide' option. The Toolbar can then be returned to view by checking it's entry in the 'View' menu.

To stop a dockable Toolbar from automatically docking when it is dragged near the edge of the window, you can hold down the <Ctrl> key whilst dragging the Toolbar.

3.4 Quick tour

This section describes a quick tour of the dScope, intended to enable new operators to find their way around, and hopefully recognise some familiar territory. This section provides instructions for a 'manual' tour of the dScope's main features; alternatively, an 'automated quick tour' feature is installed automatically with the dScope software - this is accessed by clicking the [Quick Tour] button at the right-hand side of the default User bar, and following on-screen instructions.

Preparing for the tour

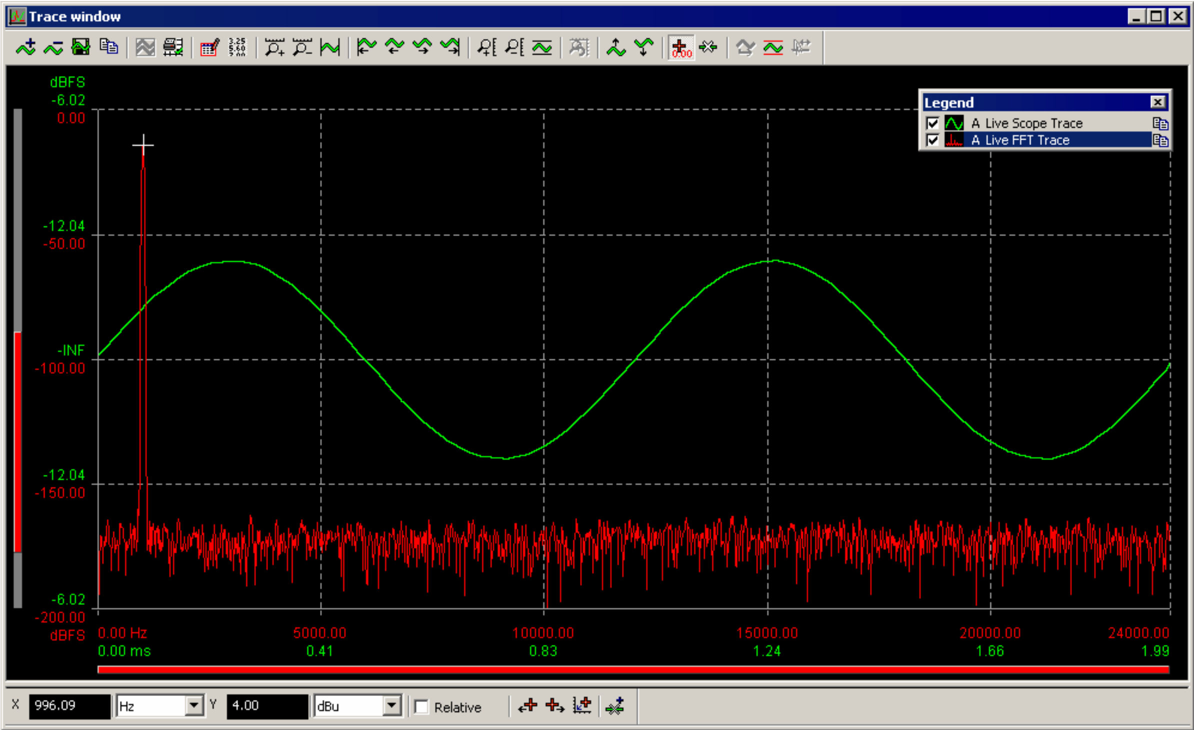
Start the dScope software, and make sure that no Configuration file has been loaded (that the 'Options' box in the 'Utility' menu has its 'Configuration to load on startup' field blank or set to use the default '~default.dsc' Configuration file provided at installation time. Make sure the dScope window is maximised.

For the purposes of this tour, the dScope analogue and digital Signal Generator outputs must be connected to their respective Signal Analyzer inputs. This can be done using three XLR leads (two for analogue, one for digital) – including a 'device under test' adds a touch of realism!

Alternatively the 'back to back' connection can be achieved using relays within the dScope, like this: Click the Toolbar icon, and select 'Source' = 'Generator'; then click the Toolbar icon, and select 'Source' = 'Generator XLR'; then close the two boxes.

Open the Trace window

Open the Trace window by clicking the Toolbar icon and turn on the FFT Analyzer trigger by clicking the Toolbar icon. A sine wave should now be displayed, both as a green 'Scope Trace' – rather like a conventional oscilloscope, and also as a red 'FFT Trace' showing the same signal in the frequency domain.



Actually, we've jumped ahead of ourselves here, since the Scope and FFT Traces are really products of the FFT Analyzer, which we're going to come to later. But it's often convenient to bring up the Trace window to inspect the Analyzer signal(s) even if you aren't using the analysis functions of the FFT Analyzer.

Try switching between the Digital Input and Analogue Input modes of the Analyzer using the and Toolbar icons, and between the individual and dual channel modes using the , and Toolbar icons. You will notice a higher noise-floor when the Analogue Input is selected compared with the Digital Input, and possibly some low-level harmonic distortion products. These differences, whilst invisible on the Scope Trace at normal levels, are easily seen on the FFT Trace – that's why a continuous FFT is such a powerful tool for detecting very many types of audio performance problems.

If the Digital Input or either channel of the Analogue Input doesn't display as a sine wave with the appropriate FFT, check that the system is correctly wired 'back-to-back' and that default settings are loaded (you can restart the software to ensure this).

The Trace window can display many other 'Live' Trace types besides Scope and FFT Traces, including Sweep Traces and Residual ('distortion') Traces, as well as non-Live Trace types such as Limit Lines, Filter Traces or copies of Live Traces.

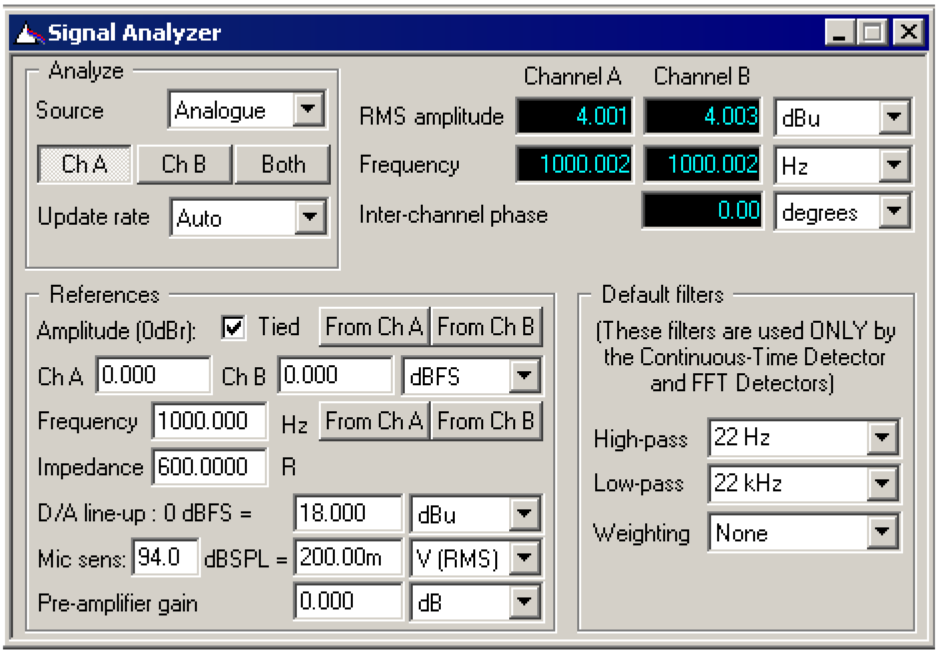
Full details of the operation of the Trace window are in the Trace window section.

Using the Signal Generator and Signal Analyzer

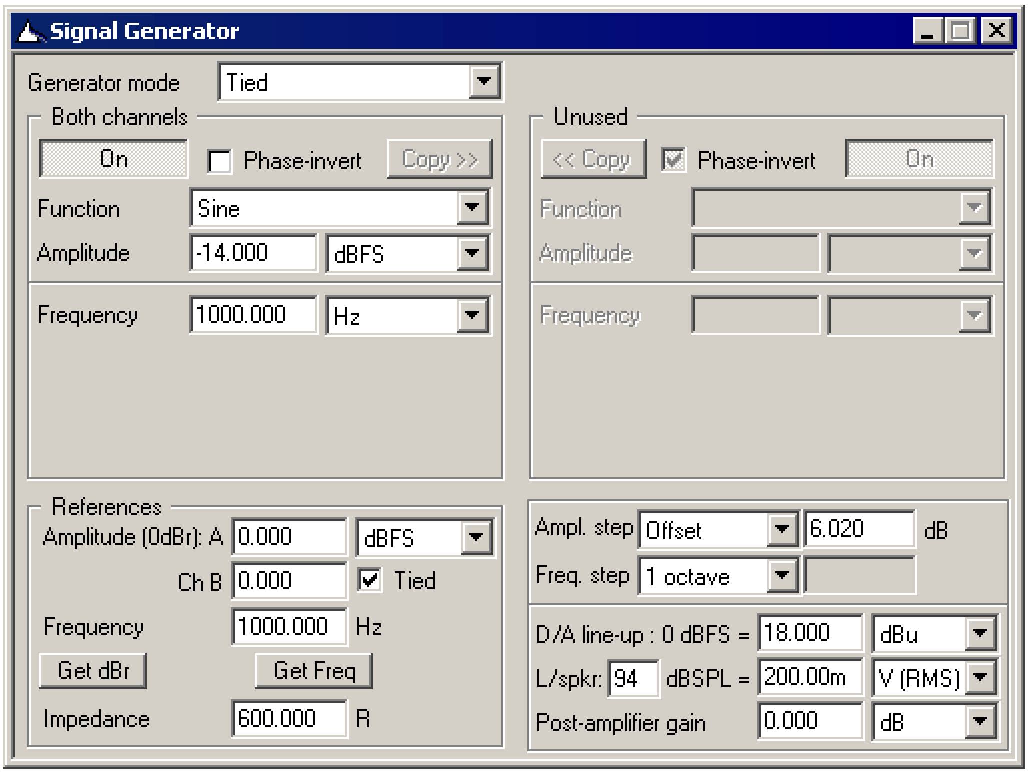
Select 'Page 2' by clicking on the appropriate tab in the bottom right-hand corner of the dScope window. dScope has five Pages available, simply to provide an extension to the desktop which would otherwise soon get full. It is useful to place related items on each Page, and then switch between Pages as required. In this case, the Trace window is still open on Page 1 – we can go back to it at any time by clicking the Page 1 tab.

Using the and Toolbar icons, bring up the Signal Generator and Signal Analyzer dialogue boxes on Page 2. Drag the boxes around by holding down the left mouse button (in the window's title bar) until they are conveniently positioned within the dScope window.

In the top part of the Signal Analyzer box, the RMS amplitude and the frequency of both channels of the selected input domain are displayed, as well as the inter-channel phase (or delay). The input amplitude can be displayed in various digital and analogue RMS units by operating the list box. The Analogue/Digital and A/B/both channel selector controls previously operated with their Toolbar icons are also resident here.



The Signal Generator box contains controls for function (waveform), amplitude and frequency. Notice that changes to the amplitude and frequency are reflected in the Signal Analyzer box. You can go back to Page 1 and inspect the results of Signal Generator changes in the Trace window. You can open another copy of the Signal Generator over the Trace window on Page 1 if you like, to save switching back and forth. Set the Signal Generator back to a sine function at –14dBFS and 1kHz afterwards since we need this later in the tour.



The Signal Generator generates both analogue and digital outputs simultaneously, and the relationship between their amplitudes is set by the 'D/A line-up' (digital/analogue line-up) control. A similar control exists in the Signal Analyzer, and these are normally ganged together. If you are measuring a dual-domain system (e.g. an analogue-to-digital converter) you should set the D/A line-up controls to match it's own line-up – then it is simple to measure analogue levels in digital units or vice-versa: thus you can set the Generator to output an analogue amplitude of –1dBFS, i.e. an analogue amplitude 1dB below the top of the ADC.

Follow these links for full details of the Signal Generator and Signal Analyzer boxes.

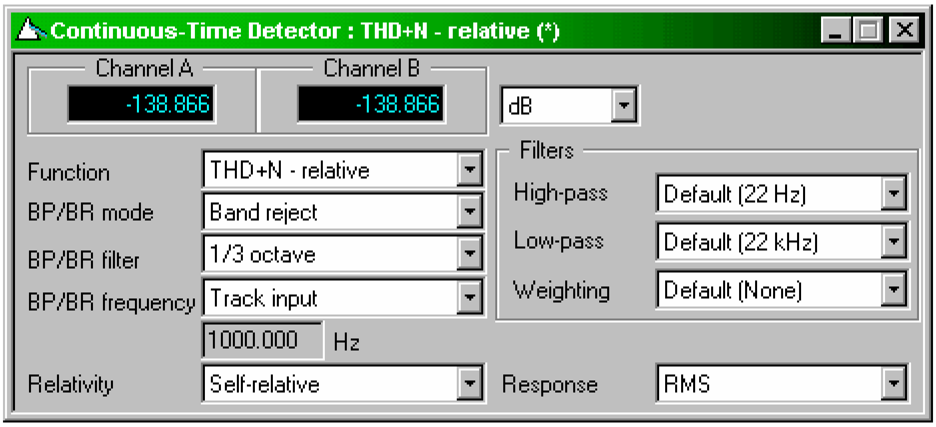
Measurements with the Continuous-Time Analyzer and FFT Analyzer

Notice that the Signal Analyzer panel doesn't have a 'function selector' for making any more sophisticated measurements such as THD+N. This is because the dScope offers two other alternative analyzers with their own dialogue boxes for performing this type of measurement:

The 'Continuous-Time Analyzer' (CTA) is like a traditional analogue signal analyzer – it can make all the 'standard' measurements, operating continuously so that any momentary change in the signal is always registered. The dScope has only one CTA (although it operates on both channels simultaneously), so only one type of measurement can be made at a time.

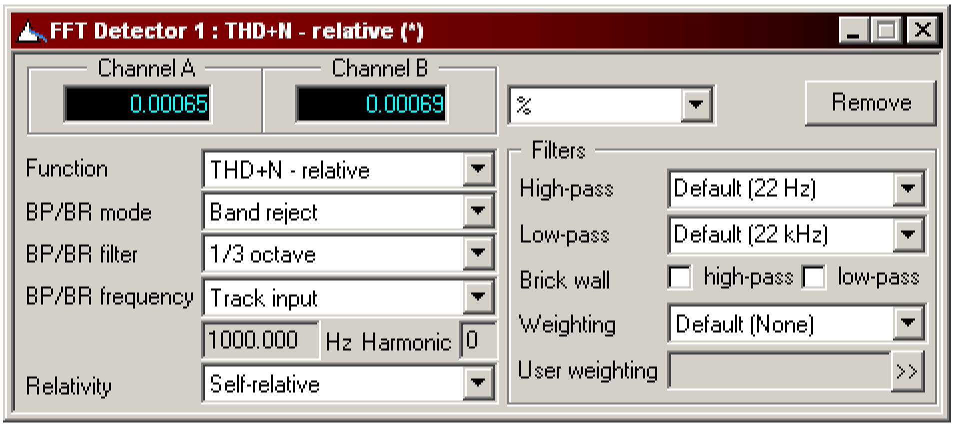
The 'FFT Analyzer' (FFTA) can also make these standard measurements, but it operates differently – by capturing a buffer of samples on activation of an oscilloscope-like trigger. Having captured the buffer of samples, the desired measurement is calculated before re-arming the trigger to capture the next buffer. The FFT Analyzer can perform many more complex functions that the Continuous-Time Analyzer (including calculating 'user-defined' measurements from VBScripts), but its trigger-based nature means that it is slower than the CTA and may miss transitory changes in the signal which happen between triggerings. The FFTA can calculate up to 40 different (two-channel) Results at once, so it is a powerful way of measuring many parameters simultaneously, for example using multi-tone stimuli as described in the Multi-tone Generation and Analysis section.

Move to Page 3, and open the Continuous-Time Detector box (select the 'Analyzer' : 'Continuous-Time Detector' menu item). The Results for both channels, in the selected units, are shown at the top of the box; immediately beneath is the Function selector, which sets the desired measurement function by forcing the states of the remaining Detector parameters. The default function is 'THD+N relative', which applies a notch filter that tracks the predominant input frequency and displays the amplitude of the residual relative to the input amplitude. Switch the units from 'dB' to '%' if you like.



High-pass, low-pass and Weighting filters can be selected, or they can be set to follow the 'default' values which are set centrally in the Signal Analyzer box – this allows the filters for all Detectors (including FFT Detectors) to be adjusted universally if required.

Open an FFT Detector box by selecting the 'Analyzer' : 'New FFT Detector' menu item. The layout of this box, and its functionality, are similar to the CTD box, the two being distinguished by the colour of their title bars. The FFT Detector has more functionality available, and can run user scripts to make customised measurements; you can also use up to 40 FFT Detectors at once, simply by opening more FFT Detectors from the Analyzer menu.



Follow these links for full details of the [Continuous-Time Detector] and [FFT Detector] boxes.

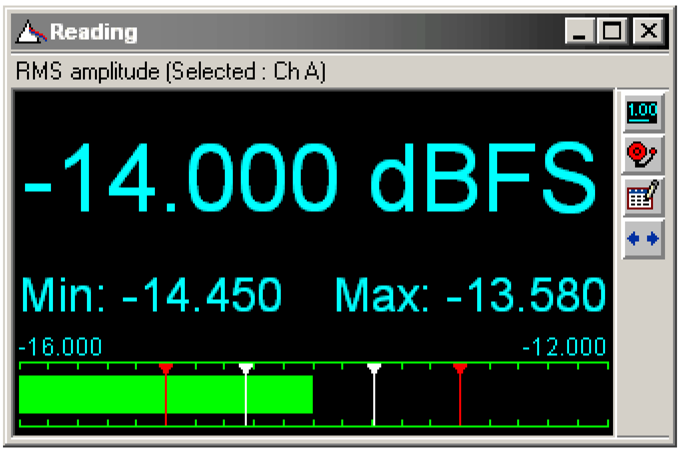
Turning Results into Readings

Return to Page 2, and locate the mouse cursor above one of the RMS Amplitude Results in the Signal Analyzer box. Hold down the left mouse button and drag the Result to an empty part of the dScope window. On releasing the mouse button, the Result is displayed as a Reading. A Reading is a versatile rendition of a Result, since its appearance (size, colour, name etc.) can be altered, bar graph output displayed, limits and alarms applied and much more.

Try resizing the Reading by dragging its edges while holding down the left mouse button. Add a bar graph to your new Reading by clicking the button to bring up the Reading Properties window, then check the 'Add bar graph' checkbox, and adjust the range of the bar graph if you like. You can also change the background and foreground colours of the Reading here.

By clicking the button to bring up the Reading Limits window, you can apply upper and lower limits to the Reading, and define the consequences of their being breached. Try setting a lower amplitude limit (below the current amplitude Reading), and enabling audible alarms – then toggle the [On] button in the Signal Generator box.

The button is used to bring up the box where the source Result for the Reading is located, in case it is not open anywhere on the current Page.

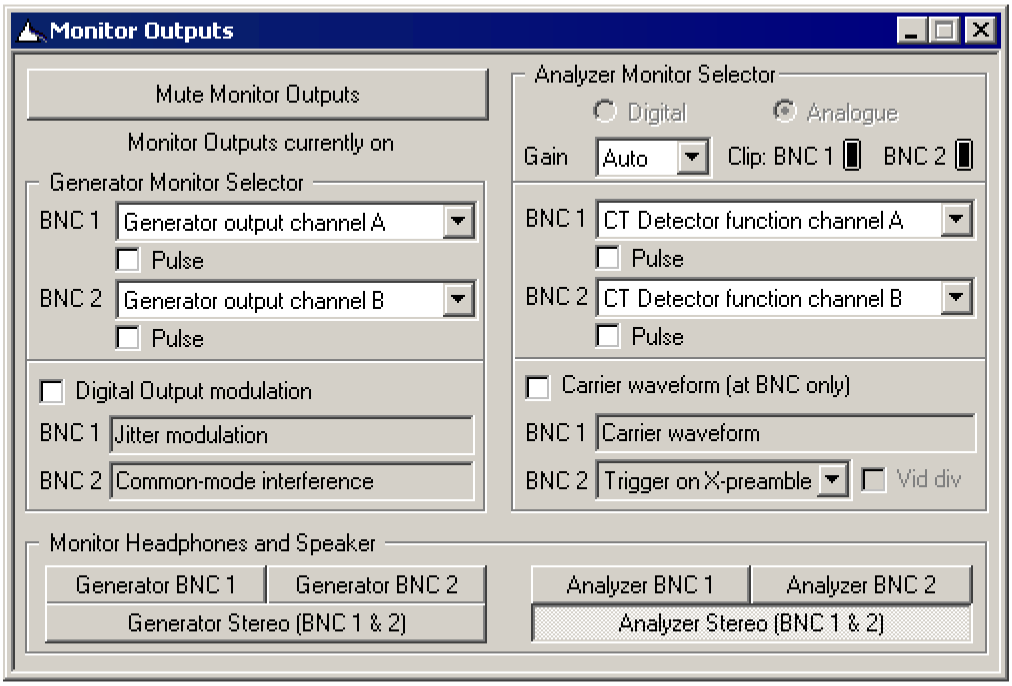


Go to the Reading window section for more details about Readings.

Using the Monitor Outputs

Move to Page 4, and open the Monitor Outputs box by clicking the Toolbar icon. Unmute the Monitor Outputs by clicking the large button in the top left-hand corner of the box, and make sure that the Volume control on the front of the dScope unit is turned up. The four Monitor BNC connectors on the front of the unit should now be monitoring the Generator A and B channels and the Analyzer A and B channels.

The assignment control for the headphone / loudspeaker is located at the bottom of the box, and allows selection of the Generator or Analyzer BNCs either singly or in pairs. Switch between the Generator and Analyzer by clicking the large buttons at the bottom of the section.

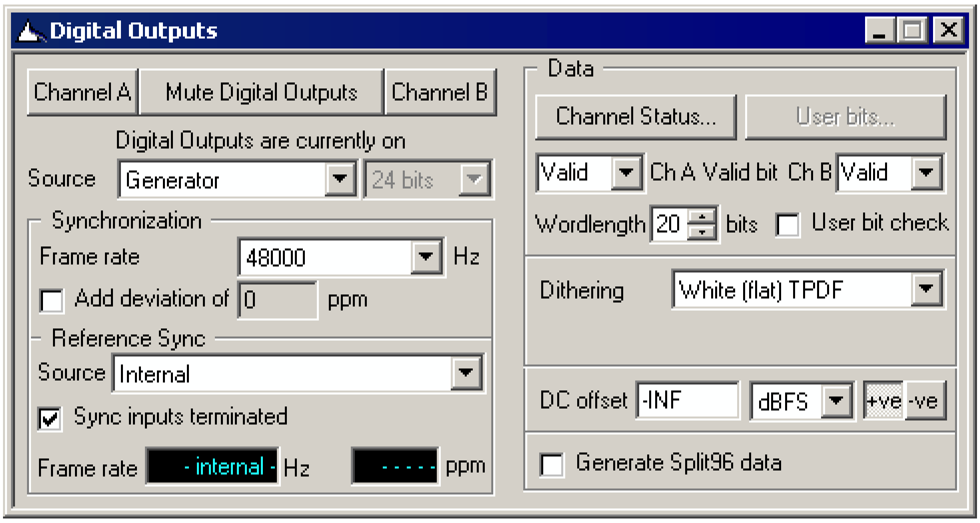


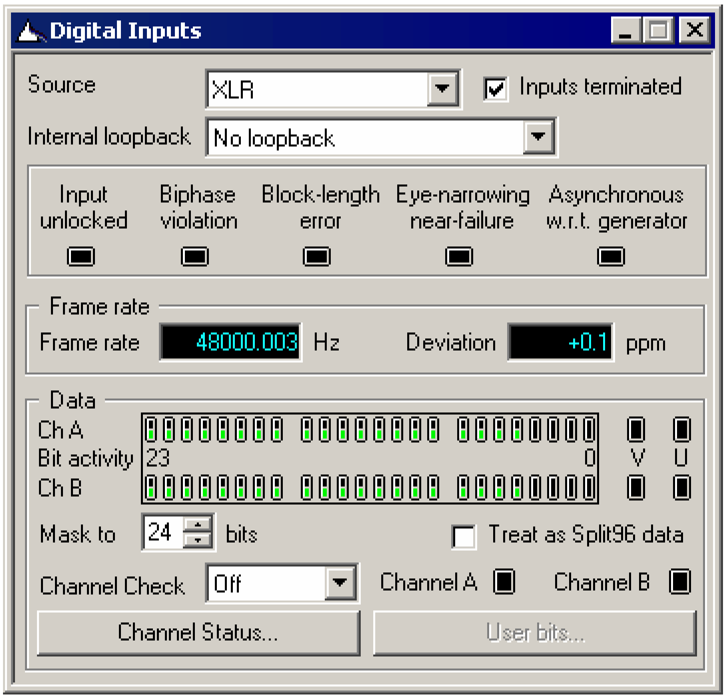
The Monitor connectors can be assigned to a wide range of functions, as fully described in the Monitor Outputs section.

Using the Digital Outputs and Inputs

Close the Monitor Outputs box, and open the Digital Outputs and Digital Inputs boxes by clicking the and Toolbar icons. Drag the boxes apart on the desktop so that both are completely visible.

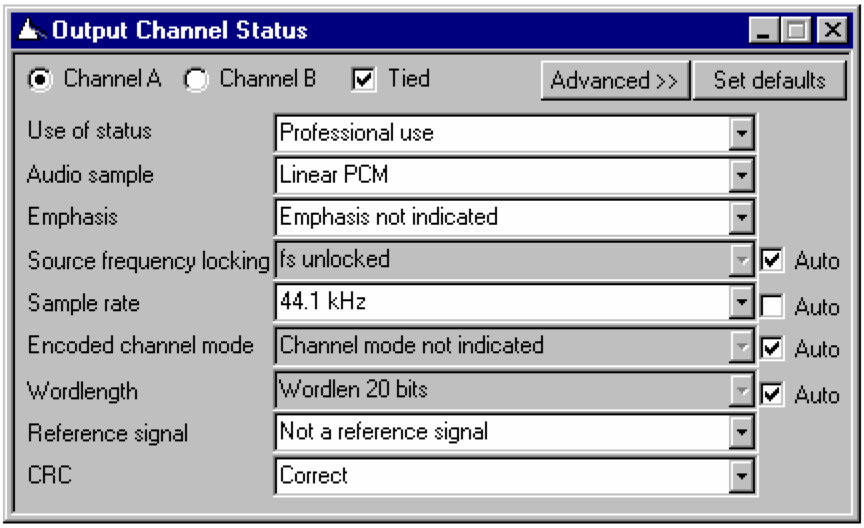
Try changing the frame rate of the Digital Outputs, and forcing a deviation of a few ppm (parts per million). Note that these changes are reflected by the measurement in the Digital Inputs box. Likewise changing other parameters such as the wordlength or Valid bits at the Digital Outputs are registered by the Digital Inputs analyzer.

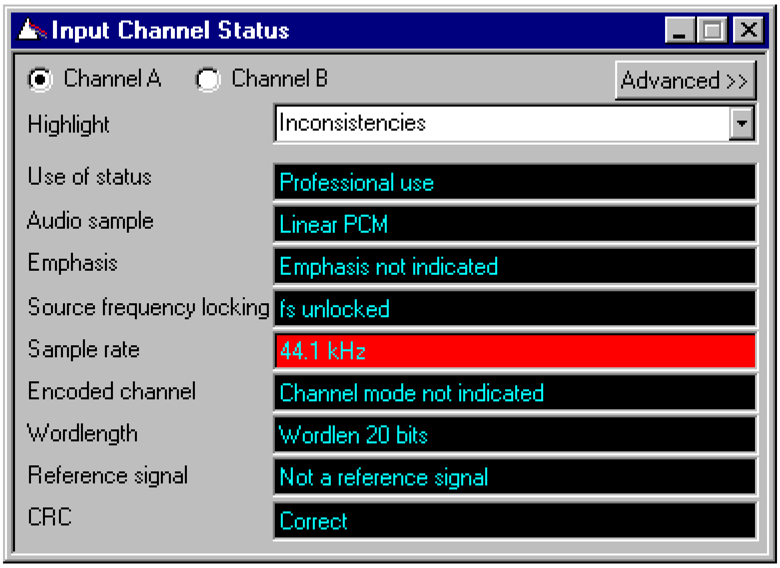




Click the [Channel Status...] buttons on each of the Digital Outputs and Digital Inputs boxes and drag the resulting boxes to the lower part of the screen so that all four boxes are completely visible. The Output and Input Channel Status boxes allow outgoing Channel Status fields to be set, and incoming Channel Status fields to be displayed and analyzed.

Notice that some of the Output Channel Status fields are set to 'Auto': this means that they automatically change to agree with the actual state of the dScope Generator. For example, try changing the Digital Output frame rate again, and note that the generated and received Channel Status boxes track the change. By unchecking their associated 'Auto' checkbox, the fields can be set manually. Try doing this and setting the Sample rate field to a value different from the actual frame rate set in the Digital Outputs dialogue box. If the 'Highlight' setting in the Input Channel Status box is set to highlight 'Inconsistencies', the inconsistent field will be displayed in red.





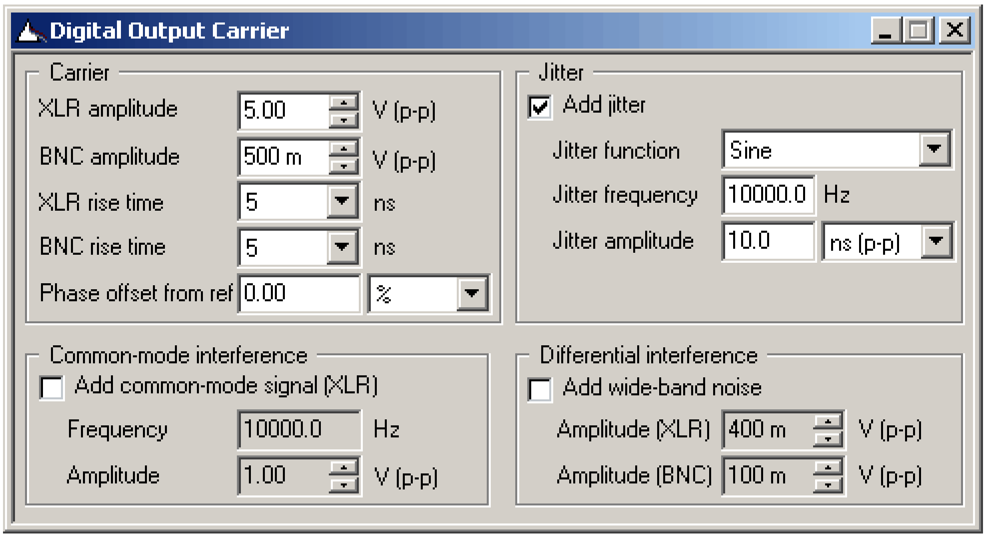
In their default 'Simple' modes, the Channel Status boxes only show the most commonly used Channel Status fields. By setting either box to 'Advanced' mode, ALL Channel Status fields are accessible.

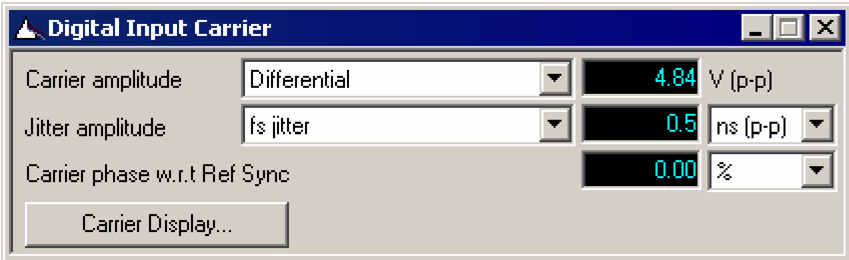
Follow these links for more information about the Digital Outputs, Digital Inputs, Output Channel Status and Input Channel Status dialogue boxes.

Digital Output and Input Carrier degradations

Open the empty Page 5 on the desktop using the 'Page' tabs, then open the Digital Output Carrier and Digital Input Carrier boxes by clicking the OUT and IN Toolbar icons. Drag the boxes apart on the desktop so that both are completely visible.

Adjust the amplitude of the generated AES3 carrier using the 'XLR amplitude' setting in the Digital Output Carrier box. Note that the changes are reflected in the 'Carrier amplitude' Result in the Digital Input Carrier box. Add jitter to the generated carrier by checking the 'Add jitter' checkbox, and note the measurement of the received carrier in the 'Jitter amplitude' Result.



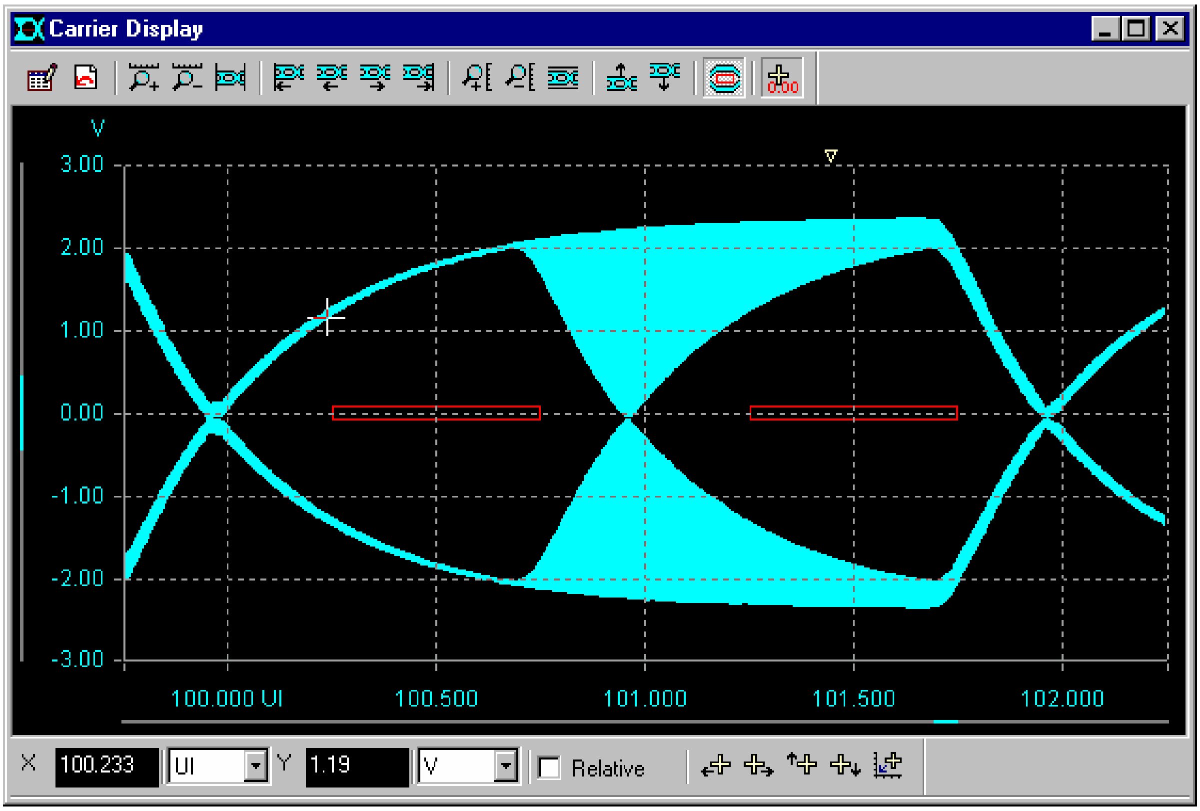


You can view the received AES3 carrier at the Digital Input by clicking the [Carrier Display...] button in the Digital Input Carrier box. The image of the designated section of the carrier waveform is built up with increasing detail as the yellow cursor makes successive passes across the window.

For a short cable (or an internal connection) the carrier should have a high amplitude and reasonably 'square' transitions. If a long or poor-quality cable is in use, the amplitude will be less, and the transitions will become rounded, leading to jitter in the recovered clock and the risk of failure of the connection.

Clicking the icon in the Carrier Display superimposes the AES3 Standard template for minimum eye-pattern opening on the display: if the carrier stays clear of the red boxes, the received carrier is within the limits set down in AES3; if the carrier encroaches into the red boxes, its eye is too closed to comply, and would probably not be reliably received by other equipment.

By reducing the 'XLR amplitude' and increasing the 'XLR rise time' in the Digital Output Carrier box, a long or poor-quality cable can be closely simulated.



Follow these links for more information about the Digital Output Carrier, Digital Input Carrier and Carrier Display dialogue boxes.

Saving and loading dScope Configurations

When you want to repeat similar tests or measurements later, it is obviously useful to be able to save the setup ('Configuration') of the dScope for later recall. In fact, this entire tour could have been provided as a Configuration file which you could have loaded immediately, but that would have defeated the object, i.e. to learn how to invoke the various functions manually.

To save the present state of the dScope as a Configuration file, click the Toolbar icon then, in the 'Save As' box, type a filename and click the [Save] button. The Configuration you saved can be loaded later in a variety of ways: either by using the Toolbar icon and then selecting the filename from the list, or by selecting the filename from the 'most recent files' list within the 'File' menu. If a Configuration will be especially useful, you can assign it to a button on the 'User bar' by entering details in the 'Customize User bar' box, accessed from the 'Utility' menu.

Close the dScope, then restart it and try loading the Configuration which you just saved.

That's the end of the tour; for more detail of any of the controls described above, check the Operation reference section.

3.5 How do I...

This section is intended to provide quick answers to "How do I..." questions concerning basic dScope features.

Many of the operations described can be performed in a number of different ways, but only the simplest is described here.

For a more detailed description of all the dScope's menus and dialogue boxes, see the Operation reference section.

The directions in the sections below assume that the user is starting from a 'default' state of the dScope (although, in general, these examples can be intermingled since they do not greatly change the dScope's Configuration).

Signal Analyzer
How do I inspect the Analyzer input signal?
How do I measure the amplitude of a signal?
How do I measure THD+N?
How can I see a 'distortion trace'?
How do I monitor an input on the loudspeaker?

Signal Generator
How do I set up the Signal Generator?
How do I generate Dolby and DTS encoded streams?

Signal Generator / Analyzer
How do I make a frequency response sweep?
How do I display frequency responses quickly without sweeping?
How do I do multi-tone testing?
How do I make acoustic measurements?
What is 'D/A line-up'?

Digital Carrier Analyzer
How do I measure the jitter of a digital input?
How do I inspect the 'eye-pattern' of a digital input?
How do I inspect the jitter spectrum of a digital input?

General
How do I save the state of the dScope?
How do I fit everything I need on the screen at once?

How do I inspect the Analyzer input signal?

Plug the input signal into the appropriate front-panel input connector, and click the Toolbar icon if it's analogue or the icon if it's digital, to select the appropriate input. Click the icon to open the Trace window (and maximise it if you want to), then click the icon to turn the FFT trigger on.

You should now see a Scope Trace (green) and FFT Trace (red) displayed. If no Traces are visible, and a red 'FFT progress' bar is not cycling at the bottom of the screen, this may be because a zero-data digital input is preventing the dScope from triggering.

Click the , or icon to select the A–channel, B–channel or two-channel mode as required. It may be necessary to click the icon on the Trace Toolbar to auto-zoom the Y-scale of the Scope Trace if its low level makes it appear small; the FFT Trace has a logarithmic Y-scale so no adjustment should be necessary.

The Scope Trace displays the amplitude of the input against time, rather like a conventional oscilloscope.

The FFT Trace shows the same information, but displayed as a frequency spectrum. This is a very easy way to see low-level distortion products which wouldn't be visible on the Scope Trace. If the input is a sine-wave, the FFT Trace should show a single peak at the sine frequency, with a flat randomly-changing noise floor beneath. Small peaks at frequency-multiples of the sine frequency are harmonic distortion products. Try double-clicking on some feature of the FFT Trace, and note that the Cursor Toolbar appears at the bottom of the screen. The coordinates of the Cursor are displayed in the Toolbar.

Further information about the Trace window can be found in the Analyzer menu section of the Operation reference.

How do I measure the amplitude of a signal?

Plug the input signal into the appropriate front-panel input connector, and click the Toolbar icon if it's analogue or the icon if it's digital, to select the appropriate input. Click the icon to bring up the Signal Analyzer dialogue box; the RMS amplitude of both input channels is shown in the upper part of the box. If the signal is periodic and of sufficient amplitude, the frequencies of the channels are also indicated along with the inter-channel phase.

Select the desired units using the list-boxes in the top right-hand corner of the dialogue box. The Signal Analyzer measures both channels at once, irrespective of the FFT Analyzer channel switch. The 'Default filters' shown in the Signal Analyzer don't affect the RMS amplitude Reading in the dialogue box – they are used by the Continuous-Time Analyzer and FFT Analyzer ONLY. To measure amplitude with a high-pass, low-pass and/or Weighting filter, use the Continuous-Time Detector (you can bring it up from the 'Analyzer' menu) in 'Amplitude' mode. You can select its own filters in it's dialogue box, but by default it uses the central selections from the Signal Analyzer dialogue box.

For further information, refer to the Analyzer menu section in the Operation reference, and the Signal Analyzer architecture section in the Hardware reference.

How do I measure THD+N?

Plug the input signal into the appropriate front-panel input connector, and click the Toolbar icon if it's analogue or the icon if it's digital, to select the appropriate input.

Bring up the Continuous-Time Detector by selecting it from the 'Analyzer' menu. The Detector's function should already be set to 'THD+N relative' by default – if not, select it from the list box. The THD+N Results for both channels should now be displayed at the top of the dialogue box. You may want to change the units from 'dB' to '%' using the list-box.

The Continuous-Time Detector measures both channels at once, irrespective of the FFT Analyzer channel switch (on the main Toolbar and in the Signal Analyzer box).

You can select high-pass, low-pass and/or Weighting filters for the measurement on the right-hand side of the Continuous-Time Detector dialogue box. If these selections are set to 'Follow defaults', the 'Default filters' settings from the Signal Analyzer box are used.

The Continuous-Time Detector can perform a wide range of measurements, as shown in the 'Function' list-box. 'FFT Detectors' are very similar to the CTD, but they can perform more complex measurements, and up to 40 may be used at the same time. However, they are slower than the CTD and, since they work on occasional buffers of data, may miss transitory input events.

For further information about the Continuous-Time Detector and FFT Detector dialogue boxes, refer to the Analyzer menu section in the Operation reference; for further information about the Continuous-Time Analyzer and FFT Analyzer architectures, refer to the Architecture section in the Hardware reference.

How can I see a 'distortion trace'?

Using a periodic (preferably sinusoidal) input signal, follow the How do I measure THD+N? and How do I inspect the Analyzer input signal? sections above to set the Continuous-Time Detector to THD+N mode, and activate the Trace window for the desired channel(s).

Click the icon on the Trace Toolbar to add a new Trace, and select 'Ch X Live CTA output' to request the distortion trace. You may need to repeat this operation for the other channel's distortion trace if you are viewing both channels at once. Now click the icon to auto-zoom the Y-scale to make the distortion trace a nice size.

It is also possible to pass the input and CTA residual signals to the Monitor BNCs on the front of the dScope, where they can be connected to an oscilloscope; however, it is an important part of the dScope's portability philosophy that residuals can be viewed without the need for an oscilloscope.

Further information about the Trace window can be found in the Analyzer menu section of the Operation reference.

How do I monitor an input on the loudspeaker?

Plug the input signal into the appropriate front-panel input connector, and click the Toolbar icon if it's analogue or the icon if it's digital, to select the appropriate input.

Click the icon to bring up the Monitor Outputs dialogue box. Click the Mute Monitor Outputs button to unmute the Monitors then, in the bottom right-hand corner of the box click the Analyzer Stereo (BNC 1 & 2) button to select a mix of both Analyzer inputs (or 'BNC 1' or 'BNC 2' for A–channel or B–channel only). Check that the volume knob on the front of the dScope is turned up – you should now be monitoring the selected input.

The Monitor Outputs are auto-ranged by default, so if the input is a continuous signal or tone, the monitoring level is adjusted automatically. However, if the signal is music, or some other intermittent signal, it may be preferable to select a fixed monitor gain in the 'Gain' list-box in the top right-hand corner of the dialogue box to avoid 'pumping'.

For further information about the Monitor Outputs dialogue box, refer to the Inputs/Outputs menu section in the Operation reference; for further information about the Monitor Outputs architecture, refer to the Architecture section in the Hardware reference.

How do I set up the Signal Generator?

Click the icon on the Main Toolbar to open the Signal Generator dialogue box.

The dScope Signal Generator generates Digital and Analogue Outputs at the same time, with a fixed line-up (see What is 'D/A line-up'? below). If you're only interested in either analogue or digital, simply select your preferred units from the 'Amplitude' units list-box, and then enter the required amplitude to the left. Frequency is entered in the box below.

The dScope Signal Generator is very versatile: apart from the basic sine function, many others can be generated, as selected by the 'Function' list-box. It is also possible to generate completely different signals on the A–channel and B–channel outputs, by selecting 'Split' in the 'Generator mode' list-box at the top of the dialogue box.

Apart from the [On] buttons in the Signal Generator dialogue box which control both the analogue and digital Signal Generators, it is also possible to mute the Analogue and Digital Outputs separately using the and icons respectively on the Main Toolbar.

For more information, refer to the Generator menu section in the Operation reference, and the Signal Generator architecture section in the Hardware reference.

How do I generate Dolby and DTS encoded streams?

dScope can generate a wide variety of multi-channel test signals in the form of Dolby Digital and DTS encoded streams. To find out how to do it, see the Encoded Stream Generation section of the dScope Applications Manual.

How do I make a frequency response sweep?

This section gives detailed instructions about how to set up a frequency response sweep manually, but a range of useful preset sweeps are instantly available by clicking the [Sweeps] button on the default User bar.

By following the How do I set up the Signal Generator? and How do I measure the amplitude of a signal? sections above, generate a sine wave (analogue or digital) of suitable amplitude, pass it through a 'device under test', and measure the amplitude of the (analogue or digital) return using the Signal Analyzer.

Select 'Sweep Setup' from the 'Sweeps' menu. Select 'Ch X RMS amplitude' in the 'Result 1' list-box, according to which channel you are analyzing. If you want to sweep both channels simultaneously, select the RMS amplitude of Ch A in Result 1 and Ch B in Result 2. In the 'Source' list-box, select 'Generator frequency (both channels)', and select the 'Start' and 'Stop' frequencies for the Sweep (say 20 and 20000 Hz) and the number of desired steps (say 30) below.

Now click the [Settings>>>] button to the right of the Result 1 list-box and, in the new dialogue box, enter the desired units, and the desired upper and lower ranges, for the Y Scale of the Sweep. Close both the dialogue boxes.

Click the icon on the Main Toolbar to start the Sweep. The Trace window will open (if it was not open already) – maximise it if you wish – and the Sweep Trace(s) will build up across the Trace area. You can repeat the Sweep by clicking the icon.

By checking the 'Append to existing sweeps' check-box in the Sweep Setup dialogue box, multiple Sweeps can be built up on the Trace area.

Further information about the Sweep Settings dialogue box can be found in the Sweeps menu section of the Operation Reference.

How do I display frequency responses quickly without sweeping?

Sweeps have been the traditional way to measure frequency response, but they are slow - and particularly awkward if you need to examine a response continuously whilst it changes. dScope offers a variety of techniques to display a 'continuous' frequency response.

Try selecting the 'Bin centres' function in the Signal Generator dialogue box, and set the Window function to 'None (Rectangular)' in the FFT Parameters dialogue box. The Live FFT Trace now shows the frequency response of the EUT in 'real time'. Check that the number of points selected for the 'Bin centres' function matches the number of FFT points (both default to 4k, so no changes should be necessary). For more information, see the paragraph about the 'Bin centres' function in the Signal Generator dialogue box section.

You can also make fast frequency response measurements using Impulse Response techniques - or with multi-tones as described in the following section.

How do I do multi-tone testing?

Multi-tone testing allows many different parameters of an EUT to be measured quickly and simultaneously. It is therefore especially useful in production-line applications.

Full details of multi-tone testing can be found in the Principles of Multi-tone Analysis section. However, you can get going right away by clicking the [Multi-tones] button on the default User bar.

How do I make acoustic measurements?

dScope can make a wide range of acoustic measurements using Impulse Response analysis. This is described in the Principles of impulse response analysis section, including a guide to how to get started fast. The dScope's Generator and Analyzer can accept calibration data to allow direct setting and reading of dBSPL amplitudes.

What is 'D/A line-up'?

'Digital/Analogue line-up' is useful when measuring 'split-domain' systems – i.e. systems which have analogue inputs or outputs as well as digital inputs or outputs. Examples include A-to-D converters, D-to-A converters, digital mixing consoles and digital tape recorders.

By telling the dScope what amplitude relationship exists between the digital and analogue ports of the equipment under test, the dScope can generate or analyze one domain in the units of the other. For example it could measure a digital signal in Volts, or an analogue signal in dBFS!

Why is this useful? Consider testing, say, an A-to-D converter: it is handy to be able to generate an analogue input to the converter of a known relationship to the digital range; for instance to measure the THD+N at –1dBFS, just set the dScope Generator to –1dBFS. If the dScope's 'D/A line-up' has been set to agree with the converter-under-test, the correct Analogue Output amplitude will be generated.

The same situation occurs when testing a D-to-A converter, but in this case the *Analyzer* 'D/A line-up' would need to be set so that it could read analogue amplitudes in dBFS. Normally, the 'D/A line-ups' for the dScope's Analyzer and Generator are locked together, since most split-domain systems have common line-up relationships for inputs and outputs. However, it is possible to make them independently adjustable by un-checking the relevant box in the 'Options' dialogue box.

How do I measure the jitter of a digital input?

Plug the input signal into the front-panel Digital Input connector, and click the Toolbar icon to bring up the Digital Input Carrier dialogue box, where the Jitter amplitude (peak-to-peak) is displayed.

The 'fs jitter' setting measures the peak-to-peak jitter at a part of the carrier which is minimally affected by high-frequency losses in interface cabling, and so its measurement is confined to jitter from the sourcing equipment.

The 'Data jitter' mode includes jitter resulting from cabling losses, so a high reading in this mode is usually an indication of a long or poor-quality cable.

The dScope's Digital Outputs can be degraded in a number of ways using the Digital Output Carrier dialogue box (accessed by clicking the Toolbar icon).

Source jitter can be added directly, or long cabling can be simulated by reducing the amplitude and increasing the rise time of the output carrier.

Further information about the Digital Input Carrier and Digital Output Carrier dialogue boxes can be found in the Inputs/Outputs menu section of the Operation reference.

How do I inspect the 'eye-pattern' of a digital input?

Plug the input signal into the front-panel Digital Input connector, and click the Toolbar icon to bring up the Digital Input Carrier dialogue box, then click the [Carrier Display] button. Alternatively, the Carrier Display can be selected directly from the 'Analyzer' menu, without opening the Digital Input Carrier dialogue box.

The Carrier Display (eye-diagram) is progressively built up, with more detail being added on successive scans. The scales can be adjusted either by accessing the Carrier Display Settings dialogue box (by clicking the icon in the Carrier Display Toolbar, or by using the other Carrier Display Toolbar icons, which offer similar zooming and scrolling actions to those icons on the Trace Toolbar.

Any section of the AES3 carrier can be displayed, by selecting the desired start and stop points in the Carrier Display Settings dialogue box. This is most easily done in 'UI' units. One UI (unit interval) is a single AES3 carrier cell duration, i.e. 1/128 of a sample period. 0UI is the start of the X–preamble (before the A–channel data), 64UI is the start of the Y–preamble (before the B–channel data) and so on.

It is also possible to pass the digital input carrier signal and a suitable sync-pulse to the Monitor BNCs on the front of the dScope, where they can be connected to an oscilloscope; however, it is an important part of the dScope's portability philosophy that carrier waveforms can be viewed without the need for an oscilloscope.

Further information about the Carrier Display window can be found in the Inputs/Outputs menu section of the Operation reference.

How do I inspect the jitter spectrum of a digital input?

The dScope is able to demodulate the jitter signal from the selected digital input, and switch it to the analogue input of the Signal Analyzer. In this way it is possible to analyze the incoming jitter signal in much more detail than simply reading its peak-to-peak amplitude. This is achieved as follows:

Plug the input signal into the front-panel Digital Input connector, and click the Toolbar icon to bring up the Analogue Inputs dialogue box, then select 'Jitter demodulator (fs jitter)' in the 'Source' list-box. It is now possible to inspect the demodulated jitter signal in just the same way as an ordinary Analogue Input, as described in How do I inspect the Analyzer input signal? above. Note that the Signal Analyzer must be set to analyze the *Analogue* Input to do this (click the Toolbar icon).

The Scope Trace shows the jitter signal in the time domain, whilst the FFT Trace shows the spectrum of the incoming jitter. The amplitude of the jitter can be confirmed on the Y–Scale of the Traces (in UI or ns), or by using the Signal Analyzer.

The 'fs jitter' demodulator mode outputs the jitter from a part of the carrier which is minimally affected by high-frequency losses in interface cabling, and so demodulates only that part of the jitter produced by the sourcing equipment. The 'Data jitter' mode includes jitter resulting from cabling losses, so a higher output in this mode is an indication of a long or poor-quality cable.

How do I save the state of the dScope?

Having set up the dScope in the state you wish to save, click the icon on the Main Toolbar to bring up the Save As dialogue box. In the 'File name' box, type the name you want the Configuration file to have, then click the [Save] button. The saved Configuration can be loaded at any time by clicking the icon on the Main Toolbar to bring up the Open dialogue box, then clicking on the file name in the list box, then clicking the [Open] button.

Note that it is possible to choose which parts of the dScope Configuration you want to save, using the hierarchical check-tree in the Save As dialogue box. Then, at load time, other parts of the dScope's Configuration will not be disturbed. Similarly, in the Open dialogue box, the save-time choice of parts can be over-ridden since, actually, the entire Configuration was saved.

You can place a button on the 'User bar' which will cause your saved Configuration to be instantly loaded, as described in the Customize User bar section.

Further information about the Save Configuration and Load Configuration dialogue boxes can be found in the File menu section of the Operation reference.

How do I fit everything I need on the screen at once?

The dScope user interface incorporates a number of features to help you fit everything you need on the screen at the same time:

Pages

The most convenient way to make the most of the available desktop space is to make use of a number of different 'Pages'. By clicking the Page tabs in the bottom right-hand corner of the dScope box, you can switch between five different 'desktops'! You can open different windows, dialogue boxes, panels, Readings etc. on each Page, and they'll still be there when you switch back to it. The easiest way to make use of multiple Pages is to use each Page for a different task, with the necessary features for the task open on its dedicated Page.

Panels

Many of the dScope's dialogue boxes are divided into ruled 'panels'. By placing the mouse cursor on a panel and then holding down the left mouse-button, a copy of the panel can be 'dragged off' the dialogue box. The original dialogue box can then be closed, leaving only the copy of the desired panel, thus avoiding the need to clutter the Page with the rest of the dialogue box which you didn't need.

Readings

By placing the mouse cursor over a 'Result' (a black box on a panel, containing a result in blue/green text) and then holding down the left mouse-button, the Result can be 'dragged off' the panel to create a 'Reading'. The Reading remains on the Page even after the original panel or dialogue box which sourced it has been closed. Actually, the main purpose of Readings is not simply to save space. They have many visual and functional features not available with Results, as described in the Reading window section of the Operation reference.

Sizing dialogue boxes and panels

Many features within the dScope user interface can be resized to make them smaller (or bigger). The dialogue boxes and panels have been designed with the most-used features towards the top-left, and the less-used features towards the bottom-right. By placing the mouse cursor over the bottom or right-hand edge of a box (or a dragged-off panel) it is replaced by a double-headed arrow, indicating that the box can be resized by holding down the left mouse-button. It is thus possible to reduce the size of dialogue boxes and dragged-off panels whilst still retaining their main functionality. Certain windows (e.g. Trace window, Script Edit window, Carrier Display) can be resized from the left or the top as well.

3.6 Model numbers

Starting from software version 1.30, the dScope software interface has some optional features depending on the version of hardware purchased.

Feature	dScope Series IIIA (Analogue)	dScope Series IIIA+ (Analogue-Plus)	dScope Series IIID (Analogue and Digital)	dScope Series IIIE (Essentials)	dScope Series III (Analogue and Digital)
Analogue I/O	✓	✓	✓	✓	✓
Soundcard I/O	✓	✓	✓	✓	✓
FFT Analysis	✓	✓	✓	✓	✓

Impulse Response	✓	✓	✓	✓	✓
Scripting	✓	✓	✓	✓	✓
Multi-tones	✗	✓	✗	✓	✓
Advanced Multi-tones	✗	✓	✗	✗	✓
FFT Detectors	Limited to 2	40 Detectors	Limited to 2	Limited to 2	40 Detectors
Scripted FFT Detectors	✗	✓	✗	✗	✓
Nested & Sensed Sweeps	✗	✓	✗	✗	✓
Sweep input on X axis	✗	✓	✗	✗	✓
Run Scripts on each Sweep step	✗	✓	✗	✗	✓
Regulation	✗	✓	✗	✓	✓
dS-NET I/O Switcher Support	✗	✓	✓	✓	✓
dS-NET VSIO Adapter Support	✗	✗	✗	✗	✓
Port access from scripts	✗	✓	✗	✗	✓
Event Manager	✗	✓	✗	✗	✓
Analogue sample rates	48kHz, 96kHz	48kHz, 96kHz, 192kHz	48kHz, 96kHz, 192kHz	48kHz, 96kHz, 192kHz	48kHz, 96kHz, 192kHz
Digital I/O	✗	✗	✓	✓	✓
Digital Carrier I/O	✗	✗	✗	✗	✓
Channel Status I/O	✗	✗	✓	Simple only	✓
Ref Sync I/O	✗	✗	✗	✗	✓
Monitor Outputs	✓	✓	✓	✓	✓
Digital Carrier & Sync Pulse Monitor Outputs	✗	✗	✓	✓	✓
Signal Generator Functions	Sine / Swept Sine / Pink & White noise / Twin-tone only	All functions	Sine / Square / Swept Sine / Twin-tone / Ramp / Pink & White noise / Twin-tone only	All functions	All functions

Analogue-Plus (A+) License files

The capabilities of the Analogue-Plus model are enabled with a license file, which must reside in the dScope program folder along with the dScope.exe file. Usually, this license will automatically be installed to the correct place from the CD shipped with the dScope.

If your software installation was not performed from this CD, or a license is added at a later date, then the license file(s) may need to be installed manually. To do this, simply copy the license file(s) into the dScope program folder (probably "C:\Program Files\Prism Sound\dScope Series III"). On the installation CD, license files are contained within a "Licenses" folder. They have file names of the form "dScopeLicense_12345.txt".

Note that multiple license files for different dScope serial numbers can all be copied into the same dScope program folder. Multiple Analogue-Plus dScope units can be used with different PCs by simply copying all relevant license files into the dScope program folder on each PC.

3.7 Compatibility of software Versions

When functional improvements or new features are added in a new dScope software version, it is sometimes inevitable that resulting changes to the user or automation interfaces of the dScope may cause compatibility issues for some customers' applications which were developed with earlier versions of the dScope software.

Consequently, when upgrading to a new dScope software Version, please review the version-specific information in the following sections.

Please also refer to the Issues with software upgrades page in the Scripting Manual for automation-related issues.

Version 1.20

Removal of Generator MLS function

From version 1.20 this function is no longer supported. Earlier versions had no capability to perfrom MLS correlation analysis in any case, whereas version 1.20 improves on the MLS technique by using Bin centre or Swept sine stimuli for correlation and impulse response analysis. Configurations and VBScripts which select the MLS function may no longer function correctly. Any users for whom this is aproblem should contact Prism Sound Technical Support.

Part 4

Operation reference

4 Operation reference

The operation reference section provides detailed descriptions of all the available menus and dialogue boxes.

Main sections are as follows:

File menu	Loading and saving setups, printing etc.
Edit menu	Context-sensitive editing
View menu	Visibility of dScope user-interface elements
Inputs/Outputs menu	Analogue and Digital I/O and Monitor controls and Results
Generator menu	Signal Generator controls
Analyzer menu	Signal Analyzer controls and Results, including Trace window
Sweeps/Regulation menu	Sweep and Regulation controls
Automation menu	Scripting and the Event Manager
Utility menu	Various options and housekeeping functions
Window menu	Window arrangement and selection
Help menu	On-line help and various 'abouts'
Reading window	A powerful way of displaying numerical results
Slider control	A control which can be used to progressively alter settings
Status bar	Area where state indicators and warnings are displayed
Icons and Hotkeys reference	
Amplitude units in dScope	Describes the way that amplitude units are applied

4.1 File menu

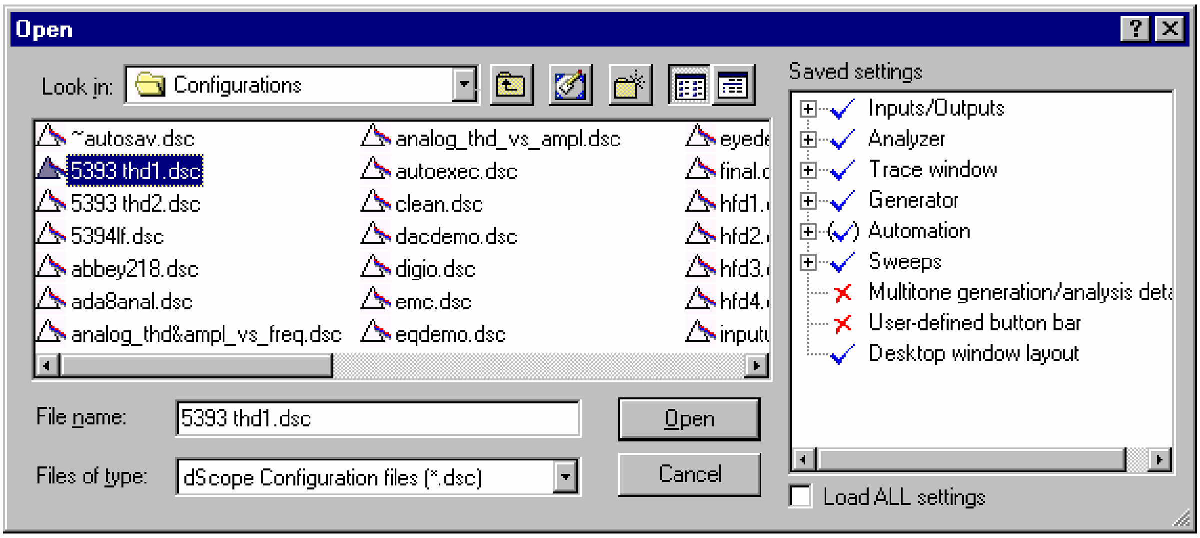
The File menu provides access to various file input/output and global operations within the dScope.

Menu options are:

Load Configuration...	Loads a dScope Configuration file from disc.
Save Configuration	Saves the current dScope Configuration to disc as existing filename.
Save Configuration as...	Saves the current dScope Configuration to disc.
Print...	Context-dependent printing of the selected dialogue box.
Print Preview...	Displays a print-preview screen of the selected dialogue box.
Print Setup...	Displays the Windows printer setup dialogue box.
Page Header/Footer Setup...	Displays a dialogue box to configure the header and footer lines for printed pages.
Graph Print/Export Setup	Displays a dialogue box to control printing and exporting of graphical output.
[List of Config Files]	Loads a recently-saved Configuration file from disc.
Exit	Closes the dScope application.

4.1.1 Load Configuration dialogue box

The Load Configuration dialogue box allows the Configuration of the dScope to be loaded from a previously-saved Configuration file.



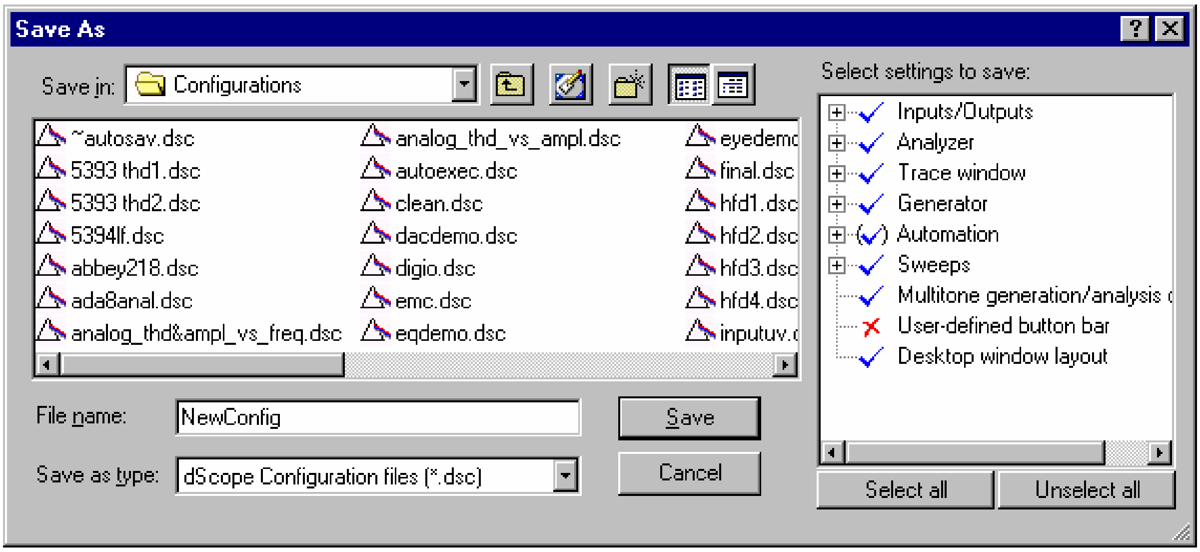
The blue ticks in the 'Saved settings' box show which sections of the dScope Configuration will be loaded, and the red crosses show which sections will not. Clicking on the ticks and crosses toggle the inclusion of that section.

In the current software version, it is not possible to select the sections to be loaded at load time; it is only possible to choose to recall either the selection specified at save time, or the entire dScope Configuration.

Note that the buttons in the upper centre of the box (used for navigating the folder structure, changing the file list view etc.) may differ from those shown, since they are inherited from the standard file open box of the particular Windows version in use.

4.1.2 Save Configuration As dialogue box

The Save Configuration As dialogue box allows dScope Configurations (setups) to be saved for later recall.



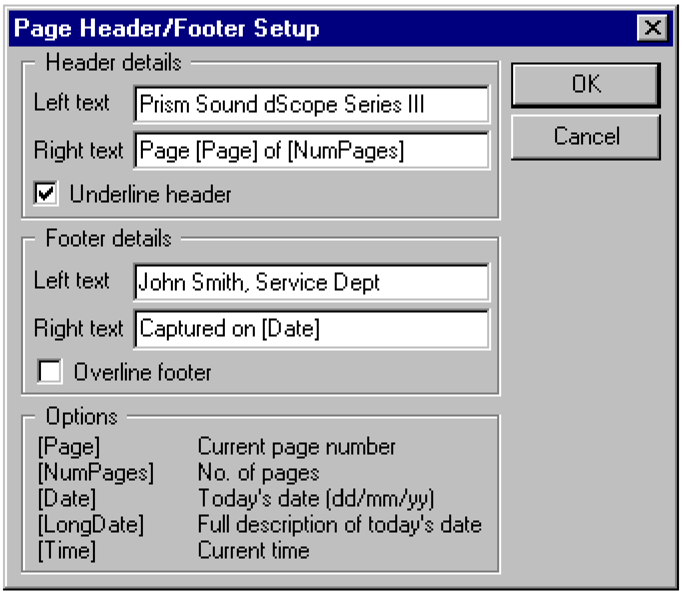
The blue ticks in the 'Select settings to save' box show which sections of the dScope Configuration will be saved for later recall, and the red crosses show which sections will not. Clicking on the ticks and crosses toggle the inclusion of that section.

Actually, the entire dScope Configuration is always stored to enable the choice of sections to be over-ridden later at load time.

Note that the buttons in the upper centre of the box (used for navigating the folder structure, changing the file list view etc.) may differ from those shown, since they are inherited from the standard file save box of the particular Windows version in use.

4.1.3 Page Header/Footer Setup dialogue box

The Page Header/Footer Setup dialogue box is used to specify the headers and footers to be included on a page printed from dScope.

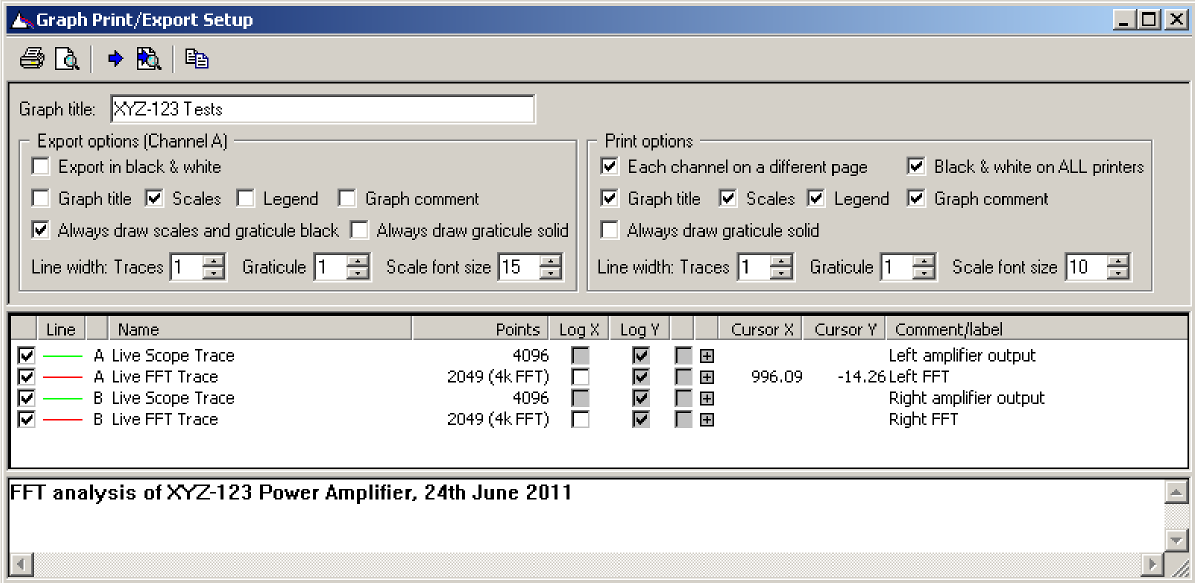


dScope has a context-sensitive printing function, i.e. selecting Print or Print Preview will act on the currently selected dialogue box. In all cases, the printed output will bear the headers and footers defined in this dialogue box. Graphical prints (prints of the Trace window) also have a title which is specified using the Graph Title button in the Trace Toolbar.

Note that some special functions can be obtained by typing any of the special option strings (in square brackets) as shown in the lower part of the box.

4.1.4 Graph Print/Export Setup dialogue box

The Graph Print/Export Setup dialogue box is used to control the way graphical data is printed and exported.



The Graph Print/Export Setup dialogue box is divided into several distinct areas; from top to bottom, these are:

Graph title	Title shown at the top of the printed or exported graph.
Export options	Options applicable to graph exports only.
Print options	Options applicable to graph printouts only.
Print/Export legend	Allows details of the Trace legend to be set. This appears beneath the graph, and provides a key to the various Traces, including details of Cursors, Marks etc.
Comment area	General text area for comments appended beneath the printed or exported graph.

Export options

A number of check boxes are available to customise graphics files exported from dScope:

Export in black and white	Exported graph is two-colour: black on a white background. This may be useful if the export is destined for a printer which attempts greyscales, producing faint lines from some colours.
Graph title	Include graph title on exported graph.
Scales	Include scales on exported graph.
Legend	Include legend on exported graph.
Graph comment	Include comment area on exported graph.
Always draw scales and graticule black	The graticule and scale numbers are exported in black, rather than their screen colour assignments.
Always draw graticule solid	The graticule is exported as solid lines, rather than dashed lines as on the screen.
Line width: Traces	Allows selection of the width of all Trace lines on the exported graph.
[Line width:] Graticule	Allows selection of the width of the graticule lines on the exported graph.
Scale font size	Allows selection of the font size for scales on the exported graph.

On clicking the [Export Graph] button, a Windows 'Save as' dialogue box appears which allows the filename and file format of the graphical export to be specified. Available formats are BMP, JPG, GIF, TIFF, PNG or EMF (Windows Enhanced Metafile). Graphical data exported in the Enhanced Metafile format retains the various elements of the graphical output as separate objects so that they may be individually modified in target applications, for example the colour or line thickness of an individual Trace can be changed if required. The other formats are 'flat' bitmaps and can only be subsequently processed as such.

Print options

dScope prints through the Windows printing system and should therefore produce acceptable results with any Windows-compatible printer. It may be necessary to force black and white output if you are using a monochrome printer which insists on attempting greyscales, since these are often hard to see, especially if lines are thin.

Each channel on a different page	If the graph to be printed in two-channel mode, each channel is printed on a separate page.
Black and white on ALL printers	Printout is two-colour: black on a white background. This may be useful if the printer is of a type which attempts greyscales, producing faint lines from some colours.
Graph title	Include graph title on printed graph.
Scales	Include scales on printed graph.
Legend	Include legend on printed graph.
Graph comment	Include comment area on printed graph.
Always draw graticule solid	The graticule is printed as solid lines, rather than dashed lines as on the screen.
Line width: Traces	Allows selection of the width of all Trace lines on the printed graph.
[Line width:] Graticule	Allows selection of the width of the graticule lines on the printed graph.
Scale font size	Allows selection of the font size for scales on the printed graph.

Print/Export legend

Each Trace in the Trace area has an entry in the Print/Export legend, which is primarily intended to provide details about the Traces when a graphical printout or export is made. The Print/Export legend is arranged in twelve columns, as detailed below (the leftmost four columns correspond to the columns in the Quick legend):

'Enabled' check box:	Controls and indicates whether or not the Trace is enabled for viewing and printing/exporting.
Line style:	Shows the colour and style of the printed Trace – can be changed by clicking. Note that until the print/export colour is changed, it follows the colour selected for screen display. Once set explicitly, the print colour is not affected by changes to the screen colour.
Channel indicator:	Indicates the source channel of the Trace.
Trace name:	Shows the name of the Trace, which can be edited by double-clicking.
Number of points:	Indicates the number of points in the Trace.
Log X, Log Y:	Indicates whether the X and Y scales of the Trace are logarithmic (when checked) or linear. When not greyed, these can be toggled by clicking.
'Show Marks' check box:	Includes a list of Marks with the print/export when checked.
'Expand Marks' tool:	Expands or contracts the list of Marks on-screen.
Cursor X, Y:	Show the X and Y Cursor positions, if a Cursor is enabled on the Trace. If the 'Show Marks' check box is checked, the list of Marks is shown beneath the Cursor, with any Mark comments appended to the right in the Comment/label field.

Comment/label:	Shows the user-defined comment for the Trace; double click to edit.

4.2 Edit menu

The Edit menu provides basic windows editing functions for use within some of the dScope windows.

Menu options are:

Cut	Deletes the currently selected text or object and copies it to the clipboard.
Copy	Copies the currently-selected text or object to the clipboard.
Paste	Inserts the text or object in the clipboard at the current cursor position.
Select All	Selects all the text in the current window (where applicable).

4.3 View menu

The View menu allows the basic elements of the dScope user-interface to be displayed or hidden as required.

Primary menu options are:

Toolbar	Toggles the main dScope Toolbar on and off.
User bar	Toggles the User bar on and off.
Status bar	Toggles the Status bar on and off.

In addition, this menu may have an additional context-sensitive view-list added at the bottom, if the currently selected window or dialogue box has suitable elements. For example, the Trace window has a number of elements, including its own Toolbar, which can be displayed or hidden using this menu, but these items only appear when the Trace window is the curently-selected window. Other windows with context-sensitive view lists include the Script Edit window and the Carrier Display window.

4.4 Inputs/Outputs menu

The Inputs/Outputs menu provides access to the dialogue boxes which control the various inputs and outputs of the dScope.

Menu options are:

Digital Outputs...	Settings and Results of the Digital Outputs (data).
Digital Output Carrier...	Settings of the Digital Output Carrier.
Analogue Outputs...	Settings of the Analogue Outputs.
Soundcard Outputs...	Settings of any Soundcard Outputs in use.
Digital Inputs...	Settings and Results of the Digital Inputs (data).
Digital Input Carrier...	Settings and Results of the Digital Input Carrier.
Carrier Display...	Opens a window showing a Trace of the Digital Input Carrier.
Analogue Inputs...	Settings and Results of the Analogue Inputs.
Soundcard Inputs...	Settings of any Soundcard Inputs in use.
Monitor Outputs...	Settings of the Monitor Outputs, headphone and loudspeaker.

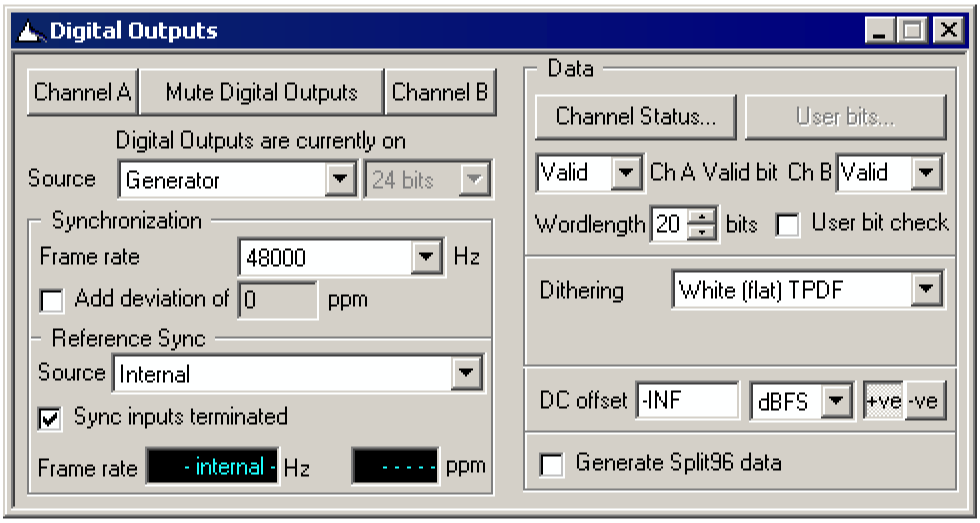
4.4.1 Digital Outputs dialogue box

NB:

This dialogue box may not be available, depending on the dScope model number.

The Digital Outputs dialogue box provides control and display of the functions associated with the Digital Outputs of the dScope. Note that carrier-related functions are dealt with by a separate Digital Output Carrier dialogue box.

For a block diagram and description of the relevant areas of the dScope hardware, see Digital Output and Carrier architecture and Reference Sync architecture.



Digital Output Source panel

The Digital Output mute buttons cause the audio part of the Digital Output channels to be muted, independently of the Analogue Outputs. When muted here, the Digital Output audio data is set to black (all zeroes), whereas muting the Signal Generator allows dither and DC offset to be maintained. The three-button layout allows the A and B channels to be controlled separately or together.

The source selector drop-list selects whether the Digital Output is sourced from the Signal Generator or looped through from the Digital Input. The latter state is useful for monitoring a digital signal 'in-line', in which case the terminating impedance would normally be switched out in the Digital Input dialogue box.

The third state of the source selector, Channel Check, sends a separate pseudo-random bit sequence (PRBS) to both Digital Output channels for the purpose of checking data channel integrity. This can be used for checking data channels such as routers, digital recorders, satellite links, wiring etc. The sequence is compatible with the Channel Check mode of the Prism Sound DSA–1 hand-held analyzer. The sequence is self-locking, so that generating and verifying equipment may be physically remote. 24, 20 or 16–bit variants of the sequence can be generated for different wordlength data channels. The Channel Check sequence can be verified in the Digital Inputs dialogue box.

Digital Output Synchronization panel

NB:

Note that, owing to the high data rates involved, use of the BNC and XLR Digital Outputs at frame rates of 176.4 and 192kHz is subject to transmission-line effects and requires the use of high bandwidth cabling and interface devices. All

cabling must be single point-to-point without splitting. Never leave a cable plugged into a Digital Output connector whilst connected 'back-to-back' internally. Use of the TOSLINK outputs and inputs at these frame rates is not supported.

The Reference Sync for the Digital Output can be selected from Internal, AES11 (on XLR or BNC), Wordclock, Digital Input (whichever is selected in the Digital Inputs dialogue box), or video. The frame rate of the selected Reference Sync is displayed, along with its deviation from the nearest standard rate.

The Reference Sync inputs termination can be switched in or out as required (75R for BNC, 110R for XLR).

When locked to external AES11, WCK or DI, the frame rate of the Digital Output can be selected either to follow the selected Reference Sync, or independently: i.e. the Digital Output can be at a different rate from the Reference Sync. In the latter case, the Digital Output is rationally related to the Reference Sync, i.e. if the Reference Sync is identified as 44.1kHz and the Digital Output is set to 48kHz, the Digital Output frame rate is set to 480/441 times the Reference Sync frame rate.

When locked to video, the relationship between audio and video frames can be complex, according to the video standard supplied and the desired output audio sample rate. The following table shows the number of audio samples per video frame in all modes:

fs	**PAL/SECAM (25fps)**	**NTSC (30/1.001 fps)**	**NTSC (30fps)**
32kHz	1280	16016/15	3200/3
44.1kHz	1764	147147/100	1470
48kHz	1920	8008/5	1600
88.2kHz	3528	147147/50	2940
96kHz	3840	16016/5	3200
176.4kHz	7056	147147/25	5880
192kHz	7680	32032/5	6400

Note that if the video reference is selected as PAL/SECAM/NTSC(29.97), then the reference rate is discriminated as either 25fps or 29.97fps, and the relationship from one of the two left-hand columns in the table is established (even if the actual rate is 30fps). If NTSC(30) is selected, the relationship from the right-hand column is established (even if the actual rate is 29.97fps). Thus pull-down audio sample rates such as 44.056kHz (i.e. 44.1kHz/1.001) can be generated from suitable video references.

Whether externally or internally referenced, a deviation from the nominal frame rate can also be set for the Digital Output, up to +/–1500ppm (parts per million) in 1ppm steps. This is useful for testing the tolerance of digital inputs.

Further details can be found in the Reference Sync architecture section.

Digital Output Data panel

The data content of the Digital Output includes auxiliary data as well as audio.

The Channel Status and User bits buttons access special dialogue boxes for setting these parameters. (NB: User bits not yet supported, except for transparency checking as described below).

The Valid bits can be set or cleared individually for each channel of the output.

The wordlength can be set anywhere between eight and 24 bits, either with the addition of TPDF dither or by direct truncation.

By checking the 'User bit check' check-box, an alternating pattern of 1's and 0's (channels in antiphase) is transmitted in the User bit of the digital output. The pattern can be verified in the Digital Inputs dialogue box in order to test User bit transparency of an EUT.

A DC offset can be added (prior to dithering or truncation) which can be specified in a variety of units.

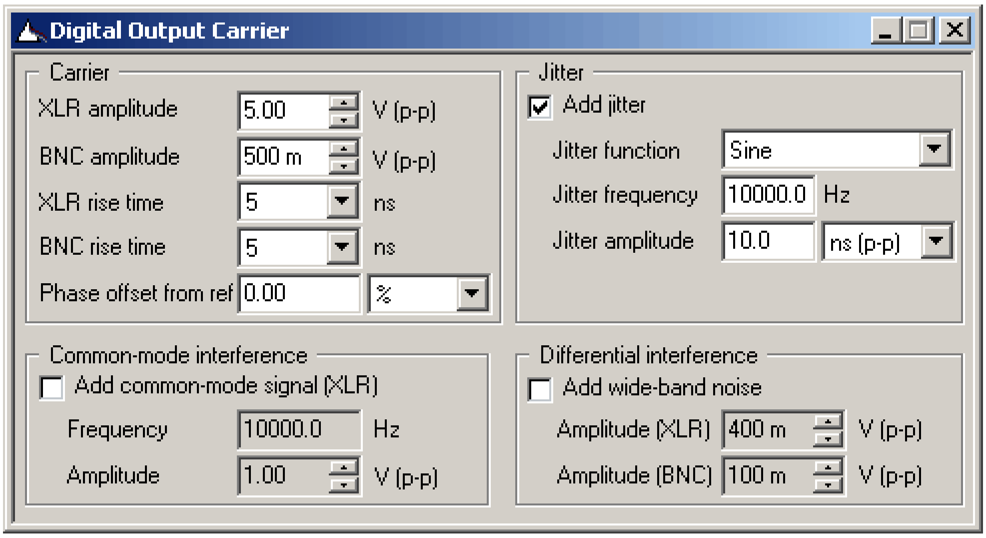
NB: **Note that the DC offset facility can be used to present fixed bit patterns on the Digital Outputs. By muting the Signal Generator, setting the Digital Output wordlength to 24 bits, and dither off, it is possible to express a DC in Hex units, which produces a corresponding static bit pattern at the output. It is a simple matter to verify bit patterns at the dScopes Digital Inputs using the bit activity indication in the Digital Inputs dialogue box.**

In Split96 mode, the Digital Output carries only one channel in 'two-wire' mode; i.e. the A and B carrier channels carry a single audio channel whose sample rate is twice the selected frame rate. Split96 mode at the supported frame rates of 32kHz to 96kHz corresponds to sample rates of 64kHz to 192kHz. In Split96, Signal Generator A–channel feeds the Digital Output, although the Signal Generator may still be placed in 'split' mode in case split signals are needed at the Analogue Outputs.

4.4.2 Digital Output Carrier dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The Digital Output Carrier dialogue box provides control and display of the functions associated with the Digital Output Carriers of the dScope. For a block diagram and description of the relevant area of the dScope hardware, go to Digital Output and Carrier architecture.



Digital Output Carrier panel

The peak-to-peak amplitudes of the XLR and BNC Digital Output Carriers are independently controlled from 120mV to 10.24V for the XLR output, and from 30mV to 2.56V for the BNC output. These assume correct termination loads of 110R and 75R respectively, and will be doubled if unterminated. The amplitude of the TOSLINK output cannot be varied.

The rise/fall times of the XLR and BNC outputs can also be independently varied between 5ns and

100ns. Note that selecting long rise/fall times for high frame rates will result in reduced carrier amplitudes. The rise/fall time of the TOSLINK output cannot be altered.

Long runs of various types of cable can be simulated by selecting low carrier amplitudes and long rise/fall times.

The carrier phase (common to all formats) with respect to the selected Reference Sync can be set in % (of a frame), degrees (1/360 of a frame) or UI ('unit intervals', 1/128 of a frame).

Digital Output Carrier Jitter panel

Various types of jitter can be added to the Digital Output Carriers (all formats), expressed in ns or UI peak-to-peak. Sine (10Hz–40kHz), audio-band noise (white, 0–40kHz) or wide-band noise (0–12MHz) jitter can be added up to 1/2 UI (3/8 UI at 88.2 or 96kHz, 1/4 UI at 176.4 or 192kHz). Low-frequency sine (10Hz–10kHz) jitter can be added up to 20UI. The latter is necessary to cover the jitter-tolerance requirements of AES3.

Digital Output Carrier Noise panel

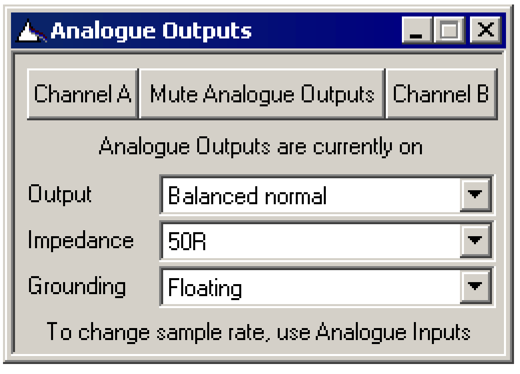
Differential wide-band noise (0–12MHz) interference can be added to the XLR and BNC Digital Output Carriers. This setting is 'ganged', with the XLR (maximum 2.56Vp–p) having four times the noise amplitude of the BNC (maximum 640mVp–p).

Digital Output Carrier CM panel

Common-mode sinusoidal interference (10Hz–40kHz, up to 20Vp–p) can be added to the XLR Digital Output Carriers. This feature is useful in testing resilience to mains or high-frequency pickup.

4.4.3 Analogue Outputs dialogue box

The Analogue Outputs dialogue box controls the parameters of the Analogue Outputs of the dScope. For a block diagram and description of the relevant area of the dScope hardware, go to Analogue Output architecture.



The Analogue Outputs can be muted independently of the Digital Outputs, although both may be muted together by muting the Signal Generator. The three-button layout allows the A and B channels to be controlled separately or together.

The output can be configured for balanced or unbalanced operation, or else for common-mode testing of inputs, where the two balanced legs of the output carry the same signal with respect to signal ground rather than a differential signal.

The XLR and BNC output connectors are both available as balanced or unbalanced outputs: when balanced mode is selected, the inner of the BNC and pin 2 of the XLR are connected to 'hot', and the outer of the BNC and pin 3 of the XLR to 'cold'. Pin 1 of the XLR is connected to signal ground. In unbalanced mode, the outer of the BNC and pin 3 of the XLR are also connected to signal ground. Note that adapters are supplied so that RCA/phono plugs can be used with the dScope's BNC connectors. For more information, see the Unbalanced operation and grounding section.

The D/A converter sample rate (which, in current software versions is locked to the Analogue Inputs' A/D converter sample rate) can be switched between 48kHz, 96kHz and 192kHz. The sample rate is adjusted in the Analogue Inputs dialogue box as noted at the bottom of the box. For more information, see the Analogue I/O sample rate section.

The maximum amplitude capability of the Analogue Outputs is normally +28dBu (balanced) or +22dBu (unbalanced), into a minimum load of 150R. Note that the maximum amplitude capability is reduced by 0.5dB to +27.5dBu (balanced) or +21.5dBu (unbalanced) when the analogue sample rate is 192kHz. In the present software versions, the sample rate of the Analogue Outputs is tied to that of the Analogue Inputs, as set in the Analogue Inputs dialogue box.

The output impedance selections vary according to the mode of the output: in unbalanced mode, the output impedance can be selected between 25R or 600R; in balanced mode (or CM test mode) the differential output impedance can be selected between 50R, 150R, 600R or asymmetric (600R in one leg and 25R in the other). The asymmetric mode is useful for testing the 'real world' common-mode rejection of an input circuit, since many input circuit designs rely on having a balanced source impedance to maintain good CMRR performance.

The 150R setting can be changed to 200R, if required, by making a jumper selection on the Analogue Board as described in PCB jumper options. In this case, the menu in the dialogue box is automatically changed, and the Analogue Input impedance option is correspondingly altered. Note that muted outputs retain the same source impedance as when they are not muted.

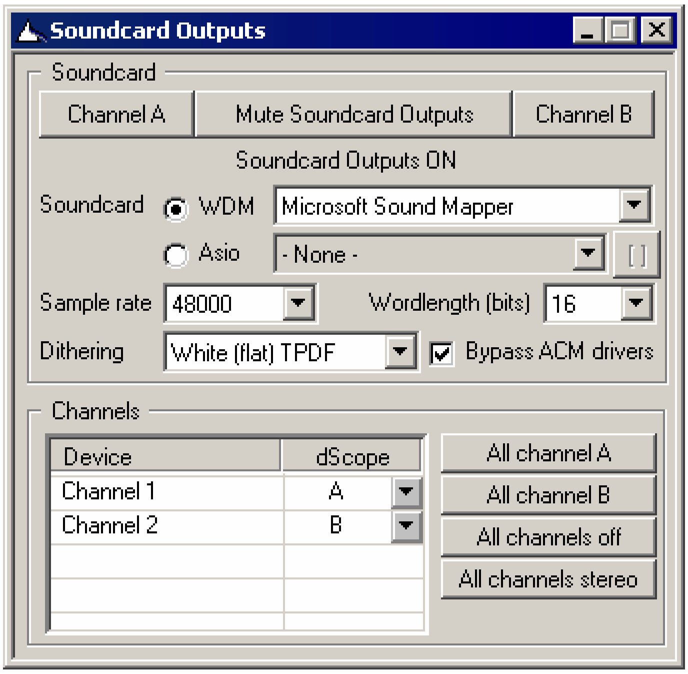
The grounding arrangement of the Analogue Outputs can be switched between floating, and XLR pin 1 (signal ground) connected to chassis. Note that the switched connection between Analogue Output ground and chassis is protected by a fuse as described in Fuses and ratings. Note that when the Analogue Output signal ground is coupled to the Analogue Input signal ground (the normal state, as described in PCB jumper options) then the 'chassis' setting also connects the Analogue Input signal ground to chassis.

4.4.4 Soundcard Outputs dialogue box

The Soundcard Outputs dialogue box allows a Windows sound device to be nominated for signal generation. This may be useful in a variety of situations: For example it opens up the possibility to use dScope with almost any media interface, for example MADI or Ethernet audio, so long as a PC card and Windows driver are available. It also enables the testing of Windows sound devices themselves, such as Bluetooth headsets or FireWire audio interfaces, since the dScope can access both the host and outside-world audio ports.

Once a Soundcard is selected, the dScope's two-channel Signal Generator feeds the nominated device, as well as continuing to feed the dScope's Analogue and Digital Outputs in the normal way.

⚠ **In order to maximise system performance, it is advisable to ensure that the Soundcard selection is set to "- None -" when no Soundcard output is required**.



Soundcard Output Source panel

At the top of the panel, the A and B channel feeds from the dScope Signal Generator to the Soundcard Outputs can be individually or collectively muted.

The 'Soundcard' drop-menus allow any installed WDM or ASIO sound devices to be selected. Choosing the WDM 'Microsoft Sound Mapper' option allows Windows to take control of Soundcard output selection (i.e. it uses the default soundcard as set up in the Sound section of the Windows Control Panel). In order to maximise system performance, it is advisable to ensure that the Soundcard selection is set to "- None -" when no Soundcard output is required.

The sample rate and wordlength can be set to any allowed by the nominated Soundcard. Note that in some circumstances a Soundcard's driver may claim to support sample rates or wordlengths which the Soundcard cannot natively support. This situation can result in sample rate conversion or truncation happening 'behind the scenes', often with a bad effect on signal quality. It is advisable to check the native capabilities of your Soundcard. Note that if the Signal Generator is set to a frequency beyond the range of the Soundcard's nominated sample rate, no signal is generated. White TPDF dither can be applied to the Soundcard feed if required.

If 'Bypass ACM drivers' is checked (WDM only), the audio samples are passed directly from dScope to the Soundcard driver, and the use of Windows' ACM drivers is prevented. If enabled, ACM drivers are applied invisibly by Windows to perform format conversion in situations where the Soundcard's native capabilities do not include the parameters of the audio being played out. See above.

When generating or analyzing using Soundcard I/O, the amplitudes of audio samples transacted with the Soundcard are treated in exactly the same way as with the dScope's Digital Inputs and Outputs. That is to say, a signal at 0dBFS corresponds to the maximum amplitude of the Soundcard. For information about using D/A line-up to express analogue amplitudes directly when using Soundcards, see the Soundcard Generation and Analysis section in Amplitude units in dScope.

NB:

If an ASIO soundcard is selected, and 'ASIO' is also selected on the Soundcard Inputs, then the output soundcard must either be the same ASIO soundcard, or '- None -'. This will be set automatically by the software, since generation and analysis cannot use different ASIO soundcards. This restriction does not apply to WDM soundcards.

Soundcard Output Channels panel

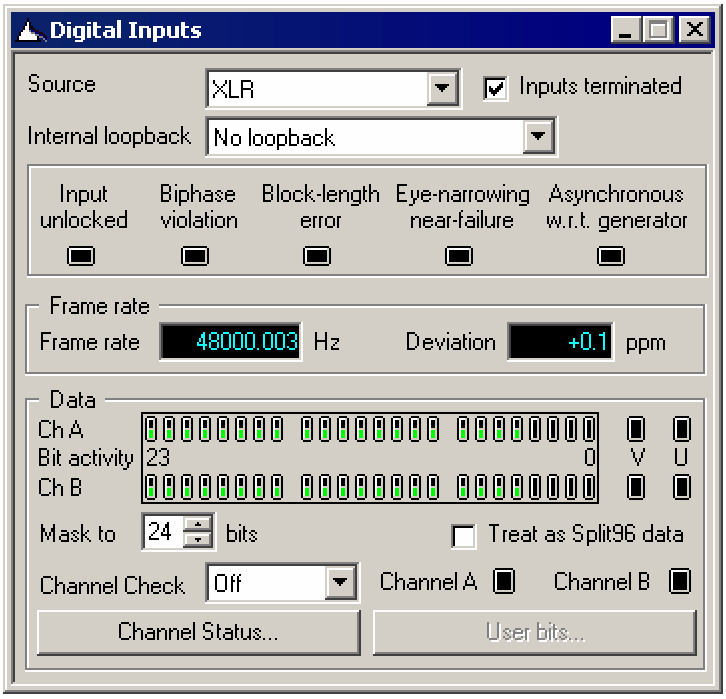
This panel controls the mapping between the dScope Signal Generator's A and B channels and the actual Soundcard channels. It allows the user to select a Signal Generator channel (or none) for each output channel of the nominated Soundcard. This facility is especially useful in the case of multi-channel Soundcards. Mappings may be selected individually for each output channel, or shortcut buttons may be used to select [All channel A], [All channel B], [All channel Off] or [All channels stereo]. The stereo selection routes all odd-numbered output channels from Generator A and even-numbered channels from Generator B.

4.4.5 Digital Inputs dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The Digital Inputs dialogue box provides control and display of the functions associated with the Digital Inputs of the dScope. Note that carrier-related functions are dealt with by a separate Digital Input Carrier dialogue box.

For a block diagram and description of the relevant area of the dScope hardware, go to Digital Input and Carrier architecture.



Digital Input Source panel

The source selector drop-list selects between the three supported input formats (XLR, BNC or TOSLINK) or enables a direct relay connection from either the XLR or BNC Digital Output, using the 'Generator XLR' and 'Generator BNC' settings. The XLR (110R) and BNC (75R) input terminations are switchable.

The internal loopback drop-list allows ONE of the digital input channels may be routed internally from the opposite Generator output channel data, whilst the other input channel is driven from the selected source as normal. This allows the two Analyzer channels to receive Generator channel A or B, both before AND after the EUT, in the same way as is possible in the Analogue Inputs dialogue box. This allows measurement of phase response and delay through a digital EUT. Possible settings are 'Channel A (Ch A Gen on Ch B)', 'Channel B (Ch B Gen on Ch A)', or 'No loopback'. NOTE: this setting should not be confused with the physical carrier loopback settings of the main source selector drop-list, described above.

Indications are provided for 'Input unlocked' (when no compliant input can be detected), 'Biphase violation' (when required transitions are missing), 'Block-length error' (when the repeat rate of the Z–preamble is not 192 frames), 'Eye-narrowing near-fail' (when the cell-duration falls below 50% of the ideal value), and 'Asynchronous w.r.t. generator' (when the input is either outside +/–90 degrees of the generator carrier phase, or is slipping with respect to it).

NB:

Note that, owing to the high data rates involved, use of the BNC and XLR Digital Outputs at frame rates of 176.4 and 192kHz is subject to transmission-line effects and requires the use of high bandwidth cabling and interface devices. All cabling must be single point-to-point without splitting. Never leave a cable plugged into a Digital Output connector whilst connected 'back-to-back' internally. Use of the TOSLINK outputs and inputs at these frame rates is not supported.

Digital Input Frame Rate panel

The incoming frame rate is displayed, along with the deviation in parts-per-million from the assumed standard rate.

Digital Input Data panel

The audio data bit activity is displayed as a bar with the most-significant bit on the left. Each bit is shown black if permanently zero, green if permanently one, and half-green if changing. It is possible to 'mask' the incoming audio word to a fixed wordlength prior to analysis, which is useful in simulating the viewpoint of an input with limited wordlength, or to check the dither of a non-truncated output.

The state of both Valid bits is also indicated: the indicators are off for the 0 (valid) state, and illuminated red for the 1 (invalid) state.

The User bit indicators are off if the User bit has continuous 0 data, and red if there is any non-zero activity - unless a continuously alternating pattern of 0s and 1s is detected, as transmitted by the dScope's digital outputs when in 'User bit check' mode, in which case the indicators are lit green. This allows the User bit transparency of the EUT to be tested. See the Digital Outputs dialogue box for more details.

Buttons provide access to dedicated dialogue boxes for displaying incoming Channel Status and User bits. (NB: User bits not yet supported, except for transparency checking as described below).

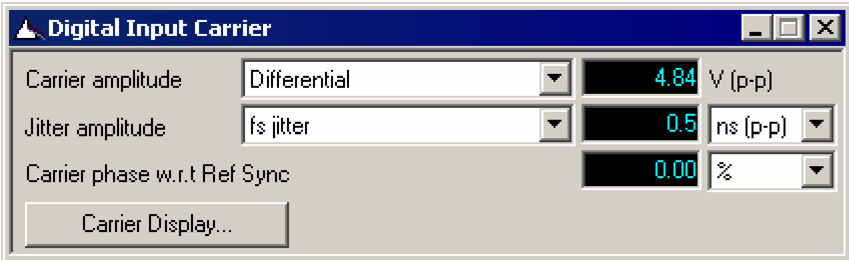
When Split96 mode is selected, the Digital Input is treated as a single channel in 'two-wire' mode; i.e. the A and B channels are assumed to be shared by a single channel whose sample rate is twice the indicated frame rate. Split96 mode at the supported frame rates of 32kHz to 96kHz corresponds to sample rates of 64kHz to 192kHz. In this case, only single channel analysis is possible with the Digital Input selected.

In the lower part of the Digital Input Data panel, Channel Check verification can be displayed. Channel Check verification checks for the presence of a separate pseudo-random bit sequence (PRBS) on both Digital Input channels for the purpose of checking data channel integrity. This can be used for checking data channels such as routers, digital recorders, satellite links, wiring etc. When Channel Check mode is enabled (for 24, 20 or 16–bit channel wordlengths) in the drop list, the 'Channel A' and 'Channel B' indicators light green to show that the sequence is recognised, and flash red if a bit error is detected. Sequence failures can be set to trigger audible alarms, entries in the event log, etc. as detailed in the Event Manager dialogue box. The sequence is compatible with the Channel Check mode of the Prism Sound DSA–1 hand-held analyzer. The sequence is self-locking, so that generating and verifying equipment may be physically remote. The Channel Check sequence can be generated in the Digital Outputs dialogue box.

4.4.6 Digital Input Carrier dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The Digital Input Carrier dialogue box provides control and display of the functions associated with the Digital Input Carriers of the dScope. For a block diagram and description of the relevant area of the dScope hardware, go to Digital Input and Carrier architecture.



The peak-to-peak amplitude of the selected Carrier Input can be measured differentially, common-mode (XLR only) or audio-band (band-limited to 20kHz to detect accidental routing/mixing of an analogue source or mains interference).

Timing degradation of the incoming carrier can be measured as fs jitter (attributable to the source), data jitter (attributable to the source and the cabling) or 'eye-narrowing' (the reduction in the duration of the eye-pattern from the 'ideal', i.e. 1UI, measured in UI or in %).Eye-narrowing can be measured either at zero-crossing, or with a 200mV threshold as specified in the AES3 standard.

The detected carrier phase with respect to the selected Reference Sync can be measured in % (of a frame), degrees (1/360 of a frame) or UI ('unit intervals', 1/128 of a frame).

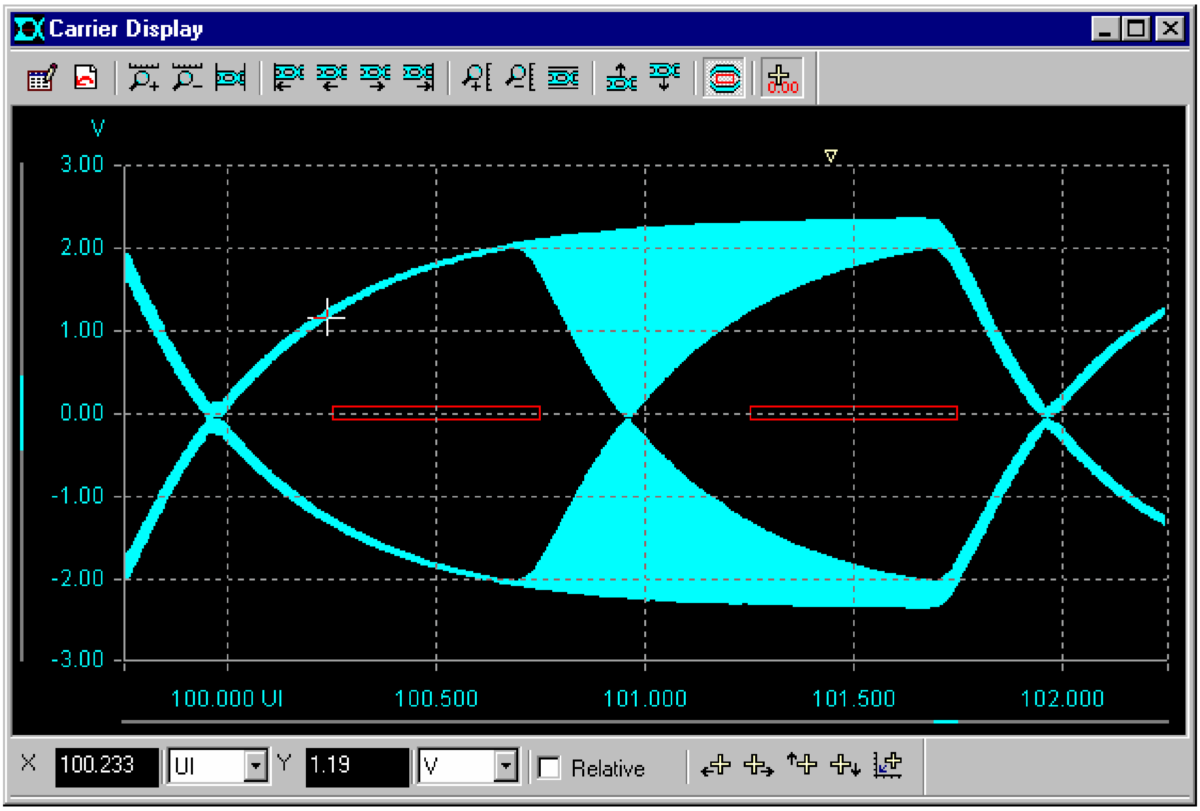
A button provides instant access to the Carrier Display window, where a Trace of the actual incoming carrier is displayed. Note that the Jitter amplitude Result is not available whilst a Carrier Display window is open.

Note that jitter on the Digital Input Carriers can be demodulated and switched to the Analogue Input of the dScope's Signal Analyzer for time-domain and spectral (FFT) analysis. This selection is made in the Analogue Inputs dialogue box.

4.4.7 Carrier Display window

NB: **This window may not be available, depending on the dScope model number.**

The Carrier Display window shows the waveform of the carrier at the selected Digital Input, rather like using a digital storage oscilloscope.



The Carrier Display of the selected part of the carrier frame is built up by a scanning yellow arrow at the top of the display. The maximum and minimum excursions of the carrier waveform above and below the 'eye' are shaded to clearly show the areas of uncertainty, for example where occasional transitions occur. In the default mode, the time-resolution is made progressively finer on each successive scan, building up a more and more detailed picture.

The Carrier Display window has its own dockable and sizeable Toolbar which docks at the top of the window, and its own dockable and sizeable Cursor Toolbar which docks at the bottom.

The Carrier Trace can be positioned either using the Trace control icons on the Toolbar (for zooming, panning, scrolling etc.), or by dragging a 'zoom box' over the desired section of the graph using the mouse.

As well as the Trace manipulation icons, the Toolbar contains some special function icons: At the left of the Toolbar are buttons for accessing the Carrier Display Settings dialogue box, for pausing, and for restarting the scan of the carrier (this happens automatically if the carrier unlocks or the frame rate changes).

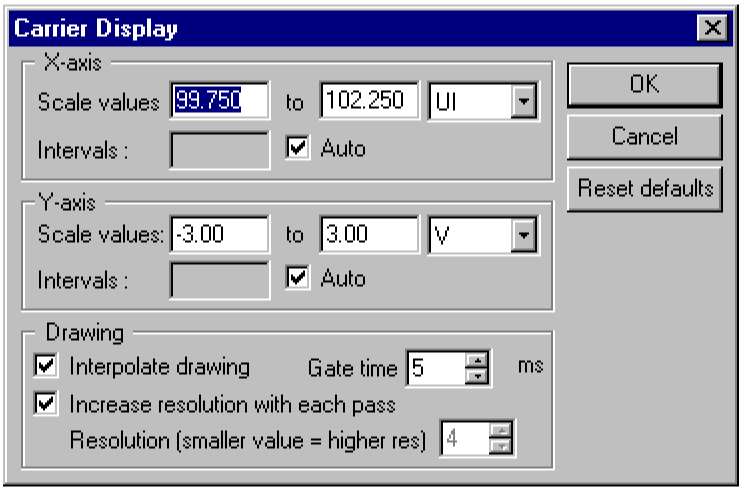
At the right of the Toolbar are icons to display or hide the AES3 template – a row of red boxes which show the worst-case eye requirement from the AES3 standard – and to display or hide the Cursor. The Cursor Toolbar ar contains an X/Y display of the current Cursor position, with selectable units, and a check-box to put the Cursor in relative mode – when it becomes effectively two cursors where the X and Y separations are displayed. There are also buttons for manipulating the Cursor position, although this can also be performed by dragging it around the graph with the mouse pointer.

Manual control of many advanced parameters of the Carrier Display window is available in the Carrier Display Settings dialogue box.

See the Carrier Display icons section of the Icons and Hotkeys reference for full details of all the Carrier Display Toolbar icons.

4.4.7.1 Carrier Display Settings dialogue box

This Carrier Display Settings dialogue box contains settings for the dScope's Carrier Display window.



The range of the X axis can be set to cover the required part of the carrier waveform, as an offset from the beginning of the AES3 frame, either in ns or in 'UI' (unit intervals, 1/128s of the frame period, the duration of a single biphase-mark 'cell', or half a bit period). As a guide, the offsets of the various data elements in the AES3 frame are:

X–preamble (or Z–preamble):	0UI to 8UI
A–channel audio data (LSB to MSB):	8UI to 56UI
A–channel Valid bit:	56UI to 58UI
A–channel User bit:	58UI to 60UI
A–channel Channel Status bit:	60UI to 62UI
A–channel Parity bit:	62UI to 64UI
Y–preamble:	64UI to 72UI
B–channel audio data (LSB to MSB):	72UI to 120UI
B–channel Valid bit:	120UI to 122UI
B–channel User bit:	122UI to 124UI
B–channel Channel Status bit:	124UI to 126UI
B–channel Parity bit:	126UI to 128UI

The range of the Y axis should be set to accommodate the required carrier voltage range. For both the X and Y axes, the number of graticule divisions can be set, or this can be left to the dScope software by selecting Auto.

The drawing parameters allow a high degree of flexibility in the way the Carrier Display is built up:

The 'Interpolate' box determines whether the graph drawing will interpolate between successive points.

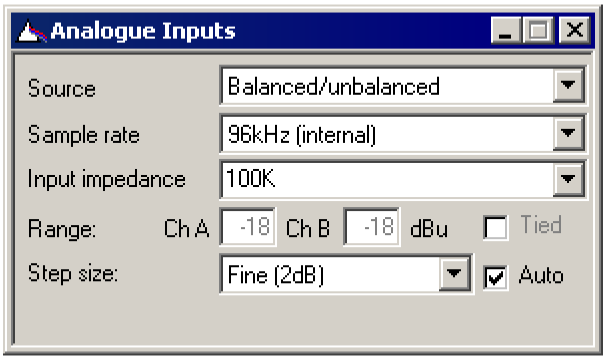
The 'Gate Time' determines how long the dScope will scan on each point – this is important in determining the certainty with which infrequent carrier transitions will be detected: for example the X–preamble is replaced by a single Z–preamble every 192 frames. This equates to every 4ms at a frame rate of 48kHz. If the Gate Time is set below 4ms, then detection of Z–preamble activity will be unreliable. Setting a short Gate Time speeds up the scan, whereas a long Gate Time improves detection of infrequent events.

The Time Resolution of the display can be set in arbitrary units between 1 and 256. The smaller the number, the finer the resolution. A setting of 1 corresponds to a time resolution of about 300ps. Low settings produce a very finely detailed graph, but very slowly. High settings are faster but reduce the level of detail.

By checking 'Increase resolution with each pass', a useful compromise is applied, where each successive pass is made with an increased time resolution. Thus it is possible to quickly see the shape of the carrier in the area of interest, and adjust if necessary, before waiting for the required degree of resolution to be attained.

4.4.8 Analogue Inputs dialogue box

The Analogue Inputs dialogue box controls the parameters of the dScope's Analogue Inputs. For a block diagram and description of the relevant area of the dScope hardware, go to Analogue Input architecture.



The Analogue Input to the Signal Analyzer can be sourced from the normal balanced / unbalanced input connectors, or can be internally fed from the Analogue Outputs, or can be set to accept the demodulated jitter signal from the Digital Input jitter demodulator. Using the latter mode, the Signal Analyzer can be used to measure RMS jitter, and the Trace window can display the waveform and spectrum of the Digital Input jitter.

Both balanced or unbalanced sources can be connected to either the BNC or XLR Analogue Input connectors. The BNC and XLR connectors are wired in parallel, so either can be used without separate selection (BNC inner and XLR pin 2 are 'hot' , BNC outer and XLR pin 3 are 'cold', XLR pin 1 is Analyzer ground); however, care should be taken to ensure that the unused input connector is not connected, since this can adversely affect results. Note that adapters are supplied so that RCA/phono plugs can be used with the dScope's BNC connectors.

The A/D converter sample rate (which, in current software versions also provides the Analogue Outputs' D/A converter sample rate) is indicated on the Analogue Inputs dialogue box. This can be switched between 48kHz, 96kHz and 192kHz. For more information, see the Analogue I/O sample rate section.

The differential input impedance of the Analogue Inputs can be set to 100kR, 600R or 150R. The 150R setting can be changed to 200R, if required, by making a jumper selection on the Analogue Board as described in PCB jumper options. In this case, the menu in the dialogue box is automatically changed, and the Analogue Output impedance option is correspondingly altered. Note that the dScope software may defeat an input impedance selection if the detected level is sufficient to damage the impedance-setting resistor. The resistors are also protected by a fuse as described in Fuses and ratings.

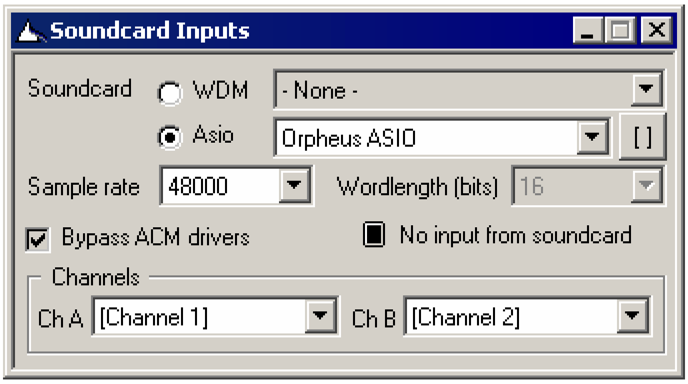
The maximum range of the Analogue Inputs can be fixed at a specified dBu amplitude, between –18dBu (97.5mV RMS) and +46dBu (154.5V RMS). This can be done on each channel separately, although by default the values are tied between the channels. More normally, the inputs can be set to auto-range depending on the amplitude of signals applied. Fixed ranging is useful if awkward waveforms or very low frequencies are being analyzed which might cause continuous hunting by the auto-range algorithm. If a fixed ranged is entered and the input amplitude exceeds it, auto-ranging takes over until the overload is removed.

The Step size of the Analogue Input auto-ranging can be selected between Fine (2dB), Medium (6dB) and Coarse (20dB). In normal circumstances, the Fine setting should always be used. The other settings can sometimes be beneficial in reducing delays due to excessive ranging, for example during Sweeps.

4.4.9 Soundcard Inputs dialogue box

The Soundcard Inputs dialogue box allows a Windows sound device to be nominated for signal analysis. This may be useful in a variety of situations: For example it opens up the possibility to use dScope with almost any media interface, for example MADI or Ethernet audio, so long as a PC card and Windows driver are available. It also enables the testing of Windows sound devices themselves, such as Bluetooth headsets or FireWire audio interfaces, since the dScope can access both the host and outside-world audio ports.

Once a Soundcard is selected, the dScope's two-channel Signal Analyzer can be selected to analyze the Soundcard's inputs using the input selector on the Signal Analyzer dialogue box, or the  icon on the Main Toolbar.



The 'Soundcard' drop-menus allow any installed WDM or ASIO sound devices to be selected. Choosing the WDM 'Microsoft Sound Mapper' option allows Windows to take control of Soundcard input selection (i.e. it uses the default soundcard as set up in the Sound section of the Windows Control Panel).

The sample rate and wordlength can be set to any allowed by the nominated Soundcard. Note that in some circumstances a Soundcard's driver may claim to support sample rates or wordlengths which the Soundcard cannot natively support. This situation can result in sample rate conversion or truncation happening 'behind the scenes', often with a bad effect on signal quality. It is advisable to check the native capabilities of your Soundcard.

If 'Bypass ACM drivers' is checked, the audio samples are passed directly from the Soundcard driver to dScope, and the use of Windows' ACM drivers is prevented. If enabled, ACM drivers are applied invisibly by Windows to perform format conversion in situations where the Soundcard's native capabilities do not include the selected audio format. See above.

When generating or analyzing using Soundcard I/O, the amplitudes of audio samples transacted with the Soundcard are treated in exactly the same way as with the dScope's Digital Inputs and Outputs. That is to say, a signal at 0dBFS corresponds to the maximum amplitude of the Soundcard. For information about using D/A line-up to express analogue amplitudes directly when using Soundcards, see the Soundcard Generation and Analysis section in Amplitude units in dScope.

It is possible to control the mapping between the dScope Signal Generator's A and B channels and the channels. The 'Channel A' and 'Channel B' drop-menus allow the user to select an input channel of the nominated Soundcard for each Signal Analyzer channel. This facility is especially useful in the case of multi-channel Soundcards.

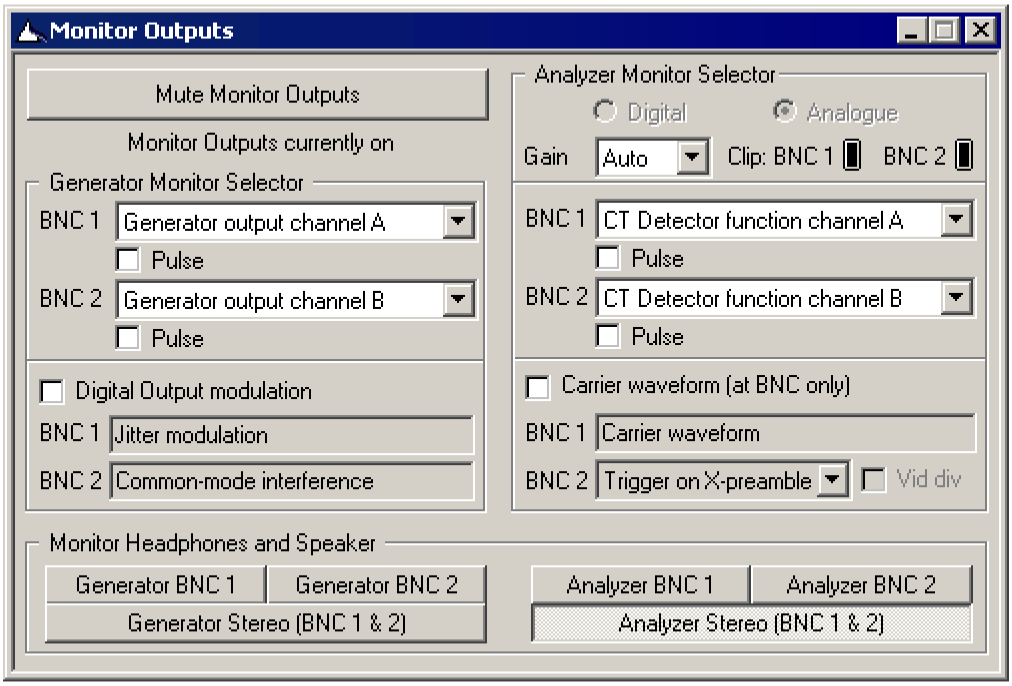
NB:

If an ASIO soundcard is selected, and 'ASIO' is also selected on the Soundcard Outputs, then the output soundcard must either be the same ASIO soundcard, or '- None -'. This will be set automatically by the software, since generation and analysis cannot use different ASIO soundcards. This restriction does not apply to WDM soundcards.

4.4.10 Monitor Outputs dialogue box

The Monitor Outputs dialogue box is used to assign various functions to the four assignable BNC Monitor Outputs on the front of the dScope unit, and also to assign the headphone socket and integral loudspeaker.

For a block diagram and description of the relevant area of the dScope hardware, go to Monitor output architecture.



The dScope's assignable monitor system is a versatile and compact way to monitor various analogue and digital signals from within the dScope hardware.

The Generator and Analyzer sections each have two assignable BNC outputs for use with oscilloscopes, external amplifiers etc. However, these will not be used as often as the auxiliary outputs of conventional measurement sets, since the dScope provides on-screen functionality for most oscilloscope-type functions, and integral audio monitoring.

The BNC outputs have a 75R output impedance, and can handle both audio-band signals and high-frequency digital carrier waveforms and sync pulses. Audio signals sent to the BNCs are automatically gain-ranged to between 2Vp–p and 4Vp–p, unterminated, (unless a manual gain has been fixed for the Analyzer Monitor from the drop-list) and digital carriers are attenuated to half their amplitude.

Loudspeaker and headphone feeds are likewise gain-ranged ahead of the volume control knob, which controls both the headphone and loudspeaker levels. Plugging in headphones causes the loudspeaker to be disabled.

'Pulse' check boxes in the main selectors allow audio signals to operate a comparator at their zero-crossing to generate TTL-compatible pulse outputs.

Monitor Outputs Mute panel

The Monitor Outputs (both the assignable BNCs and the headphone socket and integral loudspeaker) can be muted and unmuted with a single button.

Generator Monitor Selector panel

The normal mode of the Generator Monitor is to monitor the output of the Signal Generator. Normally channel A is output on BNC 1 and channel B on BNC 2, but this can be reversed.

By checking the Digital Output modulation box, BNC 1 monitors the Digital Output common-mode interference signal and BNC 2 monitors the jitter modulation signal (if it is in the audio band).

NB: **The Digital Output modulation controls may be disabled, depending on the dScope model number.**

Analyzer Monitor Selector panel

The Analyzer Monitor Outputs can each be set to follow the Analyzer input or the Continuous-Time Detector output (residuals etc.) of either channel. They can be set to follow the 'selected' (or 'unselected') channel of the Signal Analyzer so that switching the Analyzer channel automatically switches the monitor feed.

The normal auto-gain-ranging action of the Analyzer Monitor can be over-ridden and a fixed gain applied if desired. This is useful for monitoring complex waveforms, speech or music. Clip indicators are provided to aid manual gain setting.

By checking the Carrier waveform box, BNC 1 monitors the waveform of the Digital Input Carrier (whichever is selected in the Digital Input Carrier dialogue box). In this mode, BNC 2 outputs a synchronization pulse (typically for oscilloscope triggering) which can be sourced from the X or Y preamble detection, a Bitclock or the selected Generator Reference Sync.

NB: **The Carrier waveform controls may be disabled, depending on the dScope model number.**

Monitor Headphones and Loudspeaker panel

The headphones and loudspeaker are fed from a six-input selector; they can monitor either BNC 1 or BNC 2, or both BNCs at once, of either the Generator or Analyzer monitors. When monitoring both BNCs, BNC 1 is routed to the left headphone output and BNC 2 to the right, with a mono mix on the loudspeaker. Note that the Analyzer carrier mode is not reflected by the headphones and loudspeaker, which continue to reflect the selections of the main Analyzer monitor; similarly, the 'Pulse' mode does not affect them.

4.5 Generator menu

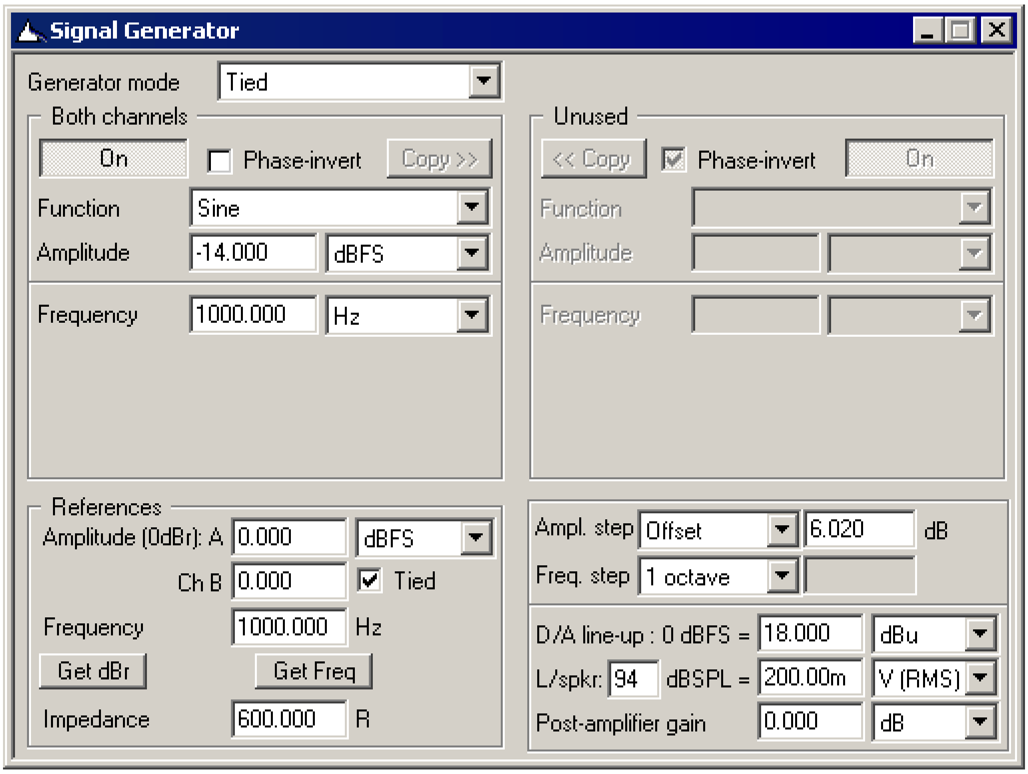
The Generator menu provides access to the dialogue boxes which control the dScope signal and data generators.

Menu options are:

Signal Generator...	Settings of the audio Signal Generator.
Channel Status...	Settings of the transmitted Channel Status.
User bits...	Settings of the transmitted User bits (NB: not yet supported).

4.5.1 Signal Generator dialogue box

The Signal Generator dialogue box controls the parameters of the dScope's Signal Generator. For a block diagram and description of the relevant area of the dScope hardware, see Signal Generator architecture.



The Signal Generator is a two-channel multi-function Generator which feeds both the Analogue and Digital Outputs simultaneously.

Signal Generator Mode panel

This panel selects between Tied mode, where the A–channel and B–channel outputs are fed with the same signal, and Split mode, where a different signal can be generated on each channel. Note that the Signal Generator cannot be operated in 'split' mode at sample rates above 96kHz. If either the analogue or Digital Outputs are operating above 96kHz, the A–channel of the Generator feeds both output channels, although they can be independently muted. An exception to this rule occurs if a table-based function is being generated; this allows multi-tone testing for both analogue channels at fs=192kHz.

Signal Generator Function panel

The Signal Generator Function panels contain the primary controls for controlling the Generator function. In Tied Mode, a single panel controls the output of both the A–channel and the B–channel, whereas in Split Mode, a separate Function panel is provided for each channel. The Copy A–B and Copy B–A buttons allow the settings of one channel to be adopted by the other channel in Split Mode.

The upper section of the panel contains an on/mute button, a phase-invert check box, a function selector list and an amplitude setting which can be made in a variety of units. Note that the phase-invert check box can be operated separately for the B-channel even when the channels are tied. In this case, the control has three states: unchecked (non-inverted), checked (inverted) and grey-checked (following A-channel, as in the picture above).

Note that the Signal Generator amplitude is 'sine peak referred', that is to say that a sine function is generated at precisely the specified level, whatever the selected units, whereas other functions are generated with the same PEAK amplitude as a sine of the specified amplitude. Whilst not strictly correct, this convention is more intuitive and thus is usual among other manufacturers' signal generators. Note that for compound stimuli such as twin-tones or multi-tones, amplitudes are specified for each element individually. Whilst the Signal Generator amplitude setting in dScope generally takes no account of source impedance (it specifies the amplitude behind the source impedance, rather than that across the terminals, in the loaded state), impedance-related units such as W and dBm (which assume a load impedance equal to the 'reference impedance') can specify terminal amplitudes in the loaded state if required, according to a setting in the Options dialogue box. For more information about the way that amplitude units are handled by the dScope, see the Amplitude units in the dScope section.

The following sections describe the available Signal Generator functions and their settings:

Sine

Settings: amplitude, frequency

This is an ultra-low-distortion synthesized sine function for use in THD+n and other single-frequency measurements.

Square (analytical)

Settings: amplitude, frequency, duty cycle, polarity

Note that this is a 'perfect' square wave, i.e. it consists of alternating runs of only two different sample values. It does not resemble a band-limited square wave such as would be created if an analogue square wave were sampled by an A/D converter.

Because of this, frequency is quite coarsely quantised (only frequencies with integer numbers of samples per cycle are available). Varying the duty cycle allows asymmetrical 'rectangular' waveforms to be output. The entered percentage is that of the 'high' part of the cycle. The polarity adjustment allows a positive-going, negative-going or bipolar output to be generated. Note that when generating a negative-going output, the entered duty-cycle applies to the negative part of the cycle.

Ramp

Settings: amplitude, frequency, duty cycle, polarity

Note that this is a 'perfect' ramp, i.e. it consists of alternating runs of only two precise gradients. It does not resemble a band-limited ramp such as would be created if an analogue ramp were sampled by an A/D converter.

Because of this, frequency is quite coarsely quantised (only frequencies with integer numbers of samples per cycle are available). Varying the duty cycle allows asymmetrical 'sawtooth' ramps to be output. The entered percentage is that of the rising part of the cycle. The polarity adjustment allows a positive-going, negative-going or bipolar output to be generated. Note that when generating a negative-going output, the entered duty-cycle applies to the falling part of the cycle.

Burst

Settings: amplitude, frequency, 2nd amplitude, t1, t2

This is a sine-burst waveform, i.e. a sinewave which switches between two preset amplitudes at a preset rate. Such a stimulus is commonly used to measure dynamics processors, among other uses.

The 2nd amplitude may be set absolutely (in the same units as selected for the main amplitude setting), or as an offset from the main amplitude (an additive offset if the main amplitude is specified in linear units, or dB if the main amplitude is specified in logarithmic units). It may alternatively be set as a ratio of the main amplitude, irrespective of the main amplitude units.

t1 (the period at the main amplitude) and t2 (the period at the 2nd amplitude) may be set in either periods (of the sinewave) or in ms. In the latter case, the values are rounded to the nearest integer number of sinewave periods.

White noise

Settings: amplitude

This noise is synthesized using a pseudo-random sequence, and is flat from DC to approximately 0.44 of the sample rate (about 21kHz at fs=48kHz), above which its response is rolled off, with a -3dB point of approximately 0.46 of the sample rate (about 22kHz at fs=48kHz).

Pink noise

Settings: amplitude

This noise is synthesized using a pseudo-random sequence, and has a pink characteristic (rolls off at 10dB per decade) from approximately 0.0008 of the sample rate (about 40Hz at fs=48kHz) up to 0.5 of the sample rate (24kHz at fs=48kHz).

Pulse

Settings: amplitude, mark-samples, space-samples, polarity

The pulse duration can be set in samples from 1 to 1000, and the interval between pulses from 1 to 512k-1. The space-period is always at zero-level, whereas the mark (pulse) may be set to be positive-going, negative going, or bipolar. In the bipolar case, a positive-going pulse of the set duration is followed immediately by a negative-going pulse of the same duration.

This function has many uses, including acoustic measurements and testing meters and digital overload indicators.

Swept sine

Settings: amplitude, start frequency, end frequency, log, samples, space period, ramp-up period, ramp-down period, repetitions

The swept sine function (sometimes known as a 'chirp') is primarily intended for use with impulse response analysis - see the Principles of impulse response analysis and Impulse Response

Parameters dialogue box sections for more details.

A sine wave is swept between the selected start and end frequencies. The progression of the sweep frequency against time can be set to be linear or logarithmic. The duration of the sweep is entered indirectly as a sample count of 2^N samples (since the FFT Analyzer must be set to this number of points for impulse response analysis to work correctly). A space period can be entered (in samples) which is a number of samples of silence to be included at the end of each sweep. This can be useful to allow reverberance to die away between sweeps in acoustic measurements. A ramp-up and ramp-down period (also in samples) can be set, which may be useful in minimising FFT discontinuities during impulse response analysis. A number or repetitions can also be entered, or the sweep be caused to run continuously by entering '0' repetitions.

Bin centres

Settings: amplitude, frequency range, samples, pink response, phases

The 'Bin centres' function is a noise-like stimulus which repeats over a duration of 2^N samples (where N is selectable). In fact, it is a special variant of a multi-tone stimulus, which contains 2^(N-1) linearly-spaced tones at equal amplitudes. So this stimulus has the useful property that a tone will occur in each bin centre if a synchronous FFT of the stimulus is performed.

By passing the Bin centres function through an EUT, and performing a synchronous FFT (i.e. using a rectangular window function and ensuring that the sample rates of the Generator and Analyzer are identical) it is possible to produce a highly accurate, high-resolution, frequency response graph in a very short time, without the need to specify a complicated multi-tone or to average multiple FFTs (as is necessary with a non-bin-centred white noise stimulus).

This function is also useful in impulse response analysis - see the Principles of impulse response analysis and Impulse Response Parameters dialogue box sections for more details.

By default, the Bin centres function contains a tone in every bin from DC to the Nyquist frequency (half the sample rate), but to address situations where it is desired to stimulate the EUT with a smaller frequency range it is possible to specify the lowest and highest frequencies in Hz. Entering '0' for the upper frequency reverts automatically to the Nyquist frequency.

A pink frequency distribution can be selected by checking the 'Pink response' check box. In this case, the non-white frequency response must be taken into account when deriving the frequency response of an EUT with a direct FFT; however, in the case of the impulse response method this is not necessary since the EUT output is correlated with the stimulus.

By default, the phases of all the tones are randomised to give the Bin centres waveform a noise-like quality, even though the amplitude of each tone is not random. This results in a high 'crest factor' for the waveform which limits the amplitude that can be set (since the specified amplitude is the amplitude for each tone). To minimise the crest factor, and so allow higher tone amplitudes without clipping, a 'Newman' phase distribution can be selected. This distribution minimises the crest factor, but causes the waveform to lose its noise-like characteristic even though the relative amplitudes in each tone bin are unchanged.

Note that the Bin centres function has replaced the MLS function in dScope software versions 1.20 and above, since correlation-based measurements in the dScope operate on a 2^N sample repetition period rather than the (2^N)-1 sample repetition period used in the MLS technique.

Twin-tone

Settings: amplitude, frequency, 2nd amplitude, 2nd frequency

The twin tone function comprises two summed tones, and is commonly used for IMD (intermodulation distortion) measurements.

The 2nd amplitude may be set absolutely (in the same units as selected for the main amplitude setting), or as an offset from the main amplitude (an additive offset if the main amplitude is specified in linear units, or dB if the main amplitude is specified in logarithmic units). It may alternatively be set as a ratio of the main amplitude, irrespective of the main amplitude units.

The 2nd frequency may be set absolutely, or as a fixed offset in Hz, or as a ratio of the main frequency.

The 'CCIF' IMD method uses two equal-amplitude high frequency tones (typically 19kHz and 20kHz) with the combined peak amplitude near to the maximum capacity of the EUT. The 'SMPTE/DIN' method uses a low- and a mid-frequency tone (typically 60Hz and 7kHz) with the upper tone 12dB below the lower, and a combined peak amplitude near to the maximum capacity of the EUT.

The dScope's Continuous-Time Analyzer and FFT Analyzer can be configured to measure IMD at the output of the EUT. For more information, see the Continuous-Time Detector dialogue box, FFT Detector dialogue box sections.

User waveform

Settings: amplitude, filename of script/wavetable

The 'User waveform' option accepts files in a number of formats: either wavetables in dScope III, dScope II or WAV file format; or User waveform scripts, as described in the Generator wavetables section of the Scripting Manual. In the latter case, the wavetable is described mathematically in a VBScript. User waveform scripts can be made to generate a wavetable file in dScope III format if required, as well as driving the Signal Generator directly.

Note that if 'User waveform' is selected from the 'Function' drop-menu when 'User waveform' was already selected, the selected 'User waveform' file is reloaded immediately into the dScope's wavetable memory. This is to ease debugging of Generator wavetable scripts. On the other hand, if a 'User waveform' has been selected, and then a non-wavetable Function (e.g. 'Sine') is selected, and then the 'User waveform' is reselected, the dScope's wavetable memory is NOT reloaded, to maximise speed.

⚠ **When a WAV file is specified, the file is loaded into the dScope's generator wavetable, up to a maximum run-length of 512k samples, and the wavetable is continuously repeated. The sample rate specified in the WAV file header is, where possible, adopted by the dScope's analogue and digital outputs. Where this is not possible, the relevant dScope outputs are muted and an error message is displayed. However the table is active, so the output can be enabled (albeit at the wrong sample rate) by unmuting the desired output.**

Multi-tones

Multi-tones are a special type of User waveform, and are specified using the Multi-tone Generation and Analysis dialogue box. See that section for details.

Other functions

A number of additional Signal Generator functions are supplied as VBScripts, which can be used as supplied or modified by the user. These include:

JTest.dss: JTest is a special digital stimulus developed by Prism Sound to expose susceptibility to cable-induced jitter.

Bong.dss: A tone burst with a finite attack and release time, useful for testing dynamics processors.

Signal Generator References panel

The Signal Generator References panel allows a reference amplitude (in a variety of units), a reference frequency and a reference impedance to be specified. The reference amplitude is used when setting the Generator amplitude in dBr or %ref units.

A separate reference amplitude is provided for each of the Signal Generator's channels. These may be tied together or split by checking or unchecking the adjacent 'Tied' check box. The reference amplitudes may be entered manually or may be 'captured' from the current Signal Generator amplitude setting by clicking the 'Get dBr' button (unless the amplitude is currently specified in dBr units). When the Signal Generator is operating in Split Mode, as described above, separate buttons are available for capturing the reference amplitude from either channel.

The Signal Generator may be made to generate different amplitudes for each channel EVEN WHEN OPERATING IN TIED MODE, if the amplitude is specified in dBr and separate reference amplitudes are set for each channel.

The reference frequency is used when setting the Generator frequency as a ratio or an offset. The reference frequency may be entered manually or may be 'captured' from the current Signal Generator frequency setting by clicking the 'Get freq' button. When the Signal Generator is operating in Split Mode, as described above, separate buttons are available for capturing the reference frequency setting from either channel.

The reference impedance is used when setting the Generator amplitude as a power in Watts or in dBm.

Note that the reference amplitudes and frequencies of the Signal Generator and Signal Analyzer can be set independently, or else they can be locked together by setting their respective check boxes in the Options dialogue box in the Utility menu. The reference impedances cannot be locked together.

Signal Generator Steps panel

Amplitude and frequency steps can be set. The Generator amplitude can be instantly changed by the offset or ratio of the step by pressing [CTRL+PAGEUP] to increase or [CTRL+PAGEDN] to decrease the amplitude. The Generator frequency can be changed by the specified offset or ratio (or in fixed octave-fraction steps) by pressing [SHIFT+PAGEUP] to increase or [SHIFT+PAGEDN] to decrease the frequency.

The D/A line-up is an important setting which locks the relationship between the amplitudes of the dScope's Analogue and Digital Outputs. It is useful to be able to express the Generator amplitude in either analogue or digital units, even if the generated signal is being used in the opposite domain; for example, when driving an A/D converter under test which has, say, an 18dBu full-scale input amplitude, the D/A line-up would be set to 0dBFS=+18dBu after which setting a Generator amplitude of –60dBFS would generate –42dBu at the Analogue Outputs, which is 60dB below full-scale of the A/D converter under test.

Note that the D/A line-up of the Signal Generator and Signal Analyzer can be set independently, or else they can be locked together by setting the appropriate check box in the Options dialogue box in the Utility menu. For information about using D/A line-up with Soundcards, see the Soundcard Generation and Analysis section in Amplitude units in dScope.

A loudspeaker sensitivity can be set for use in acoustic testing. This setting allows a sound pressure level to be equated to an electrical amplitude, to calibrate the dScope to an amplifier / loudspeaker combination connected to the Signal Generator output. Once the calibration value has been set, Signal Generator amplitudes can be set directly in dBSPL. Note that this setting can be used to enter the combined sensitivity of a loudspeaker/amplifier combination directly, or else the loudspeaker can be specified here and the amplifier gain entered separately as described below.

A gain can be entered for a post-amplifier (i.e. an amplifier between the Signal Generator output and the input of the EUT, for example a power amplifier). The dScope software takes the post-amplifier gain into account when setting the Signal Generator amplitude, so that the specified amplitude is attained at the OUTPUT of the post-amplifier. This setting is applied to any Signal Generator amplitude setting, whether manual or automatic (e.g. from a Sweep or a VB Script); it is also applied at the Digital Outputs as well as at the Analogue Outputs. The post-amplifier gain defaults to 0dB (unity gain) and in this state has no effect on Signal Generator operation. Post-amplifier gain can be entered in dB or as a simple rational gain. Maximum amplitude limits at the Analogue and Digital Outputs are unaffected.

4.5.2 Output Channel Status dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

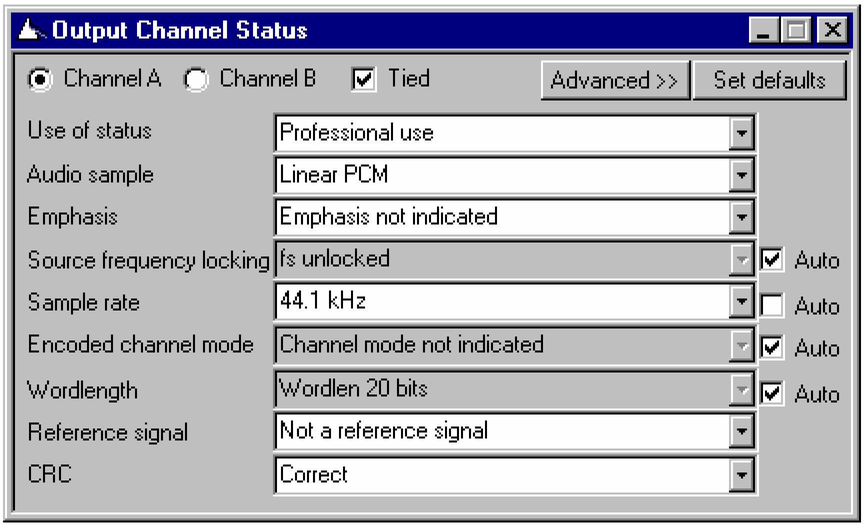
The Output Channel Status dialogue box provides field by field control of the Channel Status which is transmitted at the dScope's Digital Outputs. For a block diagram and description of the relevant area of the dScope hardware, go to Digital Output and Carrier architecture.

There are two alternative versions of the box; the first provides simple control of only the most basic Channel Status fields, whereas the alternative provides complete control of all fields, including reserved fields.

In either mode, the Channel Status data can be 'Tied', in which case it is identical in both transmitted channels, or 'Split' in which case separate Channel Status patterns can be defined for each channel. When switching between Split and Tied modes, the A–channel Status is copied to the B–channel. In either mode, the 'Auto' check boxes associated with certain fields (for example wordlength and sample rate) can be checked to make the transmitted value of the field automatically reflect the prevailing state of the dScope's Digital Output. Default settings for all the fields (including selection of all 'Autos') can be achieved by clicking the [Set defaults] button.

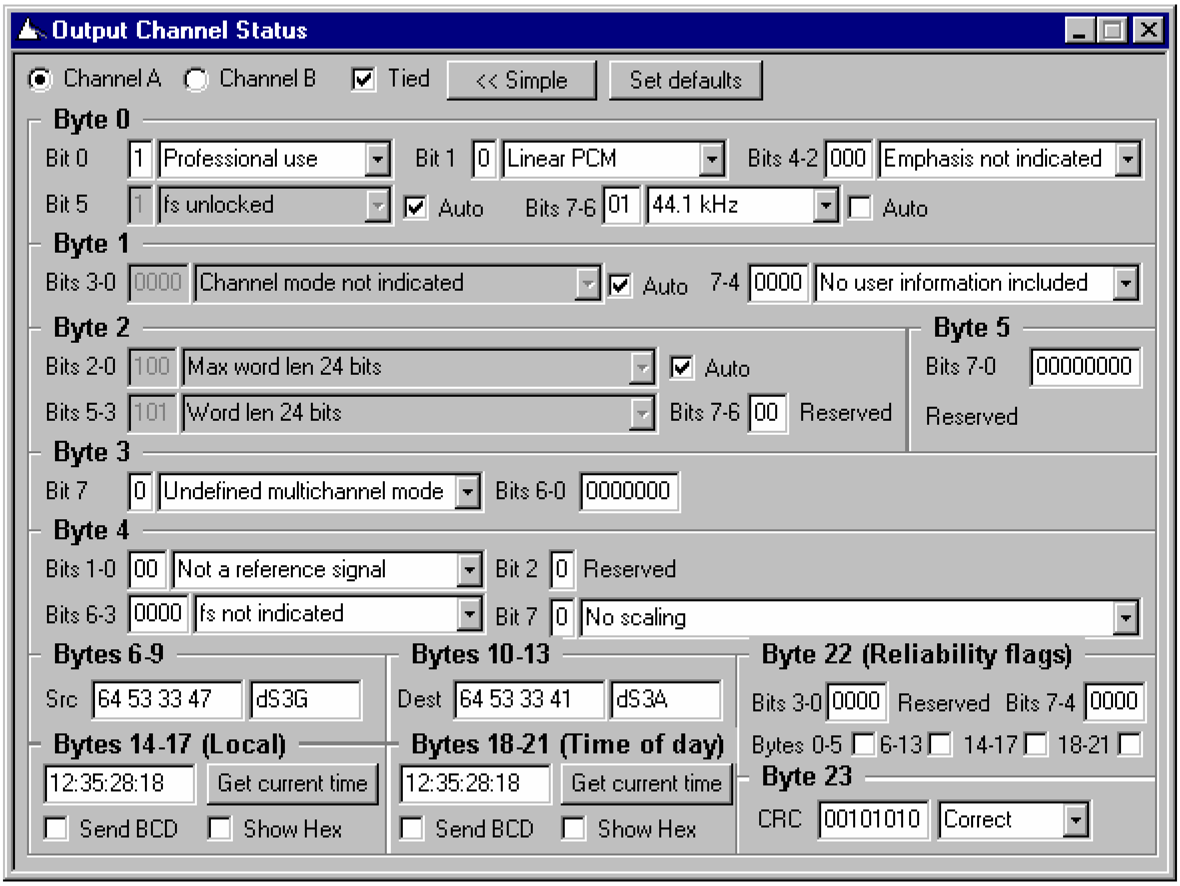
The setting of the first Channel Status bit causes the layout of the remainder of the dialogue box to change, to suit either Professional or Consumer use of the Channel Status bits.

Output Channel Status (simple)



In the simple mode, the most commonly used Channel Status fields can be set 'verbally', without recourse to the binary values of the fields. The remaining fields are not shown and must be set in the advanced mode (although some are forced by the [Set defaults] button).

Output Channel Status (advanced)



In the advanced mode, all of the Channel Status fields may be set (including reserved fields) to any binary value (including reserved values). The settings of each field can be made either verbally, by selecting from a drop-list of allowed settings, or numerically by setting the desired binary data pattern directly.

Note that the timecode fields (sample time and time of day) in Professional mode cannot currently operate dynamically. They can be manually set to static values, or can be copied from the PC's clock by clicking the Get current time button. In addition, two implementations of the timecode fields are supported. By default, the 'sample count past midnight' mode defined in the AES3 standard is used; but if the 'Send BCD' box is checked, a de-facto standard mode is applied where the 32–bit field carries eight BCD digits. In the normal mode, the actual hex count can be displayed instead of the equivalent time-of-day if required by checking the 'Show hex' box.

4.5.3 Output User bits dialogue box

The Output User bits dialogue box will provide control of the User bit stream which is transmitted at the dScope's Digital Outputs. This box is not currently implemented, and User bits are currently transmitted as zeros.

Note, however, that a User bit transparency test is available - please refer to the Digital Outputs dialogue box and Digital Inputs dialogue box sections.

4.6 Analyzer menu

The Analyzer menu provides access to the dialogue boxes which contain controls and Results for the dScope's signal and data analyzers.

Menu options are:

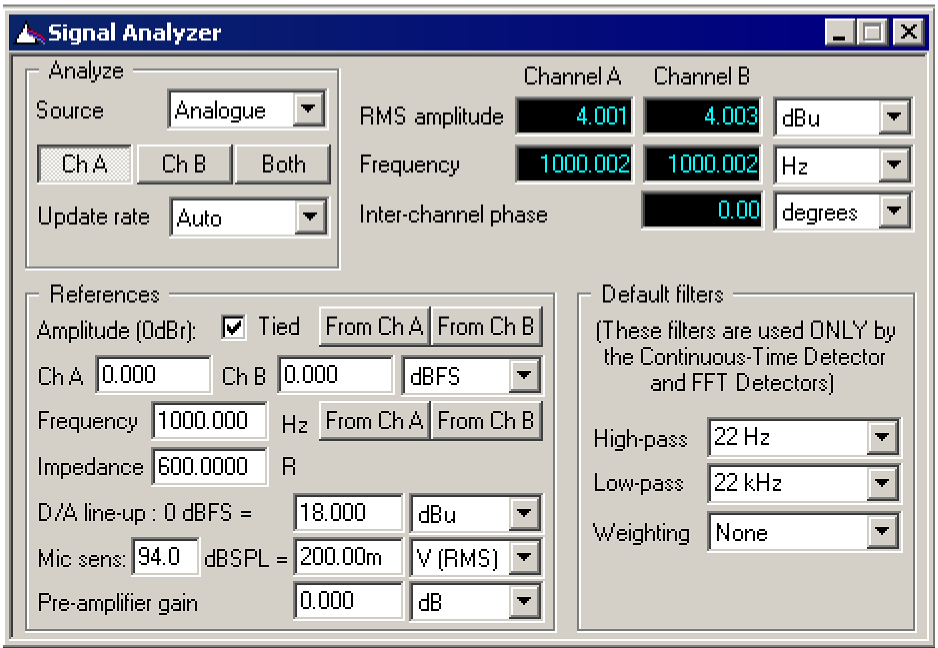
Signal Analyzer...	Settings and Results of the audio Signal Analyzer.
FFT Parameters...	Settings of the FFT Analyzer.
Impulse Response Parameters...	Settings for impulse response analysis.
Channel Status...	Results of the received Channel Status.
User bits...	Results of the received User bits.
Trace window	Displays the Trace window, where all audio data is graphed.
Continuous-Time Detector...	Settings and Results of the CTD.
[List of FFT Detectors]	Settings and Results of any of the existing FFTDs.
New FFT Detector	Creates a new FFTD, and displays Settings and Results.

4.6.1 Signal Analyzer dialogue box

The Signal Analyzer dialogue box provides control and display of the central functions associated with the dScope's Signal Analyzer.

Access to the controls and Results of the specific sub-sections of the Signal Analyzer are contained in the Continuous-Time Detector, FFT Parameters and FFT Detector dialogue boxes.

For a block diagram and description of the relevant area of the dScope hardware, go to Signal Analyzer architecture.



Signal Analyzer Source panel

The Signal Analyzer Source panel determines whether the Signal Analyzer will analyze the dScope's Digital or Analogue Input, or the input from a Windows Soundcard, and whether it will analyze the A–channel, B–channel or both simultaneously. These functions can also be made available as Main Toolbar icons. Note that the soundcard selection is greyed out and unavailable unless a soundcard has been enabled for input using the Soundcard Inputs dialogue box.

NB: **This dialogue box may not be available, depending on the dScope model number.**

The update rate can be set in binary steps between 32 per second and 4 per second, or can be set to 'Auto', which is the recommended setting for most tasks. This setting determines the rate at which Signal Analyzer and Continuous-Time Analyzer Results are collected by Sweeps and scripts. The Results displayed in the dialogue boxes and in Readings are updated at the set rate, unless it is faster than 8 per second, in which case the display rate remains 8 per second to aid readability. The update rate is automatically reduced according to the Signal Analyzer's frequency counter to allow time for measurements on low input frequencies. **This occurs even when the update rate is not set to 'Auto'.**   In the 'Auto' setting, the rate is also reduced in two further circumstances: Firstly, where residual measurements are being collected (BR mode of the CTD) with extended low-frequency response (HPF settings below 10Hz). In this case, changes to the input amplitude or frequency can take a longer time than usual to settle out. Second, when measuring amplitude of signals with frequencies very close to half the sample rate, the Result may fluctuate if the update rate is too high. 'Auto' mode slows the update rate under both these conditions to ensure reliable and stable measurements. The explicit slower rates are not usually necessary, but may help to reduce Result variations for complex input signals with low frequency content, such as noise.

NB: **Note that Signal Analyzer Results for both channels are continuously calculated and displayed even when the Signal Analyzer is set to A–channel or B–channel in the Source panel – that setting only affects the Trace window and the FFT Analyzer.**

Signal Analyzer Detectors panel

The Signal Analyzer Detectors panel contains the Signal Analyzer Results for RMS amplitude and frequency of both channels in the selected domain, along with inter-channel phase (or delay). Units for these Results can also be selected. Note that in common with all dScope Results, these can be 'dragged off' to form Reading windows if required. For more information about the way that amplitude units are handled by the dScope, see the Amplitude units in the dScope section.

Signal Analyzer Default Filters panel

The Signal Analyzer Default Filters panel is used to set default values for high-pass, low-pass and Weighting filters to be used by the Continuous-Time Detector and the FFT Detectors when these are set to use 'default' filters within these categories. Alternatively, the Continuous-Time Detector and FFT Detectors may each have their own individual filters selections set locally if required.

NB: **The default filter settings are not used in calculating the amplitude Results displayed in the Signal Analyzer Detectors panel itself. They are located centrally in the Signal Analyzer panel so that filters in the CT and FFT Detectors can be centrally switched if desired.**

Signal Analyzer References panel

The Signal Analyzer References panel allows a reference amplitude (in a variety of units), a reference frequency and a reference impedance to be specified. A microphone sensitivity setting may also be entered so that acoustic amplitudes may be shown in dBSPL units.

A separate reference amplitude is provided for each channel; these are used when displaying a measured amplitude in dBr or %ref units. The reference amplitudes can be entered manually, or captured from the current A–channel or B–channel amplitude Results by using the nearby buttons. When the adjacent 'Tied' check box is checked, the reference amplitudes of both channels are both updated to the same value if either is changed; when unchecked, they can be set independently.

The reference frequency is used when displaying the measured signal frequency as a ratio or an offset. The reference frequency can be entered manually, or captured from the current A–channel or B–channel frequency Result by using the nearby buttons.

The reference impedance is used when displaying a measured amplitude as a power in Watts or in dBm.

The D/A line-up is an important setting which locks the relationship between the amplitudes of the dScope's Analogue and Digital Inputs. It is useful to be able to express an Analyzer amplitude in either analogue or digital units, even if the signal is actually in the opposite domain; for example, when analyzing a D/A converter under test which has, say, an 18dBu full-scale output amplitude, the D/A line-up would be set to 0dBFS=+18dBu after which a -42dBu output from the converter could be displayed as –60dBFS by selecting the appropriate units. For information about using D/A line-up with Soundcards, see the Soundcard Generation and Analysis section in Amplitude units in dScope.

Note that the reference amplitudes and frequencies of the Signal Generator and Signal Analyzer can be set independently, or else they can be locked together by setting their respective check boxes in the Options dialogue box in the Utility menu. The reference impedances cannot be locked together.

The microphone sensitivity for acoustic measurements is entered by equating an electrical amplitude with a sound pressure level. It is common for measurement microphone sensitivities to be quoted in mV at 94dBSPL. Note that if an external preamplifier is used, the entered sensitivity should be the combined value for microphone and preamplifier, or else the pre-amplifier gain can be entered separately as described below. It is also possible for the dScope to take account of measurement microphone frequency response calibration data, i.e. to correct for non-flat microphone response. This is accomplished by using the FFT Analyzer, and by entering the response calibration data as a user-defined Weighting filter, either centrally in the FFT Parameters dialogue box or in each FFT Detector dialogue box.

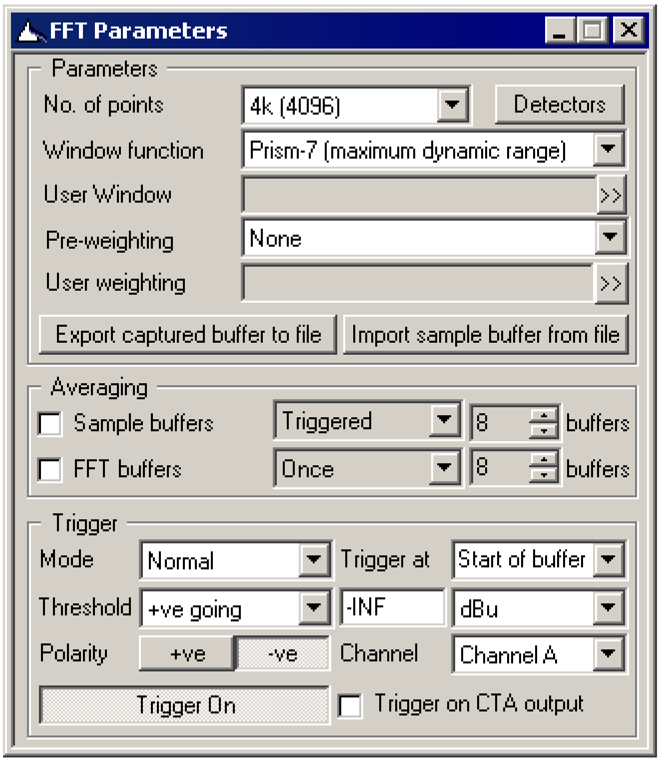
A gain can be entered for a pre-amplifier (i.e. an amplifier between the EUT output and the Signal Analyzer input, for example a microphone pre-amplifier). The dScope software takes the pre-amplifier gain into account when displaying Signal Analyzer amplitude Results, so that the displayed amplitude is the amplitude at the INPUT of the pre-amplifier. This setting is applied to any Signal Analyzer amplitude Results (including Detectors), whether manual or automatic (e.g. from a Sweep or a VB Script); it is also applied to the Digital Inputs as well as at the Analogue Inputs. The pre-amplifier gain defaults to 0dB (unity gain) and in this state has no effect on Signal Analyzer operation. Pre-amplifier gain can be entered in dB or as a simple rational gain. Maximum amplitude limits at the Analogue and Digital Inputs are unaffected.

4.6.2 FFT Parameters dialogue box

The FFT Parameters dialogue box controls the central parameters of the dScope's FFT Analyzer. The FFT Analyzer is a powerful tool which allows the spectra of signals to be viewed with very high resolution, and complex Results to be calculated.

For a block diagram of the structure of the dScope's FFT Analyzer, go to FFT Analyzer architecture. For a detailed discussion of FFT analysis, see the Applications Manual.

Controls and Results specific to each current FFT Detector are accessed through the appropriate FFT Detector dialogue box.



FFT Parameters panel

The FFT Parameters panel defines the number of FFT points, the FFT Window function, selects a pre-weighting filter if required, and allows exporting and importing of sample buffer data.

Number of FFT points

The number of FFT points determines the resolution of the frequency-domain (spectrum) display. The higher the number of FFT points, the finer the frequency resolution but the longer the time taken to acquire the sample buffer and to calculate the FFT. The frequency resolution of the calculated spectrum (the 'bin width' of the FFT) is twice the sample rate divided by the number of points. The number of FFT points is adjustable in binary powers from 1k (1024) to 256k (262144) for analogue and digital input signal analysis and to 1M (1048576) for soundcard analysis. The 'bin width' for a 48kHz sampled signal and a 256k-point FFT is about 0.37Hz.

FFT Window function

A suitable FFT Window function is usually required to enhance the available dynamic range of the FFT. This is because the length of the FFT Analyzer buffer is finite, and the discontinuities at its ends are manifest in the signal spectrum resultant from the FFT as 'skirts' around resolved frequency components which obscure the dynamic range of the FFT. To prevent this, the FFTA buffer is usually modified using a bell-shaped 'window' which emphasises samples near the middle of the buffer at the expense of those at the ends, minimising the 'skirts'. However, this is not without penalty – individual frequency components in the resulting spectrum are artificially broadened by the process; but this is usually preferable.

The 'Rectangular' window is really no window at all, and is generally not usable except in special circumstances, such as when the input signal repeats exactly over the buffer length (i.e. all frequency components fall into bin centres) – for example in 'synchronous multi-tone' testing.

The optimum windows are the Prism 5, 6 and 7 functions which have a very high dynamic range with minimal broadening (Prism7 has the widest dynamic range at nearly 150dB, but is the most broadening of the three).

The 'Prism flat-top' window minimises the 'picket fence' effect, whereby the apparent amplitude of a frequency component in the FFT display depends on how close it is to a 'bin centre'. By using the 'Prism flat-top' window, the apparent amplitude error of a component at a bin edge is less than 0.05dB.

A special kind of window function is available which allows 'windowless' FFTs (i.e. a Rectangular window) in situations where the Generator and Analyzer sample frequencies are not the same, for example in cross-domain or SRC measurements. Selection of these 'frequency correction' window functions causes the Analyzer signal to be sample-rate converted to return frequency components which were generated at bin centres to be restored to bin centres despite the sample frequency difference. Note that the stimulus MUST be generated with all components in bin centres (this is automatic if the Multi-tone Generation and Analysis tool is used), and that both Generator and Analyzer sample frequencies must be at a nominal standard rate (but may be unsynchronised). Two variants of this function are available, one which assumes that the stimulus has been sourced from the dScope's Digital Outputs - 'None (freq correct from Digital)' - and another which assumes the use of the dScope's Analogue Outputs - 'None (freq correct from Analogue)'.

Another variant of the Rectangular window is included to improve impulse response testing with single or low-number shots of the stimulus, where results can be compromised by low frequency drifting of the EUT output - this is especially common when the EUT is DC-blocked with a low corner-frequency and the 'settling' time of the blocking filter is long in comparison to the duration of the stimulus. This compromises the synchronous nature of the FFT (it must be continuous between its end and beginning, since it is notionally repeated over all time) and results in disturbances which look like sporadic periods of ringing on the impulse response. The effect is not noticeable when using a continuous stimulus, because there is no discontinuity when the DC-blocking system has settled. The solution is to select the 'None (n-shot correction)' Window function instead of the normal Rectangular window function. This function is still rectangular in shape, but applies a 'tilt' to the sample buffer in order to remove the discontinuity between its two ends.

The remaining 'standard' Window functions (Triangular, Blackman, Hann, Hamming, Blackman-Harris 4, Gaussian) are inferior to the Prism windows in most applications, since their 'skirts' prevent sufficient dynamic range for measuring modern audio systems. They are included only to provide consistency with other equipment and theoretical papers.

A 'User defined' Window function can be selected if required. The filename specified should be that of a dScope Window function file (.wnd) or a VBScript (.dss) which generates a Window function, as described in the Window function reference section of the Scripting Manual.

Pre-weighting filter

A Pre-weighting filter can be selected if required. This operates in a similar manner to the User-defined Weighting filters which can be selected on a per-FFT-Detector basis in the FFT Detector dialogue box. However, a Pre-weighting filter is applied to the FFT buffer before the FFT Trace is displayed, and affects all FFT Detectors IN ADDITION to their individual weighting filters.

A common use for a Pre-weighting filter is to apply frequency-dependent measurement microphone calibration data for acoustic measurements.

Weighting filter files can be created from dScope graphical Traces (as described in the Trace area drop-menu section of this manual), or using VBScripts (as described in the FFT Detector Weighting filters section of the Scripting Manual). An additional method is also supported, which accepts the filter data from a text file. This method is particularly useful for applying calibration data which exists at spot frequencies. The syntax of the file is described in the following example. Note that "dB" can be replaced by "Gain" if linear gain values are to be used. Unlike the other Weighting filter file formats, filter gains from a text file are interpolated between the specified points.

```
"Microphone: XYZ Company Model 1234, serial # 10777"

"Any number of comments allowed here, as long as "
"they're in double quotes!"
```

```
"Hz" "dB"
1000.00, 0.00
2000.00, 0.1
3000.00, 0.2
5000.00, 0.25
8000.00, 0.33
10000.0, 0.5
15000.0, 1.75
20000.0, 2.5
25000.0, 2.50
30000.0, 1.5
40000.0, -3.0
```

Importing and exporting sample buffers

The buttons at the bottom of the FFT Parameters panel allow the FFT Analyzer's sample buffer to be exported or imported in WAV format.

When exporting the current buffer, the channel to be saved must be selected if the FFTA is in two-channel mode. The sample buffer itself or the CTA Output buffer must be selected for saving if a CTA Output Trace is present on the Trace window.

When importing a WAV file, the channel to be loaded must be selected if the FFTA is in two-channel mode. The sample buffer itself or the CTA Output buffer must be selected for loading if a CTA Output Trace is present on the Trace window. In case the WAV file to be imported is not of length 2^N samples, it is possible to specify whether to trim the data down to the next lower 2^N length, or whether to pad it with zeroes up to the next higher 2^N length, or whether to adjust to the nearest 2^N length. If the buffer length is longer than the file length, it is possible to choose whether to pad the buffer with zero samples beyond the end of the file data, or whether to repeat the file data up to the length of the buffer.

FFT Averaging panel

The dScope can average FFT Analyzer data in both the time domain (sample buffers) and the frequency domain (FFT buffers). Each techniques has its own benefits, as described below. The dScope can perform time-domain and frequency-domain averaging simultaneously if required, which combines the benefits of both.

Time-domain (sample buffer) averaging

This technique minimises the effects of uncorrelated random noise in measurements (or background room noise in acoustic measurements) and effectively improves the signal-to-noise-ratio of the measurements.

As well as selecting the number of averages, the user may choose between 'Contiguous' and 'Triggered' modes.

In triggered mode, every time the FFT Analyzer triggers, the newly acquired data is combined with the previously captured buffers to produce an averaged buffer, until the desired number of buffers have been averaged, at which point the trigger is turned off. The FFT is displayed after each averaging pass, so the gradual improvement of signal-to-noise-ratio can be observed. The trigger must be re-armed to start another averaging series. In order for this mode to work reliably, the trigger point must be consistently positioned within the repeating test signal (e.g. sine stimulus) otherwise the signal itself will be uncorrelated between buffers and will be averaged down.

In contiguous mode, the first time that triggering occurs after averaging is turned on, the desired number of buffers are sampled immediately and contiguously (i.e. with no gaps in between), and averaged. This removes the need for successive reliable triggering, but requires that the test stimulus repeats exactly over the period of one buffer. This is the case in synchronous multi-tone analysis, and in impulse response analysis.

Frequency-domain (FFT buffer) averaging

In order to resolve frequency components hidden in the noise of an FFT display, the dScope can average a number of FFTs; this has the effect of reducing variations in the displayed FFT noise floor and so emphasizes real low-level components and artifacts. Despite revealing sub-noise-floor FFT components, this method does not in itself improve the signal-to-noise-ratio of measurements.

As well as selecting the number of averages, the user may choose between 'Once' and 'Pseudo-rolling' modes.

When 'once' mode is enabled, the dScope averages the requested number of successively-triggered FFTs and then disarms the trigger. The FFT is displayed after each averaging pass, so the gradual smoothing of the noise floor can be observed. The trigger must be re-armed to start another averaging series.

In 'pseudo-rolling' mode, the trigger is never disabled, but an effect similar to a rolling FFT average is displayed. A true rolling average is not possible, since the individual FFT buffers are not retained. However, the effect is achieved by positively weighting the effect of more recent buffers in the average.

FFT Trigger panel

The FFT Analyzer trigger is normally used to trigger the acquisition of the FFT Analyzer buffer. It generally works by applying a threshold to the incoming audio, rather like an oscilloscope trigger, but can also be linked to the Signal Generator.

Mode

The mode control determines whether trigger operation is *continuous* (buffer acquisitions follow each other with minimal delay, without waiting for the trigger condition to be met), *normal* (a buffer acquisition is made each time the trigger condition occurs), *single-shot* (dScope waits for the next occurrence of the trigger event, acquires a buffer, then disarms the trigger) or *Generator wavetable* (as the Generator wavetable wraps through its first sample).

The Generator wavetable mode is particularly used in impulse response analysis.

Trigger at

The trigger point can be set to occur at any of the quartiles of the buffer, to record events before or after the trigger, or both. It is also possible to position the trigger point at a precise position within the buffer by typing the sample offset of the required trigger point into the 'Trigger at' box.

Threshold and polarity

The trigger threshold can be entered in a variety of units, and the trigger event can be set to occur when the threshold is breached in either a positive-going or a negative-going direction, or when an incoming sample exactly equals the threshold, or when incoming samples cease being equal to the threshold.

The 'Polarity' controls are used to specify a positive or negative threshold.

Trigger source

The trigger 'Channel' setting allows triggering from either the A Channel or the B Channel input.

The 'Trigger on CTA output' check-box allows the trigger condition to be evaluated on the output of the Continuous-Time Analyzer (on the THD+N residual, for example) if required.

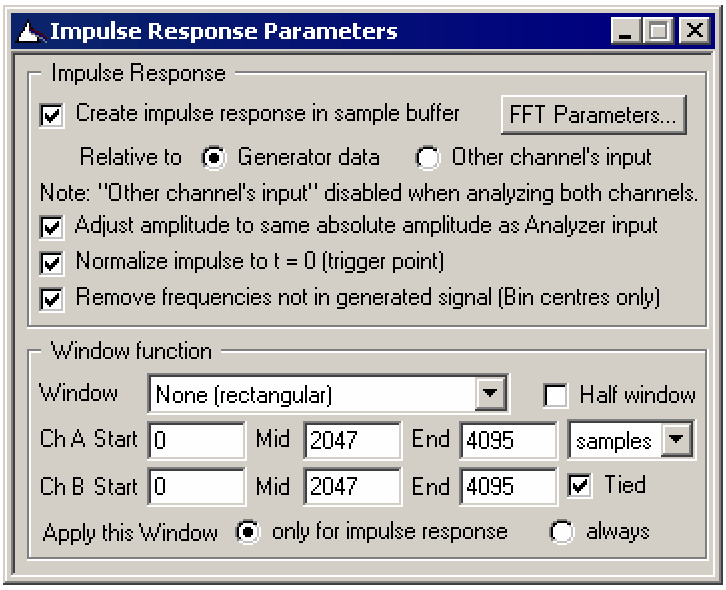
Arming the trigger

The [Trigger On] button arms and disarms the trigger; these functions can be duplicated on the Main Toolbar using the and icons respectively.

4.6.3 Impulse Response Parameters dialogue box

The Impulse Response Parameters dialogue box controls the parameters of the dScope's FFT impulse response analyzer. This is a powerful tool which can be used to measure acoustic spaces or transducers, as well providing an alternative method of analyzing the response of electronic EUTs.

For further explanation of impulse response analysis and the detailed operation of these parameters, please see the Principles of impulse response analysis section, with particular reference to the block diagram.



Impulse Response panel

This panel contains the basic controls to set up impulse response operation. Note that some parameters in the FFT Parameters dialogue box are also likely to require adjustment for impulse response measurement (e.g. the FFT Window function and trigger mode) and so a button is provided on this panel to access the FFT Parameters dialogue box.

Checking 'Create impulse response in sample buffer' changes the source of data for the Trace window live Scope and FFT Traces. In normal operation, both the Scope and FFT Traces display to the Analyzer input data (sample buffer), whereas with this box checked, the Scope Trace shows the calculated impulse response (prior to impulse response windowing - see below) and the FFT Trace shows the resultant frequency response (i.e. the FFT of the windowed impulse).

A pair of radio buttons switches the correlation reference data source between the Generator data and the Other channel's Analyzer input data, depending on the particular test setup in use.

Checking the 'Adjust amplitude to same absolute amplitude as Analyzer input' check box causes the impulse response Trace to be displayed with absolute units. This can sometimes be intuitive even though the impulse response is, of course, strictly relative. When displayed in absolute units, the impulse response is scaled so that its apparent amplitude is the same as the amplitude of the test stimulus detected at the Signal Analyzer input. When this box is unchecked, the impulse response Trace is shown with relative units.

Checking the 'Normalize impulse to t = 0 (trigger point)' check box causes the impulse response to be forced to the selected trigger-point time on the Trace window, rather than being at its actual time after triggering. It is sometimes useful in situations where triggering is unreliable to force the temporal position of the impulse, to stop it moving around (particularly since this would cause it to move with respect to the impulse Window function); however, particularly when testing acoustic spaces, it may be useful to display the impulse delayed by the actual delay time with respect to the reference data source, since that delay may need to be measured.

The 'Remove frequencies not in generated signal' check box is provided to clarify the impulse response display when using a 'Bin centres' stimulus which does not cover the whole frequency range from DC to the Nyquist frequency (half the sample rate). In this situation, the impulse response contains only noise at those frequencies, and the resultant FFT also. By checking the box, noise in these frequency ranges is removed. Note that when using 'Bin centres' stimuli which cover the entire frequency range, or when using a swept sine stimulus, no noise removal is necessary (although some noise is visible in the swept sine case). See the Signal Generator dialogue box section for more information about these stimuli.

Window function panel

This panel allows selection and time-adjustment of the impulse response Window function. Note that this Window function is not the same as the FFT Windows function accessed in the FFT Parameters dialogue box.

The time-adjustment process can also be achieved visually by dragging the Window function on the Trace window, which is generally much easier. To do this, either click the icon on the Trace Toolbar, or right-click on the appropriate Scope Trace in the Quick legend, then select 'Edit impulse response Window function' from the drop menu. The impulse response Window function is then displayed on the Trace window, superimposed on the impulse response, and can be dragged to the required position and width-adjusted using the handles provided. When editing the impulse response Window function visually, the following dialogue box appears, which contains all the relevant settings from the Window function panel:

The 'Apply this Window' radio buttons at the bottom of the Window function panel allow the manually-adjustable Window function to be applied to the FFT process even when not in impulse response mode. This is occasionally useful in generating an FFT of captured data which contains extraneous noise or interference. This feature allows the FFT to contain only such data as is required.
Be careful not to accidentally leave this feature selected.

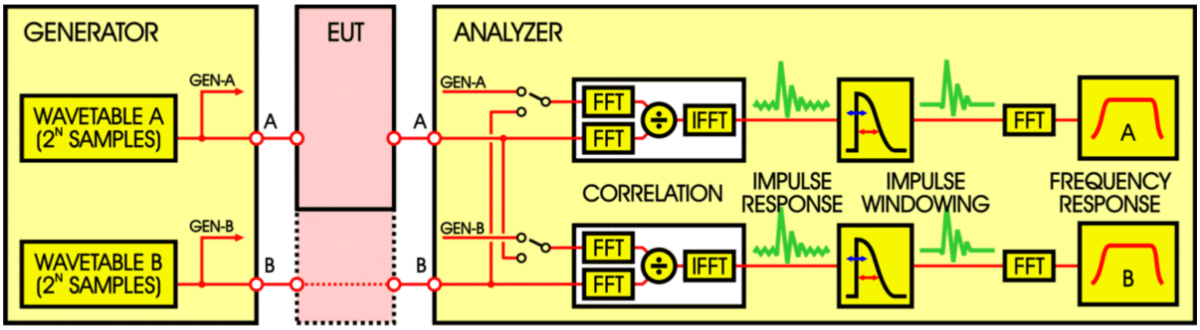
Note that the FFT of the EUT's frequency response (which is calculated from the impulse response whenever 'Create impulse response in sample buffer' is checked) is limited in resolution at low-frequencies by the duration of the time window applied. The FFT Trace is shown dotted below the limiting frequency.

4.6.3.1 Principles of impulse response analysis

Impulse response analysis is a powerful tool which can be used to measure acoustic spaces or transducers, as well providing an alternative method of analyzing the response of electronic EUTs.

Principle of operation

The principle of operation is shown in the block diagram below:



Stimulus

A wavetable stimulus is used which has a run length of 2^N samples. This is necessary because the analysis part of the process is FFT-based, and must run synchronously with the Generator, i.e. their sample rates must be identical. As usual with FFT analysis, small values of N (short buffers) produce fast measurements, but larger values of N (longer buffers) provide greater resolution.

A variety of stimuli can be contained in the wavetable; the main requirement is that the aggregated spectrum of the stimulus covers the band of interest (i.e. the audio band) reasonably evenly. For this reason, continuously-swept sine waves (chirps) or noise-like stimuli are usually favoured. Using dScope, the Signal Generator should be set to Swept Sine (logarithmic) or a Bin Centres stimulus (a noise-like signal which is actually a finely-spaced multi-tone). The two channels generally contain identical stimuli.

Analysis

The stimulus is passed through the EUT and into the Analyzer. The first stage of analysis involves correlating the signal recovered from the EUT with a reference. dScope can use either the Generator wavetable or the Analyzer input from the other channel as the correlation reference, which allows useful flexibility. By using the Generator wavetable directly, it is possible to analyzer two EUT channels simultaneously. On the other hand, by using the other Analyzer channel as the reference, it is possible to compare two points in a signal chain; also (if the measurement is analogue) the response of the dScope's own converters is cancelled from the result; however, only one EUT channel can be measured at a time.

Correlating the signal recovered from the EUT with the selected reference produces an 'impulse response' of the EUT - i.e. its theoretical output if stimulated by a perfect impulse. In dScope, the correlation is performed using an FFT-based process. It is generally more practical to derive the EUT's impulse response by this method than by stimulating it with an actual impulse, which contains very little energy and so would produce a result with poor signal-to-noise ratio.

In assessing acoustic spaces, the impulse response can be interesting in itself, whereas for measuring transducers or electronic EUTs it is mainly a means to deriving the device's frequency response. The frequency response of the EUT can be calculated as the FFT of its impulse response.

Impulse windowing

A common requirement in transducer measurement is to be able to measure devices in anechoic conditions, and free from background noise (which might compromise the reliability of the results). To do this in real anechoic chambers is expensive and inconvenient in a production environment, particularly for low-cost devices. Impulse response windowing can help to solve this problem. This involves applying a window to the impulse response prior to performing the final FFT. Basically, the windowing operation allows only the desired period of the impulse response to be included, i.e. later parts which contain reflections of the stimulus can be excluded, as well as the intervals before and after the impulse response which may contain background noise. By carefully time-positioning a suitable window function over the impulse response prior to calculating the frequency response, it is possible to simulate anechoic measurement conditions.

dScope allows any of the normal FFT Window functions to be selected for impulse windowing, in either a full or 'half-window' configuration. A half-window has only the 'right-hand' side of the specified function; i.e. it has unity gain at its start, and reducing gain thereafter. The selected window or half-window can be precisely positioned at the right time, and scaled to the desired duration, to provide optimum selectivity.

Triggering

Since the Generator and Analyzer are synchronous and their buffers of equal length, there is no fundamental reason why the Generator cannot repeat continuously, with the Analyzer capturing data buffers at arbitrary instants. The derived frequency response would be identical irrespective of when the Analyzer acquired its buffer.

However, in practice, it is important that the Analyzer can be reliably triggered at the appropriate time. One reason is that it may not be desirable or possible to cycle the Generator continuously if, for example, a high-power loudspeaker is being tested near its limit, or an automated production line needs to test each EUT in a single cycle. Reliable triggering is also required if the impulse response needs to be reliably located in time (e.g. to assess the delay through the EUT, or a room response) - although the dScope can automatically align the impulse response at t=0 if required.

It is difficult to obtain reliable triggering at the beginning of a Swept Sine stimulus (and impossible for a noise-like signal) using the normal threshold trigger. Some test systems solve this problem by generating a pulse on the second channel as the stimulus begins, and using this to trigger the Analyzer. However, this would prevent two-channel operation. The problem is solved in dScope by providing a 'Generator wavetable' trigger mode, wherein the Analyzer can be triggered each time the Generator wavetable begins or wraps.

Contiguous time-domain averaging

Another technique for minimising the impact of background noise in acoustic measurements is to average the data from a number of acquisitions prior to analysis. Thus any 'uncorrelated' content, i.e. content which does not occur in every acquisition, is averaged down compared to the stimulus, and so the signal-to-noise ratio is improved. However, the acquisitions must be accurately timed so that the phase of the stimulus is identical in each one.

Since the stimuli in use here repeat over 2^N samples, and are acquired in a 2^N sample Analyzer buffer, it is simple for the dScope to perform 'contiguous' averaging in the time-domain by sampling multiple buffers contiguously (i.e. with no gaps) from a single trigger event and averaging these to produce a buffer of data for analysis.

Setting up the dScope for impulse response analysis

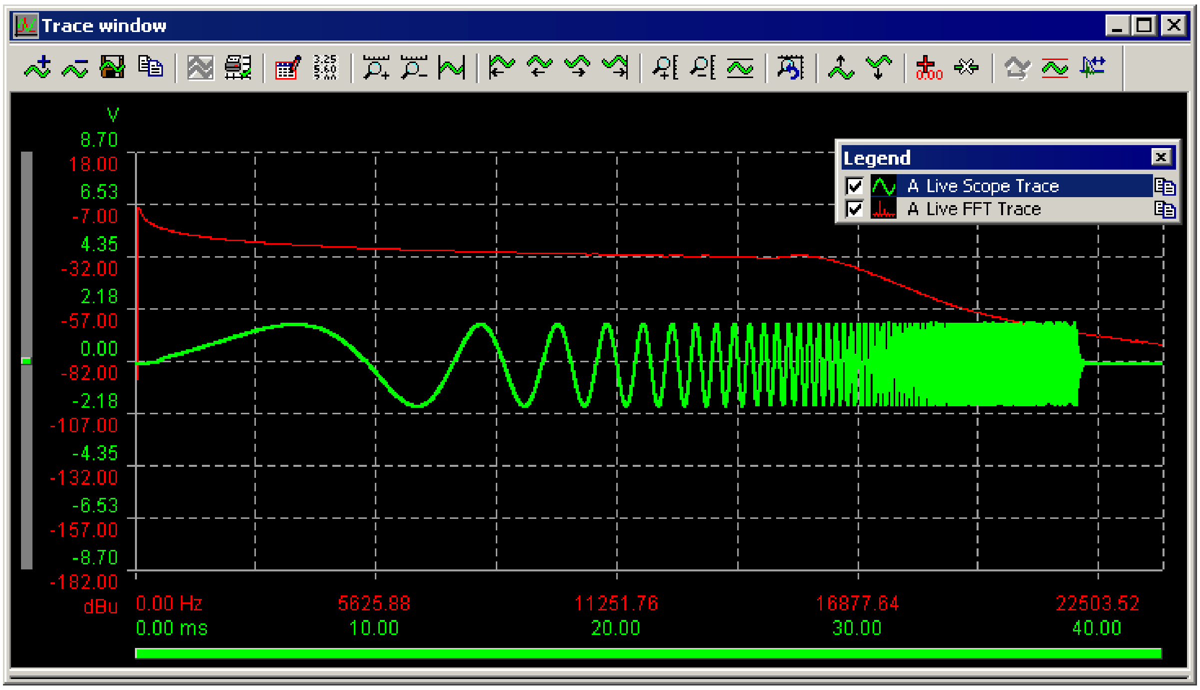
Setting up impulse response analysis for a particular EUT is initially a process of successive adjustment. This section contains various tips about how to do it.

In the first case, we will set the dScope up to make an impulse response measurement 'back to back'. To do this, load the default configuration and then connect the Analogue Outputs of the dScope to its Analogue Inputs, either using cables or by selecting 'Generator' as the 'Source' setting in the Analogue Inputs dialogue box.

In the Signal Generator dialogue box, set the 'Function' to 'Swept sine', with the default parameters (20Hz to 20kHz, log, 4k points with 250 sample space, 100 sample ramp up and down, playing continuously).

In the FFT Parameters dialogue box, make sure that the 'Number of points' is set to 4k, set the 'Window function' to 'None (Rectangular)' and 'Trigger', 'Mode' to 'Gen wavetable'.

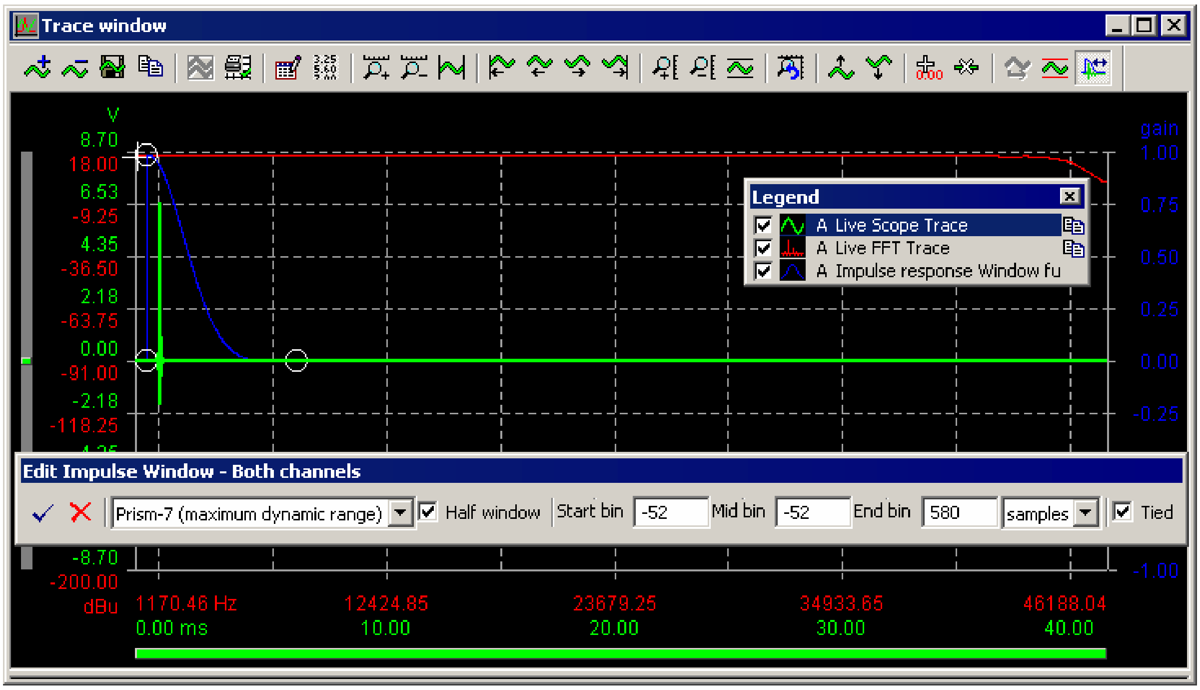
Look at the Trace window; make sure that the Scope and FFT Traces for channel A are displayed, and that the Scope Trace is fully zoomed out on X - if not, repeatedly click the Trace Toolbar icon with the Scope Trace selected. The Trace window should look something like this:



The Scope trace shows the Swept sine waveform, triggering reliably at the same point; the FFT Trace shows the FFT of the stimulus, which steadily falls in-band owing to its logarithmic progression, and then rolls off out-of-band.

Now open the Impulse Response Parameters dialogue box from the Analyzer menu and check 'Create impulse response in sample buffer' . The Scope Trace should be replaced by an impulse at the left-hand side of the Trace window. To make this a bit clearer, go back into the FFT Parameters dialogue box, click in the 'Trigger at' box, enter '100' and hit [Enter]. The impulse should move to the right a little (to sample 100, in fact).

The FFT won't yet be showing the frequency response of our wire (hopefully flat) because the impulse response is probably not yet being windowed properly. Click the icon on the Trace Toolbar, and an Impulse response Window function Trace should appear, along with the Edit Impulse Window dialogue box. You can drag the Window Trace with the handles on its left, and adjust its duration with the handle on its right. When you position it to include the impulse on the Scope Trace, the FFT Trace should begin to show a flat frequency response, like this:



When the window is satisfactorily positioned, close the Edit Impulse Window dialogue box using the tick button.

This procedure shows the basic impulse response analysis process. It should now be a straightforward matter to insert a real EUT and to measure its response.

Single-shot and n-shot impulse response analysis

It is sometimes desirable to make impulse response measurements without repeating the stimulus continuously. This is particularly the case in acoustic space and transducer testing, where a continuous stimulus night be annoying or destructive.

To make the measurement with single shots of the stimulus, set the 'Play' count in the Signal Generator dialogue box to 1 (enter 0 if you need to set it back to 'continuous') and trigger the stimulus with the [Play now] button, or add the  icon to the Main Toolbar. This 4k chirp is quite short; for acoustic measurements, it may be necessary to make the stimulus longer, and to increase the 'Space' parameter of the Swept sine to leave long enough for reverberation to die down between repetitions, especially if you are measuring in a very reverberant space.

Contiguous averaging is enabled by checking the 'Sample buffers' check box in the 'Averaging' panel of the FFT Parameters dialogue box, setting the averaging mode to 'Contiguous' and the 'Buffers' setting to the number of averages required. Note that if the Generator is not operating continuously you will have to set the number of repetitions in the Signal Generator dialogue box to at least the number of averages selected.

Sometimes, when impulse response testing with single or low-number shots of the stimulus, results are compromised by low frequency drifting of the EUT output - this is especially common when the EUT is DC-blocked with a low corner-frequency and the 'settling' time of the blocking filter is long in comparison to the duration of the stimulus. This compromises the synchronous nature of the FFT (it must be continuous between its end and beginning, since it is notionally repeated over all time) and results in disturbances which look like glitches or sporadic periods of ringing on the impulse response. The effect is not noticeable when using a continuous stimulus, because there is no discontinuity when the DC-blocking system has settled. The solution is to select the 'None (n-shot correction)' Window function instead of the normal rectangular Window function. This function is still rectangular in shape, but applies a 'tilt' to the sample buffer in order to remove the discontinuity between its two ends.

4.6.4 Input Channel Status dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The Input Channel Status dialogue box provides field by field indication of the Channel Status received at either channel of the dScope's selected Digital Input.

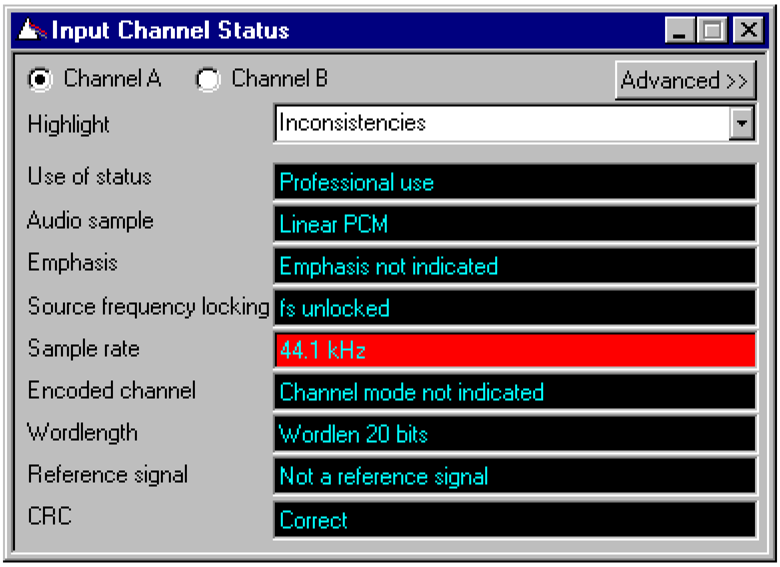
For a block diagram and description of the relevant area of the dScope hardware, go to Digital Input and Carrier architecture.

There are two alternative versions of the box; the first provides simple display of only the most basic Channel Status fields, whereas the alternative displays all fields, including the reserved fields.

In either mode, various 'Highlight' modes are available wherein the requested fields are highlighted in red. Highlighting can be set to show fields which are different from the Output Channel Status, different from the other input channel, reserved (but not zero) or inconsistent. Examples of inconsistency might include Sample Rate or Wordlength fields indicating a different value than detected, or mutually incompatible states of related pairs of fields.

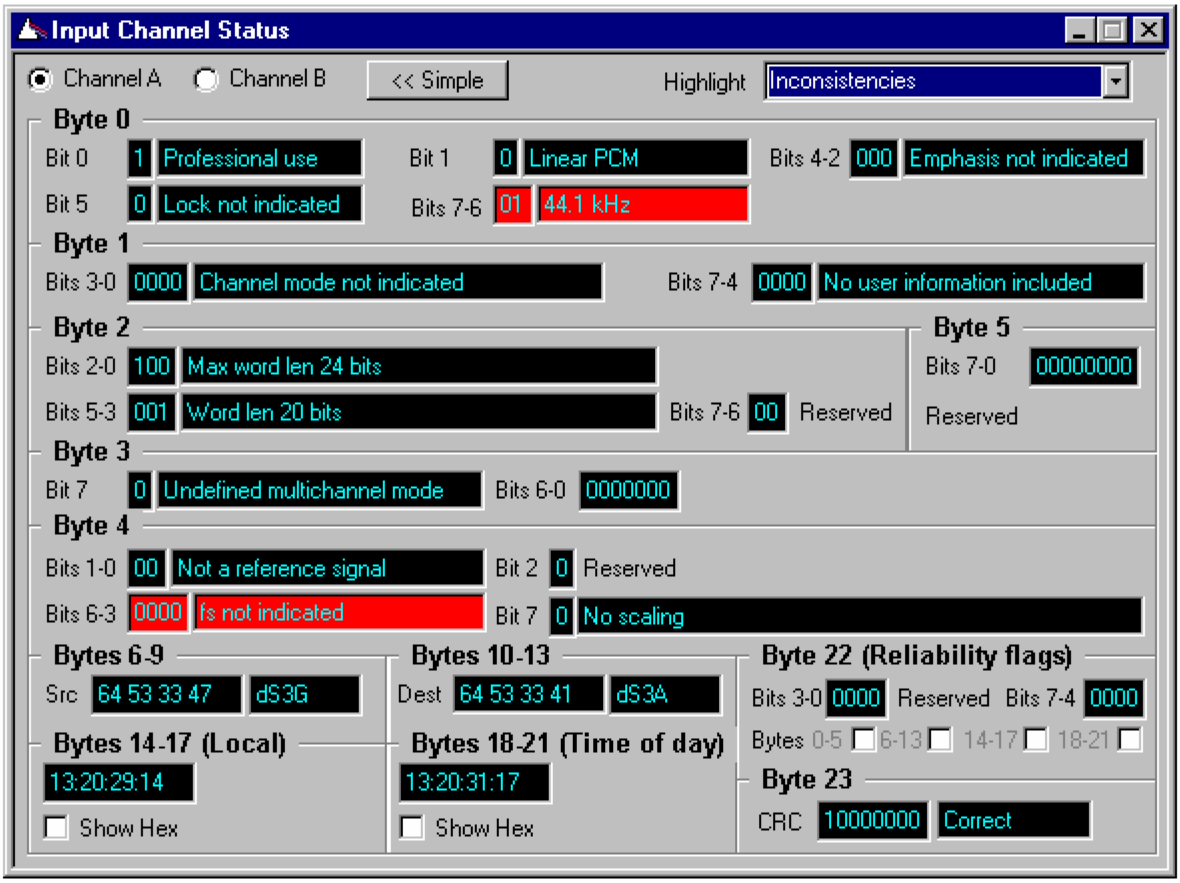
The setting of the first Channel Status bit causes the layout of the remainder of the dialogue box to change to suit either Professional or Consumer use of the Channel Status bits.

Input Channel Status (simple)



In the simple mode, the most commonly used Channel Status fields are displayed 'verbally', without recourse to the binary values of the fields. The remaining fields are not shown.

Input Channel Status (advanced)



In the advanced mode, all of the Channel Status fields are displayed (including reserved fields). The values of each field are shown both verbally and in binary format.

In normal operation, the timecode fields (sample time and time of day) in Professional mode are automatically converted from the incoming 'sample count past midnight' values (as defined in the AES3 standard) into conventional time displays. But by checking the 'Show Hex' check box, the incoming hex data is shown directly. This latter mode is also applicable when using the de-facto standard 'BCD' mode of the timecode fields (wherein eight BCD digits are transmitted instead of a sample count) since the time is thus directly displayed.

4.6.5 Input User bits dialogue box

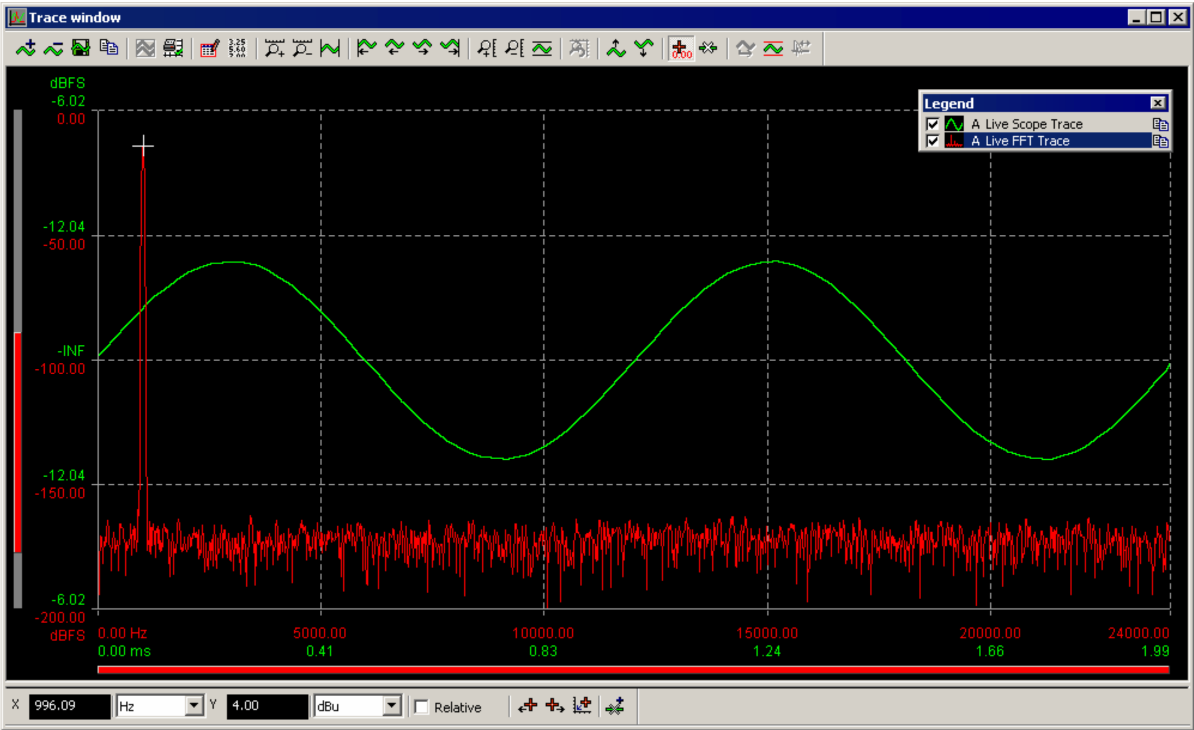
The Input User bits dialogue box will display the User bit stream received at the dScope's selected Digital Input. This box is not currently implemented, and incoming User bits are not currently displayed.

Note, however, that a User bit transparency test is available - please refer to the Digital Outputs dialogue box and Digital Inputs dialogue box sections.

4.6.6 Trace window

The Trace window displays all the main graphical results, including Scope, FFT and Sweep Traces, residuals from the Continuous-Time Analyzer, Limit Lines and filter responses, as well as containing the necessary controls and legends to operate on them.

Note that since Scope and FFT Traces are produced by the FFT Analyzer, the FFT trigger must be ON in order for the Traces to work. This is most easily achieved by clicking the icon on the Trace Toolbar.



The Trace window comprises several parts, as follows:

Trace Toolbar	Toolbar, for quick control of Trace-related functions. *Dockable Toolbar, docks at top of Trace window.*
Trace area	Graphical area for display of Traces, Scales Scale-bars etc. *May be single-channel, or two-channel.*
Quick legend	Concise list of all currently-enabled Traces. *Dockable window, docks at left or right of Trace area.*
Cursor Toolbar	Toolbar for Cursor-specific functions. *Dockable Toolbar, docks at bottom of Trace window, above Mark Toolbar. Only available when Cursor enabled on current Trace.*
Mark Toolbar	Toolbar for Mark-specific functions. *Dockable Toolbar, docks at bottom of Trace window, below Cursor Toolbar. Only available when Marks enabled on current Trace.*

Trace Toolbar

The Trace Toolbar contains a range of icons to provide quick access to many commonly used functions within the Trace window. Like the Main Toolbar, the Trace Toolbar is dockable; unlike the Main Toolbar the selection of icons on the Trace Toolbar cannot be customized. See the Trace window icons section in the Icons and Hotkeys reference for details of the icon functions. See the User-interface basics section in Operation Overview for more information about dockable Toolbars.

Trace area

The Trace area, where the graphical Traces are displayed, occupies the main part of the Trace window.

The sections below describe the many functions of the Trace area, and how to operate them:

Trace types

dScope supports a variety of Trace types:

Live Traces:
Live Traces are derived from the actual audio input to the Signal Analyzer, whereas the other Trace types detailed below are not, even though they might be copies of 'previously-live' Traces. There can only be one Live Trace of each type per channel at any one time (except for Sweep Traces).

Scope	Amplitude-vs-time Trace, similar to conventional oscilloscope.
FFT	Amplitude-vs-frequency Trace, with log Y scale, for spectral analysis.
Sweep	Trace formed by capturing a number of sequential Results, usually interspersed with progressive variations of a source parameter; e.g. Analyzer amplitude vs. Generator frequency is a 'frequency response' Sweep.
Scope of CTA output	Scope of Continuous-Time Analyzer output, e.g. residual distortion using CTA on THD+N.
FFT of CTA output	FFT of Continuous-Time Analyzer output, e.g. FFT of residual distortion.

Limit Lines:
Limit Lines are the comparison envelopes used for range-checking Live Traces. A Trace can be associated with an Upper Limit Line, or a Lower Limit Line, or both, as described below.

Copy Traces:
Instant 'scratchpad-copies' of Live Traces, made using the icon or with the Trace area drop-menu. These are permanently discarded when removed.

Saved Traces:
Disk-file copies of Live and Sweep Traces, made using the icon or with the Trace area drop-menu.

Filter Traces:
Filter Traces show the net filter action of the Continuous-Time Detector or any FFT Detectors which are currently active. The Filter Trace shows the combined effect of any high-pass, BP/BR, low-pass and Weighting filters which are active in the selected Detector.

Window Function Traces:
Show the shape of internal or user-defined FFT Window functions.

Current Trace

By clicking on a Trace's entry in the Quick legend, or on the Trace itself in the Trace area, that Trace becomes 'current', and is highlit in the Quick legend display. The current Trace may also be drawn as a bold line if the appropriate option is enabled in the 'Trace window' tab of the Options dialogue box.

A great many of the Trace window controls operate on the current Trace (for example all the Trace manipulation icons on the Trace Toolbar, Cursor and Mark operations, Trace area drop-menu operations etc.). For this reason, it is important to be familiar with the Trace selection operation.

Two-channel operation

By using the , and icons on the Main Toolbar (or the equivalent buttons on the Signal Analyzer dialogue box) the Trace area can be toggled between A–channel, B–channel and two-channel operation (many other functions within the Signal Analyzer follow this selection).

In two-channel mode, the Trace area can display the two channels either on two different axes, one above the other, or on the same axes. These alternative modes are toggled using the icon on the

Trace window.

Scales, Scale-bars, Zooming and Scrolling of Traces

Each Trace displayed in the Trace area has its own numerical X and Y Scales drawn in the same colour as the Trace itself. The spacing of the dashed 'graticule' lines follows the settings for the current Trace, which are set up in the Trace Settings dialogue box . The current Trace also has both X and Y 'Scale-bars' associated with it, which are drawn to the left of the Y Scales and below the X Scales. NB: Where a Trace shares scales with other Traces, the scale colours match the first Trace in the list.

The Scale-bars provide a quick indication of the zoom and scroll state of the current Trace: for example the overall length of the X Scale-bar represents the total length of the captured buffer of samples, and the coloured section represents the portion currently on display; thus changing the X-axis zoom and scroll positions change the length and position of the coloured part of the Scale-bar respectively.

The current Trace can be zoomed and scrolled using a variety of control options (follow the links for details):

- Clicking the Trace Toolbar icons
- Using keyboard 'Hotkeys'
- Entering axis limits directly in the Trace Settings dialogue box
- Selecting Trace area drop-menu options
- Dragging a box around the desired zoom area

The 'Auto Zoom' function is useful for automatically zooming the scales of the current Trace: dScope takes into account the amplitude (and frequency for Scope Traces) of the incoming signal.

To undo a change of the current Trace's scale, e.g. after an accidental zoom, use the ('Revert to last scale values') icon in the Trace Toolbar.

To quickly regain the default scale settings for the current Trace, click the [Reset Defaults] button in the Trace Settings dialogue box. To regain the default scale settings for ALL Traces, use the 'Set all Traces to defaults' option in the Trace area drop-menu.

Limit Lines

Limit Lines can be associated with any of the Live Trace types in order that they can be continually range-checked if desired. Either an 'Upper Limit Line' or a 'Lower Limit Line' or both can be applied to each live Trace.

Limit Lines can be created in three different ways: by manually drawing the line onto the Trace area, by copying from a Trace (and then usually shifting the resulting Limit Line to provide an operating margin), or by scripting.

To create a Limit Line to apply to the current Trace, either by drawing or Trace-copying, click the icon on the Trace Toolbar (or use the 'Create/Edit Limit Line' option on the Trace area drop-menu), and choose the preferred entry method and whether the line will be an Upper or a Lower Limit Line.

If the Limit Line has been made by copying, it appears straight away superimposed on the current Trace. If the Trace was an FFT or Scope Trace, a 'beep' will probably be heard (the default limit-violation action) if the FFT trigger is still on, as subsequent acquisitions breach the Limit Line. Scroll the Limit Line up (if it is an Upper Limit Line) or down (if it is a Lower Limit Line) using the keyboard arrow keys or Trace Toolbar icons. Reselect the desired Live Trace before adding the second Limit Line if required.

HINT: When creating a Limit Line by copying an FFT Trace, it is often useful to average the FFT Trace over a number of acquisitions as described in the FFT Parameters dialogue box section. This 'irons out' deep excursions in the noise floor of the FFT Trace and thus allows tighter limits to be set.

If the Limit Line has been selected to be made by manual drawing, the mouse cursor is replaced by a drawing cursor (an arrow with a yellow zigzag next to it). Position the drawing cursor where one end of the Limit Line is required and click the left mouse button. Move to the next desired point on the Limit Line and click again, and continue adding points until the whole Limit Line is complete; then right-click and select 'Finish' from the drop-menu. The Limit Line is now complete.

Note, when drawing a Limit Line, that the Line need not cover the entire X–range of the current Trace – the Trace will only be limit-checked over the X–range for which a Limit Line exists. In fact, using right-click then selecting 'End segment', it is possible to build a Limit Line composed of a series of discrete segments. The 'Undo' option of the same drop-menu can be used to correct misplaced points. Once the Limit Line is complete, it can be edited by selecting it as the current Trace (rather than its associated Live Trace) and clicking the  icon on the Trace Toolbar as before. The drawing cursor can now be used to drag any of the points in the line (by holding down the left mouse-button) and then to drop them wherever the button is released.

Refer to the Limit Tables section of the Scripting Manual for details of this method of Limit Line definition.

Limit Lines can be saved and later reused by entering their filenames in the Upper and Lower Limit Line windows in the Trace Settings dialogue box.

When creating Limit Lines, dScope initialises the consequence of a breach 'event' to be an audible 'beep'. However, there are a wide range of possible consequences which can be selected, using the Event Manager dialogue box.

Trace area drop-menu

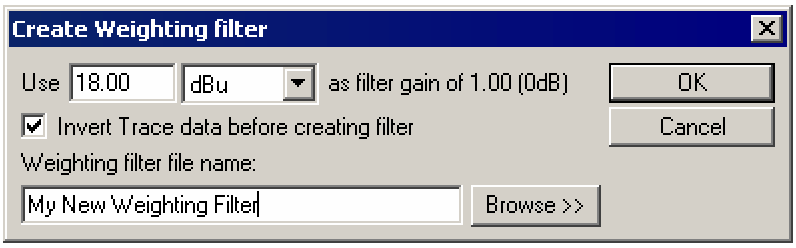
Clicking the right mouse-button over the Trace area produces a drop-menu at the mouse cursor, as shown below.

Print graph		
Print-preview		
Export graph		
Export-preview		
Copy graph to clipboard		
Auto-zoom all Traces		
Set all Traces to defaults		
Change default colours		
Add/load Trace		
Set current Trace:	[List of available Traces]	
Current Trace:	Trace settings	
	Change Trace colours	
	Remove Trace	
	Save Trace	
	Copy Trace	
	Revert to last scale values	
	X Scale:	Log X
		Zoom X
		Un-zoom X
		Auto-zoom X
		Set X axis to default
		Move to start of Trace
		Move left along Trace
		Move right along Trace
		Move to end of Trace
	Y Scale:	Log Y
		Zoom Y
		Un-zoom Y
		Auto-zoom Y
		Set Y axis to default
		Move Trace up
		Move Trace down
	Trace Cursor on	
	Trace Marks on	
	Show transformed Trace data	[checked or not checked]
	Edit Trace transforms	
	Create/edit Limit Line	

	Create Weighting filter from Trace	
	Edit impulse response Window function	
	Edit Trace comment	
	Edit print style:	[list of line styles]
Print/export options		
Printout annotation	NB: Not yet supported	
Reset all Trace colours		

This drop-menu contains all the Trace window functions, many of which are more easily accessed from the Trace Toolbar, if it is visible. Note that the Current Trace options in the central section of the table are also available for each Trace by right-clicking on the desired Trace in the Quick legend.

Selecting the 'Create Weighting filter from Trace' option in the Trace area drop-menu displays the following dialogue box which enables an FFT Detector Weighting filter to be created from the current Trace. The gain of the Weighting filter can be normalised and inverted as required.



For more information about creating FFT Detector Weighting filters, see the FFT Detector Weighting filters section of the Scripting Manual and the FFT Parameters dialogue box section of this manual.

For more information about printing and exporting graphs, see the Graph Print/Export Setup dialogue box section.

For more information about the various Trace transform operations, see the Trace transform operations section.

Quick legend

The Quick legend provides a summary of the Traces currently present in the Trace area. The Quick legend is arranged in five columns:

'Enabled' check box:	Controls and indicates whether or not the Trace is enabled for viewing and printing.
Type / Colour indicator:	Shows the basic type of the Trace (Scope, FFT, Sweep etc.) and its colour, which can be changed by double-clicking.
Channel indicator:	Indicates the source channel of the Trace.
Trace name:	Shows the name of the Trace, which can be edited by double-clicking.
'Live' flag / copy control:	Control to create a copy of a Trace - so only present on 'live' Traces.

Single-clicking on a Trace in the Quick legend (or double-clicking on the Trace itself) causes that Trace to be selected as the current Trace.

The Quick legend behaves as a dockable Toolbar which docks at either the right or the left side of the Trace area.

Quick legend drop-menu

Clicking the right mouse-button over the Quick legend produces a drop-menu at the mouse cursor. This drop-menu contains the entire 'current Trace' submenu of the Trace area drop-menu detailed above; note that this action also selects the Trace as current.

Cursors and the Cursor Toolbar

To enable a Cursor on the current Trace, click the icon on the Trace Toolbar, or use the 'Trace Cursor on' option in the Trace area drop-menu. Alternatively, a Cursor can be placed at any position on any Trace by double-clicking at the desired position on the Trace. Turning on the Cursor causes the appearance of the Cursor Toolbar at the bottom of the Trace window. NOTE: The Cursor Toolbar can also be shown/hidden using the View menu, irrespective of whether a Cursor is enabled on the current Trace.

The Cursor Toolbar contains a pair of Result boxes showing the X and Y coordinates of the Cursor in user-selectable units. By checking the 'Relative' checkbox, the Cursor drops an X–shaped Datum from which the X and Y offsets of the Cursor are now measured. The following icons appear on the Cursor Toolbar:

- Scrolls the Cursor on the current Trace to the left.
- Scrolls the Cursor on the current Trace to the right.
- Moves the Cursor on the current Trace to the left of the Trace area.
- Adds a Mark to the current Trace at the Cursor position.

The Cursor can be scrolled by using the Left and Right controls on the Cursor Toolbar, or by dragging it with the left mouse-button held down. In Relative mode, the Datum usually stays where it was originally dropped, although it can be dragged to another position with the mouse. There are also a variety of Hotkeys assigned to controlling the Cursor.

The Cursor Toolbar is a dockable Toolbar. See the User-interface basics section in Operation Overview for more information about dockable Toolbars.

Marks and the Mark Toolbar

Marks are a system of multiple reading-markers which can be applied to Traces, annotated, and legended on printed graphs if required.

To enable Marks on the current Trace, click the icon on the Trace Toolbar, or use the 'Trace Marks on' option in the Trace area drop-menu. Turning on Marks causes the appearance of the Mark Toolbar at the bottom of the Trace window (below the Cursor Toolbar). Alternatively, if a Mark is added to the current Trace using the icon on the Cursor Toolbar, the Mark bar appears automatically. NOTE: The Mark Toolbar can also be shown/hidden using the View menu, irrespective of whether Marks are enabled on the current Trace.

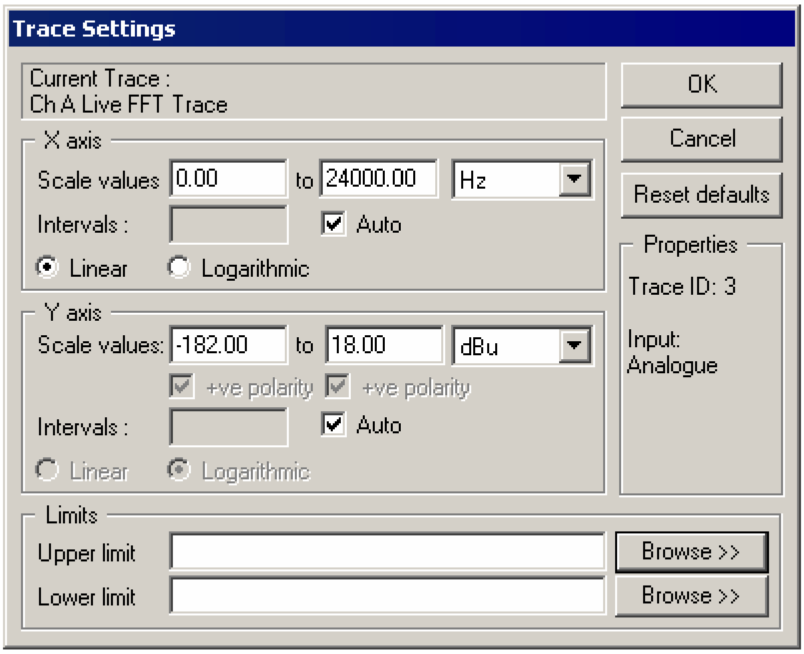
The Mark Toolbar shows the number and user-defined name of the current Mark on the current Trace. The name can be edited by placing the cursor in the name box. The following icons appear on the Mark Toolbar:

- Removes the currently-selected Mark.
- Selects the previous Mark on the current Trace.
- Selects the next Mark on the current Trace.
- Removes ALL Marks from the current Trace.
- Lists all Marks on the current Trace.
- Sums the Y-values of all Marks on the current Trace. (NB: Not supported).
- Marks and displays all harmonics of the predominant frequency on a current FFT Trace.

The Mark Toolbar is a dockable Toolbar. See the User-interface basics section in Operation Overview for more information about dockable Toolbars.

4.6.6.1 Trace Settings dialogue box

The Trace Settings dialogue box controls scaling of the current Trace on the Trace window, and also which Limit Lines are applied to the Trace (if any). The Trace Settings dialogue box is accessed from the Trace window by using the icon on its Toolbar, or by right-clicking in the window and selecting 'Current Trace' : 'Trace Settings' from the drop-menu.



For the X and Y axes of the current Trace, the start and end scale values can be entered, along with the desired units. For some Trace type / units combinations (e.g. Scope Traces in dBFS) where the Y–scale is logarithmic and also bipolar, it may be necessary to use the polarity check-boxes to properly define the desired Y–scale.

The number of graticule divisions can also be entered, or this can be left up to the dScope by checking the 'Auto' box. Note that these graticule settings only apply whilst the Trace remains current; when a different Trace is current, the scale ranges of other Traces remain as defined, but intermediate scale values are adapted to fit the graticule of the current Trace.

The X and Y scales can be selected to be linear or logarithmic, although for certain Trace type / units combinations this setting is inherent and so is shown greyed.

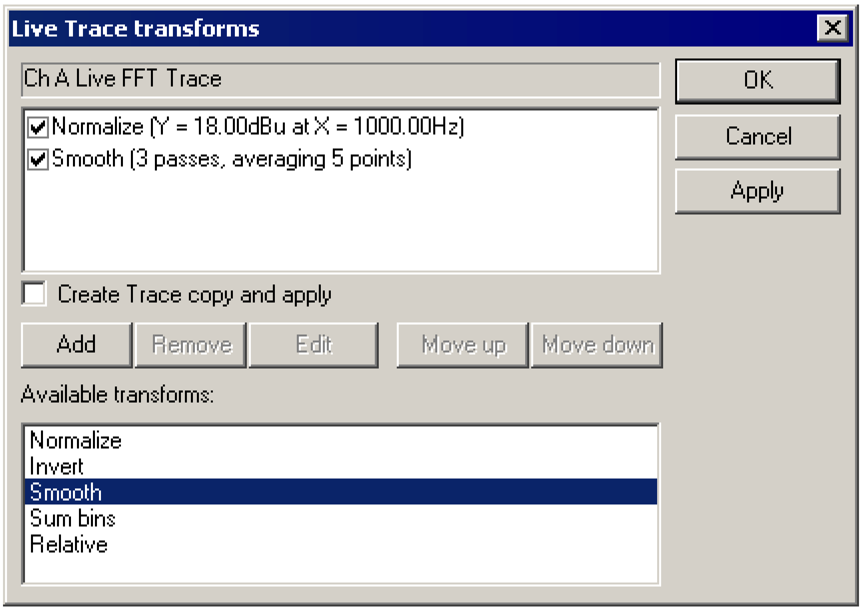
The [Reset defaults] button sets all scale-related settings back to the default settings according to Trace type.

The bottom part of the dialogue box allows saved Limit Lines to be recalled for association with the current Trace. Manual creation and association of Limit Lines is described in the Trace window dialogue box section.

The 'Properties' area beneath the [Reset defaults] button shows the domain from which the Trace data was captured (Analogue, Digital or Soundcard). This can be useful where Traces are not being updated from the current Analyzer domain. The 'Trace ID' is also shown, for the assistance of those writing VB Scripts, wherein Traces are referred to by their Trace IDs.

4.6.6.2 Trace transform operations

A variety of transform operations can be applied to Traces. The Trace transforms dialogue box for the current Trace is accessed by selecting 'Edit trace transforms' from the Trace area drop-menu (or more conveniently via the Quick Legend drop-menu, by right-clicking on the desired Trace in the Quick Legend).



The available transforms are listed at the bottom of the box, and desired transforms are added to the list of active transforms at the top of the box by highlighting the desired transform and clicking the [Add] button. Active transforms can be highlighted and removed by clicking the [Remove] button. The [Edit] button accesses a dialogue box which defines the parameters of the selected active transform, as detailed for each transform below. Note that this box is also automatically accessed when the transform is first added.

The following transforms are available:

Normalize Trace data	Adjust the Y–range of a Trace so that a specified Y–value is attained for a given X–value.
Invert Trace data	Invert the Y–values of a Trace about a specified point on the Y–axis.
Smooth Trace data	Apply a single or multi-pass moving average filter to a Trace.
Sum Trace data bins	Sums the bins of an FFT Trace over specified bandwidth.
Show Trace as relative	Shows Y-values of a Trace relative to another Trace on the same axes.

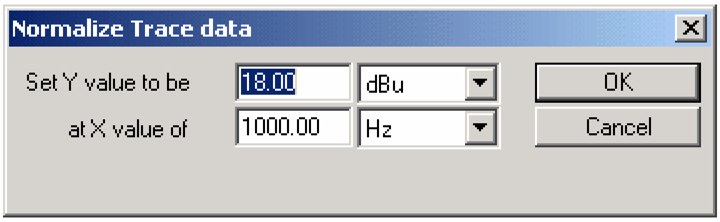
Note that the functions may be employed in combination; for example a Trace of frequency response could be smoothed and inverted, for example to create a Weighting filter. The [Move up] and [Move down] buttons allow the active transforms to be correctly ordered.

Check-boxes next to the active transforms allow them to be temporarily disabled if required.

Normally, the list of active transforms is applied to the Trace 'in-place', i.e. the Trace is displayed after the transforms have been applied. The 'Create Trace copy and apply' check-box causes a copy of the Trace to be made, to which the transforms are applied - the Trace itself is displayed as normal, without transforms.

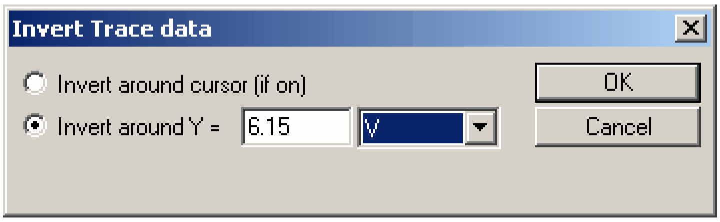
The 'Show transformed Trace data' item in the Trace area drop-menu (or the icon on the Trace Toolbar) can be used to switch the Trace to be displayed with or without its transforms. In the case of the 'Create Trace copy and apply' check-box being checked, this option causes the copy Trace to be displayed or not.

Normalize Trace data



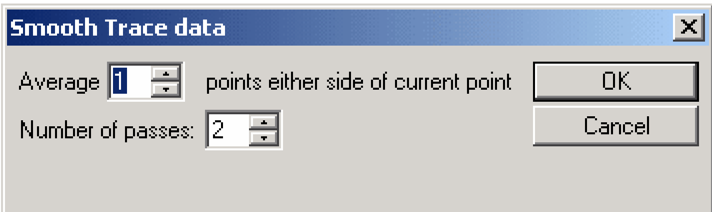
This transform causes the Trace to be shifted vertically on the Y-axis. The desired Y–value at a corresponding X–value is specified.

Invert Trace data



This transform causes the Trace to be inverted on the Y-axis. The Y–value around which to invert the Trace can be specified using the Cursor, or by entering it explicitly.

Smooth Trace data

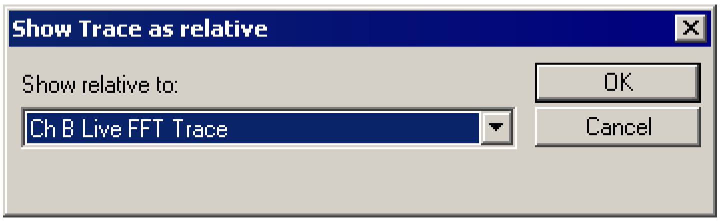


This transform causes the Trace to be smoothed by averaging each point with a number of points either side of it. For additional smoothing, the process can be repeated. The number of adjacent Trace points to be averaged is specified, along with the number of passes.

Sum Trace data bins

This transform applies only to FFT Traces, and causes the Y-value of each point to be the summation of FFT bins over a fixed bandwidth centred on that point. This allows, for example, an FFT to yield a similar Trace to a third-octave band-pass sweep, but with much higher frequency resolution. The frequency resolution is maintained: i.e. each point represents a summation centred at that bin frequency, rather than collecting bins within specific nth-octave bands to form a 'histogram'.

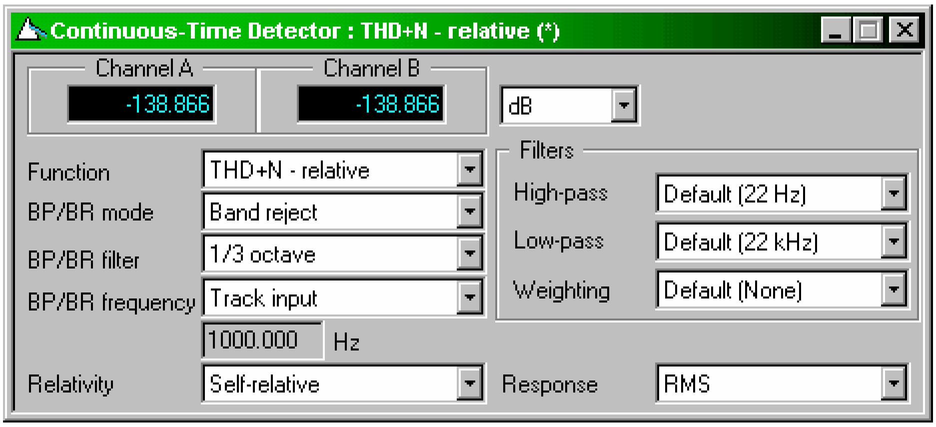
Show Trace as relative



This transform allows a Trace to be displayed relative to another Trace with the same axes. This transform is useful, for example, in showing deviation of an EUT from a 'golden unit'.

4.6.7 Continuous-Time Detector dialogue box

The Continuous-Time Detector dialogue box controls the operation of the Continuous-Time Detector and displays its Results. For a block diagram and description of the relevant area of the dScope hardware, go to Continuous-Time Analyzer architecture.



The Continuous-Time Detector is similar in appearance and functionality to the FFT Detectors, except that its title bar is green whereas those of the FFT Detectors are red. The Continuous-Time Detector is generally preferred for basic measurement tasks, because it is faster than the FFT Detectors, and runs continuously and so cannot miss transitory input events. FFT Detectors, on the other hand, are slower, operate on captured buffers of input data (and so can miss events which occur between triggerings) but are capable of more complex analyses, including user-defined Detector-calculations programmed in VBScript. Up to 40 two-channel FFT Detectors can be in use at once, measuring many different parameters simultaneously, whereas only one (two-channel) Continuous-Time Detector is available.

Note that in common with all dScope Results, the Continuous-Time Detector Results can be 'dragged off' to form Reading windows if required.

Selecting CTD Functions

The 'Function' selected from the list-box actually runs a VBScript of the same name which sets the other parameters in the dialogue box. Thus, for example, selecting the 'THD+N – relative' function sets the BP/BR filter in third-octave band reject mode, tracking the predominant input frequency, and expresses the Result relative to the input amplitude. Filters are set to follow the defaults indicated in the Signal Analyzer dialogue box. If any of the parameters is then manually altered an asterisk is displayed in the title bar to show that the settings from the script have been over-ridden. If the 'Remember changes to Detector functions' option is checked in the Options dialogue box, any such changes are recalled whenever that function is selected again within the current session. The original script has not been altered, however. It is possible for the user to include his own functions by adding his own script to the 'scripts\CT Detector Functions' folder within the dScope program folder. This is most easily done by copying an existing script, then renaming and editing the copy. Note, however, that such a script can only do what could be done manually by selecting values for the various CTD parameters. On the other hand, such a script is essentially an automation script and so can make use of other settings within the dScope. For more information on this see the Detector Functions section of the Scripting Manual.

Note that the 'Units' setting of the dialogue box is not set directly by the selected Detector function script. Instead, each Detector has an associated absolute unit for each of the analogue and digital domains, and also an associated relative unit. Switching between the absolute and relative units takes place when the absolute/relative setting is modified either manually or by selecting a function script. These selected units can then be modified for each Detector by changing the units setting. Returning to a previous relativity restores the appropriate unit set for that relativity in each Detector.

CTD Parameters

This section describes the various parameters of the CTD and lists the available options.

Units

Selects the units in which the Result is displayed.

NB: **It is important to understand how units are applied. Some units (such as 'dBu') are inherently 'RMS', others (such as '%FS') are inherently 'peak'; to obtain 'correct' results, be sure that the 'Response' setting is correct. Incompatible responses can be used with either type of units (and this is occasionally useful) but with possibly confusing results. Some units (such as V) can be correctly applied to either response. For more information about this, see the Amplitude units in the dScope section.**

Relative units are used for relative measurement modes (such as THD+N relative) and simply describe the ratio of the measured Result to the relative-reference selected by the Relativity parameter (see below).

'dBr' and '%Ref', on the other hand, are absolute units but allow Results to be expressed relative to a fixed reference amplitude set by the user.

NB: **The dScope allows Results in one domain to be displayed in the units of the other. For example an analogue amplitude can be displayed in dBFS if required – this can be useful when measuring mixed-domain systems such as A/D or D/A converters. In these cases, the 'D/A (digital-analogue) line-up' setting in the Signal Analyzer dialogue box is used to determine how to convert between the analogue and digital units.**

Supported units are shown in the following table:

ABSOLUTE UNITS			
	Analogue	*Digital*	*General*
'RMS' type	dBu, V, dBV, dBm, W, dBSPL	dBFS	dBr
'peak' type	V	Hex,FFS, %FS	%Ref
RELATIVE UNITS			
dB, %, gain, 'Per' (see text)			

The 'Per' unit is a special relative unit which allows the Detector Result to be expressed as a cross-unit normalised gain. When the relativity of the Detector (see below) is set to Generator-relative, the 'Per' unit appears in the drop list as '[Analyzer units]/[1 Generator unit]' where

[Analyzer units] are the units currently selected in the Signal Analyzer dialogue box (e.g. V, dBFS etc.) and [1 Generator unit] is the unitary value of the units currently selected in the Signal Generator dialogue box (e.g. 1V, 100%FS, 0dBu). Thus, using the 'Per' unit, the gain of the EUT could be expressed, for example, in 'dBSPL/1W' or 'W/0dBFS'. The 'Per' unit inherently assumes EUT voltage or numeric gain linearity, i.e. an Analyzer Result of 1V for a Generator amplitude of 200mW would equate to '2.236V/1W' whether or not the EUT would actually output 2.236V for a 1W input. Note that the 'Per' unit is also available when the relativity of the Detector is Channel-relative; in this case, the amplitude of the other Analyzer channel is converted into Generator units and used as the normalising reference.

Response

Selects the response of the CTD's peak-detector. See also the 'Units' parameter above. Available settings are:

RMS	Displays the root-mean-square amplitude.
Peak	Displays the peak amplitude, derived using an interpolation filter.
Peak-sample	Displays the amplitude of the largest individual incoming sample.
CCIR468 Q-peak	Displays the 'quasi-peak' amplitude according to CCIR468.

BP/BR Mode

Sets the mode of the band pass/band reject filter. Available settings are:

Off	For non-selective amplitude measurements.
Band pass	For frequency-selective measurements or for low-level measurements in order to eliminate noise.
Band reject	For residual measurements such as THD+N.
IMD demodulation	For SMPTE/DIN IMD measurements. This is measured with a stimulus containing a high and a low frequency tone. The LF tone is removed with a 2kHz high-pass filter, the remainder is then demodulated to the base band, and the IMD products isolated with a 4Hz high-pass filter and a 700Hz low-pass filter.

BP/BR filter

Sets the bandwidth (Q-factor) of the BP/BR filter. Available settings are:

1/3 octave	'Standard' setting emulates traditional analogue analyzers.
1/6 octave	
1/12 octave	
1/24 octave	Most selective setting.

BP/BR frequency

Determines the way that the frequency of the BP/BR filter is set. Available settings are:

Track input	Centres the BP/BR filter on the Analyzer input frequency. Used for standard residual measurements.
Track generator	Centres the BP/BR filter on the generator frequency of the same channel. Useful for low-level 'band pass' measurements where frequency sensing is unreliable, e.g. converter linearity.
Track generator (opp ch)	Centres the BP/BR filter on the generator frequency of the opposite channel. Used for cross-talk measurements.
Fixed	Allows manual setting of the filter to a specified frequency.

IMD differential (CCIF)	When the Signal Generator function is 'twin-tone', this setting causes the filter to be centred on the difference-frequency of the tones. Used to measure IMD by the CCIF method.

Relativity

Determines whether the Result will be displayed as an absolute amplitude or relative to some other amplitude. See also the Units section above. Available settings are:

Absolute	Displays the Result in an absolute unit.
Self-relative	Displays the Result relative to the pre-BP/BR signal of the same channel; e.g. for residual measurements such as THD+N.
Generator-relative	Displays the Result relative to the amplitude of the equivalent channel of the signal generator; e.g. for measuring gain of the EUT.
Channel-relative	Similar to self-relative, but uses the pre-BP/BR signal from the opposite Analyzer channel as a reference; e.g. for cross-talk measurements.

High-pass filter

Selects a high-pass filter if required. Available settings are:

Follow defaults	Follows the default setting in the Signal Analyzer dialogue box.
Off	Disables the high-pass filter (not available for DC-blocked Analogue Inputs).
DC-block	Approximately 1.2Hz cut-off frequency for Analogue Inputs.
10Hz	
22Hz	
100Hz	
400Hz	

In addition to the preset frequencies in the drop-menu, it is also possible to manually enter any desired frequency between 10Hz and half the sample frequency, after clicking in the box.

NB: **This setting can be very critical to the settling time of some measurements, for example THD+N. The reason is that the dScope offers a very extended low-frequency response with the high-pass filter off or set to DC-block. In this state, a step change in frequency or amplitude at the Analyzer input may produce a disturbance in the measurement which can take a long time to settle. This is unavoidable and is a consequence of the extended LF response. This extended settling time may make Sweeps slow or cause timeouts or unstable results in Sweeps unless settling parameters are suitably modified. To ensure fast accurate THD+N Sweeps, make sure that a high-pass filter of at least 10Hz is included (as it is in the default settings). Note that even if a 22Hz HPF is selected, no noticeable variation in the measured THD+N occurs even at 20Hz because the default 1/3 octave band reject filter causes significant attenuation extending above 22Hz, which is not significantly increased by including the 22Hz HPF. To guarantee optimum speed and accuracy of THD+N Sweeps, the extended LF response of the dScope should not be used unless it is needed, in which case the Sweep settling must be extended by increasing the number of points or reducing the reading rate.**

Low-pass filter

Selects a low-pass filter if required. Available settings are:

Follow defaults	Follows the default setting in the Signal Analyzer dialogue box.
20kHz (AES17)	An especially steep 20kHz LPF as specified in AES17

22kHz	
30kHz	
40kHz	
80kHz	
Off	Disables the low-pass filter.

In addition to the preset frequencies in the drop-menu, it is also possible to manually enter any desired frequency between 100Hz and half the sample frequency, after clicking in the box.

The AES17 filter is especially recommended when measuring EUTs with significantly rising ('shaped') noise beyond the audio band, for example digital power amplifiers and 1-bit devices and systems, e.g. SACD/DSD. Use of the AES17 filter prevents noise and distortion results being worsened by the partial inclusion of out-of-band noise, which occurs even when the conventional 22kHz LPF is selected.

Note that at sample frequencies above 96kHz, the stop-band attenuation of the CTD's AES17 filter may be less than the -60dB recommended in AES17. However, even at high sample frequencies the AES17 filter provides improved rejection of out-of-band noise.

Note that settings above half the sample frequency of the selected input may be selected **but obviously do not function**. Even in the 'off' setting, high-frequency response is limited to half the sample frequency (0.5fs) for Digital Inputs, and about 0.49fs for Analogue Inputs (about 47kHz at fs=96kHz and 95kHz at 192kHz, –3dB points). See the Specifications section for more details.

Weighting filter

Selects a Weighting filter if required. Available settings are:

Follow defaults	Follows the default setting in the Signal Analyzer dialogue box.
None	Disables the Weighting filter.
A–wtg	Selects an ANSI/IEC A–Weighting filter.
C–wtg	Selects an ANSI/IEC C–Weighting filter.
CCIR468–1k	Selects a CCIR468 Weighting filter, normalized for unity gain at 1kHz.
CCIR468–2k	Selects a Weighting filter of the CCIR468 shape, but normalized for unity gain at 2kHz, as specified for AES17 and Dolby measurements.

'Factory' CTD Functions

A number of 'factory' CTD functions are included. Note that these provide a convenient starting point for most measurements, by loading the Detector's parameters with default values (see the table below); however, all parameters can be adjusted manually if required. Further Detector functions can be added to the drop-menu by adding scripts to the 'scripts\CT Detector Functions' folder within the dScope program folder as described above.

Factory CTD functions are as follows:

Amplitude measures the absolute RMS amplitude at the analyzer inputs. This Result is similar to the RMS Amplitude Result of the Signal Analyzer, except that default high-pass and low-pass filters are applied. Also, unlike the Signal Analyzer case, the response can be set to 'Peak', 'Peak Sample' or 'CCIR468 Q-peak' rather than RMS if required.

Balance measures the RMS amplitude at the analyzer inputs in the same way as the 'Amplitude' function, except that the Result for each channel is displayed relative to the other, rather than absolutely. Thus if the A-channel is 1.5dB louder than the B-channel, its Result would be '+1.5dB' whilst the B-channel's would be '-1.5dB'.

Band Pass makes a 'selective' measurement of the absolute RMS amplitude at the analyzer inputs via a third-octave band pass filter. This type of measurement is often used to distinguish small incidences of a particular spurious frequency, for example cross-talk, unrejected common-mode or mains interference, since it is able to exclude to an extent the wider-band noise which would otherwise usually dominate and obscure the desired Result. The band-pass frequency is set to follow the dScope's Signal Generator by default, since tracking the input frequency is impractical when trying to measure small components hidden in noise. Note that this function is usually unsuitable for measuring small frequency components within large signals (e.g. harmonic distortions) since the limited Q-factor of the CTD's band-pass filter (maximum at 1/24 octave bandwidth) would not usually allow sufficient attenuation of loud frequency component(s). If the residual (band reject) method is too unselective for a particular task, the FFT Detector can provide band pass filtering with extremely narrow bandwidth and no leakage from outside the band.

Band Reject measures the absolute RMS amplitude of the 'residual' signal at the analyzer inputs, i.e. the total signal which remains after the predominant frequency has been removed by a third-octave band reject filter. This is the traditional way to measure 'THD+N' (actually, this function is the same as the 'THD+N absolute' function). However such measurements, whilst providing close agreement with traditional analysers, are slightly compromised in accuracy by the low Q-factor of the band reject filter, which causes the effects of residual components close to the predominant frequency to be understated. This may be important if low-frequency modulation effects such as sampling jitter (which produce close-spaced distortion side-bands) are being assessed. In these cases, the FFT Detector can provide band reject filtering with extremely narrow bandwidth and unity gain at all frequencies outside the rejection band.

Cross-talk makes a 'selective' measurement of the RMS amplitude at the analyzer inputs via a 1/24-octave band pass filter. Each channel's filter is tuned to the frequency of the opposite channel of the dScope's Signal Generator, and each channel's Result is expressed relative to the generated amplitude of the opposite channel.

Gain expresses the RMS amplitude of the analyzer inputs relative to the generated amplitude of the same channel, thus showing the gain of the EUT. The measurement is actually made 'selectively' via a third-octave band pass filter tuned to the generator frequency to maintain accuracy at low levels by excluding the effects of wider-band noise.

IMD CCIF measures intermodulation distortion by the 'difference frequency' method. This is usually performed by stimulating the EUT with a mix of two high-frequency tones at high amplitude and close together in frequency (e.g. 19kHz and 20kHz). This function measures the resulting component at the 'difference frequency' (1kHz in the above example) using a 1/24 octave band pass filter. Note that the 'IMD diff CCIF' setting of the band pass filter frequency causes it to be automatically tuned to the difference frequency of the dScope's Signal Generator when it is generating a 'twin tone'. The Result for each channel is expressed relative to the total analyzer RMS signal amplitude for that channel.

IMD SMPTE-DIN measures intermodulation distortion by the traditional SMPTE-DIN method. This is performed by stimulating the EUT with a mix of low- and mid-frequency tones, usually with the higher frequency at a lower amplitude (a 7kHz tone 12dB below a 60Hz tone is usual, this being the 'default' setting of the dScope Signal Generator in 'twin tone' mode). Any sidebands on the higher tone as a result of modulation by the lower frequency within the EUT are measured by filtering off the lower frequency and then demodulating the remainder so that its spectrum is reproduced with the higher frequency at DC, which is then also removed along with the original higher frequency. The demodulated IMD thus remains, and is measured relative to the total analyzer RMS signal amplitude for each channel. The SMPTE-DIN method was devised to suit analogue analysis, and this CTD function allows compatible measurements to be made. However, the dScope's FFT Analyzer can provide much greater insight into intermodulation effects since it can resolve individual modulation components.

Noise (unweighted) measures the absolute RMS amplitude at the analyzer inputs, over a bandwidth defined by the default high-pass and low-pass filters. Alternative filter frequencies can be selected if desired.

Noise (A-weighted) is similar to the 'Noise (unweighted)' function, but is measured via an A-weighting filter.

Noise (CCIR-468) is similar to the 'Noise (unweighted)' function, but is measured via a special weighting filter, and using a special quasi-peak detector response, as described in CCIR468-2.

THD+N absolute reproduces the settings of 'Band Reject', as described above.

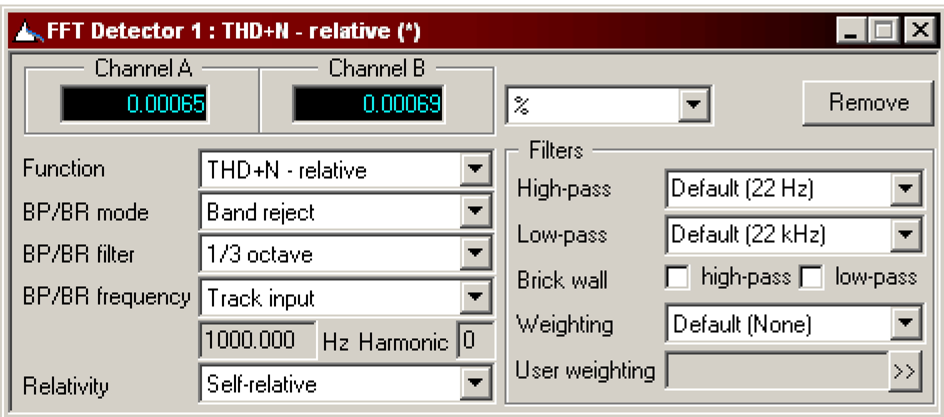
THD+N relative is similar to 'Band Reject', described above, but presents the residual Result relative to the RMS amplitude of the total analyzer input signal.

The following table lists the available 'factory' CTD functions and their associated parameters. Note that high-pass and low-pass filters are set to 'follow defaults' in all functions.

Function	**Band pass / reject**			**Relativity**	**Weight'g**	**Resp**
	Mode	**Filter**	**Freq**			
Amplitude	Off			Absolute	Default	RMS
Balance	Off			Chan-rel	Default	RMS
Band pass	BP	3rd oct	Track gen	Absolute	Default	RMS
Band reject	BR	3rd oct	Track input	Absolute	Default	RMS
Cross-talk	BP	24th oct	Track gen (opp chan)	Chan-rel	Default	RMS
Gain	BP	3rd oct	Track gen	Gen-rel	Default	RMS
IMD CCIF	BP	24th oct	IMD diff CCIF	Self-rel	Default	RMS
IMD SMPTE-DIN	IMD demod			Self-rel	Default	RMS
Noise (A-weight'd)	Off			Absolute	A-weight	RMS
Noise (CCIR-468)	Off			Absolute	CCIR-468 1k	CCIR-468 Q-peak
Noise (unweighted)	Off			Absolute	None	RMS
THD+N absolute	BR	3rd oct	Track input	Absolute	Default	RMS
THD+N relative	BR	3rd oct	Track input	Self-rel	Default	RMS

4.6.8 FFT Detector dialogue box

The FFT Detector dialogue boxes control the function of the FFT Detectors and display their Results. Up to 40 two-channel FFT Detectors can be active simultaneously, measuring different parameters. For a block diagram and description of the relevant area of the dScope hardware, go to FFT Analyzer architecture.



The FFT Detectors are similar in appearance and functionality to the Continuous-Time Detector, except that their title bars are red whereas that of the Continuous-Time Detector is green. The Continuous-Time Detector is generally preferred for basic measurement tasks, because it is faster than the FFT Detectors, and runs continuously and so cannot miss transitory input events. FFT Detectors, on the other hand, are slower, operate on captured buffers of input data (and so can miss events which occur between triggerings) but are capable of more complex analyses, including user-defined Detector-calculations programmed in VBScript. Up to 40 two-channel FFT Detectors can be in use at once, measuring many different parameters simultaneously, whereas only one (two-channel) Continuous-Time Detector is available.

Note that in common with all dScope Results, the FFT Detector Results can be 'dragged off' to form Reading windows if required.

Selecting FFTD Functions

The 'Function' selected from the list-box actually runs a VBScript of the same name which sets the other parameters in the dialogue box. Thus, for example, selecting the 'THD+N relative' function sets the BP/BR filter in third-octave band reject mode, tracking the predominant input frequency, and expresses the Result relative to the input amplitude. Filters are set to follow the defaults indicated in the Signal Analyzer dialogue box. If any of the parameters is then manually altered an asterisk is displayed in the title bar to show that the settings from the script have been over-ridden. If the 'Remember changes to Detector functions' option is checked in the Options dialogue box, any such changes are recalled whenever that function is selected again within the current session. The original script has not been altered, however. It is possible for the user to include his own functions by adding his own script to the 'scripts\FFT Detector Functions' folder within the dScope program folder. This is most easily done by copying an existing script, then renaming and editing the copy. Note, however, that such a script can only do what could be done manually by selecting values for the various FFTD parameters. On the other hand, such a script is essentially an automation script and so can make use of other settings within the dScope. For more information on this see the Detector Functions section of the Scripting Manual.

Note that the 'Units' setting of the dialogue box is not set directly by the selected Detector function script. Instead, each Detector has an associated absolute unit for each of the analogue and digital domains, and also an associated relative unit. Switching between the absolute and relative units takes place when the absolute/relative setting is modified either manually or by selecting a function script. These selected units can then be modified for each Detector by changing the units setting. Returning to a previous relativity restores the appropriate unit set for that relativity in each Detector.

For more complex analyses it is possible for the user to program the functionality of FFT Detectors in a way which transcends merely setting the other parameters in the dialogue box. By selecting a 'User' function, a VBScript can be nominated which actually processes the bins of the FFT buffer and calculates Results according to any algorithm the user wishes. This method is used to process many Results from a single multi-tone, for example. For more details of this process, see the FFT Detector Calculation scripts section in the Scripting Manual.

⚠ **When the FFT Detector Function is set to 'User' to select an FFT Detector Calculation script, the other parameters in the FFT Detector dialogue box remain unaffected, i.e. they stay as before. If an FFT Detector Calculation script requires that FFT Detector parameters be set to particular states, this must be done within the FFT Detector Calculation script.**

FFTD Parameters

This section describes the various parameters of the FFTD and lists the available options.

Units

Selects the units in which the Result is displayed.

Relative units are used for relative measurement modes (such as THD+N) and simply describe the ratio of the measured Result to the relative-reference selected by the Relativity parameter (see below).

'dBr' and '%Ref', on the other hand, are absolute units but allow Results to be expressed relative to a fixed reference amplitude set by the user.

NB: **The dScope allows Results in one domain to be displayed in the units of the other. For example an analogue amplitude can be displayed in dBFS if required – this can be useful when measuring mixed-domain systems such as A/D or D/A converters. In these cases, the 'D/A (digital-analogue) line-up' setting in the Signal Analyzer dialogue box is used to determine how to convert between the analogue and digital units.**

Supported units are shown in the following table:

ABSOLUTE UNITS			
	Analogue	*Digital*	*General*
	dBu, V, dBV, dBm, W, dBSPL	dBFS, Hex, FFS, %FS	dBr, %Ref
RELATIVE UNITS			
			dB, %, gain, 'Per' (see text)

The 'Per' unit is a special relative unit which allows the Detector Result to be expressed as a cross-unit normalised gain. When the relativity of the Detector (see below) is set to Generator-relative, the 'Per' unit appears in the drop list as '[Analyzer units]/[1 Generator unit]' where [Analyzer units] are the units currently selected in the Signal Analyzer dialogue box (e.g. V, dBFS etc.) and [1 Generator unit] is the unitary value of the units currently selected in the Signal Generator dialogue box (e.g. 1V, 100%FS, 0dBu). Thus, using the 'Per' unit, the gain of the EUT could be expressed, for example, in 'dBSPL/1W' or 'W/0dBFS'. The 'Per' unit inherently assumes EUT voltage or numeric gain linearity, i.e. an Analyzer Result of 1V for a Generator amplitude of 200mW would equate to '2.236V/1W' whether or not the EUT would actually output 2.236V for a 1W input. Note that the 'Per' unit is also available when the relativity of the Detector is Channel-relative; in this case, the amplitude of the other Analyzer channel is converted into Generator units and used as the normalising reference.

Response

The 'Response' setting of the Continuous-Time Detector is not present in the FFT Detectors, since all FFT-derived Results are RMS owing to the nature of the FFT process.

BP/BR Mode

Sets the mode of the band pass/band reject filter. Available settings are:

Off	For non-selective amplitude measurements.
Band-pass	For frequency-selective measurements or for low-level measurements in order to eliminate noise.
Band-reject	For residual measurements such as THD+N.

BP/BR filter

Sets the bandwidth (Q–factor) of the BP/BR filter. Available settings are:

1/3 octave	'Standard' setting emulates traditional analogue analyzers.
1/6 octave	
1/12 octave	

1/24 octave	
Window notch	Most selective setting. Applies a rectangular notch with infinite attenuation inside or outside the BP/BR frequency and unity gain elsewhere. The width of the notch is the minimum dictated by the leakage characteristic of the selected Window function.

BP/BR frequency

Determines the way that the frequency of the BP/BR filter is set. Available settings are:

Track input	Centres the BP/BR filter on the Analyzer input frequency. Used for standard residual measurements.
Track generator	Centres the BP/BR filter on the generator frequency of the same channel. Useful for low-level 'band pass' measurements where frequency sensing is unreliable, e.g. converter linearity.
Track generator (opp ch)	Centres the BP/BR filter on the generator frequency of the opposite channel. Used for cross-talk measurements.
Fixed	Allows manual setting of the filter to the frequency specified below.
IMD differential (CCIF)	When the Signal Generator function is 'twin-tone', this setting causes the filter to be centred on the difference-frequency of the tones. Used to measure IMD by the CCIF method.
IMD side-bands (SMPTE/DIN)	When the Signal Generator function is 'twin-tone', this setting causes the filter to be centred on the upper side-band frequency, i.e. HF+(HF –LF) and set to window-width. Used to measure second-order IMD by the SMPTE/DIN method.
All harmonics	Creates a multi-frequency BP/BR filter at harmonics of the input frequency, excluding the fundamental. Forces Window notch for BP/BR bandwidth. Maximum harmonic set in box below, or 0 for all harmonics.
All harmonics + fundamental	Creates a multi-frequency BP/BR filter at the input frequency and harmonics of the input frequency. Forces Window notch for BP/BR bandwidth. Maximum harmonic set in box below, 1 for fundamental only, or 0 for all harmonics.
Nth harmonic	Centres the BP/BR frequency on the Nth harmonic of the input frequency. N is entered below, or 0 for off, 1 for fundamental.

NOTE: When 'All harmonics' or 'All harmonics + fundamental' is selected, and BP/BR mode is set to BR (band reject), the residual result is automatically modified to add the average bin-noise into each of the excluded bins. In this way, for example, the accuracy of the 'Noise (residual)' function is improved.

Relativity

Determines whether the Result will be displayed as an absolute amplitude or relative to some other amplitude. See also the Units section above. Available settings are:

Absolute	Displays the Result in an absolute unit.
Self-relative	Displays the Result relative to the pre-BP/BR signal of the same channel; e.g. for residual measurements such as THD+N.
Generator-relative	Displays the Result relative to the amplitude of the equivalent channel of the signal generator; e.g. for measuring gain of the EUT.
Channel-relative	Similar to self-relative, but uses the pre-BP/BR signal from the opposite Analyzer channel as a reference; e.g. for cross-talk measurements.

High-pass filter

Selects a high-pass filter if required. Available settings are:

Follow defaults	Follows the default setting in the Signal Analyzer dialogue box.
Off	Disables the high-pass filter (not available for DC-blocked Analogue Inputs).
DC-block	Approximately 1.8Hz cut-off frequency for Analogue Inputs.
10Hz	
22Hz	
100Hz	
400Hz	

In addition to the preset frequencies in the drop-menu, it is also possible to manually enter any desired frequency between 10Hz and half the sample frequency, after clicking in the box.

Note that low-frequency resolution of FFT Traces and FFT Detectors is dependent on the number of FFT points. If the number of points is insufficient, low-frequency resolution is reduced so that, for example, high-pass filters may not operate as expected.

Low-pass filter

Selects a low-pass filter if required. Available settings are:

Follow defaults	Follows the default setting in the Signal Analyzer dialogue box.
20kHz (AES17)	An especially steep 20kHz LPF as specified in AES17
22kHz	
30kHz	
40kHz	
80kHz	
Off	Disables the low-pass filter.

In addition to the preset frequencies in the drop-menu, it is also possible to manually enter any desired frequency between 100Hz and half the sample frequency, after clicking in the box.

The AES17 filter is especially recommended when measuring EUTs with significantly rising ('shaped') noise beyond the audio band, for example digital power amplifiers and 1-bit devices and systems, e.g. SACD/DSD. Use of the AES17 filter prevents noise and distortion results being worsened by the partial inclusion of out-of-band noise, which occurs even when the conventional 22kHz LPF is selected.

Note that settings above half the sample frequency of the selected input may be selected **but obviously do not function**. Even in the 'off' setting, high-frequency response is limited to half the sample frequency (0.5fs) for Digital Inputs, and about 0.49fs for Analogue Inputs (about 47kHz at fs=96kHz and 95kHz at 192kHz, -3dB points). See the Specifications section for more details.

Brick wall

Check-boxes are provided to allow the high-pass and low-pass filters to be specified as 'brick wall' if desired. In 'brick wall' mode, the filters are idealised, with flat pass-bands transitioning suddenly (over a single bin width) into infinitely attenuative stop bands.

Weighting filter

Selects a Weighting filter if required. Available settings are:

Follow defaults	Follows the default setting in the Signal Analyzer dialogue box.

None	Disables the Weighting filter.
A–wtg	Selects an ANSI/IEC A–Weighting filter.
C–wtg	Selects an ANSI/IEC C–Weighting filter.
CCIR468–1k	Selects a CCIR468 Weighting filter, normalized for unity gain at 1kHz.
CCIR468–2k	Selects a Weighting filter of the CCIR468 shape, but normalized for unity gain at 2kHz, as specified for AES17 and Dolby measurements.
User weighting	Selects a user-defined Weighting filter, either in the form of a VBScript (.dss) or a table (.wtg). See the FFT Detector Weighting filters section of the Scripting Manual, and the FFT Parameters dialogue box and Trace area drop-menu sections of this manual for more details about creating user-defined Weighting filters.

'Factory' FFTD Functions

A number of 'factory' FFTD functions are included. Note that these provide a convenient starting point for most measurements, by loading the Detector's parameters with default values (see the table below); however, all parameters can be adjusted manually if required. Further Detector functions can be added to the drop-menu by adding scripts to the 'scripts\FFT Detector Functions' folder within the dScope program folder as described above.

Factory FFTD functions are as follows:

Amplitude measures the absolute RMS amplitude at the analyzer inputs. This Result is similar to the RMS Amplitude Result of the Signal Analyzer, except that default high-pass and low-pass filters are applied.

Balance measures the RMS amplitude at the analyzer inputs in the same way as the 'Amplitude' function, except that the Result for each channel is displayed relative to the other, rather than absolutely. Thus if the A-channel is 1.5dB louder than the B-channel, its Result would be '+1.5dB' whilst the B-channel's would be '-1.5dB'.

Band Pass makes a 'selective' measurement of the absolute RMS amplitude at the analyzer inputs via a third-octave band pass filter. This type of measurement is often used to distinguish small incidences of a particular spurious frequency, for example cross-talk, unrejected common-mode or mains interference, since it is able to exclude to an extent the wider-band noise which would otherwise usually dominate and obscure the desired Result. The band-pass frequency is set to follow the dScope's Signal Generator by default, since tracking the input frequency is impractical when trying to measure small components hidden in noise. Note that the third-octave filter is specified for compatibility with other analysers and with the CTD, but may not be optimal for measuring small frequency components within large signals (e.g. harmonic distortions) since its limited Q-factor may not provide sufficient attenuation of loud frequency component(s). The 'Window width notch' setting provides band pass filtering with extremely narrow bandwidth and no leakage from outside the band.

Band Reject measures the absolute RMS amplitude of the 'residual' signal at the analyzer inputs, i.e. the total signal which remains after the predominant frequency has been removed by a third-octave band reject filter. This is the traditional way to measure 'THD+N' (actually, this function is the same as the 'THD+N absolute' function). However such measurements, whilst providing close agreement with traditional analysers, are slightly compromised in accuracy by the low Q-factor of the third-octave filter, which causes the effects of residual components close to the predominant frequency to be understated. This may be important if low-frequency modulation effects such as sampling jitter (which produce close-spaced distortion side-bands) are being assessed. In these cases, the 'Window width notch' setting provides band reject filtering with extremely narrow bandwidth and unity gain at all frequencies outside the rejection band.

Cross-talk makes a 'selective' measurement of the RMS amplitude at the analyzer inputs via a highly selective 'Window width notch' band pass filter. Each channel's filter is tuned to the frequency of the opposite channel of the dScope's Signal Generator, and each channel's Result is expressed relative to the generated amplitude of the opposite channel.

Gain expresses the RMS amplitude of the analyzer inputs relative to the generated amplitude of the same channel, thus showing the gain of the EUT. The measurement is actually made 'selectively' via a third-octave band pass filter tuned to the generator frequency to maintain accuracy at low levels by excluding the effects of wider-band noise.

IMD CCIF measures intermodulation distortion by the 'difference frequency' method. This is usually performed by stimulating the EUT with a mix of two high-frequency tones at high amplitude and close together in frequency (e.g. 19kHz and 20kHz). This function measures the resulting component at the 'difference frequency' (1kHz in the above example) using an extremely selective 'Window width notch' band pass filter. Note that the 'IMD diff CCIF' setting of the band pass filter frequency causes it to be automatically tuned to the difference frequency of the dScope's Signal Generator when it is generating a 'twin tone'. The Result for each channel is expressed relative to the total analyzer RMS signal amplitude for that channel.

IMD SMPTE-DIN measures intermodulation distortion by the traditional SMPTE-DIN method. This is performed by stimulating the EUT with a mix of low- and mid-frequency tones, usually with the higher frequency at a lower amplitude (a 7kHz tone 12dB below a 60Hz tone is usual, this being the 'default' setting of the dScope Signal Generator in 'twin tone' mode). Any sidebands on the higher tone as a result of modulation by the lower frequency within the EUT are measured by filtering off the lower frequency and then demodulating the remainder so that its spectrum is reproduced with the higher frequency at DC, which is then also removed along with the original higher frequency. The demodulated IMD thus remains, and is measured relative to the total analyzer RMS signal amplitude for each channel. Note that the FFT Detector implementation does not need to use successive filtering and demodulation, but separates the sideband frequency range directly using a highly selective notch filter. This allows much lower levels of IMD to be measured, since there is no leakage of the stimulus tones into the measurement whatsoever, which is not the case in the time-domain implementation. NOTE: it is necessary to use an adequate number of FFT points in order for the IMD SMPTE-DIN result to be able to include the first sideband. The minimum number of points depends on the sample rate, the 'width' of the FFT window function and the frequency of the lower tone. At 96kHz, with a Prism7 window and a 60Hz lower tone, a minimum of 16k FFT points are required.

Noise (unweighted) measures the absolute RMS amplitude at the analyzer inputs, over a bandwidth defined by the default high-pass and low-pass filters. Alternative filter frequencies can be selected if desired.

Noise (A-weighted) is similar to the 'Noise (unweighted)' function, but is measured via an A-weighting filter.

Noise (residual) measures unweighted noise in the presence of a sine stimulus, by excluding the fundamental and first ten harmonics with window-width notch filters and compensating the result by adding the average bin-noise for each excluded bin. Thus it is possible, for example to measure noise and distortion at the same time.

THD+N absolute reproduces the settings of 'Band Reject', as described above.

THD+N relative is similar to 'Band Reject', described above, but presents the residual Result relative to the RMS amplitude of the total analyzer input signal.

THD measures the total amplitude of all harmonic distortion components by placing highly selective 'Window width notch' band pass filters at all multiples of the detected input frequency. The Result for each channel is expressed relative to the total analyzer RMS signal amplitude for that channel.

2nd Harmonic Distortion measures the amplitude of the second harmonic distortion component by placing a highly selective 'Window width notch' band pass filter at twice the detected input frequency. The Result for each channel is expressed relative to the total analyzer RMS signal amplitude for that channel.

3rd Harmonic Distortion is as per '2nd Harmonic Distortion', but with the filter at three times the input frequency.

4th Harmonic Distortion is as per '2nd Harmonic Distortion', but with the filter at four times the input frequency.

User employs an 'FFT Detector Calculation Script' to perform a custom measurement. By selecting a 'User' function, a VBScript can be nominated which actually processes the bins of the FFT buffer and calculates Results according to any algorithm the user wishes. For more details of this process, see the FFT Detector Calculation scripts section in the Scripting Manual.

The following table lists the available 'factory' FFTD functions and their associated parameters. Note that high-pass and low-pass filters are set to 'follow defaults' in all functions.

Function	**Band pass / reject**			**Relativity**	**Weight'g**
	Mode	**Filter**	**Freq**		
2nd Harmonic Distortion	BP	Window notch	Nth harm N=2	Self-rel	Default
3rd Harmonic Distortion	BP	Window notch	Nth harm N=3	Self-rel	Default
4th Harmonic Distortion	BP	Window notch	Nth harm N=4	Self-rel	Default
Amplitude	Off			Absolute	Default
Balance	Off			Chan-rel	Default
Band pass	BP	3rd oct	Track gen	Absolute	Default
Band reject	BR	3rd oct	Track input	Absolute	Default
Cross-talk	BP	Window notch	Track gen (opp chan)	Chan-rel	Default
Gain	BP	3rd oct	Track gen	Gen-rel	Default
IMD CCIF	BP	Window notch	IMD diff CCIF	Self-rel	Default
IMD SMPTE/DIN	BP	Window notch	IMD s'band SMPTE/DIN	Self-rel	Default
Noise (unweighted)	Off			Absolute	None
Noise (A-weighted)	Off			Absolute	A-wtg
Noise (residual)	BR	Window notch	All harms+F up to N=10	Absolute	None
THD+N absolute	BR	3rd oct	Track input	Absolute	Default
THD+N relative	BR	3rd oct	Track input	Self-rel	Default
THD	BP	Window notch	All harms up to N=10	Self-rel	Default
User	(FFT Detector Calculation script mode - see text)				

4.7 Sweeps/Regulation menu

The Sweeps/Regulation menu provides access to the dialogue boxes which control the dScope's Sweeping and Regulation functions.

Menu options are:

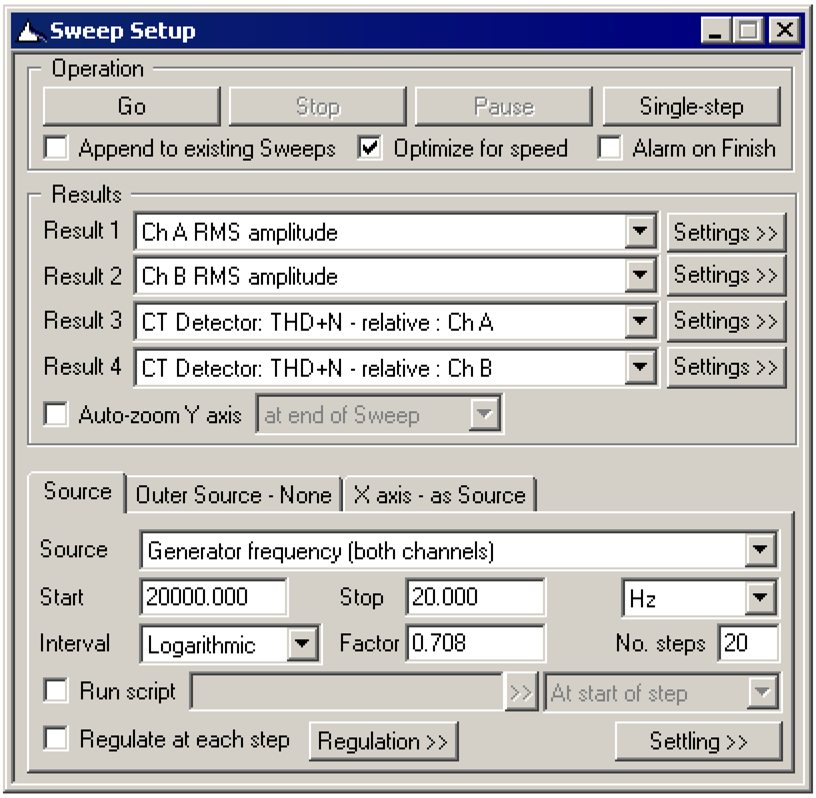
Sweep Setup	Settings for a Sweep.
Sweep Settling	Settling times and algorithms for various Result types.
Sweep Data Table Entry	Data entry utility for table-based Sweeps.
Regulation	Settings for Regulation.

4.7.1 Sweep Setup dialogue box

⚠ The dScope is capable of producing many graphical measurements which are not Sweeps; for example Scope Traces, FFT Traces, Carrier Displays etc. are nothing to do with the dScope's Sweep feature. In the dScope, Sweeps are specifically measurements where many individual scalar Results are sequentially obtained and assembled into a graphical output with the 'Sweep Source' variation shown on the X–axis. This is a common source of confusion to users familiar with other Audio Analyzers which use 'sweeps' to derive any graphical output. FFT Traces, Scope Traces etc. are very fast on the dScope because the Sweep engine is not involved.

⚠ Many measurements which are traditionally made by sweeping on other Audio Analyzers (e.g. frequency response) can be made much faster on the dScope by using other methods. For example, frequency response can be measured very fast using FFT Analysis in conjunction with the 'Bin centres' stimulus, as described in the Signal Generator section. By using multi-tone testing, a single acquisition of the multi-tone stimulus, passed through the equipment under test, can produce many simultaneous measurements (both scalar, and plots vs frequency) for both channels. See the Multi-tone Generation and Analysis section for more details.

The Sweep Setup dialogue box contains the main settings which control the dScope's Sweep function.



Sweep Operation panel

The [Go], [Stop], [Pause] and [Single–step] buttons control the progress of the dScope's Sweep function. Note that unavailable button functions are greyed and inactive. For example [Go] and [Single–step] are greyed if no Sweep is defined, or one is already in progress. [Stop] and [Pause] are greyed if no Sweep is in progress. The [Single–step] function is useful when setting up or debugging Sweeps.

Note that the [Go] and [Stop] functions are more conveniently accessed with the  and  Toolbar icons respectively.

When 'Append to existing sweeps' is not checked, each new Sweep replaces the Trace of any previous Sweeps in the Trace window. If the box is checked, successive Sweeps causes the previous live Sweep Trace to be copied and retained in the Trace window, whilst the new Sweep is added.

An audible alarm sounds at the end of each Sweep when 'Alarm when finished' is checked.

The 'Optimize for speed' check-box allows the dScope to improve the speed of Sweeps by making some assumptions based on the type of Sweep being undertaken. For example:

- If no FFT-derived Results are being swept, the FFT Trigger is then turned off for the duration of the Sweep; the pre-Sweep state of the Trigger is restored at the end of the Sweep.
- The Sweep engine takes control of the Analogue Input auto-ranging controls for the duration of the Sweep, modifying them according to the type of Sweep being undertaken; the pre-Sweep state of the auto-ranging controls is restored at the end of the Sweep.
- If any Continuous-Time Detector or Signal Analyzer Results are being swept, the Sweep engine takes control of the Signal Analyzer update rate for the duration of the Sweep, modifying it according to the type of Sweep being undertaken; the pre-Sweep state of the update rate is restored at the end of the Sweep.

If any of these actions produces undesirable results in particular circumstances, the 'Optimize for speed' box should be unchecked.

Sweep Results panel

A wide variety of Results can be swept, up to four at a time, as selected in the Result 1..4 drop-menus. Since the dScope can read Results from both Analyzer channels simultaneously, it is possible to sweep two Results from each channel simultaneously, as in the example screenshot above.

Note that any of the functions of the Continuous-Time Detector or FFT Detectors can be swept. In this case, the Detector(s) should be set up first to measure the desired function(s), after which their functions become available in the Sweep Results drop lists.

Clicking the [Settings>>] button for any of the Results opens a dialogue box in which the Y scale and Limit parameters for the resulting Sweep Traces can be set up prior to performing the Sweep. These parameters can be adjusted after (or during) the Sweep by selecting the Trace as current, and accessing the Trace settings as described in the Trace Settings dialogue box section.

Checking 'Auto-zoom Y axis' causes the Y axis to be automatically scaled to accommodate the range of the Sweep Result points. This can happen 'after each step' or 'at end of Sweep' as selected from the drop-menu.

Sweep Source tab

The Sweep source is the parameter which is to be varied, which normally forms the X-axis of the Sweep; for example a frequency response Sweep might define the Generator frequency as the source, varying it between 20Hz and 20kHz in, say, 30 linear steps.

So for most Sweep sources, the Start and Stop values must be specified (in the desired units), as must the number of steps and the linear/logarithmic nature of the desired progression. For linear progressions, an offset (step) value is specified; for logarithmic progressions, a multiplying factor. Note that the X-axis of the resulting Sweep Trace may be specified to be linear or logarithmic (in the Trace settings) independently of the Sweep source. For example, a frequency response with linearly-spaced Sweep points could nonetheless be plotted on a log scale (although the points would not be equally spaced).

If 'Run script' is checked, a script can be selected which will be automatically run between each Sweep step. This facility can be used for implementing complex manual Regulation operations, or for constructing a wide range of custom Sweeps. The script may be selected to run 'At start of step', 'After setting Source', 'After Regulation' or 'At end of step'.

If 'Regulate at each step' is checked, a Regulation is performed before each Sweep Result is recorded. This might be used, for example, to produce a Sweep such as "THD+N vs Frequency at constant power".

The Sweep Source tab also contains shortcut buttons to the Sweep Settling and Regulation dialogue boxes. These can also be opened from the Sweeps menu.

Certain 'special' Sweep source types require individual explanation:

'Sense' Sweeps may be made where the frequency or amplitude points which form the X–axis are sensed by observing the frequency or amplitude of the specified Analyzer channel. Each time a new frequency or amplitude is detected, the specified Results are plotted in the usual way. This is useful, for example, where replay characteristics of a tape or disc player are to be measured. In this case, a series of frequencies or amplitudes are recorded on a test tape or disc, with adequate duration to allow settling. On replay, the Sweep is made by sensing each frequency or amplitude, and plotting the measured Results. Note that to record such a tape or disc, a dummy Sweep can be set up, sourced from the dScope's Generator frequency or amplitude, with the settling time set to provide sufficient duration at each point. To ensure reliable operation of Sense Sweeps, a 'variation' and a 'threshold' must be specified. The variation (defined by offset or factor) is the amount by which the sensed parameter must be seen to change in order for a new point to be captured. This need only be set large enough to prevent spurious sensing of new points owing to sensing errors or variations. The threshold is an amplitude below which sensing does not occur, for example to exclude the possibility of spurious points resulting from gaps between recorded tones. An end value must also be specified to define the end of a Sense Sweep. Note that Regulation is not supported between the steps of a Sense Sweep.

'Table' Sweeps allow arbitrary source progressions (i.e. not simply linear or logarithmic). For example, a frequency response could be swept in great detail between 10Hz and 50Hz, and between 17kHz and 25kHz, without any time being wasted making measurements in between. The X–axis values for Table Sweeps are defined by either using the Sweep Data Table Entry dialogue box, or by writing a VBScript as described in the Sweep Data Tables section of the Scripting Manual.

'dS-NET Channel Array' Sweeps cause the channels of a Channel Array to be cycled through between Sweep steps. This allows a Result or Sweep to be repeated for each channel in a Channel Array. For more information on setting up and using dS-NET Switcher Channel Arrays, see the dS-NET Peripherals sub-menu.

'Manual' Sweeps have merely a numerical X–axis – a new point is acquired each time the [F3] key is pressed.

Sweep Outer Source tab

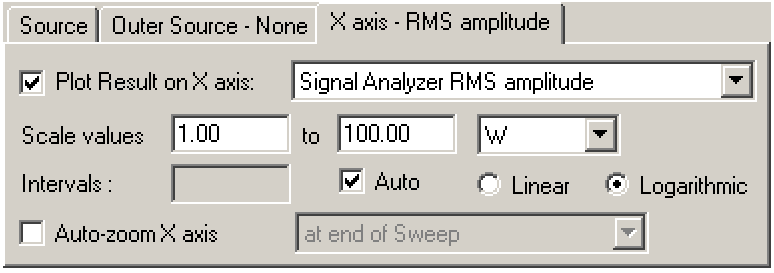
When performing basic Sweeps, the Source tab is used to define the Source of the Sweep and the Sweep Outer Source tab is set to '- None -'. Selecting an Outer Source initiates 'Nested Sweep' mode, wherein the basic Sweep is repeated for every step of the Outer Source.

The Outer Source tab is functionally identical to the Source tab, except that it can be set to '- None -'.

Sweep X axis tab

The Sweep X axis tab allows a Result rather than the Source to be plotted on the X axis of a Sweep. For example, it might be desired to plot THD+N vs output power for a power amplifier whilst varying the input amplitude to the amplifier. In such a case, it is convenient if the X axis can be plotted using the power Results measure by the Signal Analyzer.

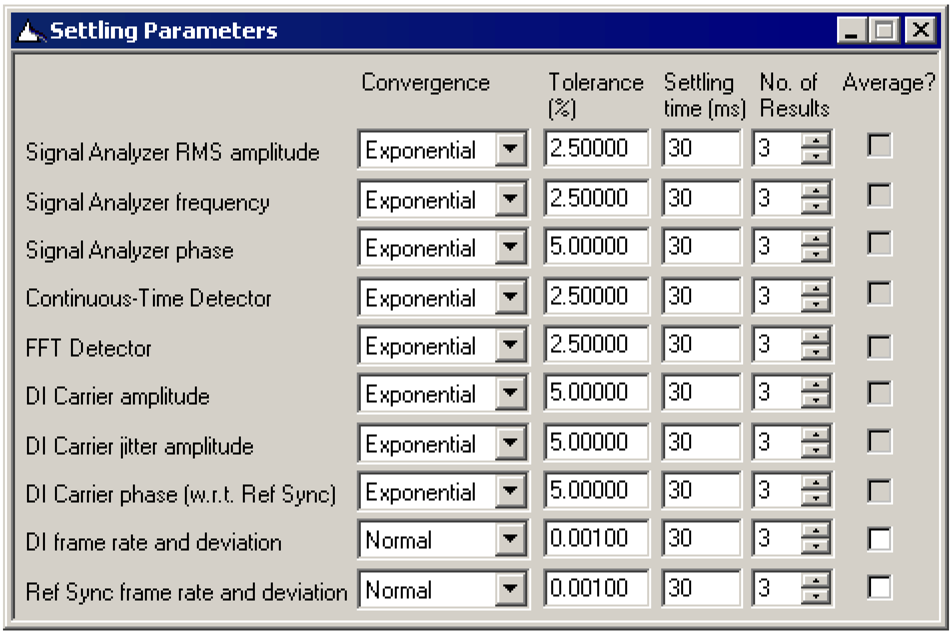
When 'Plot Result on X axis' is checked, the Result parameter, range, units and scale intervals of the X axis can be defined, along with a linear or logarithmic progression. When the box is not checked, these entries are greyed out and the Source is plotted on the X axis in the usual way.



Checking 'Auto-zoom X axis' causes the X axis to be automatically scaled to accommodate the range of the X axis values (usually only necessary when Results are placed on the X axis, or in the case of a Sense Sweep). This can happen 'after each step' or 'at end of Sweep' as selected from the drop-menu.

4.7.2 Sweep Settling dialogue box

The Sweep Settling dialogue box controls the settling times and algorithms used by the Sweep function. Note that these settings are also used by the Regulation function and, optionally, when collecting Results in a VBScript.



Different settings of each settling parameter can be set up for each type of Sweep Result. The defaults, as shown in the screenshot above, reflect likely starting values based on the accuracy and stability of each type of Result.

The settling sequence is followed each time a new point is to be captured in a Sweep (i.e. whenever the source parameter has been advanced by the dScope, or its advance has been sensed in the case of a Sense Sweep):

1. Wait for the specified settling time (to allow for delay or settling in the device under test).

2. Begin gathering Results – Results are gathered by the dScope at a nominal 32/sec, or one per

period of the incoming frequency, whichever is the longer. This allows for maximised sweep speed across high and low frequencies. Note that maximum speed is obtained if unnecessary processes (e.g. FFT triggering or drawing of Carrier Displays) are disabled during Sweeps.

3. Wait until the specified number of Results have met the specified Convergence criteria. If Convergence is set to 'None', the first n Results will suffice, whatever they are. If Convergence is set to 'Normal', the first group of n Results which are all within the specified Tolerance of the last will be deemed to have settled. If Convergence is set to 'Exponential', a convergence 'funnel' is applied where progressive convergence is quickly detected. This algorithm usually provides the best speed/accuracy trade-off: the first group of n Results where the Result(n–1) is within Tolerance of Result(n), Result(n–2) is within 2xTolerance of Result(n), etc. are deemed to have settled.

4. If Convergence is 'None' or 'Normal', and 'Average' is checked, plot the average Result of the n Results which converged, otherwise plot the last Result.

Note that when a VBScript reads a Result, it is possible to specify whether these Sweep Settling criteria are used to provide a Result to the script, or whether a single instantaneous Result is provided instead. This depends on a setting in the Options dialogue box.

⚠ Hints for maximising Sweep speeds

Here are a number of points to bear in mind when optimising the speed of dScope Sweeps:

Use the Optimize for speed setting

The Sweep Setup dialogue box has a check-box called 'Optimize for speed'. Selecting this function allows the dScope to automatically control certain operating parameters during the Sweep, including several of those detailed below. However, even if this function is not used, many of the following measures can be applied manually to improve Sweep speed.

Sweep CTD and not FFTD Results if possible

Where a Result can be measured with either Continuous-Time or an FFT Detector, use the Continuous-Time Detector -it's faster.

Turn off unnecessary background processes

Unless you are sweeping an FFT Detector Result, turn off the FFT Analyzer trigger, at least while the Sweep is in progress.

Some other 'background processes' such as gather Carrier Display data can also slow down Sweeps a little.

Disable Analogue Input Auto-ranging if it's not needed

The dScope normally tries to optimise the sensitivity of its Analogue Inputs by continually Auto-ranging. However, this can slow down Sweeps and it's surprising how often it's not necessary - set a fixed range for the Analogue Inputs instead - choose a range which slightly exceeds the maximum amplitude which will be presented to the Analogue Inputs during the Sweep.

For Sweeps of an amplitude Result, such as frequency response, Auto-ranging can always be disabled since accurate amplitude measurements don't rely on making the most of the dScope's instantaneous dynamic range. For Sweeps of residuals where amplitude is nominally constant (e.g. THD+N vs frequency) it is obviously fine to disable Auto-ranging. Even when sweeping residuals with changing amplitude (e.g. THD+N vs amplitude) it is usually possible to disable Auto-ranging since the instantaneous range of the dScope is often comfortably greater than that of the EUT. Even in the case of a very high-performance EUT with wide dynamic range, when sweeping residuals with changing amplitude, it is STILL often faster to disable Auto-ranging. In this case, set a range which is high enough to measure the residuals at the lowest amplitudes of the Sweep, even if it's too low to accommodate the higher amplitudes. dScope will Auto-range when overloaded EVEN IF AUTO-RANGING IS DISABLED, so the majority of the Sweep will be fast since no ranging occurs, but Auto-ranging will cut in automatically at high amplitudes.

Don't use unnecessary analyzer LF extension

The dScope Analogue Inputs have a flat response down to very low frequencies, unless limited by selecting a high-pass filter (for example 22Hz). The LF response of the Digital Inputs without a HPF extends to DC. Whilst this is useful for measuring LF amplitude responses of EUTs, it can slow down residual Sweeps such as THD+N, because step changes in amplitude or frequency entering the Signal Analyzer can produce disturbances in the residual measurement which may take a long time to settle - this is an inevitable consequence of the extended LF response. The solution is to make sure that a high-pass filter is selected, preferably 22Hz although 10Hz largely solves the problem. For residual measurements there is usually no point in turning the 22Hz filter off since a third-octave band reject filter, even when tuned to very low frequencies, causes significant attenuation at frequencies well above 22Hz, which is not significantly increased by the application of the 22Hz filter. If it is absolutely required to perform residual Sweeps without high-pass filtering, the Sweep settling must be extended by increasing the number of points or reducing the reading rate.

Since all CTD Results' update rates are necessarily slowed down at low frequencies, where the period of the frequency exceeds the specified update rate, don't Sweep down to lower frequencies than you need - the last few points may end up taking most of the time.

Use sensible Signal Analyzer update rate and Sweep Settling parameters

This goes without saying - if the Signal Analyzer update rate is set to be slow, then Sweeps of Signal Analyzer or CTD will be slow; if settling parameters are set too tightly, extended settling times and repeated timeouts will occur. The best advice is to leave the update rate on 'Auto', and the Sweep Settling parameters at their defaults.

Sweep direction

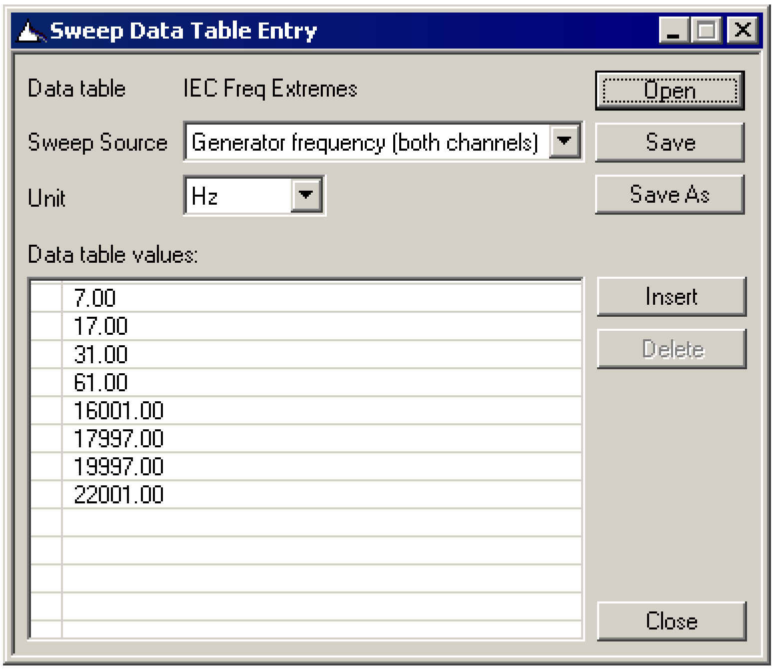
The direction of a Sweep can sometimes make a difference to Sweep speed. For example, if the dScope's Analogue Input Auto-ranging is active, it's usually better to Sweep downwards in amplitude - the reason is simple: if ranging is caused by the signal amplitude getting too low, the dScope knows which is the best range to choose; if the signal amplitude gets too high, the dScope only knows that it's too high and must apply a little experimentation to find the right range.

Consider using multi-tone testing instead

Many measurements which traditionally require Sweeps against frequency can be better performed using a multi-tone method, since a single stimulus and measurement provides every frequency point at the same time. If many different parameters are to be measured against frequency, e.g. gain, distortion, cross-talk etc. then multi-tones will be MUCH faster since these can all be calculated from the same single stimulus and capture! For more information, see the Multi-tone Generation and Analysis section

4.7.3 Sweep Data Table Entry dialogue box

The Sweep Data Table Entry dialogue box is a simple way to enter Sweep data tables. It is also possible to generate Sweep data tables using VBScripts if desired. Table-based Sweeps can be a useful way of speeding up Sweeps by only collecting specific relevant data points, rather than a set of points which are linearly or logarithmically spaced. The pictured example shows a data table for a frequency response Sweep concentrated at the ends of the audio band.

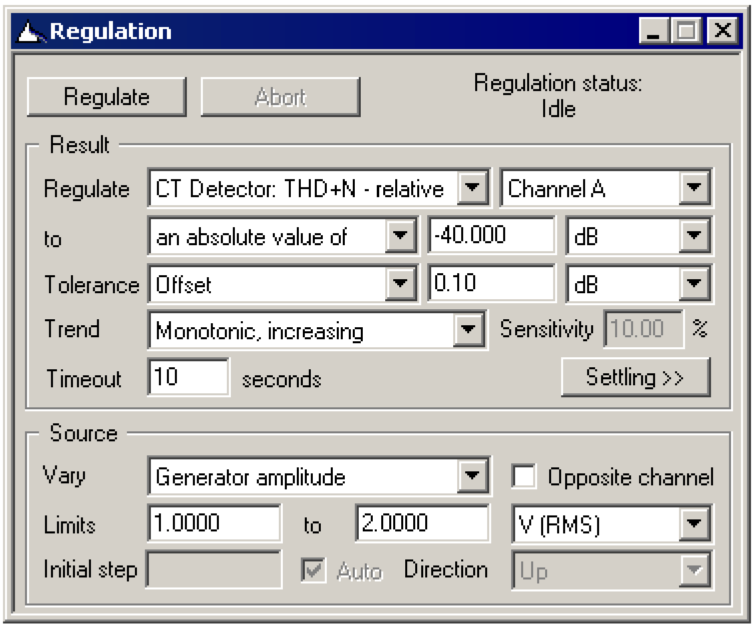


To create a new data table, it is only necessary to specify the desired Sweep source, its units, and a list of values. This can then be saved to an appropriate filename and loaded using the Sweep Setup dialogue box. Existing tables can be easily loaded, edited and resaved.

4.7.4 Regulation dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The Regulation dialogue box controls the algorithms of the dScope's Regulation function.



Regulation is the process by which a nominated Result is continuously collected from the EUT, whilst some nominated parameter of the dScope's Signal Generator is varied progressively until the Result falls within some tolerance of a nominated target value.

The Result panel allows entry of the nominated Result, target, tolerance and alternative algorithms, whilst the Source panel is for entry of the varying parameter and its allowed range.

The [Regulate] button causes Regulation to take place, and the [Abort] button aborts a running regulation which has not yet settled or timed out.

Regulation is commonly applied between the steps of a Sweep, as described in the Sweep Setup dialogue box section. An example of this might be to produce a Sweep such as "THD+N vs Frequency at constant power".

The sections below briefly describe the function of the various settings within the Regulation dialogue box. Note that regulation operation also takes note of the settings of the Sweep Settling dialogue box. Optimising regulation for speed and accuracy requires a detailed understanding of the principles and algorithms employed (as well as a degree of trial and error). Refer to the Principles of Regulation section for an in-depth discussion.

Regulation Result panel

Regulation can be applied to the Signal Analyzer's amplitude, frequency and inter-channel phase Results, as well as to the Results of the Continuous-Time Detector and any existing FFT Detectors. The drop-menu is context-dependent, so it is necessary to set up any desired Detector prior to completing the Regulation Result panel.

The selected Result can be regulated to an absolute target value, or to its maximum or minimum value over the range of the nominated Source. If a maximum or minimum target is selected, regulation can be applied to the 'A channel', 'B channel', 'Selected channel', 'Non-selected channel' or 'Both channels' as selected from the drop-menu. If the target is an absolute value, it is also possible to regulate 'Both channels (tied)' or 'Either Channel'. The absolute target can be entered in any desired units. Absolute regulation also allows specification of whether the relationship between the Source and the Result is 'Monotonic, increasing', 'Monotonic, decreasing', 'Monotonic, direction unknown' or 'Non-monotonic'. Monotonic means that the Result moves in the same direction as the Source is increased over its whole range. Regulation of monotonically related Results is usually faster if the relationship can be specified. A maximum or minimum target requires the specification of a 'Sensitivity' parameter which governs how sensitive the regulation algorithm is to local maxima or minima.

For both absolute and maximum/minimum regulations, a tolerance must be set which determines how close to the target the regulation tries to get before it completes. For absolute targets, this can be specified as a ratio or an absolute offset; for a maximum or minimum target the tolerance must be set as a ratio.

Finally, a timeout parameter determines a failsafe time after which regulation is concluded even if the target has not been achieved to the specified tolerance. In such cases, the Source is left at its last setting during the attempted regulation when regulation is timed out.

Regulation Source panel

The regulation Source can be selected from the Signal Generator's amplitude or frequency, or the Digital Output's DC offset, jitter amplitude or jitter frequency. The limits over which the dScope is allowed to vary the Source in its attempts to achieve regulation must also be specified.

The 'Opposite channel' check-box allows the Result to be regulated by the Source of the opposite channel during two-channel regulation, for example when regulating a cross-talk related Result.

For maximum or minimum regulation, the initial step size and direction of the Source can be specified to over-ride the automatic values if desired. This can be used to improve the speed and accuracy of such regulations.

4.7.4.1 Principles of Regulation

This section explains the operation of the dScope's Regulation feature, and is intended to aid the user in selecting appropriate algorithms and tolerances for optimal Regulation in any situation. A summary of the terminology used in describing the regulation action is provided at the end of the section.

What can be regulated?

The following Result types can currently be regulated: Signal Analyzer amplitude, frequency and phase Results; Continuous-Time and FFT Detector Results.

Regulation can be performed on a single channel or on A and B channels simultaneously.

In the case of a two-channel regulation, both channels' regulation Result is of the same type (e.g. RMS Amplitude or THD). The channels will share a single stimulus. For each regulation step a pair of measurement Points will be found, one per channel, assuming the same Source value for both. An example is finding input amplitude (common to both channels) where both channels of an amplifier show a THD of 1%, within a certain tolerance. This kind of measurement is well suited for regulating a stereo device, is faster and has a different meaning then performing two separate regulations, first for the Channel A and then for the Channel B.

Phase is measured between channels A and B but is considered as a single-channel measurement (because there is only one result to measure).

Types of regulation

The Absolute regulation algorithm tries to get the final Result within the specified Tolerance of the Goal Result. Goal is an absolute magnitude, such as amplitude in Volts or THD in %.

The Peak regulation algorithm tries to locate the maximum or minimum value of the Result Function by varying the Source within its Range.

Source Range

The user specifies the range of values that the Source can take. During regulation, the stimulus will never go outside these limits. For the Peak algorithm, if the maximum (or minimum) result lies exactly at one of the extremes of the Range, this point will be found (hopefully) and returned as the final Answer even if the Function continues increasing (or decreasing) beyond the Range.

If the current stimulus value lies outside of the Range before a regulation starts, the first step of the regulation will always be to bring the stimulus into Range. The first step's Source will be set to the Range's extreme that lies closest to the current stimulus.

Otherwise, regulation will always start at the current point. The selection of the initial point can be important for the outcome of regulation.

Regulation Goal and Tolerance

Ideally, regulation should stop when the result is found exactly, that is for Absolute algorithm the Result is equal to the Goal Result and for the Peak algorithm the result Point lies at the true maximum or minimum of the function. In practice, the exactness of the result is limited not only by noise, drift, and system resolution but also by the time available to reach the answer.

Tolerance is simply a way to terminate the regulation before it has achieved its best result, in order to get a compromised result earlier. Tolerance can be set to zero if the user is interested in the best possible answer at the cost of slowness. In this case, regulation will proceed until the Step Size goes down to the system resolution.

In any case the regulation's duration is also limited by a time out value. If regulation fails for any reason the current stimulus will remain at the value set by the last regulation step.

Absolute mode

For Absolute regulation the user can specify the goal's Y coordinate; i.e. the Goal Result value. If the Function crosses this level at various points within Range, there may be more then one Goal point for this result. The algorithm succeeds if it finds any of these points (however, regulation parameters can be set to help locating a specific answer). Tolerance is defined as the maximum deviation from this goal for an answer to be considered satisfactory.

For a single channel, Absolute regulation is successful when the result lies within the Tolerance from the Goal Result. For example, if Goal is 1V and Tolerance is 1% then the Result should lie between 0.99 and 1.01V. Note that if the Function curve is very horizontal at the Goal point, the Answer's Source will not be well defined. In this case a tighter tolerance should be used. If the Goal Result is 0.0 (in the current unit) and Tolerance is expressed in percent, the resulting error margin is null. This is equivalent to having a zero tolerance.

For two-channel absolute regulation, the user can specify whether the Goal must be met for both channels or for either channel. If 'Both Channels' is specified then both channels must fall within the Tolerance margin from the common goal. In the course of regulation, the algorithm will take the larger channel's deviation from the goal in order to make a guess for the next step. If 'Either Channel' is selected, it is enough that one of the channels meets the Tolerance. The algorithm will take the smaller of the deviations for its calculations.

Note that if Tolerance is tight (a low value), and the channels differ considerably, it will be more difficult to find a point where both channels are within tolerance. Also note that because at each step the algorithm will take the larger (or smaller) of the deviations, and for each step this may correspond to a different channel, the resulting data uncertainty will increase (as in the case of excessive noise) and in extreme cases the algorithm may fail.

Peak mode

For Peak regulation, both X and Y coordinates of the Goal point unknown at the start since they are both to be found by the Regulator. It's best to think of the Goal's X coordinate as the answer: we're looking for the Source value for which the Result is maximum or minimum.

If Tolerance is non-zero, regulation is considered successful when the last three Points used to locate the peak are within the specified Tolerance of each other (see the description of the Peak algorithm below). The criterion used is the same as for Settling Tolerance. Of the last three measurement points, one point will be considered the "true" peak. If the remaining two points are within the Tolerance from the "true" peak, expressed as percentage from the peak value, then the regulation is finished with success. For example, if the results of the last three points during a Maximum regulation are 0.99, 1.0 and 0.995 then if Tolerance is 1% the regulation finishes with a result of 1.0.
Note that if the Function curve is very flat, so the peak is not well pronounced, applying a relaxed Tolerance (larger value) may result in the Source of the answer point being not well defined.

As with the Absolute mode, the user may set Tolerance to zero. In this case the Peak algorithm will narrow the answer using a smaller and smaller step until it gets down to the system resolution. At this point it will declare success.

Absolute mode algorithms

Finding an absolute Goal Result within a tolerance for an arbitrary Function can be always done with a "brute force" approach. A brute force algorithm would use a very fine step size to scan the function over the entire source range, and for each step it would measure the Result and apply tolerance to determine whether the Goal was hit.

For reasons of speed, a more optimised approach is also needed. In dScope two specialized absolute mode algorithms can be used to find an answer much more quickly.

The first one, the Stepping Algorithm, does not make any assumptions about the nature of the Function. This is a variation of the "brute force" method with a variable step size. The second algorithm, Monotonic, takes advantage of additional information provided by the user about the Function. If the Function is known to be monotonic (always increasing or always decreasing), including a linear ramp case, the answer can be found much faster.

Stepping Algorithm

The Stepping Algorithm will try to roughly locate the goal point by stepping through the Function with a big step size and then using finer and finer steps in order to increase accuracy. The Stepping Algorithm is applied when the Result in the Regulation dialog box is an absolute value and the Trend is set to Non-monotonic. The user can specify the initial direction of stepping (Up, Down or Unknown), the initial step size (as a value or Automatic) and time out value in seconds.

If the initial stepping direction is set to Unknown the algorithm will make a guess based on the relation between the current Result and the Goal Result, assuming that the Function is rising from the current point. If the step size is set to Auto, the algorithm will divide the distance from the current Source to the Range limit, in the direction of the first step, by four and use this value.

The Stepping Algorithm will start at the current point, then it will advance the Source, step by step, using the initial step size. If the Result crosses the Goal (falls below or goes above, depending on whether the starting value lay above or below the goal level) the step size is halved and the direction is reversed.

Regulation will succeed when the result is within the Tolerance from the Goal Result or, if Tolerance is zero, the step size is reduced to the smallest possible value (system resolution).

The regulation will fail if Tolerance is non-zero and the step size is reduced down to the system resolution, the Range limit is reached without the Result crossing the Goal or the time out period elapses.

The actual algorithm used in dScope uses a few optimisations in addition to the basic behaviour. First, to reduce the number of times that the same point is revisited, if the next step after reversing the direction does not cross the goal level, the step size is further halved. Second, if after stepping with the initial fixed step size the goal level was not crossed and the Range limit is reached, the algorithm will go back to the initial starting point and try to scan in the opposite direction. This is especially useful if the initial step direction was set to Auto and the program made a wrong guess.

Observations

The Stepping algorithm performs well if it can quickly locate a crossing point between the Goal level and a line drawn between the current point and the previous one. A crossing is detected when the deviation from the Goal changes sign from step to step. Once a crossing point is found the algorithm is reasonably fast because it uses a concept similar to binary search.

If the Function is relatively smooth, the crossing point can be located easily even with a very coarse initial step size. For this reason the Auto step size divides the travel distance only by four. A finer initial step increases the probability of finding the Result but at the expense of using more steps.

A problem for the Stepping algorithm occurs when the crossing point is never detected, even if the Goal point does exist on the curve. This can happen when the Function has a relatively narrow peak (like a resonance curve) and the Goal point is located on the slope, near the peak point. If the initial step size is not small enough the algorithm may skip the peak point, and never register any Result that is higher than the Goal Result. The outcome depends heavily on the step size and also on the location of the starting point, which determines the relative phase between the sampling points and the function peak. If this situation occurs, the user should use a finer initial step size and/or try to shift the starting point by less then the current step size.

Monotonic Algorithm

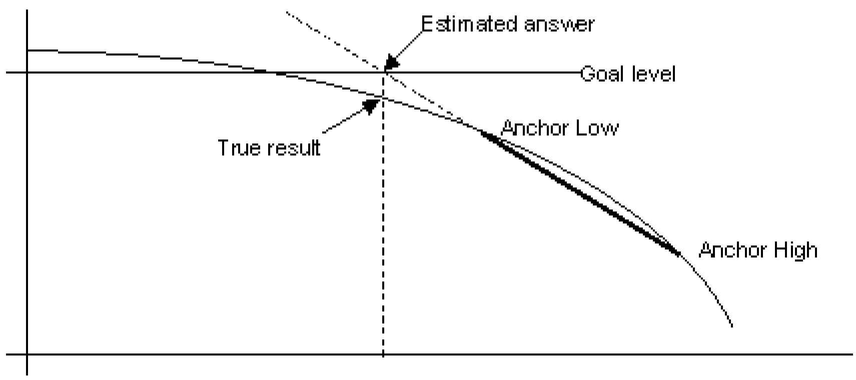
If the Function is known to be either always increasing or always decreasing, a possibility arises to reduce the searching time considerably. In the simplest case, the Function is just a linear ramp starting from the point 0,0. A typical case is the gain curve of a linear operational amplifier. Assuming the function's linearity, provided that the current point does not lie at 0,0, in theory the answer can be found in just one step by calculating the gain and extrapolating. The Monotonic Algorithm extends this concept to any monotonic function, not necessarily linear.

To use this algorithm the user should select one of the Monotonic trends in the Regulation dialog box. Options are 'Monotonic Increasing', 'Monotonic Decreasing' and 'Monotonic, direction unknown'. There are no parameters specific to the algorithm, other then the usual tolerance and goal values.

The idea behind the Monotonic algorithm is to try to approximate the real Function with a number of straight-line segments. Once a segment is worked out then linear interpolation or extrapolation is used from the segment in an attempt to locate the crossing point with the Goal Result level. The two end points of an approximation segment are called Anchor Low and Anchor High. If the segment intersects the Goal Result level then Anchor Low is located below the goal level and Anchor High is above. The algorithm can remain in one of two phases. The Tangential phase is used when there is no crossing point between Anchors. Otherwise, the Intersecting phase is in effect. The Monotonic Algorithm can freely switch between the two phases, based on the outcome of the previous step.

Tangential phase

In the Tangential phase the goal is assumed to lie on the straight line extending the current segment, at the point of intersection with the Goal Result level. If the segment is below the goal level then its upper endpoint is Anchor Low and the lower one is Anchor High. Otherwise, if the segment is above the goal line then the lower endpoint is Anchor High and the other one is Anchor Low.

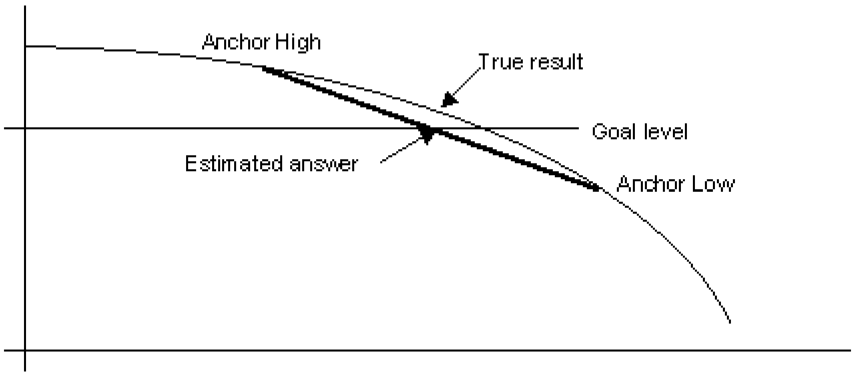


Tangential phase of the Monotonic Algorithm

After calculating the estimated answer position the next step is performed to actually set the Source at the point. Then, when the new Current Point settles the true Result is read. If the true Result does not meet the Tolerance and it did not cross the Goal level the procedure is repeated, remaining in the Tangential phase.

Intersecting phase

This phase is entered when as a result of the last step the Current Point has crossed the Goal Result level, that is, the deviation changed sign. At this stage, the Anchor points are ordered so the Anchor High lies above the goal and Anchor Low lies below it.

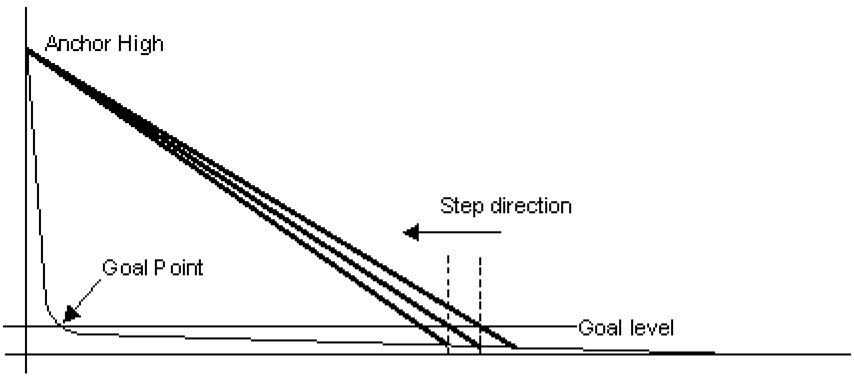


Intersecting phase of the Monotonic Algorithm

In the Intersecting phase the goal lies between the Anchor points. The estimated answer is calculated using linear interpolation. Then, in the next step, the real Result is obtained and one of the Anchor points is updated by moving it onto the Current Point. If the result does not meet the Tolerance and the goal level still lies between the Anchors the procedure repeats. Otherwise, the algorithm switches back to the Tangential phase.

The Monotonic Algorithm usually works very fast but it may sometimes not converge or converge very slowly. This can happen if the curve is locally non-monotonic or is very non-linear over the entire range. Also, excessive noise can destabilise the algorithm. Because step size can as well decrease or increase from step to step, noise can cause the algorithm to wander until timeout, without ever settling.

A slow convergence can occur if the curve has two regions: a relatively low-level flat area and a near-vertical region. For example, if the Goal level lies slightly above the flatness region and the Anchor High point is located high at the steep portion of the curve the Intersecting phase could advance in very tiny steps for a long time.



Slowly converging Intersection Phase

To deal with the slow convergence case, the intersecting phase algorithm detects whether the next step did not advance significantly towards the Goal. If this is the case, instead of using interpolation the next step is located halfway between the Anchors to cut distance rapidly. Of course, it may happen that this move was an overshoot, but even in that case the algorithm will simply adjust the anchors and continue.

Estimating the first step

The Monotonic algorithm operates on straight-line segments and therefore it needs to know two points on the Function to evaluate the next step. However, at the start, only one point is known – the initial Current Point. The algorithm needs to make a guess about the location of another point on the curve.

Usually this guess is of little importance. As soon as the algorithm gets two measurement points it can very quickly interpolate the final Answer. However, selecting a good second point can save one step. To help in the decision, the user can select either Monotonic Increasing or Monotonic Decreasing trend for the Result. If the direction of the slope is known, the algorithm can evaluate the direction of the first step based on in which quadrant the Current Point is and the distance between the Current Result and the Goal Result.

Based on the slope information and the location of the current point, the algorithm may make further assumptions that speed up processing in many cases. For example, if the function is know to be increasing and the current point lies in the first or third quadrant (both Source and Result are positive or both are negative), the algorithm assumes that the function might be linear and passing through the 0,0 point. In many cases this results in getting the answer in just one step.

If it cannot be assumed that the Function passes through the 0,0 point or if the initial Current Point lies on one of the axis (one or both of its coordinates is 0), the algorithm must make a guess about the direction and arbitrarily select an appropriate step size for the first move. After evaluating the direction, the initial step size is then calculated to be half of the distance from the current point to one of the Source Range limits.

If the user selects a Monotonic trend with direction unknown the algorithm assumes the best case and tries to use linear extrapolation for the first step.

Observations

If the Function is truly monotonic the algorithm proves to be very efficient in most typical cases. Usually the answer is found in one to four steps. However, even a small non-monotonic region may cause the algorithm to fail, but only if it was not lucky enough to skip this region when stepping.

Since the Monotonic algorithm is by far the fastest and usually also the most exact one, the user is encouraged to try to locate monotonic regions in the Function and set the regulation limits accordingly in order to be able to use the Monotonic algorithm wherever possible.

Noise can also act to produce momentary non-monotonic regions in the otherwise perfectly monotonic function. This depends on the amount of noise but also on the inclination of the function's slope. More vertical curves are more immune to noise. The user can try to alleviate the problem by selecting a larger 'Number of Results' in the Sweep Settling dialogue box and by using the Averaging option.

The Peak algorithm

Finding a maximum or minimum value of an unknown function over a specific range can always be done by using a "brute force" approach, as in the case of Absolute regulation. Obviously, a more intelligent method is needed in order to reduce the number of necessary steps.

For any non-brute algorithm, a problem appears: If the entire range is not scanned with the finest possible step, there is a chance that a small local peak is found instead of the biggest one. The Peak algorithm used in dScope tries to address both the issues of efficiency and local peaks.

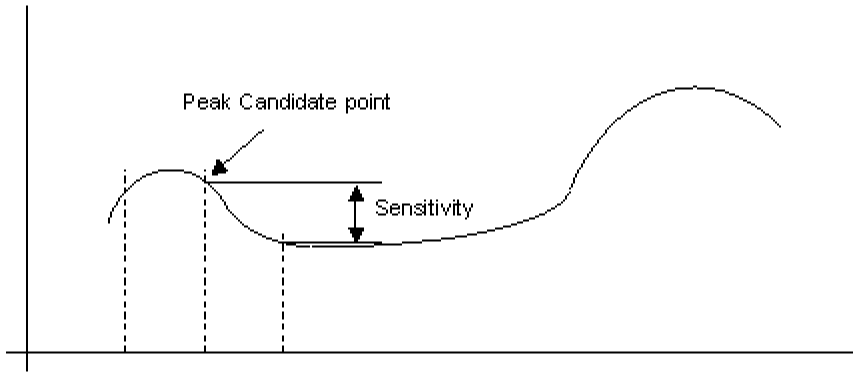
To invoke the Peak algorithm, the user should select the 'Maximum Value' or 'Minimum Value' option in the Regulation dialog box. There are additional parameters which help to find the best answer in the shortest time. The results of the regulation are sensitive to the values of the parameters and to the location of the starting point.

The 'Initial Step Direction' option lets the user select the direction in which the Function will be scanned. The algorithm will only look in this direction and will therefore ignore any possible answers lying in the Range on the opposite side of the starting point. This is important because it lets the user decide which peak should be located if there are several candidates on the curve. The Unknown step direction is included only for completeness (and for inexperienced users) and it is equivalent to the

'Up' direction.

The 'Initial Step Size' determines the step size for the first phase of the algorithm, which scans the function for the best peak candidate using a coarse step. If 'Automatic' is selected, the calculated step size will be one tenth of the distance to travel. This factor is arbitrary and can be adjusted with experience.

The 'Sensitivity' parameter controls the minimum size for a peak (or a bump in the curve) to be considered as the focus for the peak search algorithm. It's an important optimisation parameter. This is a self-relative ratio, expressed in percent, which tells by what amount the Result must extend above the "floor" to gain the focus. The parameter's value is relative to the currently found coarse peak point, called a Peak Candidate.



The Sensitivity parameter

The Peak Algorithm has two phases. In the first phase the Function is scanned using the fixed initial step size, in search for a "bump" of big enough proportions. Once a peak candidate is found, the next phase starts in order to locate the most exact location of the peak.
The following description assumes that a Maximum is sought. The Minimum case is a mirror image of the Maximum case, except where stated otherwise.

The Bump Scanning phase

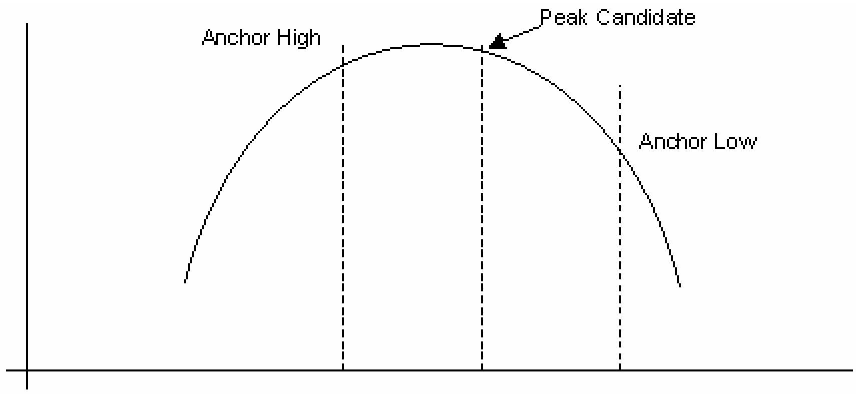
This phase starts from the initial point. Using the initial step size, either user-specified or calculated, it advances in the specified direction step by step. For each step the Current Point result is examined to determine whether or not it has beaten the current Peak record. If the current point becomes a new Peak Candidate the algorithm continues scanning. If the current point falls below the highest watermark seen so far, the Sensitivity parameter is applied to see whether the curve dropped sufficiently to consider the current Peak Candidate as big enough. To do so, the current highest result is multiplied by the Sensitivity (converted from percent to ratio), thus producing a sensitivity limit line. If the point to be tested lies below this limit the bump is considered high enough and the second phase is entered. Otherwise the bump is considered too low and the scanning phase continues. If the Range limit is reached without finding a decent bump then the second phase is entered anyway, with the current Peak Candidate considered the winner.

The difference between the Maximum and Minimum algorithms is that in the latter, to determine the Sensitivity limit the Result value of the current Peak Candidate is divided by the Sensitivity parameter instead of being multiplied. This implies that the regulated Function lies above the X-axis (is always positive).

The Peak Location phase

When this phase is entered, the current Peak Candidate point is considered to lie somewhere on a slope of the bump that should be examined to find its true Peak point. Depending on the Sensitivity parameter and the initial step size, this may or may not be the true maximum over the entire Range.

At entry, the previous Bump Scanning phase has provided three measurement points: the Peak Candidate and its two closest neighbour points, called the Anchor High and Anchor Low. The Anchors are always ordered, so the Anchor High is closer to the Peak Candidate.



Peak Location phase starting conditions

Each Peak Location algorithm step starts by assessing whether the three measurement points lie so close to each other (i.e. the peak is so flat) that further efforts would not improve the result significantly. To do this, the Tolerance is applied to the Anchor Low point (which is guaranteed to lie the farthest from the peak). If the Result value for Anchor Low is within the specified Tolerance of the current Peak Candidate then the algorithm finishes by returning the current Peak Candidate as the overall winner. Otherwise it continues.

Tolerance checking is skipped if the Tolerance value is set to zero, wherein the user wishes to get the most exact answer possible. Also Tolerance is omitted during the first step of the Peak Location algorithm, i.e. at least one narrowing step is performed even if the three points are within tolerance immediately after finishing the first Bump Scanning phase.

At each step, if tolerance checking does not produce a conclusion, the algorithm proceeds to narrowing the distance between points in order to locate the peak more precisely. To do this it locates new sampling points mid way between the current Peak Candidate and the anchors.
The first sampling point is put in between Anchor High and the Peak Candidate. If the new point becomes a new Peak then the old Peak Candidate becomes Anchor High. Anchors are always sorted so the Anchor High is always the closest to the Peak. Only if the new point located between the Anchor High and the Peak Candidate did not beat the watermark is a second sampling point tried between the Peak Candidate and Anchor Low. Similarly, if that new point becomes a new Peak Candidate then the current Peak becomes Anchor High.

At this stage, if the step size decreases down to the system resolution the algorithm finishes with success, returning the current Peak Candidate as the winner.

Observations

The Peak Algorithm usually performs well but sometimes it may be tricky to set the optimum parameters. Also, the chances are that if the initial step size is comparable to (or larger than) the width of the peak to be found, and the current location point results in an unfavourable sampling phase (the location of the sampling points relative to the location of the peak sought) then the algorithm may end too quickly with a wrong result. This is a similar problem to 'aliasing' in any sampled system.

For example, it may happen that the sampling phase and step size cause the sampling to skip the true peak altogether, return a poor Peak Candidate and the Anchor points already lying within tolerance. Although it is counterintuitive, relaxing the Tolerance in this case (setting it to a higher value) would only make things worse. If this happens the user should decrease the step size and/or slightly shift the initial starting point. in such cases, tweaking the Tolerance and Sensitivity is of no help and the only remedy is to decrease the step size or move to a better starting point. In general, the initial step size should be considerably smaller then the expected width of the peak.

The Sensitivity parameter plays an important role when the step size is correct (that is, not too small). The smaller the Sensitivity value the more undulations on the curve (including noise) are filtered out when searching for the best Peak candidate. If Sensitivity is set to zero the algorithm will always scan the whole way from the starting point to the end of Range and find the highest peak, as allowed by the step size and sampling phase (note that, as already explained, this may or may not be the true Peak). Conversely, when the Sensitivity is 100% the scanning will stop at the first sample that is found below the current Peak (or above, for the Minimum case). This may be sometimes caused by noise, if the curve is flat and noisy. Note that the Sensitivity parameter only applies to the first phase – the Bump Scanning. It has no meaning in the second phase. So, if the bump is wide and has undulations on top of it, the Sensitivity setting does not filter out these undulations.

The Bump Scanning phase scales the height of the peak using its relative distance from the X-axis. Because the measurement is self-relative using a ratio, then if the curve lies close to the axis (the values of the Function are small) the bumps must be small (in absolute units) to be filtered out. If the Function has high values then the bumps should be proportionally bigger to have the same effect on the algorithm. This can manifest as unexpected failures (spurious settling on tiny local peaks) if the Function has a long region where its value is very close to zero. Near zero, even a small bump or additive noise looks big compared to the local "floor". The remedy in this case is to decrease the Sensitivity, possibly even to zero.

There is also another implication of the method used for bump filtering: the function must not cross the X-axis. If it does, then assessing the height of a bump very close to the zero level would be very problematic. If it happens that the current Peak Candidate is right at the zero level then the notion of self-relative sensitivity expressed as a ratio becomes meaningless.

The Peak Algorithm takes this into account so when the value of a sample is zero a special case is created. If Sensitivity is also zero the algorithm continues scanning, but if it is non-zero then it stops. The algorithm cannot currently deal with negative or bipolar functions.

Currently, all Result values that can be regulated in dScope are positive, except Inter-channel Phase. For Phase regulation, the algorithm adds 180 to the measured value to ensure that it is always positive. This has no impact on the Answer.

Additional notes

Any Regulation algorithm will fail if the input data it receives as it changes the stimulus does not reflect the true behaviour of the EUT. This means that the Settling parameters used for regulation must be correctly set.

One common problem for Regulation (as well as for Sweeps) is the latency of the Result. When the Regulation algorithm issues a command to change an stimulus, it also starts a settling detector to monitor the input and determine when the results are settled and reliable. However, some time passes before the EUT's output reacts to the new stimulus. In case of digital devices this can be quite a long time. The total amount of time between changing the stimulus and observing the changed Result is called Result Latency. The user can specify an appropriate Settling Delay time in the Sweep Settling dialog box. This time should be guaranteed to be larger then the Result Latency otherwise the regulation algorithms will certainly fail.

Terminology

Result: the value to be regulated. Usually, the output of an EUT. Not to be confused with the Answer.
Source (A.K.A. Stimulus): the value to vary in order to change the result. Usually, an input to the EUT.
Point (A.K.A Sample): a single measurement. A Point's X coordinate is a Source value and the Y coordinate is the Result measured for this stimulus, after the value has settled.
Result Function (A.K.A Function or Curve): the relationship between Source and Result – a characteristic curve of the EUT that is examined (sampled) during regulation. Can be seen as a collection of Points.

Range (A.K.A Input Limits): the range of values that can be taken by the Source.
Goal Point (A.K.A. Target): the ideal Point that should be obtained through regulation. The point has a Goal Result (Y) and Goal Source (X).
Answer Point: the actual result of the finished regulation; the point found as the final answer. There are Answer Result (Y) and Answer Source (X).
Tolerance: the maximum acceptable deviation of an Answer Result from the Goal Result. If regulating to Peak, Tolerance is defined as a measure of convergence between the last three Points used to locate the peak (see below).
Regulation Step: a single step of a regulation algorithm. For each Step, the regulator produces the Current Point. The regulation process is a sequence of Steps.
Step Size: the distance between two most recently used Source values (or X coordinates of the two last Points).

4.8 Automation menu

The Automation menu provides access to the dialogue boxes which control the dScope's extensive range of automation and scripting features.

Menu options are:

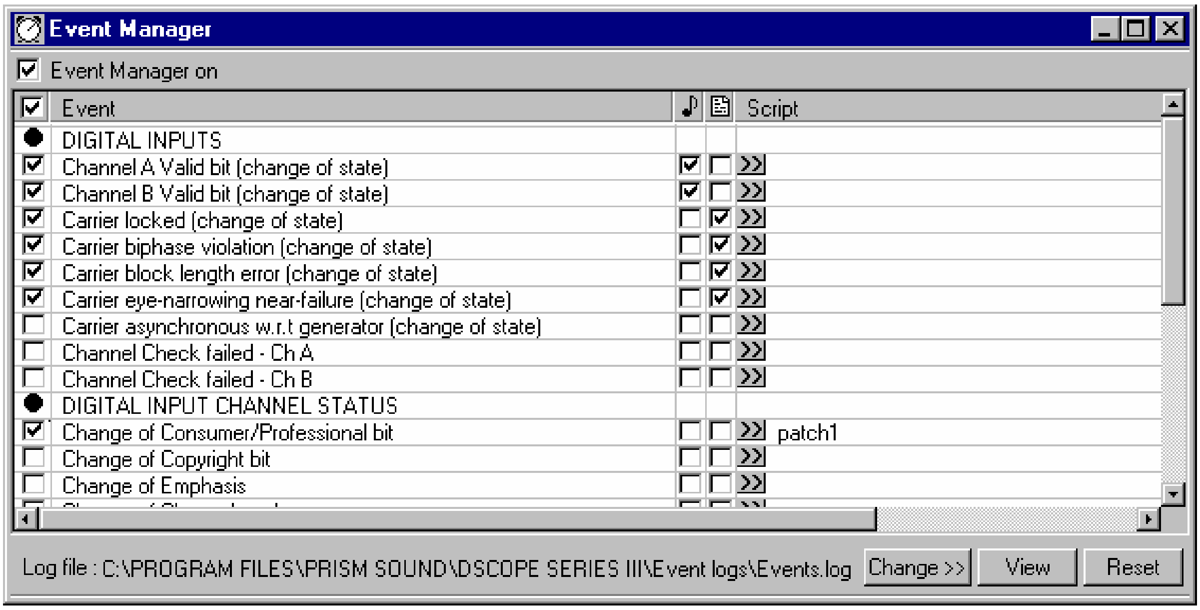
Run script	Runs an Automation script.
Stop script	Stops a currently-running Automation script.
Edit script...	Opens an editing window for dScope scripts.
Record script	Begins recording an Automation script from user-interface actions. (NB: not yet supported).
Event Manager...	Settings for the Event Manager, which links various actions to dScope Events.
View Event Log File...	Displays the contents of the Event Log File.
Reset Event Log File...	Clears the contents of the Event Log File.

For more information about VBScripting and other ways of automating dScope, see the Scripting Manual.

4.8.1 Event Manager dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The Event Manager dialogue box allows the user to specify various 'Effects' to take place in response to various causal 'Events' which may occur within the dScope. The Event Manager generally deals with advanced operational modes, and should not need to be used for normal operation.



The Event Manager is enabled and disabled with a checkbox at the top of the dialogue box.

Individual Events are enabled by checking the appropriate box on the left of the Event descriptions.

The simplest (and most commonly used) Effects are the sounding of an alarm beep, or the logging of the Event to the dScope Events log file. These are enabled in the first two columns of Effect check boxes.

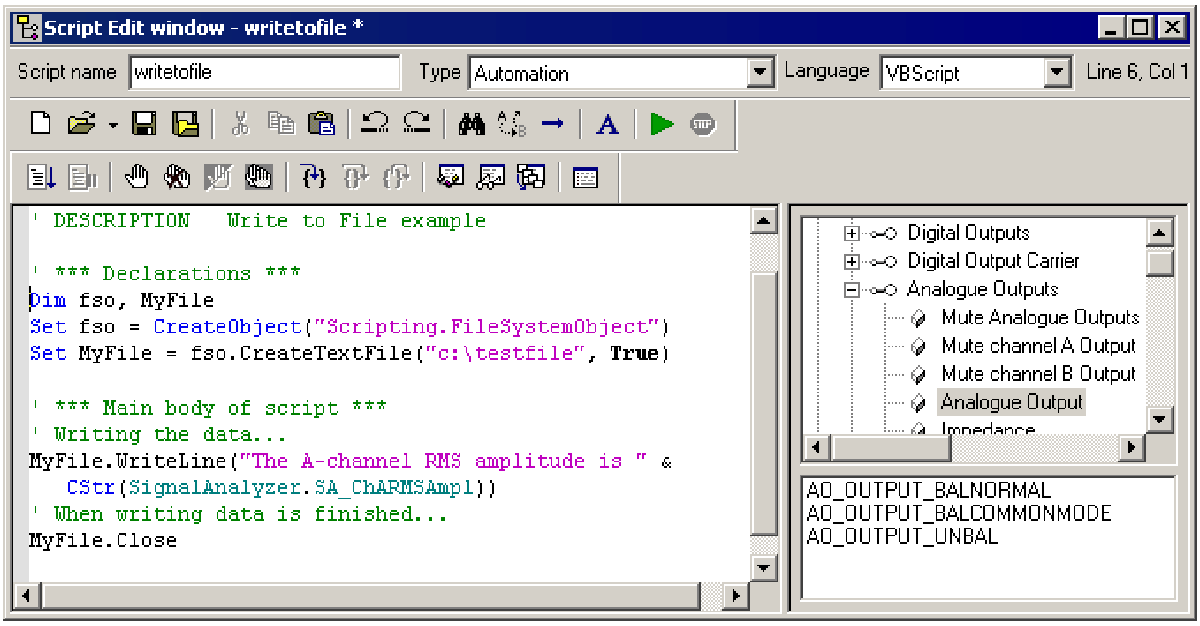
More complex causal links can be brought about by causing VBScripts to be run in response to Events. These are entered in the right-hand column. This feature might be used, for example, to turn off the FFT trigger after acquiring an FFT Trace which breaches a Limit Line.

At the bottom of the Event Manager dialogue box, the name of the Event log file may be specified, and the log file viewed or cleared.

⚠ **Some Events types are not listed in the Event Manager table unless they are already configured in the dScope. For example, breaches of Result Limits or Trace Limit Lines do not appear in the table unless the appropriate Limit has been applied.**

4.8.2 Script Edit window

The Script Edit window can be used to edit and test any of the various scripts which can be used within the dScope. Since all dScope scripts are simple text files, they can also be edited with any other preferred text editor.



Full details of the operation of the Script Edit window can be found in the Script Edit window section of the Scripting Manual.

For more information on scripting in general, see the Scripting and OLE Automation section of the Scripting Manual.

4.9 Utility menu

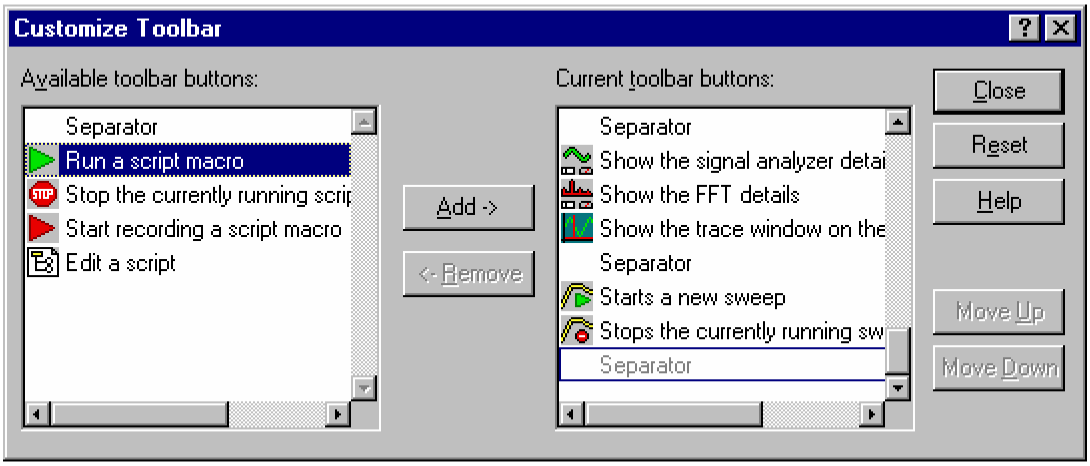
The Utility menu provides access to various miscellaneous functions within the dScope.

Menu options are:

Customize Toolbar...	Sets up the Toolbar to the user's requirements.
Customize User bar...	Sets up scripts and Configurations to appear on the User bar.
Multi-tone Generation and Analysis...	Automatically configures dScope for multi-tone generation and analysis.
Memos	Accesses the Memo sub-menu
Print Display	Prints the current dScope screen.
Save Display as Bitmap	Saves the current dScope screen as a bitmap.
dS-NET Peripherals	Accesses dS-NET Peripheral sub-menu
Options...	Sets various miscellaneous options.

4.9.1 Customize Toolbar dialogue box

The Customize Toolbar dialogue box is used to select which icons appear on the dScope's Toolbar. The Toolbar is the bar of icons which normally appears immediately below the dScope's Menu bar; it is useful in providing fast access to commonly used dScope operations.



The list of available Toolbar icons and their functions are detailed in the Main Toolbar icons section.

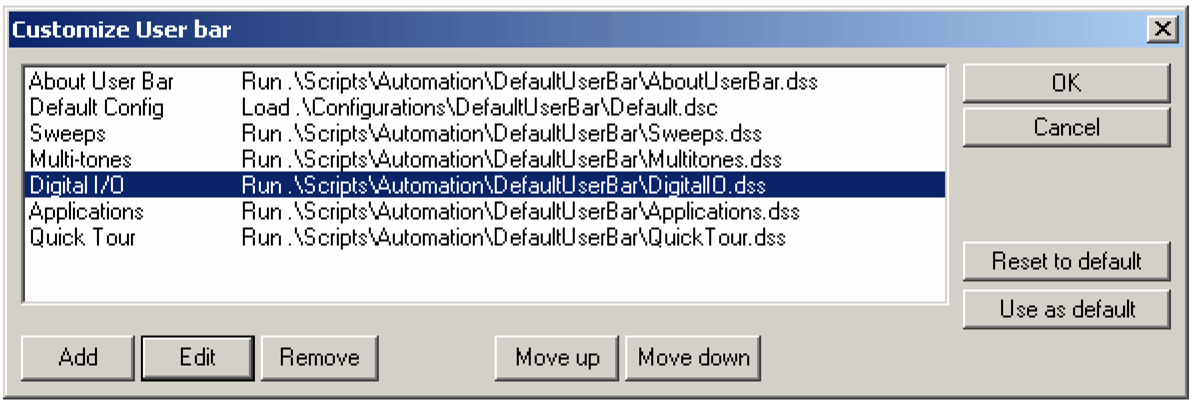
The currently-displayed Toolbar icons are shown in the right-hand list, and those not currently-displayed, plus 'Separator' are shown in the left-hand list. Icons are added by selecting the desired icon in the left-hand list, selecting the icon before which (i.e. to the left of which) the addition is to be made in the right-hand box, and clicking [Add–>]. Separators are added in the same way. Icons are removed from the Toolbar by selecting the icon in the right-hand list and clicking [<–Remove]. Icons are repositioned in the list (and hence on the Toolbar) by selecting the icon in the right-hand list and using the [Move Up] and [Move Down] buttons to adjust its position as required. The [Reset] button reverts to the default Toolbar layout (which actually includes all the possible icons).

The [Help] button opens the on-line help at this page. The provision of help buttons on dialogue boxes is not a feature of the dScope (the F1 key is used to access context-sensitive help); this example is because the Customize Toolbar dialogue box is a standard Windows control.

4.9.2 Customize User bar dialogue box

⚠ After first installation, a 'default User bar' is installed containing shortcuts to a variety of common tasks. Details of the default User bar are available by clicking the [About User bar] button on the left of the bar.

The Customize User bar dialogue box is used to select which user scripts and Configuration files can be accessed directly from the dScope's User bar. The User bar is the bar of buttons which normally appears immediately below the dScope's Toolbar; it is useful in providing fast access to commonly used dScope automation scripts and Configurations.



The main window of the dialogue box shows a list of button captions and their actions (i.e. to run a specified script or to load a specified Configuration). The specified buttons are arranged from left to right on the User bar.

The [Add] button is used to add new buttons to the bar; on clicking [Add], a box is displayed in which the user can specify a caption for a new button, and a script to be run or a Configuration to be loaded when it is pressed. The [Edit] button allows the caption and associated action of a selected (highlit) button entry in the list to be modified. [Remove] removes the selected button.

[Move up] and [Move down] allow the buttons to be re-ordered on the bar.

The [Use as default] button causes the current list of User bar buttons to be retained as a 'default' setting, which is reloaded whenever [Reset to default] is clicked.

4.9.3 Multi-tone Generation and Analysis dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

Analysis using synchronous multi-tones is fast and powerful. It allows many parameters of the equipment under test to be measured simultaneously, using a single stimulus waveform, and yields results much more quickly than sweeps or sequences of spot measurements.

Until now, multi-tone analysis has been difficult: it has been possible only with expensive and specialised equipment, which has been difficult to program. Merely generating appropriate multi-tone stimuli with such equipment has been complicated enough, but tailoring the required analysis functionality has been next to impossible. dScope provides the solution, bringing multi-tone techniques within the grasp of any operator, without the need for ANY programming whatsoever. Using the 'Multi-tone Generation and Analysis' dialogue box, you simply tell the dScope the range of tones you want to generate, and what numerical or graphical results you need – dScope does the rest.

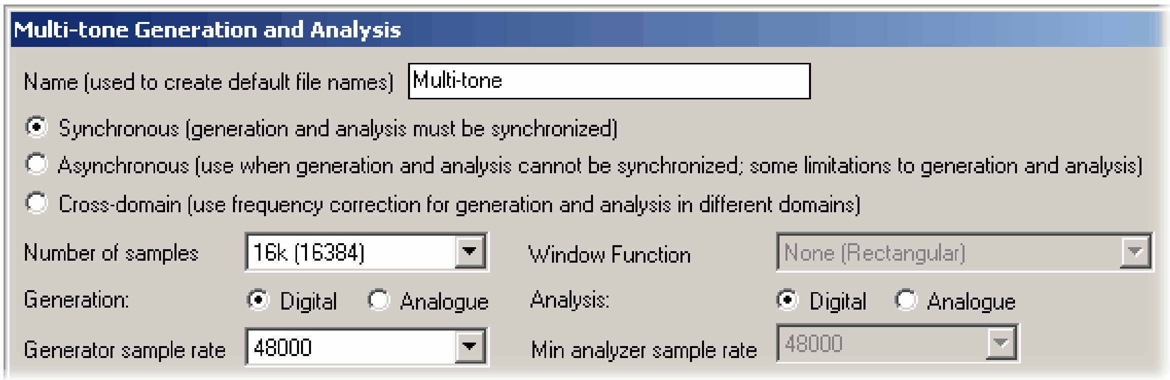
All this works thanks to versatility of dScope's VBScripting capabilities within the Signal Generator and Analyzer, but when you use the 'Multi-tone Generation and Analysis' dialogue box, dScope writes and loads the scripts automatically, sets up all required parameters, and arranges the Readings and Traces on the screen.

Using multi-tone analysis, it is possible to make many different measurements at the same time – for example you could measure: Distortion, Noise, Total Distortion + Noise, Frequency Response, Ripple, Gain, Channel Balance and Cross-talk on both channels at the same time, view the results in either graphical or numerical form (or both), and check them against your acceptable limits, all in a few seconds!

Some of the capabilities of dScope's multi-tone analysis rely on the generator and analyzer being 'synchronous', i.e. running at identical sample rates. However this is not always possible, particularly when running cross-domain tests. For this reason, dScope also provides an 'Asynchronous' mode of operation with some functional limitations

For more information, see Principles of Multi-tone Analysis.

General settings

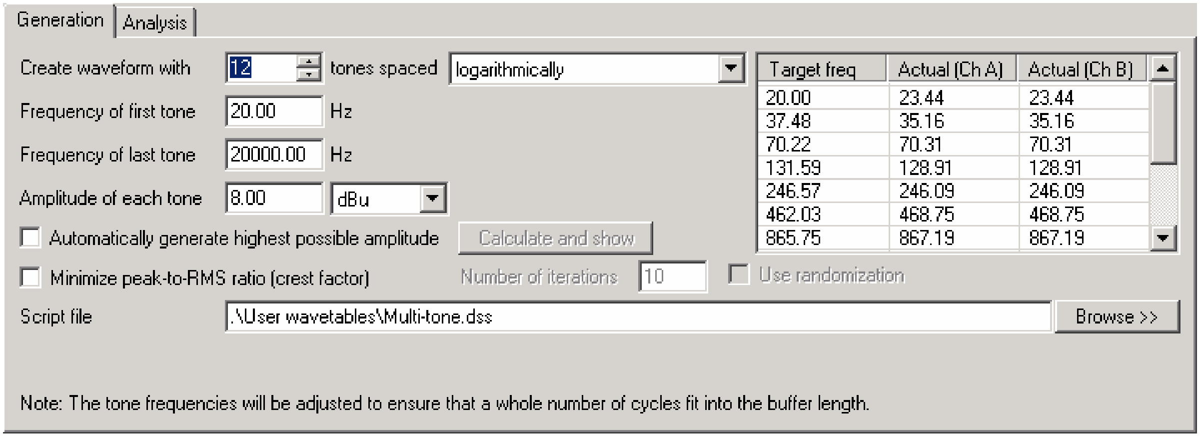


The upper section of the dialogue box contains some general settings as follows:

Name	When the dialogue is applied, a Generator wavetable script and a number of FFT Detector Calculation scripts are created. The entered name becomes the name of the Generator wavetable script, and the name of a folder which is created to contain the FFT Detector Calculation scripts.
Synchronous / Asynchronous / Cross-domain	Set to 'Synchronous' if the Generator and Analyzer will be locked together at identical sample rates, 'Cross-domain' if Generator and Analyzer will operate asynchronously in different domains, but will be resynchronised by the dScope, and 'Asynchronous' if true asynchronous operation is required.
Number of samples	Sets the number of samples in the Signal Generator and FFT Analyzer buffers. The larger the number of samples in the buffer, the greater the frequency resolution, but the slower the analysis.
Generation, Analysis	Sets the Generator and Analyzer domains. Thyese are tied together in 'Synchronous' mode, and forced apart in 'Cross-domain' mode.
Generator sample rate	Sets the Generator sample rate (also Analyzer if 'Synchronous').
Window function	In 'Asynchronous' mode only, sets analyzer window function. Should be set to window function with minimum bin-spreading for required dynamic range, so probably 'Prism 7' or 'Prism 5'.
Min analyzer sample rate	In 'Asynchronous' and 'Cross-domain' modes only, the minimum sample rate at which the analyzer will be operated must be set so that separation of tones can be ensured.

The lower section contains two tabs: 'Generation' controls the range of tones to be generated, 'Analysis' is for entry of the desired analysis functions.

Generation tab



The upper part of the tab allow the number of individual tones to be specified, along with the lower and upper frequency bounds. The spacing of the tones can be linear or logarithmic. Note that the maximum number of tones increases with buffer length, and is substantially reduced in asynchronous mode since tones spread across several bins in the analyzer but must not overlap.

All tones are equal in amplitude; this amplitude can be set in a variety of units. A check box allows the maximum possible amplitude to be set instead, depending on the number of tones and the resulting crest factor. When this option is selected, the [Calculate and show] button calculates the maximum tone amplitude and displays it in the amplitude setting box.

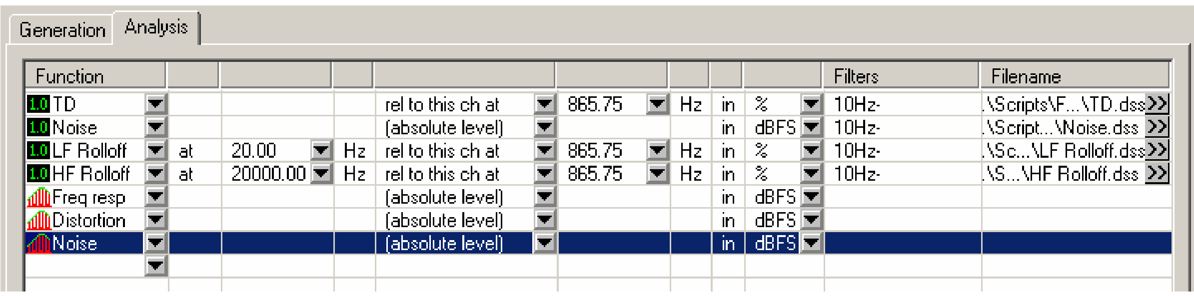
The 'Minimize peak-to-RMS ratio' check box causes the phases of the individual tones to be varied in order to achieve the lowest crest factor, so that the amplitude of each tone can be maximized without exceeding the peak handling of the equipment under test. This can be done either by a deterministic algorithm or, by checking the 'Use randomization' check box, with a random process which often gives improved results. In both cases, the number of iterations can be varied to trade off crest-factor minimization against calculation time.

The actual frequencies of the individual tones are modified from the requested values in order to ensure that a precise number of cycles of each tone occurs over the duration of the specified buffer length. On the right-hand side of the box, a table of target and actual frequencies is displayed. The target frequencies can be modified manually if desired. Note that the A and B channel frequency sets are different if cross-talk analysis has been selected.

Error messages are displayed if too many tones or too high an amplitude for each tone is selected. In this case, the entry must be modified before the dialogue can be applied.

The name of the Generator wavetable is specified in the lower box. This defaults to the root name supplied at the top of the dialogue.

Analysis tab



Many different analysis functions can be specified for simultaneous calculation from the captured FFT buffer. These are selected using the drop-list under the 'Function' heading. Functions can produce Readings (numerical results) or Traces (graphical results) as designated by the and symbols respectively. Readings are arranged on Page 1, and Traces are added to the Trace window which is opened on Page 2. Note that unlike Sweeps, Traces produced by multi-tone analysis are calculated from the same single FFT buffer as the Reading results, and do not require accumulation of spot measurements. They are thus extremely fast.

Traces produced by multi-tone analysis have the 'sum bins' transform applied over third-octave bands, thus accurately emulating the results of a third-octave sweep. For more information about this transform, see Sum Trace data bins in the Trace transforms section.

For each function, a separate Weighting filter can be selected under the 'Weighting' heading (including user-scripted weightings) if the default weighting specified in the Signal Analyzer dialogue box is not appropriate. Note that high and low-pass filters are automatically applied to all results as per the defaults specified in the Signal Analyzer dialogue box. The filename for the resulting FFT Detector script can be specified under the 'Filename' heading, if the default name is not appropriate. The remaining columns may contain other parameters depending on the function selected.

The available analysis functions are described in the following table:

TD	Total distortion excluding noise. This unique measurement sums the even bins (excluding original tones) to total the harmonic and intermodulation distortion. The sum of the odd bins is subtracted to try to present a distortion measurement which excludes any noise. Where the distortion products are low in comparison to the noise floor, this measurement may fluctuate widely, and may even occasionally read '–INF' owing to random variations in the noise content of the bins. In this case, TD+N may be a preferable measurement.
Noise	Total noise, excluding harmonic and intermodulation distortion products, calculated by summing the odd bins and doubling the result to include inferred noise in the even bins.
TD+N	Total distortion plus noise, calculated by summing all the bins which do not contain original tones, and correcting to account for inferred noise in the original tone bins.
Cross-talk	Measured at a selected frequency. The selected frequency is separated slightly for the A and B channels, enabling cross-talk to be measured in the bin corresponding to the opposite channel's tone frequency. If no cross-talk analysis is requested, the tone frequencies are identical for both channels.
Gain	Measured at a selected frequency. The amplitude of that tone is expressed relative to its generated amplitude.
LF Rolloff	Measured by comparing the amplitude of a selected (low frequency) tone with that of a nominal (mid-frequency) tone.
HF Rolloff	Measured by comparing the amplitude of a selected (high frequency) tone with that of a nominal (mid-frequency) tone.
Ripple	Measured by calculating the difference in amplitude between the loudest and quietest tones over the frequency range between two specified tones.
Amplitude	Measured at a selected frequency.
Balance	Calculated as the difference in amplitudes between the selected frequency tone in each channel.
Tone ratio	A general-purpose function to compare the amplitude of one tone to another.
Lowest	Measures the amplitude of the quietest tone within a specified range of tones.
Highest	Measures the amplitude of the loudest tone within a specified range of tones.
Freq resp	Plots the amplitude response against frequency.
Distortion	Plots total distortion, excluding signal and noise, against frequency.
Noise	Plots noise, excluding signal and distortion, against frequency.
Dist+Noise	Plots the sum of distortion and noise against frequency.
Cross-talk	Plots inter-channel cross-talk against frequency.

Certain analysis functions are not available in asynchronous mode, since it is not possible in this mode to distinguish noise and distortion. See Principles of Multi-tone Analysis for further information.

Any tone frequencies specified in the Analysis tab are selected from a drop-list of the actual 'modified' tone frequencies, or can be specified as the 'lowest' or 'highest' AMPLITUDE tone. Amplitude measurements can be specified as 'absolute', 'relative to generator' (same or opposite channel), 'relative to opposite channel', 'relative to total RMS amplitude' or 'relative to this channel at (specified frequency)'

Note that the function names on each line can be edited once they have been selected. The various frequency and amplitude-specifying fields can also be modified. Thus a wide variety of customised measurement functions can be made, even without any modification of the VBScripts, and these can be named by the user. The function name appears in the title bar of the resulting FFT Detector Reading.

An analysis function can be removed from the list of selected functions by selecting the row with the left mouse button, and then pressing 'delete'.

Applying the dialogue

When the Generation and Analysis tabs have been completed as required, clicking 'Apply and Close' exits the dialogue box, and initiates the selected measurements. A number of actions take place:

- A wavetable script is created to produce the desired multi-tone stimulus, and is stored in the 'User Wavetables' folder, with the root name entered at the top of the dialogue box.
- The wavetable script is loaded into the Signal Generator, and the amplitude set as requested in the Generation tab.
- The appropriate Analogue or Digital Output is turned on, and the other is muted.
- A number of FFT Detector Calculation scripts are created, one per analysis function requested, and stored in a folder created within the 'Scripts\FFT Detector Calculations' folder, with the name entered at the top of the dialogue. FFT Detectors are created to run these scripts. If the Signal Analyzer was set to channel A or B when the dialogue is applied, a single set of Detectors is created which subsequently switch with the Analyzer channel selection. If the Signal Analyzer was in two-channel mode, two sets of Detectors are created to measure the designated results for both channels simultaneously.
- The FFT Analyzer is set to the appropriate number of points, with a rectangular Window function for synchronous mode or the specified Window function for asynchronous mode.
- The Signal Analyzer is placed in Analogue or Digital mode according to the dialogue setting.
- If the Analogue Input is selected, its input range is fixed on the appropriate range to accommodate the generated level. This may be necessary because some combinations of multi-tones produce complex waveforms which may defeat the auto-ranging. If the set range is not appropriate for the equipment under test, you can either change it or try returning to auto-ranging.
- The analysis Readings are placed on Page 1, the Trace window is opened on Page 2, and the Signal Generator dialogue box is opened on Page 3.
- The FFT Analyzer trigger is turned on.

It is possible to exit the dialogue box without 'applying' the settings. If the 'Close' button is used, changes made within the dialogue are remembered but not applied. If the 'Cancel' button is pressed, all changes are lost.

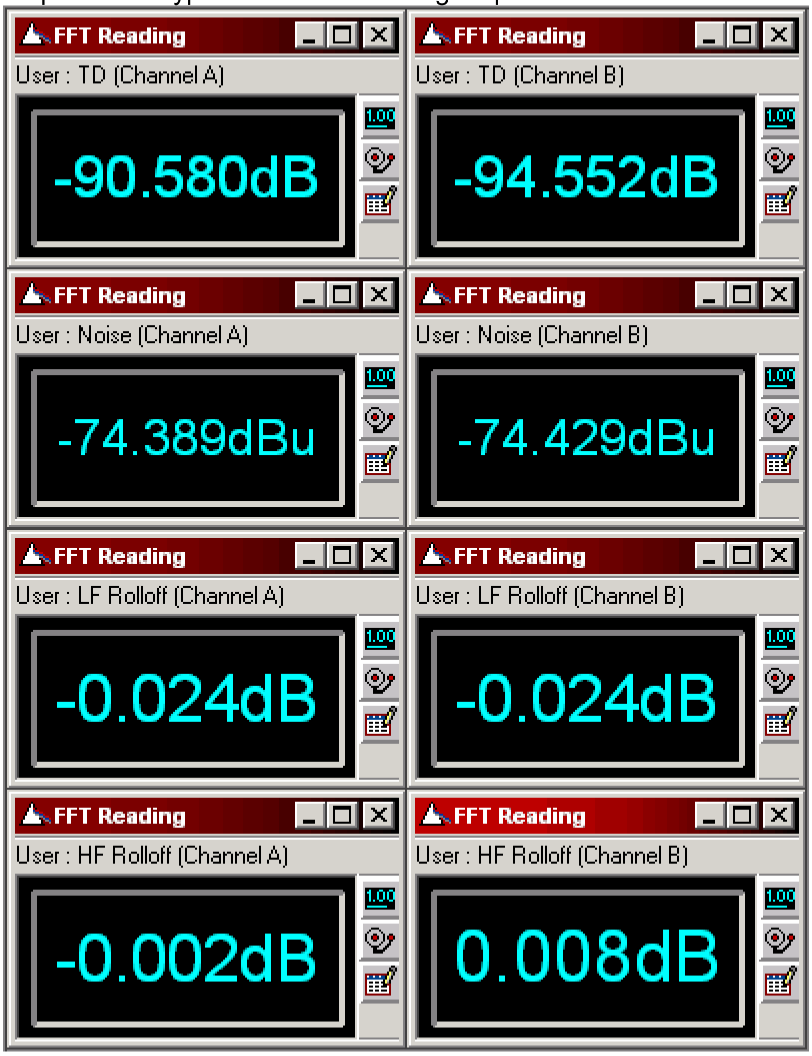
If, having run the multi-tone dialogue, the results are not as expected it is a simple matter to reopen the dialogue, change parameters and reapply. The amendments become active immediately. If the dScope's Configuration is saved after applying the multi-tone dialogue, it can be recalled later without the need to run the dialogue again. Obviously any parameters altered from their scripted settings, any additional windows opened etc. are also saved.

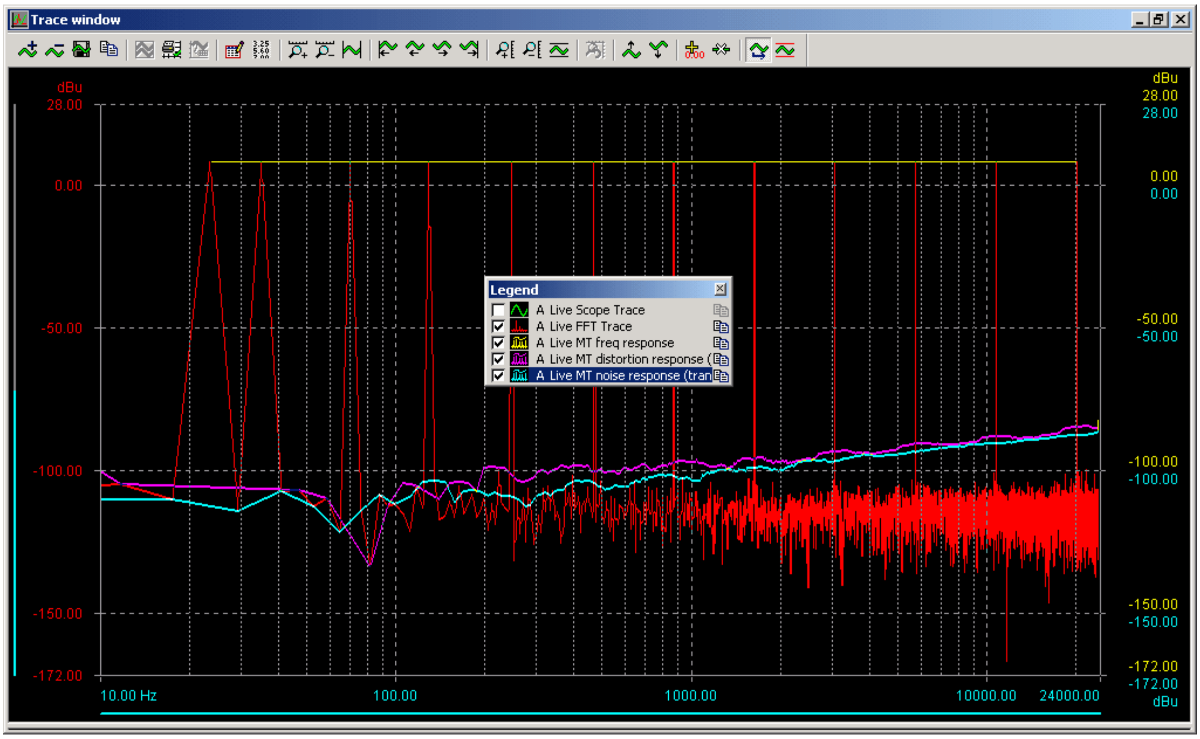
NB: Applying the dialogue may take some time, particularly if long buffer lengths, high numbers of tones, large numbers of analysis functions or high crest-factor-minimization iterations have been set. The delay is the result of dScope having to make many optimisations and write and deploy many VBScripts. **However, note that this delay**

need only be incurred once – as soon as the multi-tone test parameters are finalised, the Configuration can be saved for later reload (possibly as part of a more extensive automation script). Reloading the Configuration takes only a very short time.

Reading and Trace window examples

The following examples show typical multi-tone testing outputs.





Further flexibility

Because the multi-tone analysis scripts produce standard dScope Readings and Traces, all the usual features of Readings and Traces are supported: Reading limits or Trace Limit Lines can be applied, colours changed, units modified etc. without having to re-run the multi-tone dialogue.

The operation of the Multi-tone Generation and Analysis dialogue box is based on the versatility of the dScope's VBScripting capabilities. A wide variety of measurements can be made using this dialogue without writing a single line of VBScript. However, if you require any very unusual measurements based on arbitrary waveforms and scripted FFT Detectors, you may well find that it is easier to generate initial wavetable and FFT Detector Calculation scripts using the Multi-tone Generation and Analysis dialogue box, and then modify the VBScripts, rather than starting to script from scratch.

4.9.3.1 Principles of Multi-tone Analysis

Synchronous multi-tone analysis is based on calculating an FFT of a captured multi-tone stimulus after passing it through the equipment under test. The complex nature of the stimulus means that many different properties of the equipment under test can be measured from a single, short acquisition of a buffer of audio data. The results are calculated by means of a frequency-domain analysis of the captured buffer, using a Fast Fourier Transform (FFT) algorithm.

In synchronous multi-tone analysis, each tone in the generated multi-tone waveform is arranged to repeat exactly over the number of samples in the FFT buffer. This means that a rectangular FFT Window function can be used, and each tone will occupy a single 'bin' in the resultant FFT, without leaking into adjacent bins. To achieve this, the requested tone frequencies may need to be slightly modified in order to correspond to the bin centres. The larger the size of the FFT buffer, the smaller will be the maximum necessary frequency modification. Furthermore, by arranging that each tone occupies an 'even-numbered' bin in the FFT, it is guaranteed that all harmonic and intermodulation products which result from the tones also occupy other even-numbered bins, and can thus be measured independently from the general noise floor, which occurs in the odd-numbered bins.

In asynchronous multi-tone analysis, the generator and analyzer do not share identical sample rates and so some features of the synchronous case are lost. The tones must still be confined to frequencies which repeat exactly over the duration of the generator buffer, but since the analyzer buffer is captured over a different time period, the frequencies no longer lie at analyzer bin centres. For this reason, a rectangular Window function cannot be used and each tone spreads across a number of bins. Thus fewer tones can be accommodated, and it is no longer possible to distinguish distortion and noise by their occupation of even and odd bins. However, the total distortion and noise can still be distinguished from the tones.

Cross-domain mode is similar to synchronous mode, except that the sample rate of the analyzer is corrected to precisely match that of the generator before analysis is performed. It allows windowless FFT analysis in asynchronous situations without the limitations of asynchronous mode.

In all cases, frequency response, ripple etc. can be measured by comparing the relative amplitudes of the recovered tones. Balance-related measurements can be made by comparing FFTs from the two channels. By generating certain tones at slightly different frequencies for each channel, it is possible to measure inter-channel cross-talk at these frequencies.

The power of the technique lies in the fact that all these measurements can be calculated simultaneously and for both channels, after acquiring a short buffer of the recovered multi-tone signal from the device under test.

For a detailed discussion of multi-tone analysis, see the Applications Manual.

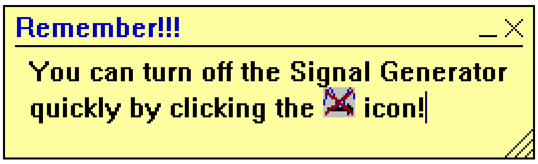
4.9.4 Memo sub-menu

The Memo sub-menu provides access to the dScope's Memos.

Menu options are:

New Memo	Creates a new Memo.
Memo List...	Accesses the Memo List for control of all Memos.

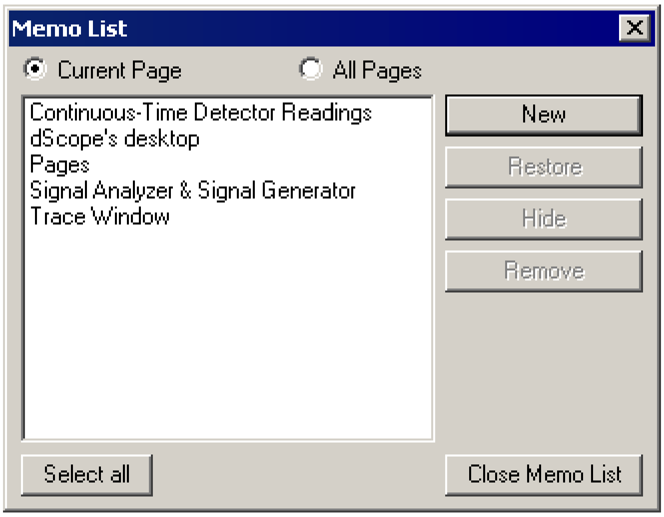
Memos are useful yellow sticky notes which can be applied to dScope's desktop, on any Page. They can be used to describe the Configuration, or to explain what's happening on a particular dialogue box or Page, or just as a useful reminder. The content of a Memo is rich text, so the font, colour etc. can be changed, or bitmaps can be included:



Left-clicking over a Memo or its Title Bar allows the text to be edited. Right-clicking over a Memo shows a drop-down menu containing basic editing cut/copy/paste functions, and options to hide or remove the Memo, or enter the . Note that the 'X' in the top right-hand corner hides the Memo; it doesn't remove it permanently.

4.9.4.1 Memo List dialogue box

The Memo List dialogue box allows Memos to be added, removed, hidden and restored.



The [New] button causes a new Memo to be created on the current Page, and exits the Memo List immediately.

The radio buttons at the top of the box select a listing of all the Memos or only those on the current Page.
Memos are listed alphabetically by their title bar text. Memos are selected using the left mouse button in the usual way, using <CTRL> and <SHIFT> keys to select multiple Memos if required. [Select all] selects all currently-listed Memos. The [Remove], [Hide] and [Restore] buttons then act on the selected Memos.

'Removed' Memos cannot be retrieved, whereas 'Hidden' Memos can be 'Restored' to their original positions.

4.9.5 dS-NET Peripherals sub-menu

NB: **This sub-menu may not be available, depending on the dScope model number.**

The dS-NET Peripherals sub-menu provides access to the dScope's dS-NET Peripherals.

Menu options are:

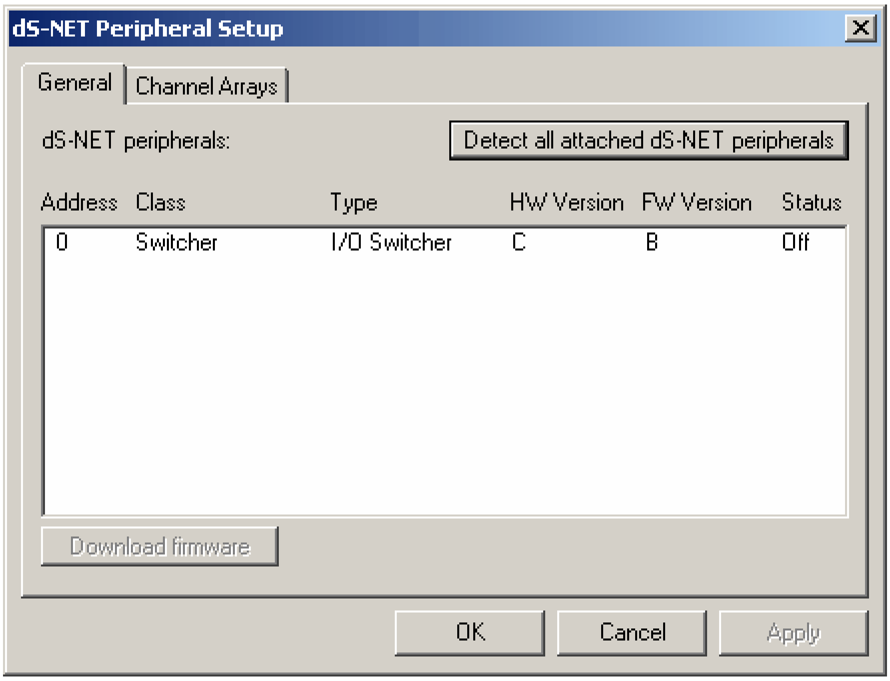
dS-NET Peripherals Setup...	Setup of all dS-NET Peripherals.
Switcher Channel Array Control	Control of Switcher Channel Arrays.
I/O Switcher Diagnostic Control	Diagnostic control of I/O Switchers.
VSIO Adapter Control	Control of VSIO Adapters.

4.9.5.1 dS-NET Peripherals Setup dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The dS-NET Peripherals Setup dialogue box contains general settings for any dS-NET Peripherals.

For general hardware and architecture information about individual dS-NET peripherals, and the dS-NET interface in general, refer to the dS-NET Peripherals section.



General tab

The [Detect all attached dS-NET peripherals] button populates the list with details of all attached dS-NET peripherals.

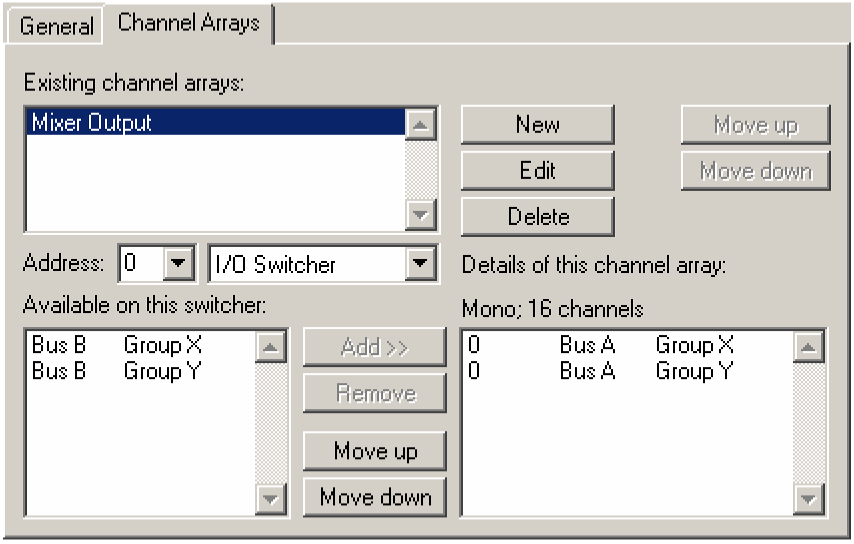
On selecting a device (by clicking on its list entry) it may be possible to update ('flash') the firmware of the device using the [Download firmware] button. This button is greyed out unless a firmware-downloadable device is selected.

To update the device's firmware, the left-most DIP switch on the rear panel of the device should be set to the down position before clicking the [Download firmware] button. Firmware downloading is indicated by the device's front-panel 'On' LED flashing red and green. This operation may take several minutes, during which the device must remain powered with its dS-NET cable connected to the dScope. The dScope user interface should not be operated during downloading, nor should the dScope application be closed. When downloading is complete, the left-most DIP switch should be returned to the up position, after which normal operation is resumed.

It is VERY IMPORTANT that the dS-NET peripheral remains powered and connected to the dScope, and that the dScope user interface is not operated nor the dScope application closed, during firmware downloading. Failure to observe this precaution may result in the firmware of the dS-NET peripheral becoming corrupted resulting in the peripheral becoming unusable.

Channel Arrays tab

The Channel Arrays tab allows Channel Arrays to be defined for use elsewhere in the dScope.



Channel Arrays are a user-friendly way of controlling switcher devices. They allow mono or stereo Arrays to be defined which may contain up to 1024 switched Channels cascaded over many physical switcher devices. Once defined, the Channel Arrays can be controlled directly without the user having to be concerned with sending control data to individual switcher devices.

All the members of a Channel Array may be 'Independently controlled', or alternatively Channel Arrays may be controlled as 'Exclusively on' or 'Exclusively off' if desired. Nominating the selected channel of a Channel Array 'exclusively' is achieved in a single operation (whether from a control panel or a VBScript): unselecting the previous channel and distributing control data to the various physical switchers comprising the Channel Array is all handled invisibly by the dScope. For more information, see the Switcher Channel Array Control dialogue box.

The upper part of the Channel Arrays tab shows a list of defined Channel Arrays. Arrays can be added or deleted using the [New] and [Delete] buttons. Existing Arrays can be modified via the [Edit] button, and the list can be reordered using [Move up] and [Move down].

The lower right-hand area of the tab shows the complement of crosspoints which currently comprise the selected Array, by dS-NET address, bus and group. Each of these crosspoint groups can be removed with the [Remove] button, or moved up and down the list with the lower [Move up] and [Move down] buttons. Note that the position in the list defines the numbering of the groups elements within the Array: elements are numbered upwards from 1, starting at the top of the list.

To add crosspoint groups to the Array, it is necessary to nominate the device containing the group by its dS-NET address in the drop-menus below the Arrays list. Available groups within the nominated switcher device are then listed in the bottom left-hand corner of the dialogue box. By selecting a desired group, and clicking the [Add>>] button, the group is added to the Channel Array.

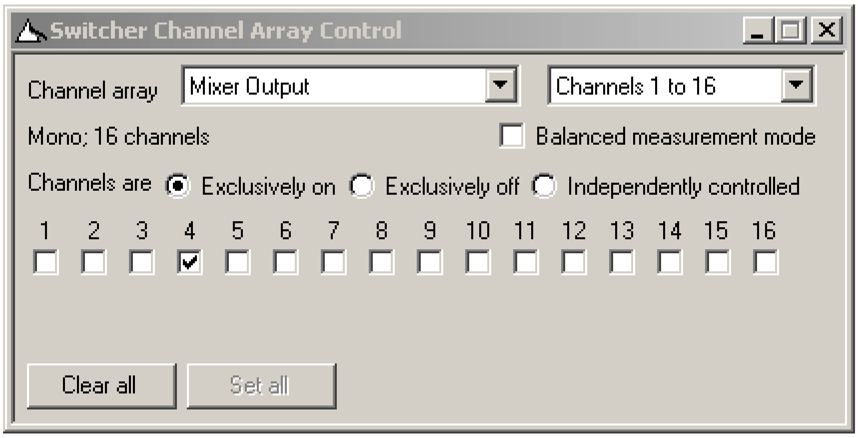
Stereo Channel Arrays can be created, wherein both buses of the included switchers are used. In this case, when defining the Channel Array, stereo bus groups are selected rather than individual mono bus groups, e.g. 'Group X' is selected for a stereo Channel Array, rather than 'Bus A Group X' . Having defined a stereo Channel Array, all odd numbered Channels switch to the A bus and even-numbered Channels switch to the B bus. dScope automatically selects/deselects adjacent Channel pairs, i.e. 1&2, 11&12 etc. when controlling stereo Channel Arrays.

4.9.5.2 Switcher Channel Array Control dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The Switcher Channel Array Control dialogue box is used to control switcher Channel Arrays.

Channel Arrays are a user-friendly way of controlling switcher devices. They allow mono or stereo Arrays to be defined which may contain up to 1024 switched Channels cascaded over many physical switcher devices. Once defined, the Channel Arrays can be controlled directly without the user having to be concerned with sending control data to individual switcher devices. For more information about Channel Arrays, see the dS-NET Peripherals Setup dialogue box.



The 'Channel Array' drop-menu is used to select a previously-defined Channel Group to be controlled. The size of the selected Array is described in the line of text beneath. The drop-menu in the top right-hand corner is used to select the part of the Array currently shown in the dialogue box: if the Array has more than 32 crosspoints, they cannot all fit onto the box at once and so a subset must be selected.

Crosspoints are closed or opened simply by clicking on their respective check-boxes. If 'Exclusively on' is selected, each Channel closed automatically opens the previously closed Channel. If 'Exclusively open' is selected, the same applies but with only one Channel being open at a time. If 'Independently controlled' is selected, the crosspoints can be opened and closed independently. In 'exclusive' modes, open-before-close sequencing is applied in order to protect EUT outputs.

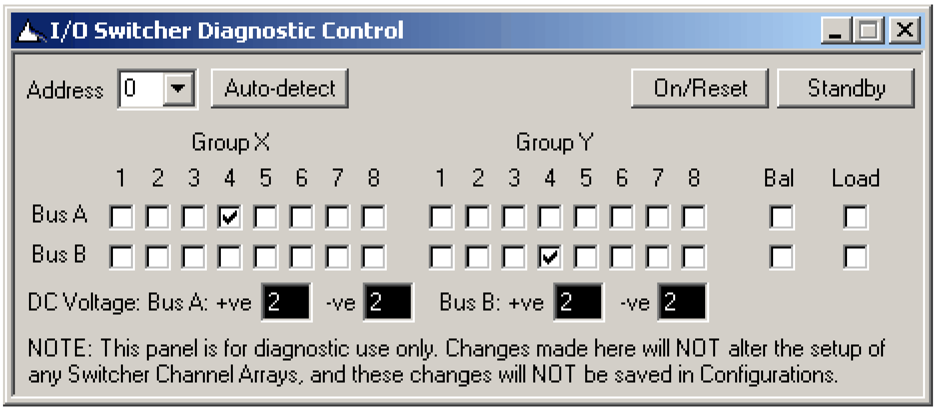
If a Stereo Channel Arrays is being controlled, all odd numbered Channels switch to the A bus and even-numbered Channels switch to the B bus. dScope automatically closes and opens adjacent Channel pairs, i.e. 1&2, 11&12 etc. together when controlling stereo Channel Arrays.

Checking 'Output balance mode' causes any I/O Switchers in the Array to be set to their 'output balance' mode which is intended for measuring the quality of balance of balanced outputs. See the I/O Switcher section for more details.

4.9.5.3 I/O Switcher Diagnostic Control dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The I/O Switcher Diagnostic Control dialogue box provides direct control of a dS-NET I/O Switcher, normally for diagnostic purposes. The preferred way to control switchers of all types is by using Channel Arrays as decribed in the dS-NET Peripherals Setup dialogue box and Switcher Channel Array Control dialogue box sections.



This dialogue box has controls which directly access each physical part of the I/O Switcher. For a full explanation of all of these, refer to the I/O Switcher section.

As highlighted on the dialogue box itself, changes made to I/O Switchers using this box do not affect the the state of any Channel Arrays, and are not memorised in saved Configurations.

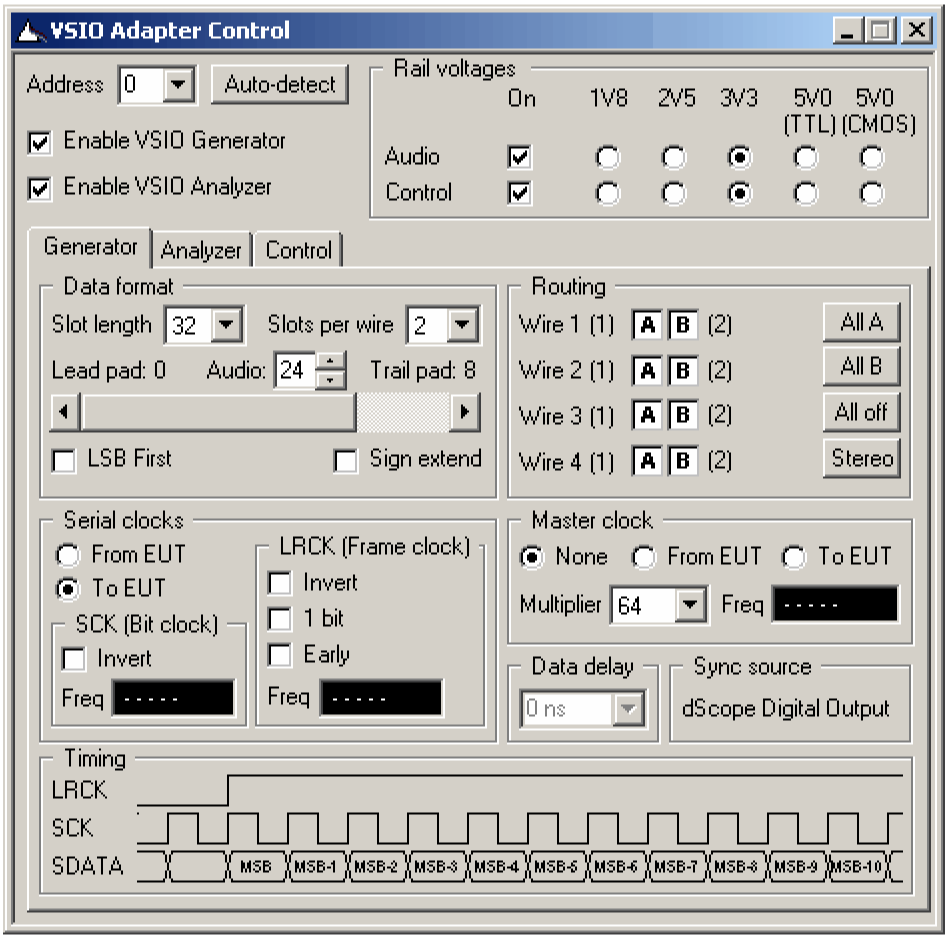
4.9.5.4 VSIO Adapter Control dialogue box

NB: **This dialogue box may not be available, depending on the dScope model number.**

The VSIO Adapter Control dialogue box is used to control any dS-NET VSIO Adapters connected to the dScope. VSIO Adapters enable the dScope to interface with chip-level serial audio multiplex formats such as I2S. For detailed information about the VSIO Adapter's capabilities, and how to connect it, see VSIO Adapter in the "dS-NET peripherals" section.

The VSIO Adapter's functions can also be automated, as described in the Scripting Manual.

General settings



The upper part of the dialogue box contains some general settings:

The 'Address' drop-menu allows selection of which VSIO Adapter is to be controlled by the dialogue box. The drop-menu is automatically populated with a list of all the VSIO Adapters present if the [Auto-detect] button is clicked, or if VSIO Adapters have already been detected using the dS-NET Peripherals Setup dialogue box. Alternatively, the address of the desired VSIO Adapter can be entered directly (in which case the dS-NET bus is interrogated to discover a VSIO Adapter at that address - if none is present, it is still possible to enter settings for a 'virtual' VSIO Adapter at that address, subject to receipt of a warning box).

The 'Enable VSIO Generator' and 'Enable VSIO Analyzer' check-boxes control the pass-through relays on the VSIO Adapter's AES3 and AES11 ports. When 'Enable VSIO Analysis' is unchecked, the dScope's AES3 Digital Input XLR is not driven by the VSIO Adapter, but is instead connected to the loop-through XLR. Similarly, when 'Enable VSIO Generation' is unchecked, the dScope's AES3 Digital Output XLR (and its Reference Sync XLR) are diverted to their respective pass-throughs. This facility is useful, for example, when testing an EUT which converts bi-directionally between I2S and AES3: in this case both signal paths can be tested without replugging.

The 'Rail voltages' panel allows the VSIO Adapter's logic levels to be set to match those of the EUT.

In the case of the 5V rail voltage option, either TTL or CMOS input thresholds can be selected. The VSIO Adapter's Audio and Control ports are powered by checking the 'On' check boxes. Note the following safety feature: Changing either rail voltage value results in those rails being automatically turned off until the 'On' setting is manually reselected. Similarly, the rails are automatically switched off whenever the dScope application is started or a new configuration is loaded.

It is VERY IMPORTANT that the correct voltages are selected for any ports connected to the EUT BEFORE powering the ports by checking the 'On' check boxes. Failure to observe this may result in permanent damage to the VSIO Adapter, or the EUT, or both.

The remainder of the panel is tabbed for access to Generator, Analyzer and Control port settings. Note that the VSIO Generator and Analyzer ports are essentially independent. They can operate with different data and clock formats, and if necessary at different sample rates and may be subject to different clock masters.

Generator tab

The settings in the Generator tab control the VSIO Adapter's output parameters, i.e. those which affect the serial audio multiplex driven out of the VSIO Adapter into the EUT. The audio data on the multiplex is derived from the dScope's Signal Generator.

Data format panel

The 'Slot length' drop-menu is used to select the number of bit periods in one 'slot' (channel) of the serial audio multiplex. Supported values are 8, 16, 24 or 32 bit periods. The 'Slots per wire' drop-menu allows either 2, 4, 8 or 16 slots (channels) to be selected in the multiplex. The VSIO Adapter can generate different channel data on each of up to four wires, but only up to a total of 16 channels, as described in the 'Routing panel' section below.

The 'Audio' spin control allows the number of active audio bits in the multiplex to be entered or adjusted. This can be any number from 8 to 24. The timing of the audio word within the slot is controlled using the scroll bar beneath. This updates the 'Lead pad' and 'Trail pad' values, which are respectively the number of unoccupied bit periods in the slot before and after the audio word. Checking 'LSB first' reverses the order of the audio bits from its normal 'MSB first' orientation. Checking 'Sign extend' causes the pad at the MSB end of the audio word to be filled with copies of the audio word's MSB (sign bit), instead of the normal zero value.

Note that the effects of modifying the various 'Data format' settings can be observed in the 'Timing' panel at the bottom of the dialogue box.

NOTE: Depending on the sample rate, some combinations of 'Slot length' and 'Slots per wire' can exceed the maximum supported multiplex bit rate of 24.576Mbps. In this case (denoted by the SCK rate being indicated in red in the 'Serial clocks' panel) the outputs of the VSIO Adapter are indeterminate.

Routing panel

This panel controls the generated contents of each of the slots (channels) defined in the 'Data format' panel.

The VSIO Adapter can generate up to 16 distinct slots, which can be spread over up to four physical wires:

Slots per wire	Max active wires	Max channels
2	4	8
4	4	16
8	2	16

16	1	16

By clicking in the routing grid, each slot on each wire can be assigned to Channel A or Channel B of dScope's Signal Generator, or can be muted. Repeatedly clicking in any box cycles the routing from 'A' to 'B' to ' ' (mute). Alternatively, buttons are provided to quickly route [All A], [All B], [All off] or [Stereo]. When [Stereo] is clicked, the earlier of each pair of Slots is routed from A, and the later from B.

Serial clocks panel

This panel controls parameters of SCK (the serial bit clock) and LRCK (the wordclock or frame clock) as transmitted (or received) by the Generator section of the VSIO Adapter.

If`'To EUT' is selected, SCK and LRCK are driven by the VSIO into the EUT, which should be configured to accept these clocks. Thus the EUT is slaved to the VSIO (and by inference to the dScope). If 'From EUT' is selected, the VSIO receives SCK and LRCK from the EUT, which is thus clock-master. In this case, the dScope's Digital Output must be slaved to its AES11 Reference Sync, provided by the VSIO. NOTE that it is also possible for an EUT-sourced MCK to be clock-master (if selected as 'From EUT' in the 'Master clock' panel whilst SCK/LRCK are set to 'To EUT'). The dScope's Digital Output must likewise in this case be slaved to the VSIO's AES11 reference Sync. For more information, see the VSIO Adapter architecture section.

In its default state, LRCK is transmitted as an equal mark/space square wave and marks the beginning of the multiplex frame with its rising edge. Selecting '1bit' causes the duty cycle to be modified so that LRCK remains high for only a single bit period. Selecting 'Early' causes the active edge to occur one bit period BEFORE the beginning of the frame's data. Selecting 'Invert' causes the LRCK waveform defined above to be logically inverted in its entirety.

In its default state, SCK is transmitted as an equal duty-cycle square wave, with a rising edge at the start of each bit period. SCK can be inverted by checking the 'Invert' box.

Note that the effects of modifying the various LRCK and SCK parameters can be observed in the 'Timing' panel at the bottom of the dialogue box. The 'Sync source' panel indicates the Generator clock master at all times.

The nominal frequencies of LRCK and SCK are displayed. Note that these are 'discriminated' rather than precisely measured, i.e. LRCK is displayed as the nearest standard sample rate to its measured frequency, and SCK is displayed as the product of that rate, the 'Slot length' and the number of 'Slots per wire'.

⚠ NOTE: Depending on the sample rate, some combinations of Slot length and Slots per wire can exceed the maximum supported multiplex bit rate of 24.576Mbps. In this case, the SCK rate is indicated in red and the outputs of the VSIO Adapter are indeterminate.

Master clock panel

If 'None' is checked, no MCK is either produced or expected by the VSIO Adapter. If 'To EUT' is checked, the VSIO outputs an MCK to the EUT. If 'From EUT' is checked, the EUT's MCK is used by the VSIO as the clock-master, providing that SCK/LRCK are set to 'To EUT' as described above. Note that the 'Sync source' panel indicates the Generator clock master at all times. For more information, see the VSIO Adapter architecture section.

The required (or supplied) MCK multiplier (ratio of MCK frequency to the sample rate) must be entered. This may be 64, 128, 192, 256, 384 or 512. The MCK frequency is displayed. Note that the value displayed is simply the product of the discriminated standard sample rate and the entered multiplier.

⚠ NOTE: Depending on the sample rate, higher MCK multipliers can result in a target MCK frequency in excess of the maximum supported frequency of 24.576Mbps. In this case, the MCK rate is indicated in red and the outputs of the VSIO Adapter are indeterminate.

Data delay panel

The timing relationship between serial clock and data wires can be extremely critical in ensuring correct transaction of a serial multiplex, especially where the bit period is very short. Where serial clocks and data are travelling in the same direction (i.e. Generator serial clocks TO the EUT or Analyzer serial clocks FROM the EUT) the problem is manageable, since the time delay effects of cabling, buffering etc. affect clocks and data similarly so that the accuracy of the timing relationship ('skew') is not worsened. However, when serial cloaks and data are travelling in opposite directions (i.e. Generator serial clocks FROM the EUT or Analyzer serial clocks TO the EUT) the problem is worse, since cabling and buffering delays tend to progressively worsen skew between clock and data.

Under normal circumstances, where both the breakout cable and extension ribbon are used to connect Generator or Analyzer to the EUT, these delays are automatically compensated by the VSIO. However, the facility exists to adjust the timing of the data output or input at the VSIO with respect to the serial clocks when clocks and data are travelling in opposite directions. This feature is needed only if non-standard cabling is used, or if buffering delays within the EUT must be compensated. The data delay is adjustable in nominal 7ns steps. Non-zero settings are highlighted in red to provide a warning of non-standard operation.

Setting a non-zero 'Data delay' value when using standard cabling, or setting an incorrect value in any case, is very likely to result in data transmission failure. For this reason, the 'Data delay' setting should only be used by experienced operators in exceptional circumstances.

For more information on cabling and termination issues, see the VSIO Adapter architecture section.

Sync source panel

The 'Sync source' panel indicates the Generator's clock master, according to the 'Serial clocks' and 'Master clock' panel settings as described above. Possible masters are 'dScope Digital Output', 'EUT (SCK/LRCK)' or 'EUT (MCK)'.

Timing panel

The Timing panel is a useful summary of some of the other settings in the Generator tab. It shows the relationship between LRCK (the frame clock), SCK (the serial bit clock) and the early audio data bits in the first slot of the multiplex. This allows a quick comparison to be made between the VSIO Adapter settings and, for example, a device data sheet.

I2S operation

Note that the serial multiplex format known as 'I2S' is easily supported by the VSIO Adapter. To set up the Generator for I2S operation, the following settings should be entered:

Setting	State
Slot length	As required, usually 16, 24 or 32
Slots per wire	2 (8 for 'I8S' etc.)
Audio	Wordlength as required, 16..24 bits
	Lead pad = 0
	LSB first: UNchecked, Sign extend: UNchecked
LRCK	Early: CHECKED, Invert: CHECKED, 1bit: UNchecked
SCK	Invert CHECKED
MCK	As required
Sync master	As required
Routing	Stereo

Analyzer tab

The settings in the Analyzer tab control the VSIO Adapter's input parameters, i.e. those which affect the interpretation of the serial audio multiplex driven into the VSIO Adapter from the EUT. The audio data is passed on to the dScope's Signal Analyzer.

The operation of the Analyzer tab is almost identical to that of the Generator tab except, of course, that the settings describe the way the data will be interpreted instead of how it is generated.

There are two detail differences:

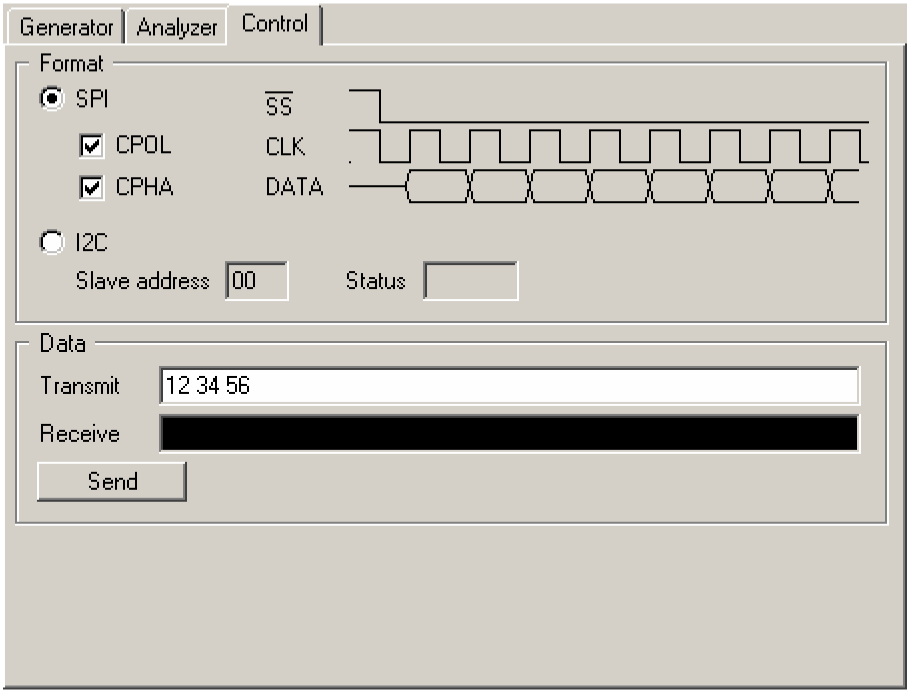
First, there is no 'Sign extend' check box for the Analyzer; the VSIO takes no notice of the pad bits.

Second, there is no need for an elaborate routing matrix. It is only necessary to define, for each of the two dScope Analyzer channels, which Slot of which Wire will be analyzed.

Additionally, there are slight differences in the way the Analyzer is synchronised - these are described in the VSIO Adapter architecture section.

Control tab

The VSIO Adapter also contains a serial Control port, which is capable of controlling, for example, devices on the EUT so that they can be placed in the appropriate mode for testing. The serial port can operate as either an SPI or an I2C 'master'. Sequences of bytes can be sent to the EUT, and status can be read. The data rate is arbitrarily fixed at 86.4kbps.



Either SPI or I2C mode must be selected using the radio buttons.

In SPI mode, the polarity and phase of the CLK can be adjusted to cater for varying EUT requirements using the CPOL and CPHA check-boxes. The timing diagram shows how the clock timing is affected.

In I2C mode, the slave address must be entered in the box provided.

For either mode, the control string is entered as a list of hexadecimal digits, which are automatically arranged into a list of bytes as they are entered or edited. The string is transmitted by clicking the [Send] button, after which the EUT device's response is displayed. In SPI mode, the response string displayed in the 'Receive' box is constructed from serial data clocked into the DIN pin as the transmitted data is clocked out - obviously this is only valid for SPI devices which return a response. In I2C mode, the displayed response is restricted to the ACK/NACK status of the transaction, and is shown in the 'Status' box.

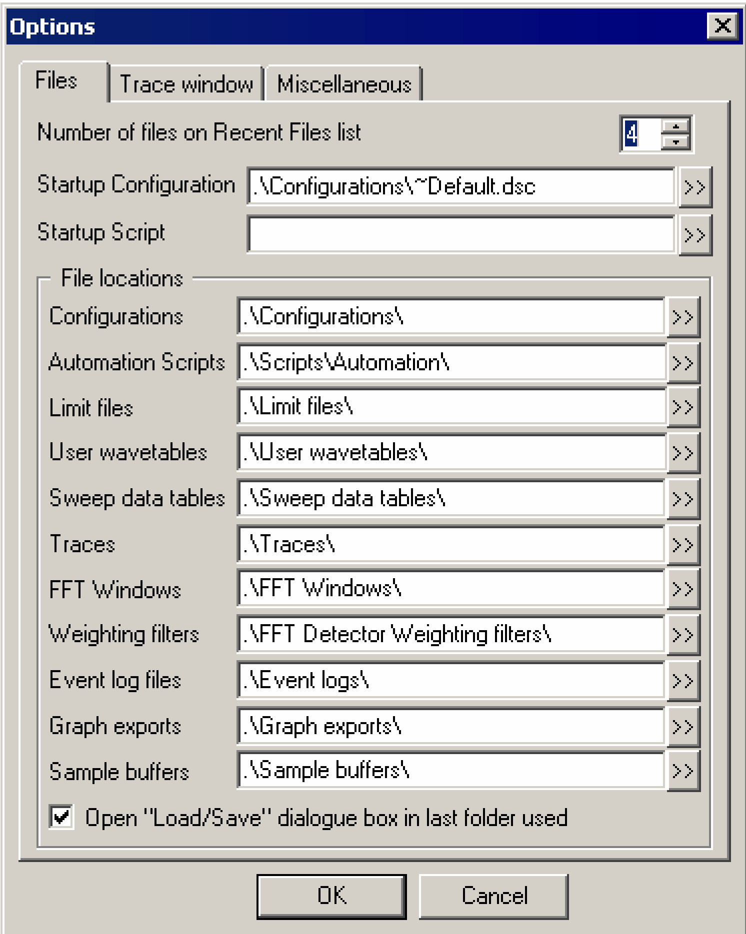
It is beyond the scope of this manual to provide details of the operation of I2C and SPI control buses. For more information, see manufacturers' data sheets for the devices to be controlled.

4.9.6 Options dialogue box

The Options dialogue box allows the user to specify a variety of miscellaneous operating options for the dScope. These are stored in the Windows registry, and are thus retained for all future sessions.

The following sections describe the functions of the settings in the Options dialogue box, and define their default state after a new installation.

Files tab



Number of files on Recent Files list: Selects the number of recently-accessed Configuration files which appear at the bottom of the Files menu. The default is four.

Startup Configuration: Specifies the Configuration file to be loaded at startup each time the dScope is run. If a simple filename is specified, it is assumed to occupy the 'Configurations' location specified beneath. If a relative path is specified with the filename (e.g. starting '.' or '..') then the path is assumed relative to the dScope installation location. A complete path to the filename can also be entered. If no file is specified, or the specified file is not found, default settings are loaded.

Reserved Configuration filenames:

Some Configuration filenames have been predefined as having special functions. Use of these names for general Configurations should therefore be avoided.

{install folder}\Configurations\~default.dsc is the default Configuration created at installation time and entered as the default Startup Configuration in the Options dialogue box.

{install folder}\configurations\~autosav.dsc is saved each time the dScope application is closed, so nominating this file to be loaded at startup causes the dScope to start in the same state as when it was last shut down.

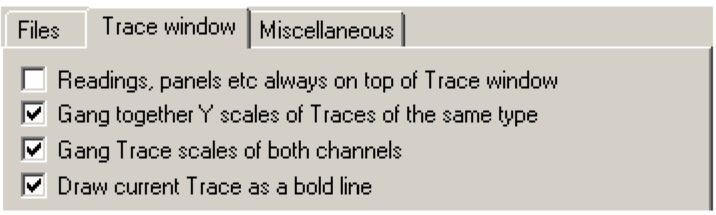
{install folder}\configurations\~powroff.dsc is saved by the dScope application on receiving a message from Windows that the host computer is about to enter a 'hibernation' power-saving mode. dScope closes after saving the Configuration to avoid possible consequences if the USB interface is shut down by the host, as is often the case during hibernation. The dScope cannot automatically restart when hibernation is over, but next time the dScope is started, the powroff.dsc file is detected and the user is offered the choice of reloading that Configuration.

Startup Script: Specifies an optional Script to be run when the dScope is started, after the Startup Configuration is loaded. If a simple filename is specified, it is assumed to occupy the 'Automation Scripts' location specified beneath. If a relative path is specified with the filename (e.g. starting '.' or '..') then the path is assumed relative to the dScope installation location. A complete path to the filename can also be entered. If no file is specified, or the specified file is not found, no Script is run.

File Locations: Specifies the default locations used for dScope files of various types. In general, these locations are accessed by default by the appropriate loading and saving dialogue boxes, but can be over-ridden by browsing to any location. The default settings are shown.

Open Load/Save dialogue box in last folder used: Causes all loading and saving dialogue boxes to point initially at the location where the last file of the same type was loaded or saved (even in a previous session), rather than at the location specified in the appropriate 'File Locations' setting. the default setting is OFF.

Trace window tab



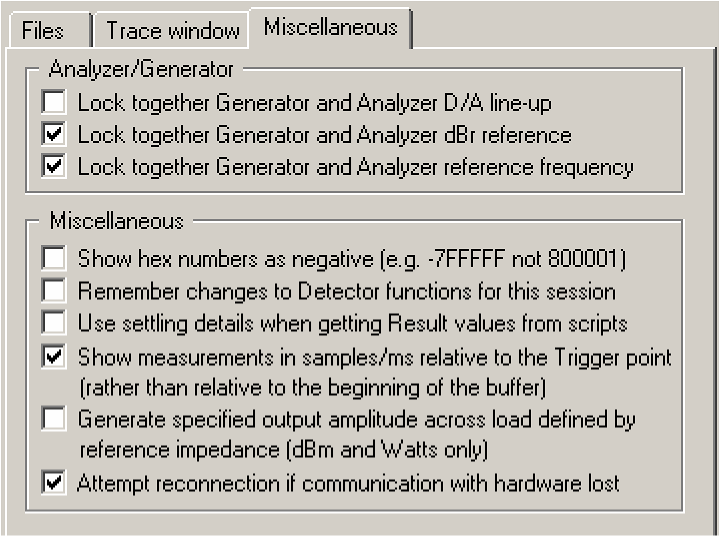
Readings, panels etc. always on top of Trace window: If this setting is OFF, all Readings, dialogue boxes, panels etc. are covered by the Trace window if it has had the focus more recently than they. With this setting on, they are always displayed 'on top' of the Trace window. The default setting is ON.

Gang together Y scales of Traces of the same type: This setting causes the Y scales of Traces of the same type to have their Y scales tied together for each channel. The default setting is ON.

Gang Trace scales of both channels: This setting causes the X and Y scales of both Trace channels to be tied together. The default setting is ON.

Draw current Trace as a bold line: If this setting is ON, the current Trace is displayed as a bolder than normal line on the Trace window. The default setting is OFF.

Miscellaneous tab



Lock together Generator and Analyzer D/A line-up: Causes the D/A line-up settings of the Generator and Analyzer to be locked together, so that changing either one also changes the other. The default setting is ON.

Lock together Generator and Analyzer dBr reference: Causes the dBr reference amplitude settings of the Generator and Analyzer to be locked together, so that changing either one also changes the other. The default setting is ON.

Lock together Generator and Analyzer reference frequency: Causes the reference frequency settings of the Generator and Analyzer to be locked together, so that changing either one also changes the other. The default setting is ON.

Show hex numbers as negative: Causes negative hex numbers to be displayed as such. For example, with this setting OFF, the smallest 24–bit negative number would be displayed as FFFFFF, whereas with this setting ON it would be displayed as –000001. The default setting is ON.

Remember changes to Detector functions for this session: Continuous-Time and FFT Detector functions are generally defined by scripts which are run when the function is selected, setting up the various Detector parameters. If any of the settings are manually altered after selection, the script is not modified, so if the function is changed and later restored the manual alterations are not recalled, assuming this setting is OFF. If this setting is ON, manual changes to each Detector function setting are remembered for the duration of the session, so that returning to a modified function also reloads the manual modifications. The default setting is OFF.

Use settling details when getting Result values from scripts: When Results are swept, the Sweep waits until user-defined settling criteria are met before each point in the Sweep is plotted. These are defined in the Sweep Settling dialogue box. When a VBScript reads a Result, this setting determines whether an instantaneous single Result is used (OFF) or whether the script will pause until the same Sweep Settling criteria are met (ON). The default setting is ON.

Show measurements in samples/ms relative to the Trigger point: When ON, this setting causes time (or sample count) indications on the X axis, at cursors etc. to have their origin at the Trigger point rather than at the start of the captured FFT Analyzer buffer. The latter was the only option in previous software versions. The default setting is OFF.

Generate specified output amplitude across load defined by reference impedance (dBm and Watts only): Whilst the Signal Generator amplitude setting in dScope generally takes no account of source impedance (it specifies the amplitude behind the source impedance, rather than that across the terminals, in the loaded state), impedance-related units such as W and dBm (which assume a load impedance equal to the 'reference impedance') can specify terminal amplitudes in the loaded state if this setting is ON. The default setting is OFF.

Attempt reconnection if communications with hardware lost: When ON, this setting causes the dScope software to poll the hardware, in the event of failure of USB communications, until the connection is resumed. Otherwise, a failure message is displayed and the application terminates. The default setting is OFF.

NB: **Since the Options settings are stored in the registry, and affect all future sessions, their states are NOT saved or recalled with dScope Configuration files. It is therefore possible to recall a saved Configuration which may behave differently than when it was saved because of changes which have been made in the Options dialogue box. However, since the Options settings generally define operational aspects of the user interface, this feature allows general personal preferences to apply even to previously-stored Configurations**

4.10 Window menu

The Window menu provides control of the various windows which are open within the dScope application. Menu options are:

Cascade	Stacks all the currently-open windows within the dScope window.
Auto-tile	Tiles the currently-open windows within the dScope window.
Close All	Closes all currently-open windows.
[window list]	Selects any one of the currently-open windows.

4.11 Help menu

The Help menu accesses the dScope's on-line help facility and various statuses.

Menu options are:

Help Contents...	Accesses the contents page of the on-line help file.
Help Index...	Accesses the index page of the on-line help file.
Help Search...	Accesses the search page of the on-line help file.
Tip of the Day...	Displays the dScope 'tips' viewer.
About dScope...	Displays a box describing the current dScope software release.
About dScope Hardware...	Displays a box describing various hardware statuses.
Show Readme.txt...	Displays a text file containing latest information about the software release.

4.11.1 About dScope dialogue box

The About dScope dialogue box displays useful information about the current version of the dScope software, the version of the operating system, and the usage of various system resources.

4.11.2 About dScope Hardware dialogue box

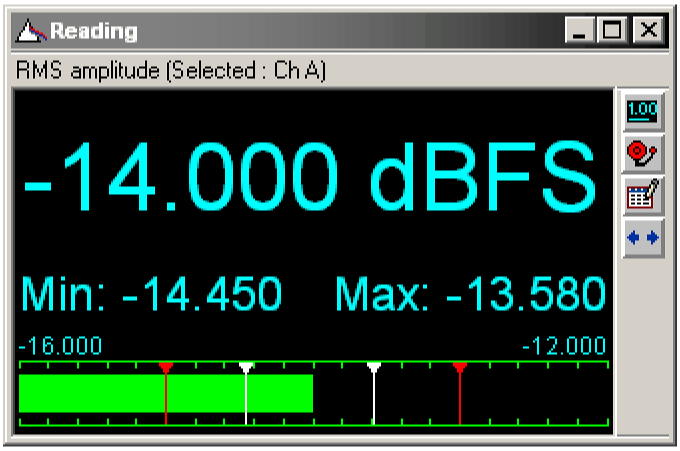
The About dScope Hardware dialogue box displays useful information about the dScope hardware. This includes hardware revisions and calibration history of the various internal subassemblies, as well as power supply and temperature status.

4.12 Reading window

A Reading window can be created from any dScope Result by placing the mouse cursor over the Result, holding down the left mouse button, dragging the Result off its dialogue box or panel, and then releasing the left mouse button.

Reading windows are a powerful way of displaying any of the dScope's Results. They allow the Result to be displayed independently of the dialogue box or panel in which it is normally displayed, so that if the box or panel is closed or minimized, the Result may remain.

In addition, the creation of a Reading window allows the display of the Result to take on many other powerful properties.

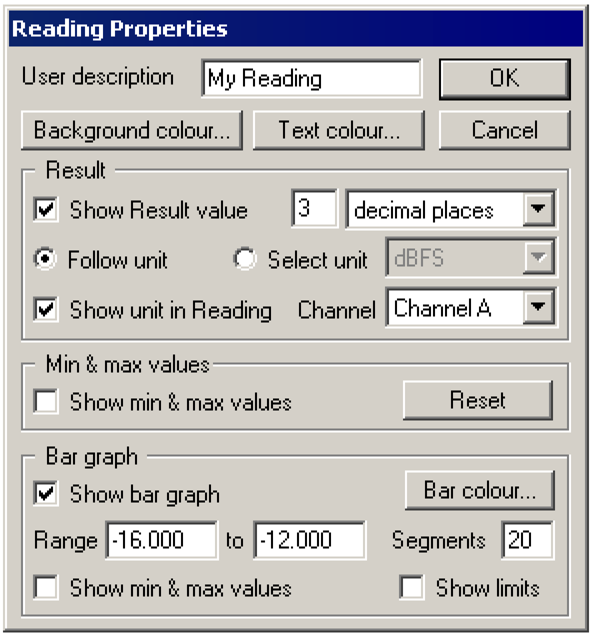


An important property of a Reading is that it can be resized: by placing the mouse cursor on one of the edges of the Reading, it is replaced by a double-headed arrow, indicating resizing mode; by holding down the left mouse-button, the edge of the Reading can be dragged in or out to resize the Reading. Thus Readings can be made large for long-distance viewing, or small to conserve screen space. In addition, the Reading can be maximised to fill the whole Page, or minimised to a small bar. In the minimised state, the Result remains visible, displayed in the Reading's title bar.

Other properties of the Reading are accessed by clicking the icon to open the Reading Properties dialogue box, or the icon to open the Reading Limits dialogue box. The icon opens the dialogue box where the Result was originally resident. The icon causes the maximum and minimum excursion values to be reset, in the same maner as the [Reset] button on the Reading Properties dialogue box.

4.12.1 Reading Properties dialogue box

The Reading Properties dialogue box controls the display attributes of its associated Reading window.



A 'User description' of the Reading can be entered at the top of the box. When the Reading is displayed (except when minimised), it carries a general indication of its source in the title bar, e.g. 'CT Reading', or 'FFT Reading' and just below, by default, a more detailed description of its function. If the user enters anything in the 'User description', then this text is substituted for the functional description.

Clicking the [Background colour...] button opens a palette for selection of the Reading's background colour. The colour of the text can be modified using the [Text colour...] button.

Below, a ruled area controls the appearance of the Reading text itself. The Result text can be hidden by unchecking the appropriate check-box – this is not as silly as it sounds, since bar graph and limit-checking operation continues. Next to this, the number of significant figures or decimal places can be specified as required. The units of the Result can be tied to unit changes made in the source dialogue box, or can be fixed to a desired unit; units can be displayed or hidden. The channel to which the Reading applies can be allowed to follow the Signal Analyzer selection ('selected'), or the opposite channel to the Signal Analyzer selection ('unselected') or can be locked on the A or B channel.

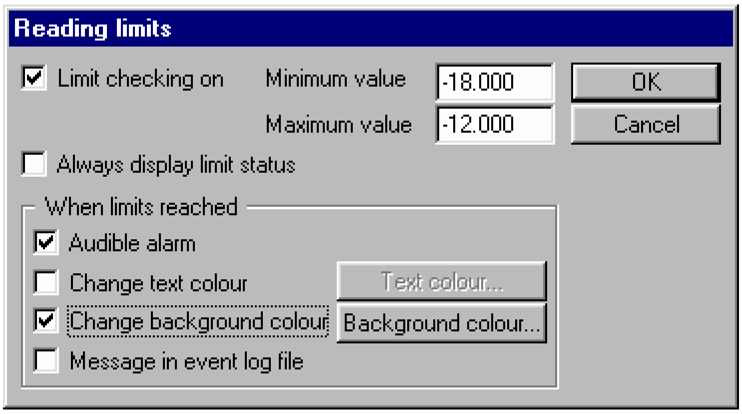
The central ruled area controls display of the maximum and minimum values. These are the maximum and minimum values attained by the Reading since it was created or last reset using the [Reset] button. Checking 'Show min & max values' causes these values to be shown below the current value in the Reading, in a small font, with minimum on the left and maximum on the right. Note that the max/min values are retained but not updated whilst 'Show min & max' is unchecked, and not reset when it is rechecked. This allows, for example, a VBScript to collect minimum and maximum excursions over a range of different tests, e.g. over many channels.

The ruled area at the bottom of the box controls the Reading's optional bar graph display; the range and number of segments can be set, and the colour can be changed using the [Bar colour...] button. Checking the 'Show min & max values' check-box causes the maximum and minimum excursions of the Reading (as discussed above) to be displayed as white markers on the bar graph. Checking 'Show limits' causes the upper and lower limits applied to the Reading (using the Reading Limits dialogue box) to be displayed as red markers on the bar graph.

Note that the box has to be closed with the [OK] button before any adjustments are adopted by the Reading. Alternatively, close the box with [Cancel] to discard the changes.

4.12.2 Reading Limits dialogue box

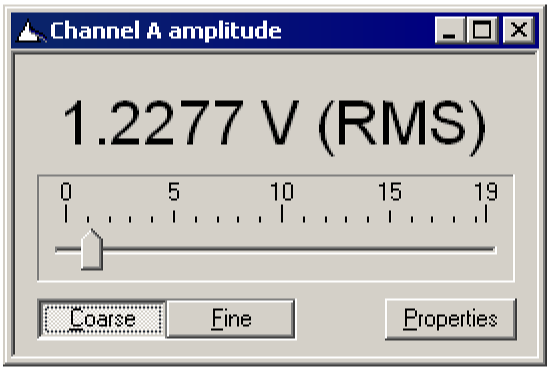
The Reading Limits dialogue box is used to place limits on the Result displayed in its associated Reading window, and to control the action of the dScope in the event of those limits being exceeded.



With limit checking enabled, limits can be entered, and the limit status can be displayed continuously on the Reading ('high', 'OK', or 'low') if required. If either limit is breached, a variety of actions can be selected: the background and/or text colour can change to pre-arranged alternatives, an audible alarm can sound, and an entry can be made in the event log file (including time of breach). In addition, the breach event can provide a causal input to the Event Manager, allowing much more complex responses to be programmed.

4.13 Slider control

A Slider control can be created from a dScope setting by right-clicking over the setting and selecting 'Show Slider control' from the drop-menu.



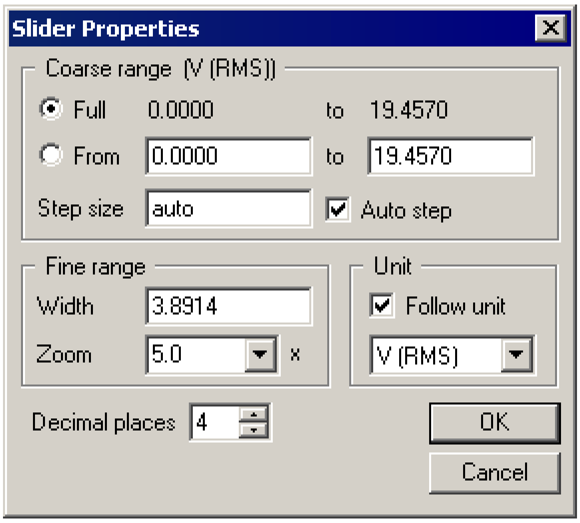
The setting can then be progressively changed by dragging Slider, or in steps by clicking on either side of the Slider. The [Coarse] and [Fine] buttons switch between the coarse and fine steps/ranges.

Control over various of the Sliders properties are gained by opening the Slider Properties dialogue box using the [Properties] button.

NB: Slider controls can currently only be applied to the Signal Generator Amplitude and Frequency settings.

4.13.1 Slider Properties dialogue box

The Slider Properties dialogue box controls the attributes of its associated Slider control.



The coarse range can be set to cover the entire permissible range of the setting, or can be limited as desired. Its step size can be entered manually or can be automatically set - the step size controls the resolution of the adjustment when the slider is dragged or stepped.

The fine range is defined by its range in the selected units, and also by a zoom factor over the coarse range; the two displays are ganged together. The width or zoom factor can be entered directly, or some round zoom factors can be selected from a drop-menu.

The units of the slider contro can be set to follow the units of the parent setting, or can be over-ridden to any other appropriate unit depending on the setting.

The number of decimal places to which the slider is resolved and displayed can be set between 0 and 4.

4.14 Status bar

The Status bar is where the dScope displays various indicator 'tiles' and also warnings about the current state of the instrument; it is also possible to click on the various tiles to quickly open the appropriate dialogue box in which the warning can be further explained, or removed by changing the appropriate settings. The various Status bar tiles are explained below, running across the Status bar from left to right:

Message tile

The left-most section of the Status bar displays messages at various times and according to the state of the dScope. In general, there are two types of message:

Help messages (shown in black) describe the function of the main menu drop-list items as they are hovered over.

Warning messages (shown in red) describe unusual or erroneous modes of operation which the user may have selected accidentally and which may be preventing normal operation of the instrument.

The warning messages which appear on the Status bar message tile form part of a more extensive warning system within dScope, which is controlled by the Warnings dialogue box.

Analogue Outputs Range tile

This tile indicates the signal status of the Analogue Outputs; possible indications are:

(grey background) Indicates that the Analogue Outputs are operating normally.
(red background) Indicates that the Analogue Outputs are muted because the selected amplitude or frequency of the Signal Generator exceeds their range.

Double-clicking on the Analogue Outputs Range tile brings up the Signal Generator dialogue box.

Digital Outputs Range tile

This tile indicates the signal status of the Digital Outputs; possible indications are:

(grey background) Indicates that the Digital Outputs are operating normally.
(red background) Indicates that the Digital Outputs are muted because the selected amplitude or frequency of the Signal Generator exceeds their range.

Double-clicking on the Digital Outputs Range tile brings up the Signal Generator dialogue box.

Digital Outputs Source tile

This tile indicates the selected source for the Digital Outputs; possible indications are:

Indicates that the Digital Outputs are sourced normally from the Signal Generator.
Indicates that the Digital Outputs are 'looped through' from the Digital Inputs, for in-line testing.
Indicates that the Digital Outputs contain the Channel Check sequence, operating at a wordlength of 24 bits.
Indicates that the Digital Outputs contain the Channel Check sequence, operating at a wordlength of 20 bits.
Indicates that the Digital Outputs contain the Channel Check sequence, operating at a wordlength of 16 bits.

Double-clicking on the Digital Outputs Source tile brings up the Digital Outputs dialogue box. Refer to this section for more information.

Analogue Inputs Source tile

This tile indicates the selected source for the Signal Analyzer's Analogue Inputs; possible indications are:

Bal/unbal	Indicates that the Analogue Inputs are sourced normally from the front-panel connectors.
fs jitter	Indicates that the Analogue Inputs are sourced from the Digital Input jitter demodulator, operating in 'fs jitter' mode.
Data jitter	Indicates that the Analogue Inputs are sourced from the Digital Input jitter demodulator, operating in 'data jitter' mode.
Gen	Indicates that the Analogue Inputs are sourced directly from the Analogue Outputs.
Ch A	Indicates that the Analogue Inputs are connected 'pre and post' the EUT's A–channel. Channel A is sourced from the Channel A front-panel connectors, while Channel B is sourced from the dScope's Channel A Analogue Output.
Ch B	Indicates that the Analogue Inputs are connected 'pre and post' the EUT's B–channel. Channel B is sourced from the Channel B front-panel connectors, while Channel A is sourced from the dScope's Channel B Analogue Output.

Note that the tile's background is grey to indicate normal operation of the Analogue Inputs, but is red if the selected terminating impedance has been over-ridden to 100kR because excessive amplitude has been detected which would risk damage to the terminating resistor.

Double-clicking on the Analogue Inputs Source tile brings up the Analogue Inputs dialogue box. Refer to this section for more information.

Digital Inputs Source tile

This tile indicates the selected source for the Signal Analyzer's Digital Inputs; possible indications are:

XLR	Indicates that the Digital Inputs are sourced normally from the front-panel XLR connector.
BNC	Indicates that the Digital Inputs are sourced normally from the front-panel BNC connector.
TOSLINK	Indicates that the Digital Inputs are sourced normally from the front-panel TOSLINK connector.
Gen XLR	Indicates that the Digital Inputs are sourced directly from the XLR Digital Output.
Gen BNC	Indicates that the Digital Inputs are sourced directly from the XLR Digital Output.

Note that the tile's background is grey to indicate normal operation of the Digital Inputs, or red if no digital input carrier is detected on the selected source ('input unlocked'). A yellow background indicates that one or more of a number of warning conditions of the Digital Input have been detected. These are 'biphase violation', 'block-length error', 'eye-narrowing near-fail', 'asynchronous wrt generator' or an error in Channel Check operation if selected.

Double-clicking on the Digital Inputs Source tile brings up the Digital Inputs dialogue box. Refer to this section for more information.

Digital Inputs Channel Status tile

This tile contains the legend "CS", normally with a grey background. The background is red if either the Digital Input is unlocked or if 'inconsistencies' in the incoming Channel Status are detected (as defined in the 'highlight inconsistencies' function of the Input Channel Status dialogue box).

Double-clicking on the Digital Inputs Channel Status tile brings up the Input Channel Status dialogue box. Refer to this section for more information.

FFT Progress tile

This tile shows a red progress bar for acquisition and calculation of the FFT buffer. If the FFT Analyzer is operating in two-channel mode, two bars are shown – the upper bar indicates the A–channel and the lower the B–channel. If FFT averaging is enabled, the tile also contains an indication of the number of averages completed.

Sweep Progress tile

This tile shows a yellow progress bar for Sweeps. If the Sweep is 'sensing', i.e. Sweep points are generated by detecting changes in amplitude or frequency of the Analyzer signal, the message "Sensing..." is displayed in place of the bar, since the dScope is not aware of the number of points to completion.

Page tabs

The main part of the dScope window contains the currently open dialogue boxes, Readings, Trace window etc. This area is notionally arranged as five different 'Pages', one of which is selected for viewing using the Page tabs on the right-hand side of the Status bar.

This facility allows different objects to be arranged on different Pages to alleviate the limits of the screen size. In general, any dialogue box, Reading etc. can be opened on more than one Page if desired.

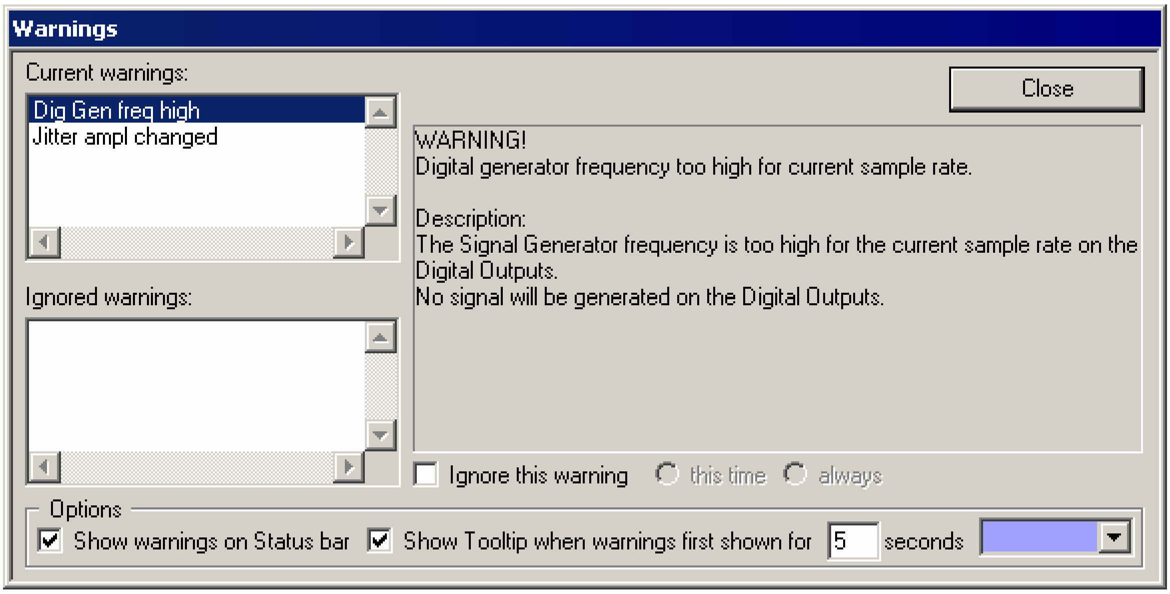
Page tabs are designated in a bold font to show that the Page has some content; empty Pages are designated in a lighter font.

4.14.1 Warnings dialogue box

The Warnings dialogue box works in conjunction with the Status bar to provide a comprehensive warning system, protecting the user against the possibility of unnoticed error conditions, incorrect settings etc.

The dScope is continuously checking for incorrect or inconsistent operating conditions. When new problems are encountered, they are generally flagged by the appearance of a 'balloon' Tooltip, containing a brief description of the condition, at the left-hand end of the Status bar. The Tooltip disappears after a few seconds, but a short description of the condition remains in red on the Status bar whilst the condition persists.

The Warnings dialogue box can be accessed by double-clicking on the message tile (left-and end) of the Status bar.



The Warnings dialogue box contains a list of all current warnings, along with a more detailed description of their meaning and consequences.

By highlighting individual warnings, it is possible to cause them to be ignored (not appear on the Status bar, nor generate a Tooltip). The warning can be set to be ignored either just for the duration of this occurrence, or each time it occurs from now on.

The 'Options' area at the bottom of the dialogue box contains several global settings which apply to all warnings. Check boxes control whether warnings are displayed on the Status bar, and whether they cause the appearance of a Tooltip. The default state of each of these check boxes is ON. The duration of the Tooltips can also be controlled. The coloured drop-list in the bottom right-hand corner allows control of the colour of the Tooltips.

4.15 Icons and Hotkeys reference

The following reference tables of icons and Hotkeys are available:

Hotkeys (shortcut keys)
Main Toolbar icons
Trace Toolbar icons
Carrier Display Toolbar icons

4.15.1 Hotkeys (shortcut keys)

The following Hotkeys are available in the dScope application:

General

F1	On-line help for the current window, dialogue box or panel
F2	Keypress for manual triggering of Event Manager
F3	Keypress for manual triggering of Sweep point
F4	Turns FFT Analyzer trigger on/off
F5	Toggles Digital/Analogue selection for Signal Analyzer source
F6	Toggles A/B/both channel for FFT Analyzer & Trace window
F7	Starts Sweep (with 'Append' turned OFF)

F8	Ends Sweep
F9	Starts Sweep (with 'Append' turned ON)
F11	Mutes Monitor Outputs
F12	Mutes Analogue and Digital Outputs
Shift + F2	Turns FFT averaging on/off
Shift + F3	Turns time domain averaging on/off
Shift + F4	Play user wavetable
Shift + F12	Stops a running Automation script
Ctrl + 1/2/3/4/5	Switch to Page 1/2/3/4/5
Ctrl + Tab	Cycle to next Page

Trace window / Carrier Display

The following Hotkeys apply to the Trace window and, where appropriate, to the Carrier Display; they are only functional when that window is selected.

Ctrl + F2	Marks harmonics on an FFT Trace
Ctrl + F3	Auto-zoom X–Range for current Trace
Ctrl + F4	Auto-zoom Y–Range for current Trace
Ctrl + F5	Turns Cursor on/off for current Trace
Ctrl + F6	Turns Cursor relative-mode on/off for current Trace
Ctrl + F7	Cycles current-selection through all enabled Traces
Plus	Adds a new Trace
Shift + Plus	Adds a Mark at the Cursor position for the current Trace
Alt + Plus	Turns the Cursor on for the current Trace
Minus	Removes current Trace
Shift + Minus	Removes selected Mark
Alt + Minus	Turns the Cursor off for the current Trace
Ins	Adds a new Trace
Shift + Ins	Adds a Mark at the Cursor position for the current Trace
Alt + Ins	Turns the Cursor on for the current Trace
Del	Removes current Trace
Shift + Del	Removes selected Mark
Alt + Del	Turns the Cursor off for the current Trace
Ctrl + Shift + Del	Removes all Marks from the current Trace
Up	Scrolls current Trace up
Alt + Up	Shifts Carrier Display Cursor up
Down	Scrolls current Trace down
Alt + Down	Shifts Carrier Display Cursor down
Home	Moves current Trace X–Range to start of buffer
Shift + Home	Selects first Mark on current Trace
Alt + Home	Moves Cursor of current Trace to start of buffer
End	Moves current Trace X–Range to end of buffer
Shift + End	Selects last Mark on current Trace
Alt + End	Moves Cursor of current Trace to end of buffer
Left	Scrolls current Trace left
Shift + Left	Selects previous Mark on current Trace
Alt + Left	Moves Cursor left on current Trace
Right	Scrolls current Trace right
Shift + Right	Selects next Mark on current Trace
Alt + Right	Moves Cursor right on current Trace
Ctrl	Speeds up steps for Cursor and Trace movement

Windows

The following standard Windows Hotkeys apply as normal within the dScope.

F1	On-line help for the current window, dialogue box or panel
Alt + F4	Quits the application
Shift + F10	Views the shortcut menu for the selected item
Ctrl + Esc	Displays the Start menu
Alt + Tab	Switches to the last selected window
Ctrl + X	Cut
Ctrl + C	Copy
Ctrl + V	Paste
Del	Delete
Ctrl + Z	Undo

User-assignable keys

Function key combinations Alt + F1 ... Alt + F12 have been reserved for future use as Hotkeys for buttons on the User bar (with the exception of Alt + F4 which is used by Windows).

4.15.2 Main Toolbar icons

The following icons are available to be placed on the dScope Main Toolbar. The contents of this Toolbar can be customized using the Customize Toolbar dialogue box available from the Utility menu.

Loads a saved dScope Configuration file
Saves the current dScope Configuration to exiting filename
Saves the current dScope Configuration to new filename
Context-dependent printing of the selected dialogue box
Displays a print-preview screen of the selected dialogue box
Opens the Digital Outputs dialogue box
Opens the Digital Output Carrier dialogue box
Opens the Analogue Outputs dialogue box
Opens the Soundcard Outputs dialogue box
Opens the Output Channel Status dialogue box
Mutes / unmutes the Digital Outputs
Mutes / unmutes the Analogue Outputs
Mutes / unmutes the Soundcard Outputs
Opens the Digital Inputs dialogue box
Opens the Digital Input Carrier dialogue box
Opens the Analogue Inputs dialogue box
Opens the Soundcard Inputs dialogue box
Opens the Input Channel Status dialogue box
Opens the Monitor Outputs dialogue box
Mutes / unmutes the Monitor Outputs
Opens the Signal Generator dialogue box
Mutes / unmutes the entire Signal Generator
Plays a single cycle of a User wavetable or Swept sine

[Continued on following page]

[Continued from previous page]

Selects Digital Inputs to the Signal Analyzer
Selects Analogue Inputs to the Signal Analyzer
Selects Soundcard Inputs to the Signal Analyzer
Sets the FFT Analyzer (Trace window and Detectors) to A–channel only
Sets the FFT Analyzer (Trace window and Detectors) to B–channel only
Sets the FFT Analyzer (Trace window and Detectors) to 2–channel mode
Turns FFT Analyzer trigger on
Turns FFT Analyzer trigger off
Turns time-domain averaging on / off
Turns FFT averaging on / off
Opens the Signal Analyzer dialogue box
Opens the Continuous-Time Detector dialogue box
Opens the FFT Parameters dialogue box
Opens the Impulse Response Parameters dialogue box
Turns impulse response mode on / off
Opens the Trace window
Starts a Sweep, or resumes a paused Sweep
Stops a running Sweep
Opens the Sweep Setup dialogue box
Opens the Sweep Settling dialogue box
Opens the Sweep Data Table Entry dialogue box
Opens the Regulation dialogue box
Opens the Event Manager dialogue box
Runs an Automation script
Stops a running Automation script
Starts recording an Automation script from user operations (NB: not yet supported)
Opens the Script Edit window
Opens the Multi-tone Generation and Analysis dialogue box
Adds a new Memo to the desktop
Opens the Memo List
Opens the Contents page of the on-line Help file

4.15.3 Trace window icons

The following icons appear on the Trace Toolbar:

Opens the 'Add Trace' dialogue box, to add a new Trace or load a saved Trace
Removes the current Trace
Saves the current Trace to a file
Makes a snapshot copy of the current Trace in the Trace window
Toggles two-channel display mode between dual and single axes
Sets the graph printing and exporting parameters
In-place annotation of the current Trace for printing (not currently supported)
Changes current Trace settings (X&Y scales and Limit Line associations)
Presents the current Trace as a numeric list for export to a csv or tsv file (Note: displayed data has limited resolution to aid readability; exported data has full double-precision resolution)
Zooms in X–range of current Trace
Zooms out X–range of current Trace
Auto-zooms the X–range for the current Trace

Moves the X–range of the current Trace to the start of the buffer
Scrolls the current Trace left
Scrolls the current Trace right
Moves the X–range of the current Trace to the end of the buffer
Zooms in Y–range of current Trace
Zooms out Y–range of current Trace
Auto-zooms the Y–range for the current Trace
Reverts to previous X&Y scales
Scrolls the current Trace up
Scrolls the current Trace down
Turns the Cursor (and Cursor Toolbar) on/off for the current Trace
Turns the Mark (and Mark Toolbar) on/off for the current Trace
Displays the current Trace after selected transform operations
Creates an upper or lower Limit Line for the current Trace
Enters Impulse Windows Edit mode (see Principles of impulse response analysis)

4.15.4 Carrier Display icons

The following icons appear on the Carrier Display Toolbar:

Changes Carrier Trace settings (X&Y scales and drawing modes)
Restarts acquisition of Carrier Trace
Zooms in X–range of Carrier Trace
Zooms out X–range of Carrier Trace
Auto-zooms X–range of Carrier Trace
Moves X–range to start of AES3 frame
Scrolls Carrier Trace left
Scrolls Carrier Trace right
Moves X–range to end of AES3 frame
Zooms in Y–range of Carrier Trace
Zooms out Y–range of Carrier Trace
Auto-zooms Y–range of Carrier Trace
Scrolls Carrier Trace up
Scrolls Carrier Trace down
Turns AES3 eye-opening template on/off
Turns Cursor on/off

4.16 Amplitude units in dScope

This section describes the way amplitude units are handled by the dScope's Signal Generator and Signal Analyzer.

There are aspects of the dScope's treatment of amplitude units which work intuitively rather than strictly correctly – there are places where either no correct treatment is possible, or where the correct treatment proves counter-intuitive.

These cases mostly result from conflicts between peak and RMS responses and units – some units are inherently 'peak' units (i.e. %FS, FFS, Hex, Vpeak) and others are inherently 'RMS', i.e. dBFS, V(RMS), dBu, dBV, dBm, dBSPL and W. Note that for the purposes of this discussion, 'peak sample' and 'Q–peak' responses behave in the same way with respect to units as the 'peak' case.

Signal Generator

The amplitude setting of the Signal Generator defines the peak amplitude of the signal. When entering the amplitude in 'peak' or 'peak-to-peak' units, operation is straightforward; when entered in an 'RMS' unit, it is 'sine peak referred' – i.e. the peak output of the Generator is set to the same peak amplitude as a sine function with the entered RMS value. Clearly, this interpretation means that for non-sine functions, the actual RMS amplitude of the output does not usually reflect the value entered.

The main reason for this is that if the generated RMS amplitude were to actually reflect the entered RMS value, this would be counter-intuitive in most circumstances. Changing the generated function would result in a change in the peak amplitude of the generated signal which, whilst strictly correct, could be confusing. This problem is exacerbated by the flexibility of the Generator which allows, for example, the duty cycle of the pulse function to be varied. Entering the amplitude in 'RMS' units, and correctly applying the pulse's crest factor would result in the peak amplitude of a pulse varying with the duty cycle.

Finally, it was desired that operation be consistent with other equipment as far as possible. Equipment with versatile dual-domain function generation tends to work in a 'sine peak referred' manner.

⚠ **NOTE: An exception to the above is the case of compound stimuli such as multi-tones, twin-tones and 'Bin centres'. In this case, the amplitude of EACH TONE is specified separately in the desired units. Whilst the RMS amplitude of the compound is the RMS summation of the amplitudes of each of the tones, the peak or peak-to-peak amplitude of the compound is arbitrary depending on the frequency and phase combination of the tones.**

Whilst Signal Generator amplitude settings in dScope generally take no account of load impedance (they specify the amplitude behind the source impedance, rather than that across the terminals in the loaded state), impedance-related units such as W and dBm (which assume a load impedance equal to the 'reference impedance' - see below) may specify terminal amplitudes in the loaded state if required, according to a setting in the Options dialogue box.

Signal Analyzer

The Signal Analyzer generally works 'correctly' in that RMS, peak and peak-to-peak Results are true RMS, peak and peak-to-peak measurements respectively. However, there are two situations in which confusion may arise.

Firstly, it is possible to set the response and the units of the Continuous-Time Detector explicitly and independently. This means that it is possible to select an 'incompatible' unit, for example peak response may be selected along with dBu units. This selection is, strictly speaking, meaningless, and dScope deals with the problem by simply converting the value into the relevant unit, ignoring the fact that it may be using the wrong response. This was felt to be preferable to preventing selection of incompatible response and units since this can on occasion be useful.

Secondly, there is the question of the Y–scales of Scope Traces. Applying the oscilloscope analogy, it would not be possible to select an 'RMS unit' since an instantaneous point on the waveform has no RMS value. However, such selections are allowed; again, the dScope simply converts the sample value into the selected unit.

Confusion may result from the odd definition of dBFS: "a 0dBFS signal has the same RMS amplitude as a sinewave whose peaks exactly reach +/– digital full scale". Because of this, peak *OR* RMS values can be displayed in dBFS, with sine-peak-referral being used to convert Peak values to dBFS. For example, if dBFS units are selected, a 0dBFS sine function is displayed with its peaks aligned with the 0dBFS points on the Y–scale, i.e. 0dBFS is treated as the instantaneous full-scale sample point. A full-scale square function would also be displayed with its peaks aligned with the 0dBFS scale marks, but its amplitude is actually 3.01dBFS! In the case of analogue RMS units, a different rule is applied in the interests of intuitive behaviour: if a sine function is displayed, the Y–scale value which corresponds to its RMS amplitude occurs at 0.7071 of the peak height of the waveform.

Note that the amplitude Results in the main Signal Analyzer are always measured with an RMS response. If a 'peak unit' is selected, the RMS amplitude will be directly converted – for example, a digital sine function with a full-scale peak amplitude will show an RMS amplitude that is 3.01dB below this, or 70.7%FS. The same applies to FFT Detector Results and FFT Traces, which are inherently RMS because of the FFT process.

Soundcard Generation and Analysis

When generating or analyzing using Soundcard I/O, the amplitudes of audio samples transacted with the Soundcard are treated in exactly the same way as with the dScope's Digital Inputs and Outputs. That is to say, a signal at 0dBFS corresponds to the maximum amplitude of the Soundcard.

In the case of an analogue Soundcard, it is possible to use analogue amplitude units for Soundcard I/O by setting the 'D/A line-up' parameter of the Signal Generator and/or Signal Analyzer to the analogue amplitude corresponding to the Soundcard's maximum. However, it should be noted that these parameters also control the alignment between the dScope's own Analogue Inputs and Analogue Outputs and their Digital counterparts, so if D/A line-up is used for analogue Soundcard alignment, then the dScope's Analogue I/O will become similarly aligned.

Part 5

Hardware reference

5 Hardware reference

For a description of the dScope's hardware module go to Hardware layout.

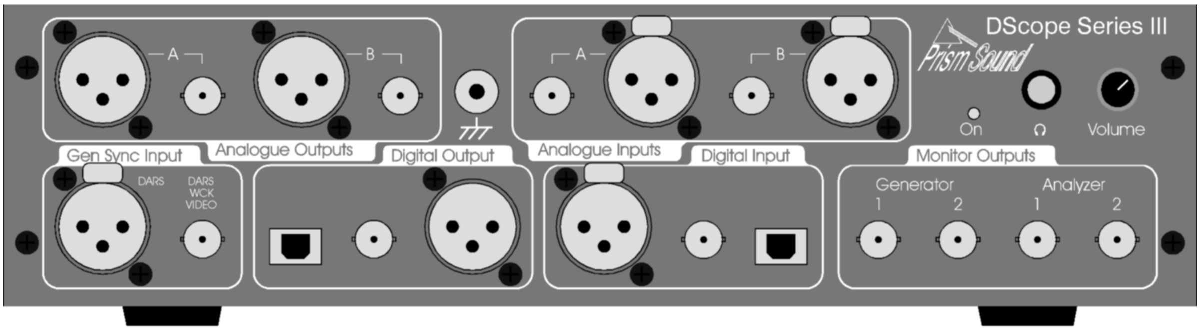
For a detailed description of the dScope architecture, including block diagrams of all sections, go to Architecture.

For details of jumper options, go to PCB jumper options.

For locations and type details of fuses, go to Fuses and ratings.

5.1 Hardware layout

The dScope hardware module is a compact unit which interfaces to the host PC (desktop or notebook) via a USB interface. A custom flight case is available which holds the dScope and a notebook PC along with test cables and accessories. For rack mount applications, a 2U 19" mounting kit is available.



Front Panel

The Analogue Outputs and Inputs for the A and B channels are arranged along the upper section of the front panel. Both XLR and BNC connectors are provided, which are connected in parallel. RCA/phono connections can be made using the adapters supplied.

NB: **The outer conductors of the Analogue Input and Output BNC connectors are connected to the inverting leg of the balanced input or output circuit (i.e. to pin 3 of the associated XLR) and not to chassis or signal ground (although the Analogue Outputs can be switched into this mode). See the Unbalanced operation and grounding section for more information**

A chassis post for grounding equipment under test is positioned between the Outputs and the Inputs.

The Digital Outputs and Inputs occupy the lower section of the panel, with the Digital Output Reference Sync inputs on the left. The Digital Outputs are to the AES3 or S/PDIF two-channel standard, and are provided on XLR, BNC (or RCA/phono via adapter supplied), and TOSLINK connector formats. Whilst these all carry the same data, their carrier parameters are adjustable separately. The Digital Input can be accepted in any of the same formats, but the desired input connector must be selected in the dScope software. A Reference Sync for the Digital Outputs can be input as AES11 (DARS) on the XLR connector, whilst the BNC can accept AES11, S/PDIF, Wordclock or video references.

In the lower right-hand corner of the panel are four assignable monitor BNC connectors, two associated with the Signal Generator and two with the Signal Analyzer. These can be assigned many different functions in software, as may be required for different tasks. They have a 75R output impedance, and can carry either analogue audio and digital carrier bandwidths interchangeably.

Above the monitor BNCs are the audio monitor volume control and headphone socket. The audio monitor loudspeaker is located in the right-hand side panel of the unit, and is automatically cut when headphones are connected. Like the monitor BNCs, the audio monitor routing is assignable within the dScope software.

Near the headphone socket is a bi-coloured power LED. This illuminates red when the dScope is in 'standby' mode – i.e. when power is applied but the dScope software has not switched the unit on. When the unit is active, the LED lights green.

Rear Panel

On the left of the rear panel (viewed from the rear) is the mains inlet, incorporating a switch, fuse and voltage selector.

The voltage selector must be appropriately set for the regional supply voltage, otherwise damage to the unit may result.

A cooling fan is located in the middle of the rear panel – this must not be covered or else the unit may shut down due to overheating.

On the right of the fan is the host connection panel, which incorporates the USB socket for connection to the host PC, the 'dS–NET' serial connector for use with switching boxes and other peripherals, plus a bank of four DIP switches with, as yet, no function.

On the extreme right of the panel are the Digital Output Reference Sync output connectors: an AES11 output on an XLR connector and a Wordclock on BNC. These are driven with the same frame rate as the Digital Outputs, but remain unaffected by variations in amplitude, jitter etc. which may be applied to the Digital Outputs themselves.

5.1.1 Changing the mains voltage or fuse

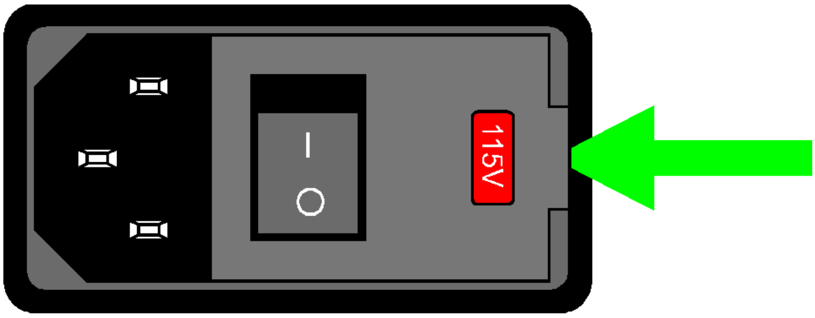
The mains (line) fuse and the mains voltage selector are carried within the IEC inlet on the rear of the dScope. The required fuse is a 2AT 20x5mm type. The voltage selector has two positions: '115V' covers the range 90VAC to 125VAC, and '230V' covers the range 180VAC to 250VAC. The diagram below shows how to replace the fuse or change the voltage selection.

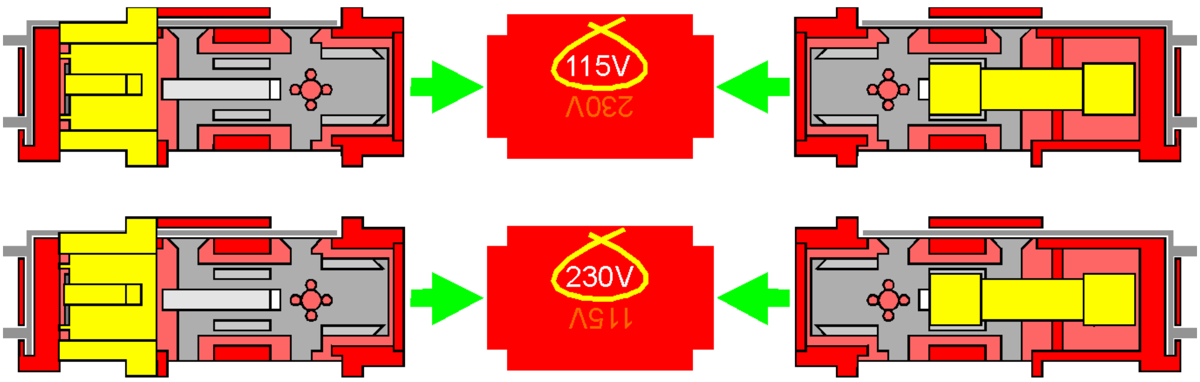
Changing the fuse

To access the fuse holder, first remove the IEC plug from the IEC inlet, then lever up the voltage selector / fuse holder cover by inserting a small, flat-bladed screwdriver into the slot as shown in the diagram below. Remove the red plastic holder. Use a 20mm x 5mm fuse of the same type and rating, placing it in the same location as the old fuse. Replace the red plastic holder in the same orientation as before (with the selected voltage AWAY from the switch) and close the cover.

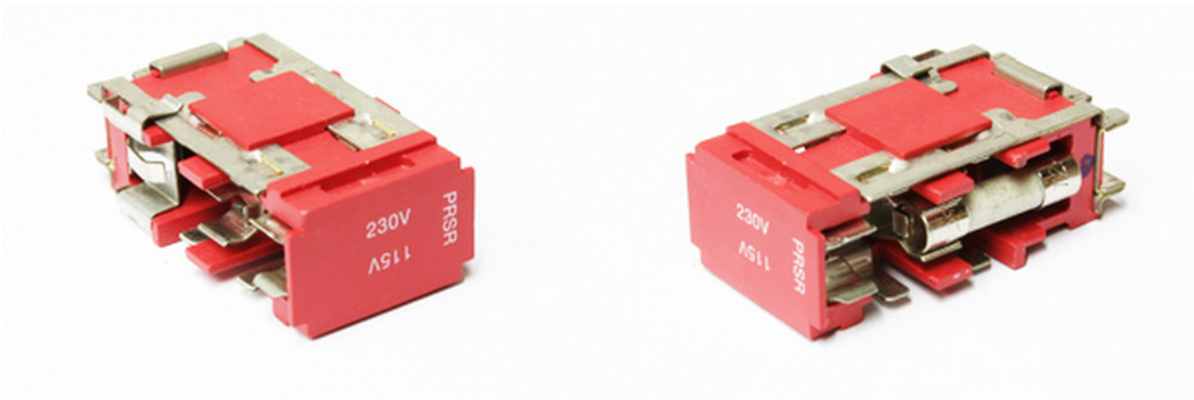
Changing the voltage selection

Following the instructions above for accessing the fuse holder, orient the red plastic holder with the desired voltage uppermost as shown in the diagram below. Swap the positions of the fuse and the small metal clip so that the clip is on the left and the fuse is on the right. Replace the red plastic holder in the new orientation (with the selected voltage AWAY from the switch) and close the cover. Check that the desired voltage is visible through the window. Note that unless the holder is inserted in the right orientation to match the positioning of the fuse and clip, the cover cannot be fully closed.

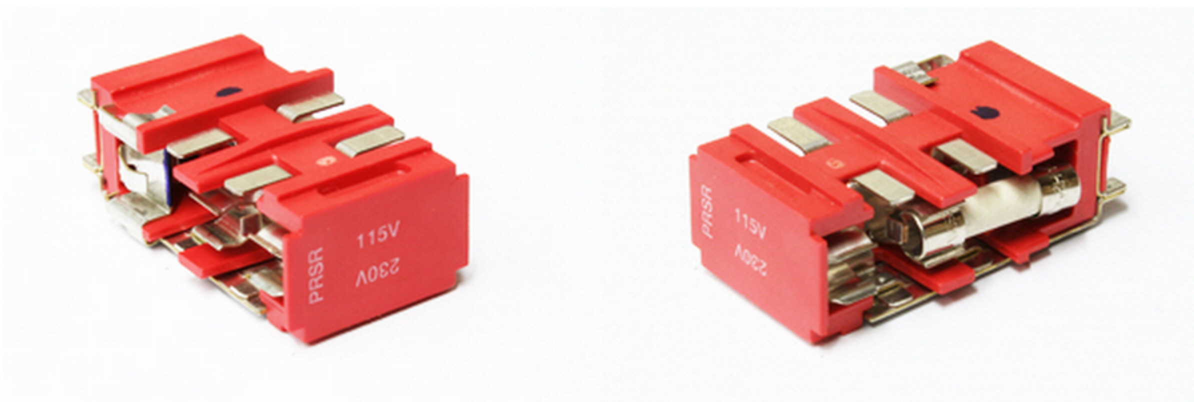




Configured for 230V operation



Configured for 115V operation



5.1.2 Unbalanced operation and grounding

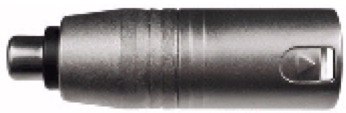
The outer conductors of the Analogue Input and Output BNC connectors are connected to the inverting leg of the balanced input or output circuit (i.e. to pin 3 of the associated XLR) and not to chassis or circuit ground (although the Analogue Outputs can be switched into this 'unbalanced' mode).

Depending on the input and output configurations and grounding arrangements of the equipment under test (EUT), it may be necessary to vary the settings of the dScope's Analogue Output, or to provide additional grounding at the dScope's inputs, if optimal measurements are to be made.

This most often applies to EUTs with unbalanced inputs and/or outputs, and especially to EUTs which are not earthed, e.g. consumer equipment with 2–core mains leads. In this case, a large voltage at mains (line) frequency can be present between the signal grounds of the EUT and the dScope which can degrade measurements with residual hum.

When driving an unbalanced EUT input, the dScope's Analogue Outputs should be set to 'unbalanced' mode. In this mode, the outer conductor of the BNC connector and pin 3 of the XLR connector are connected to the Generator's signal ground (pin 1 of the XLR). It may also be advantageous to set the grounding of the dScope's outputs to 'chassis'.

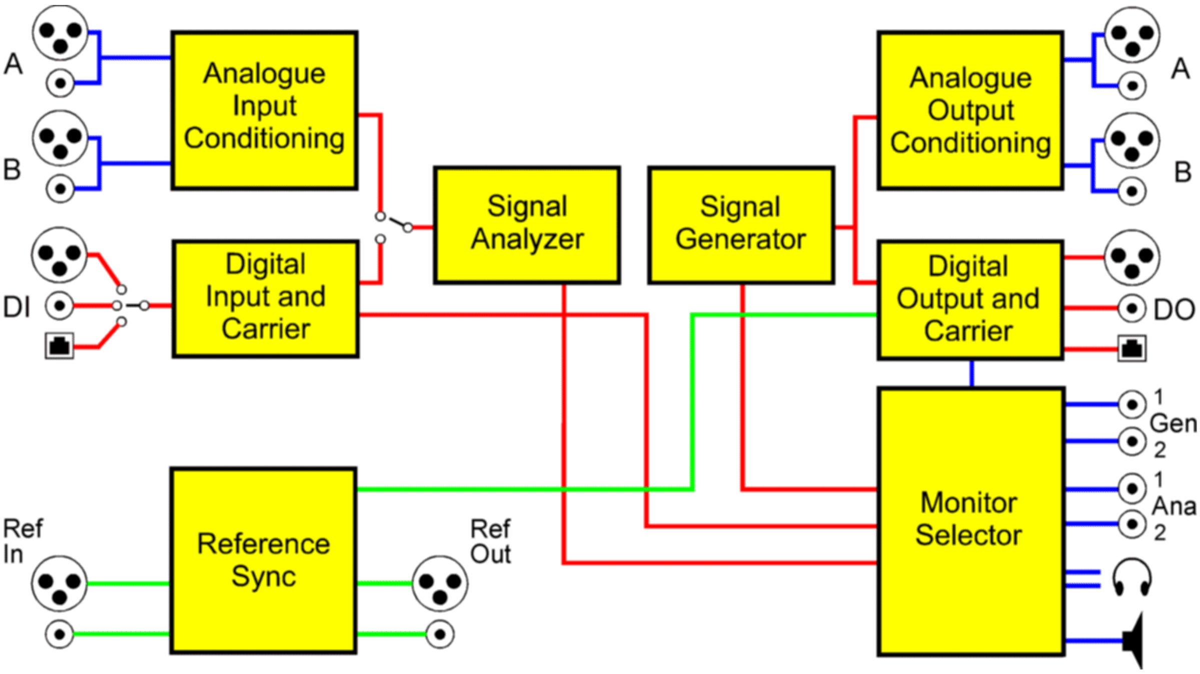
When analyzing an unbalanced EUT output, the dScope's Analyzer can usually operate successfully in the normal balanced mode. However, where a significant difference in ground potentials exists (if the EUT is floating), it may be necessary to provide additional grounding if measurements are not to be compromised by excessive hum. This is most simply achieved by connecting the EUT to the dScope's **XLR** inputs, using the RCA/phono–to–XLR adapters provided.



These adapters connect the outer conductor of the RCA/phono to both pin 1 and pin 3 of the dScope's XLR input, referencing the inverting leg of the Analyzer's input to its signal ground. In unusual cases, it may also be necessary to connect the chassis of the EUT to the dScope's front-panel chassis terminal.

The dScope's analogue Generator and Analyzer have separate signal grounds which are normally commoned with a jumper. Neither of these signal grounds is directly coupled to the chassis, which is connected to mains earth. In unusual circumstances, it may be necessary to separate the Generator and Analyzer signal grounds, as described in the PCB jumper options section.

5.2 Architecture

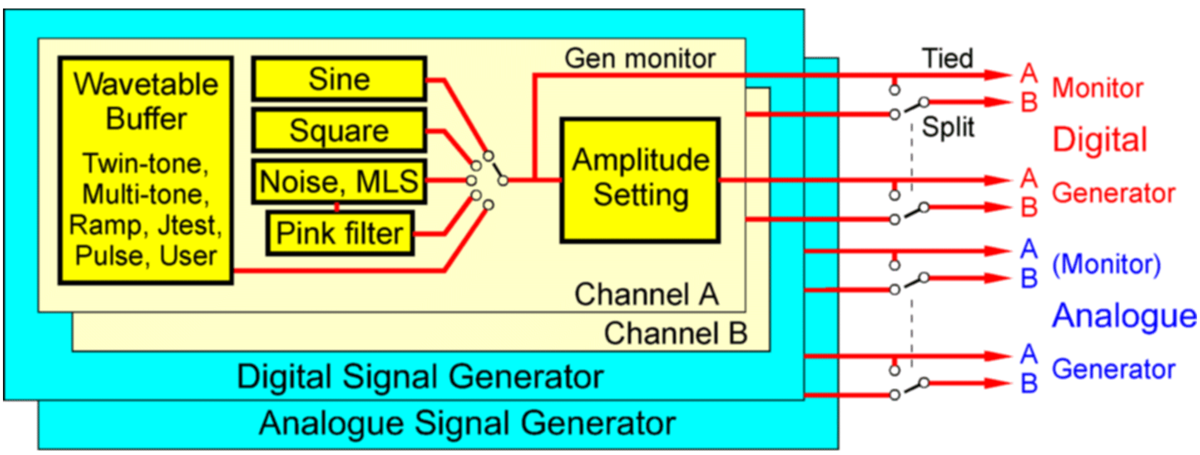


The figure above is a simplified block diagram of the dScope architecture. See the following sections for more detail of the individual blocks.

NB: **Various signal types are represented in this and the other block diagrams as follows:**

Digital audio signals are shown in RED
Analogue audio signals are shown in BLUE
Digital audio carrier signals are shown in BROWN
Synchronization and timing signals are shown in GREEN
Control and status signals are shown in BLACK

5.2.1 Signal Generator architecture



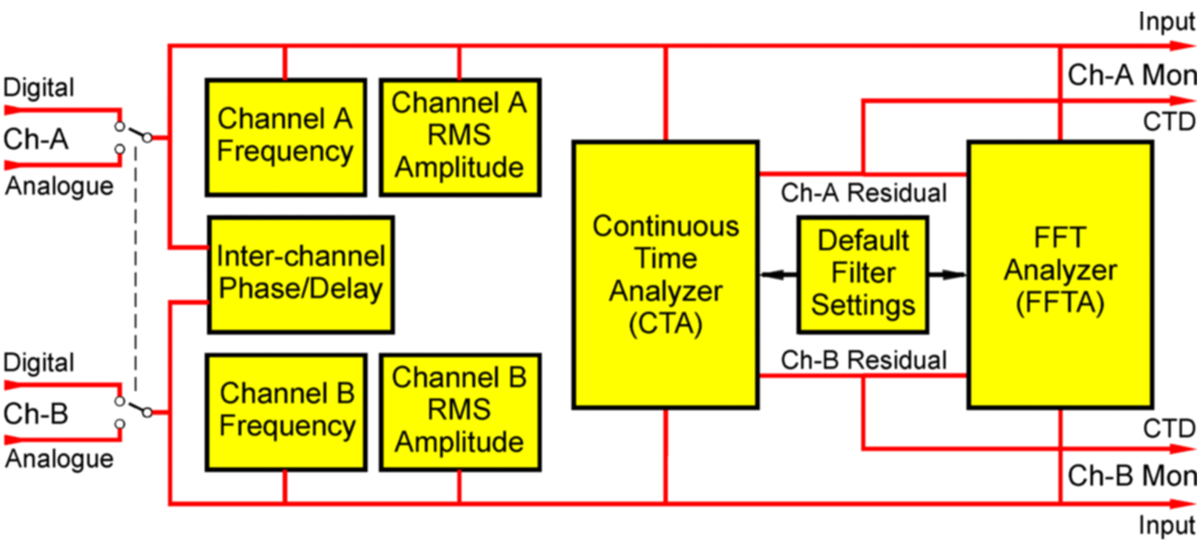
The figure above illustrates the functionality of the dScope's Signal Generator. The various settings are accessed through the Signal Generator dialogue box.

The dScope Signal Generator may be considered as four separate generators: one for each Analogue Output channel and one for each Digital Output channel. In 'Tied' mode, the A and B channel outputs are the same (although they may be turned on and off individually in both domains). In 'Split' mode, each channel may be drive with different amplitude and frequency, or even with entirely different generated functions. Analogue and Digital Outputs are generated simultaneously, and presently the software forces the outputs in the two domains to be driven with equivalent signals, i.e. 'domain split' mode is not possible. When either the analogue or Digital Outputs are set to sample above 96kHz, Split mode operation is not possible.

The Signal Generators use real-time signal processing to generate sine, square and noise-based signals, whereas wavetables are used to generate other functions. The user can generate any arbitrary waveforms by loading the wavetables from files (dScope III '.wfm' format, dScope II '.usr' format, or Windows '.wav' format) or by writing a VBScript to fill the table.

The various Signal Generator outputs are available to the Monitor Outputs, as well as at the main Digital and Analogue Outputs.

5.2.2 Signal Analyzer architecture



The figure above illustrates the functionality of the dScope's Signal Analyzer. Architectures of the Continuous-Time Analyzer (CTA) and FFT Analyzer (FFTA) are detailed in separate sections.

The settings and Results of the Signal Analyzer are accessed through the Signal Analyzer, Continuous-Time Detector, FFT Parameters and FFT Detector dialogue boxes.

The Signal Analyzer is a two-channel analyzer which is switched to analyze either the analogue or Digital Inputs. There is no hardware restriction which prevents cross-domain (i.e. simultaneous analogue and digital) analysis, but no such functions are currently supported in the dScope software. Cross-domain analysis may be offered in the future.

Measurements of signal frequency and RMS amplitude for both channels, as well as inter-channel phase (or delay) are continuously available in the Signal Analyzer dialogue box. A two-channel Continuous-Time Analyzer (CTA) is also continuously available, and an FFT Analyzer (FFTA) which may be enabled for single or dual channel measurements.

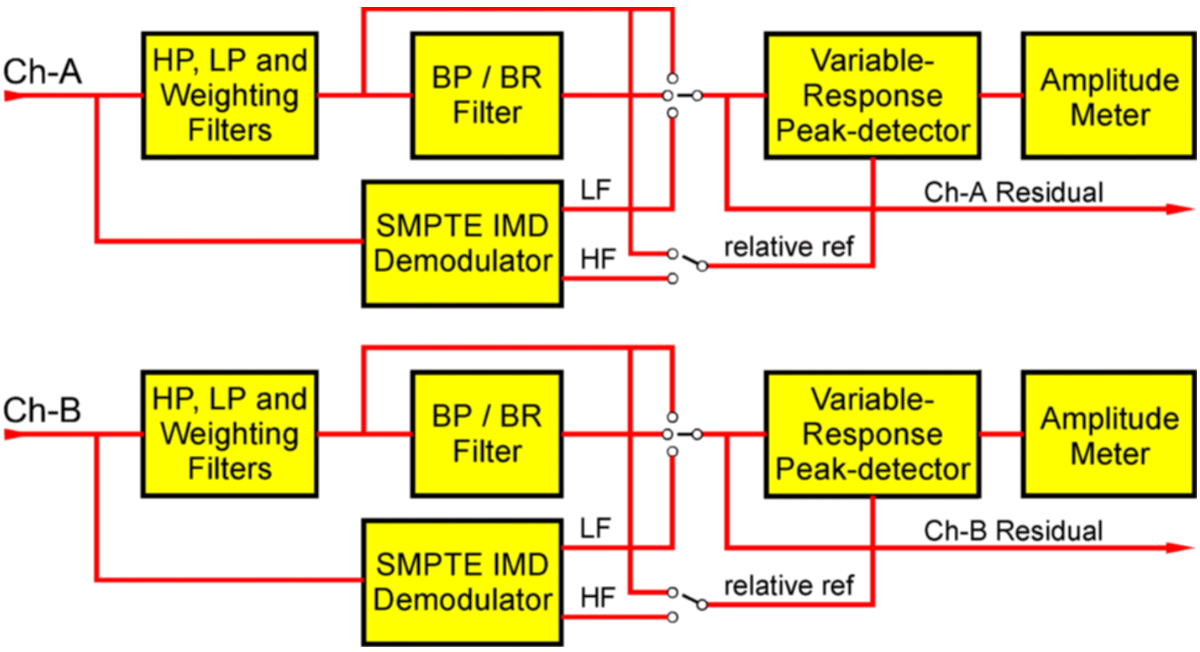
The 'Continuous-Time Analyzer' (CTA) is like a traditional analogue signal analyzer – it can make all the 'standard' measurements, operating continuously so that any momentary change in the input signal is always registered. The dScope has only one CTA (although it operates on both channels simultaneously), so only one type of measurement can be made at a time.

The 'FFT Analyzer' (FFTA) can also make these standard measurements, but it operates differently – by capturing a buffer of samples on activation of an oscilloscope-like trigger. Having captured the buffer of samples, the desired measurement is calculated before re-arming the trigger to capture the next buffer. The FFT Analyzer can perform many more complex functions than the Continuous-Time Analyzer (including calculating 'user-defined' measurements from VBScripts), but its trigger-based nature means that it is slower than the CTA and may miss transitory changes in the input signal which happen between triggerings. The FFTA can calculate up to 40 different (two-channel) Results at once, so it is a powerful way of measuring many parameters simultaneously, for example using multi-tone stimuli as described in the Multi-tone Generation and Analysis section.

Another feature of the FFTA is that its buffers can be displayed graphically in the form of Scope Traces in the time domain or FFT Traces in the frequency domain, continuously and without the need to perform Sweeps. This is a powerful diagnostic tool – the provision of a continuous FFT display with up to 256k points and a large dynamic range makes many fault conditions instantly recognisable, where a simple numerical reading would not.

The inputs of the FFTA can be switched to analyze the residual outputs of the CTA so that, for example, Scope or FFT Traces of distortion residuals can be displayed.

5.2.2.1 Continuous-Time Analyzer architecture



The figure above illustrates the functionality of the dScope's Continuous-Time Analyzer (CTA).

The CTA operates continuously on both Analyzer channels. On entering the CTA, the input signal is filtered according to the selection of high-pass, low-pass and Weighting filters. According to the selected function, the signal is then passed to the peak-detector either directly, or via a band pass or band reject filter, or via the SMPTE IMD demodulator.

The direct feed is used, for example, in basic amplitude measurements, the band reject mode for residual measurements such as THD+N, and the band pass mode for frequency selective measurements or for the exclusion of noise. The selectivity of the BP/BR filter can be selected from a number of different bandwidths (Q–factors). Its frequency can be set to track the detected Analyzer frequency, or the generator frequency, or it can be fixed at a preset frequency.

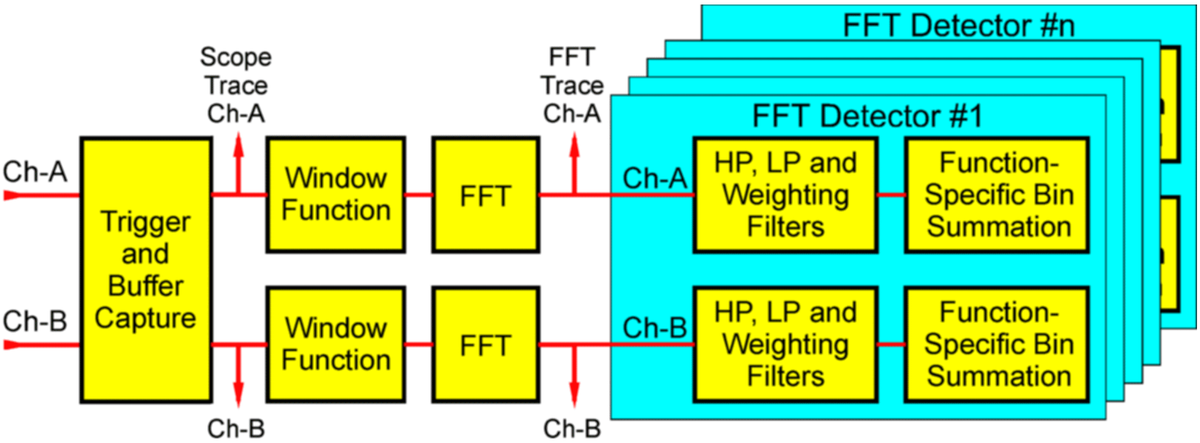
The SMPTE IMD demodulator is used for intermodulation distortion measurements, only in the SMPTE/DIN method where the stimulus is a combined low and high-frequency tone. It comprises a high-pass filter to remove the LF tone, followed by a demodulator circuit which shifts the IMD components around the HF tone to the base-band. The HF tone is then removed with a further high-pass filter, leaving only the demodulated IMD products.

The peak-detector can operate in RMS, peak or peak-to-peak modes, or in a special quasi-peak mode as defined by CCIR468. The latter mode is usually used for noise measurements in conjunction with a CCIR468 Weighting filter. A 'peak-sample' mode is also provided which measures the peak amplitude of the discrete audio samples rather than the interpolated peak amplitude measured in 'peak' mode. The peak detector can work in either absolute or relative mode. In relative mode, the Result is normally expressed relative to the pre-band-reject signal amplitude, as in the THD+N case. However, a number of other options for the relative-reference exist, for example the output of the Signal Generator (for gain measurements), or the Analyzer input of the opposite channel (for cross-talk measurements) or the SMPTE HF tone extraction for use in SMPTE IMD measurements.

The channel A and B residual signals are passed to the FFT Analyzer allowing, for example, display of distortion traces or FFTs of distortion residuals.

For further details of the various operating modes, settings and Results of the CTA, refer to the Continuous-Time Detector dialogue box section.

5.2.2.2 FFT Analyzer architecture



The figure above illustrates the functionality of the dScope's FFT Analyzer (FFTA).

The FFTA operates by capturing a buffer of audio samples which are then processed to display the Scope and FFT Traces, and to calculate Results for up to 40 simultaneous FFT Detectors (FFTDs). Each FFTD can perform different measurements, with different filter settings if required. The FFTA can operate in two-channel mode, or in single-channel mode (which is faster) if required.

On entering the FFTA, the input signal is compared against the user-defined trigger condition. When the trigger condition is satisfied the FFT buffer is filled, with the trigger point located at the selected point in the buffer. Triggering can be continuous or threshold-activated, and can be repetitive or single-shot. A manual setting is also provided where sampling takes place when a key is pressed. The unprocessed buffer is used to display the Scope Trace, if enabled. The buffer is then windowed with the selected FFT Window function, and an FFT is calculated. The FFT data is used to display the FFT Trace, or spectrum, of the audio if enabled, and also to calculate any FFT Detectors which are active. The length of the buffer (and subsequent FFT) can be set between 1k and 256k points. The FFT data can also be successively averaged from a user-defined number of successively captured buffers if required; this allows noise to be reduced and small components to be distinguished in the FFT Trace.

Each FFT Detector may apply high-pass, low-pass and Weighting filters prior to bin summation. The selection of filters may be different for each FFT Detector if required, or all may use a common set of filters. Bin summation then produces the selected FFTD Result according to the summation mode selected. Note that all FFTD Results have an RMS response, owing to the nature of the FFT process.

The bin summation process essentially emulates the BP/BR function of the CTA. For wide-band measurements, all FFT bins are summed; for band pass (selective) measurements, only the bins in a narrow range of frequencies are included; for band reject (residual) measurements, such as THD+N, bins within a frequency range are excluded from the summation. The BP/BR frequency can be set to track the detected Analyzer frequency, or the Generator frequency, or it can be fixed at a preset frequency. A range of different BP/BR bandwidths (Q–factors) is provided in the same way as in the CTD, and in addition there is a 'window-width' mode where the BP/BR filter includes or removes ONLY the specified frequency, with essentially infinite Q, covering only the number of bins which may contain the specified frequency, as defined by the selected Window function. Bin summation Results can be displayed in absolute or relative modes. In relative mode, the Result is normally expressed relative to the pre-band-reject signal amplitude, as in the THD+N case. However, a number of other options for the relative-reference exist, for example the output of the Signal Generator (for gain measurements), or the Analyzer input of the opposite channel (for cross-talk measurements).

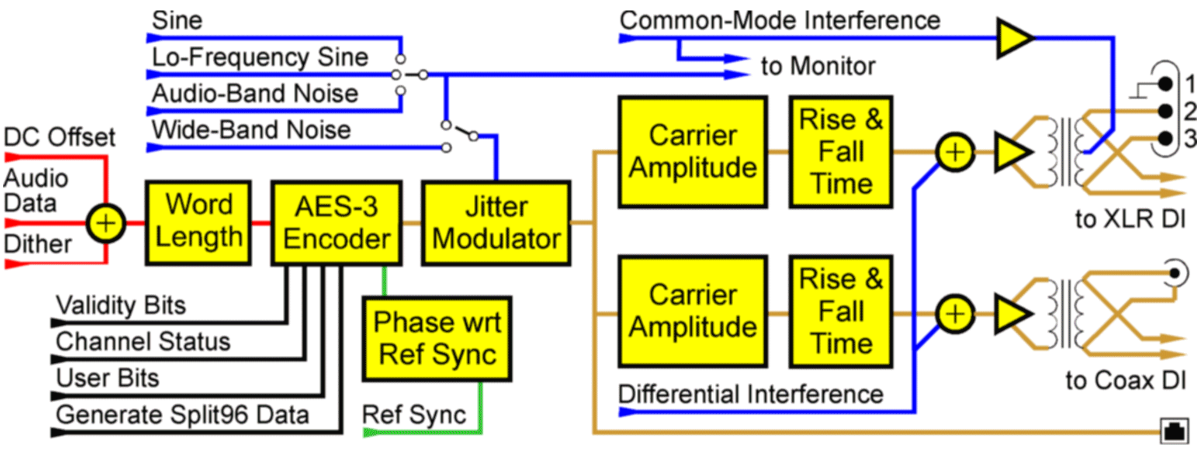
The versatility of the FFTD's bin summation process allows it to perform many measurements which the CTD cannot, for example THD can be measured without including noise by summing only the harmonics of the input frequency, or individual harmonic distortion (e.g. 3rd harmonic) can be measured. By writing a customised 'FFT Detector Calculation script', the user can generate his own FFTD Results by scripting the summation of the FFT (and/or input buffer) bins as he wishes. This facility offers a level of flexibility unrivalled in other instruments. The process is described in the FFT Detector Calculation scripts section of the Scripting Manual.

By using the Multi-tone Generation and Analysis dialogue box, it is possible to automatically create and script a large number of simultaneous FFT Detectors which will extract various Results from a synchronous multi-tone stimulus, which is scripted within the Signal Generator.

For further details of the various operating modes, settings and Results of the FFTA, refer to the FFT Parameters and FFT Detector dialogue box sections.

5.2.3 Digital Output and Carrier architecture

NB: **This section may not be relevant, depending on the dScope model number.**



The figure above illustrates the functionality of the Digital Output and Carrier circuits. The Digital Outputs of the dScope are very versatile, allowing variation of a wide variety of interface parameters and controlled degradation of many aspects of the digital audio carrier.

Digital Output parameters

Output wordlength is adjustable between eight and 24 bits, with optional addition of DC offset and dither.

Transmitted flags including Valid bits, User bits and all Channel Status fields can be set as required.

At high sample-rates, the Digital Output can be configured to operate in 'Split96' (or 'two-wire') mode, where a single channel is transmitted over a single AES3 carrier at a frame rate of half the sample rate.

These settings are controlled through the Digital Outputs dialogue box. Refer to this section for further detail.

Digital Output Carrier parameters

Output Carrier phase with respect to the Reference Sync can be adjusted.

Output Carrier jitter can be added, with sine, wide-band noise or audio-band noise functions. The low-frequency sine function with up to 20UI amplitude is available for testing jitter tolerance to the AES3 template.

Carrier amplitudes and rise/fall times can be set independently for the XLR and coaxial outputs.

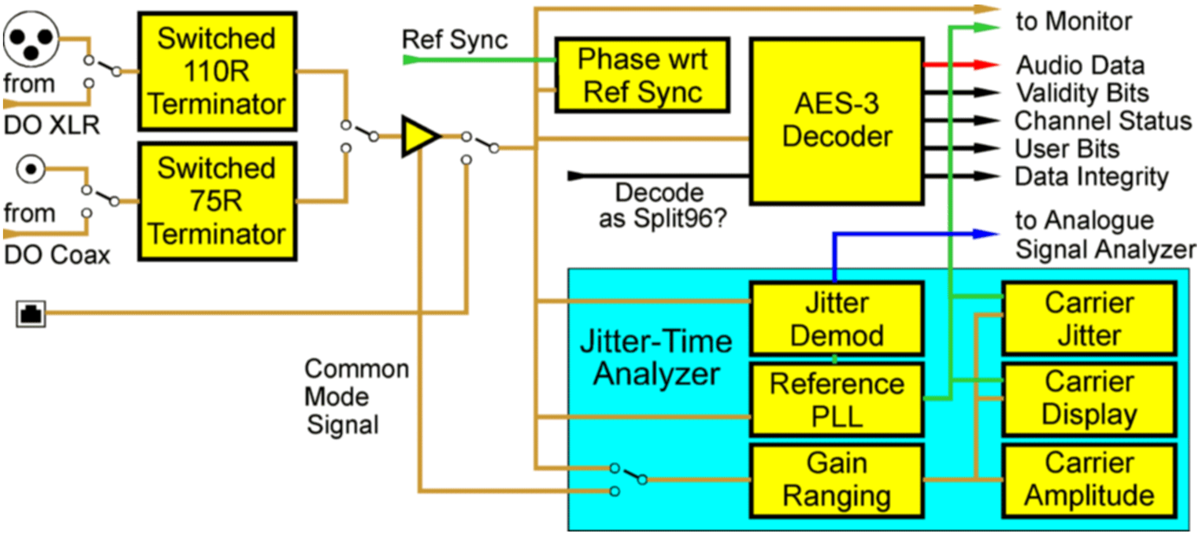
Differential wide-band noise interference can also be added to these outputs. Common-mode interference can also be added to the XLR output.

Many of the degradation functions can be output from the Monitor Outputs if required.

These settings are controlled through the Digital Output Carrier dialogue box. Refer to this section for further detail.

5.2.4 Digital Input and Carrier architecture

NB: **This section may not be relevant, depending on the dScope model number.**



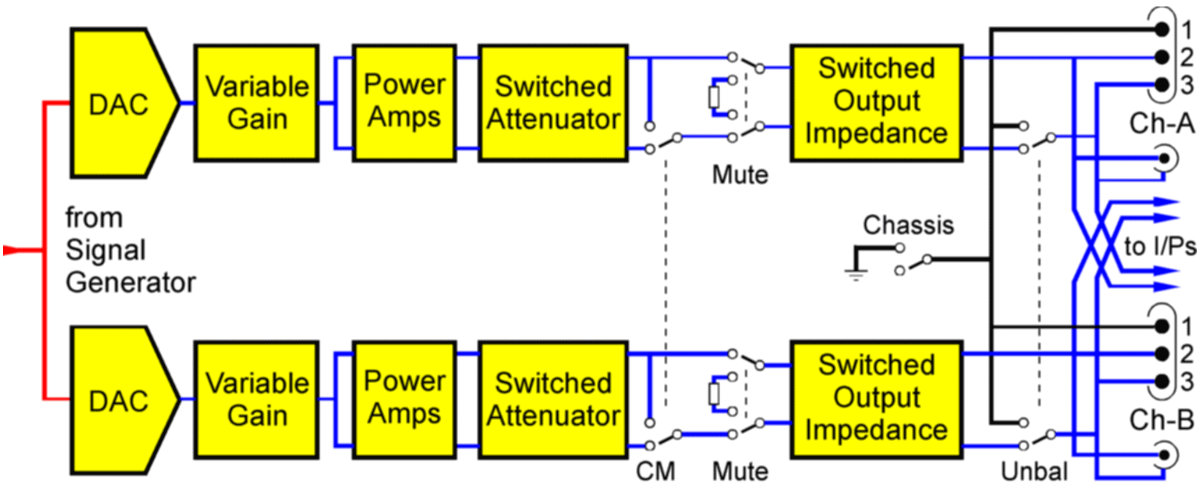
The figure above illustrates the functionality of the Digital Input and Carrier circuits. The settings and Results for these sections are accessed through the Digital Inputs dialogue box and the Digital Input Carrier dialogue box.

The digital audio input is selected from the XLR, BNC or TOSLINK front-panel DI connectors, or the XLR or BNC Digital Outputs can be looped back. The XLR and BNC options may be terminated appropriately if required. In 'loop-through' modes, the termination can be lifted to allow the dScope to analyze 'in-line'.

The selected input is routed to the AES3 decoder, where its various data components are distributed for audio and status analysis. It is also compared with the Digital Output reference for the 'Phase wrt Ref Sync' Result. The selected input is also fed to the monitor section (for carrier and sync-pulse output modes) and to the JTA (Jitter Time Analyzer).

The JTA is a sophisticated processing block which performs carrier timing and amplitude measurements, and can be used to plot a Carrier Display or eye-diagram. Note that the JTA also demodulates any jitter present on the Digital Input, and this signal can be switched to the Analogue Input of the dScope's Signal Analyzer for time-domain and spectral (FFT) analysis. This selection is made in the Analogue Inputs dialogue box.

5.2.5 Analogue Output architecture



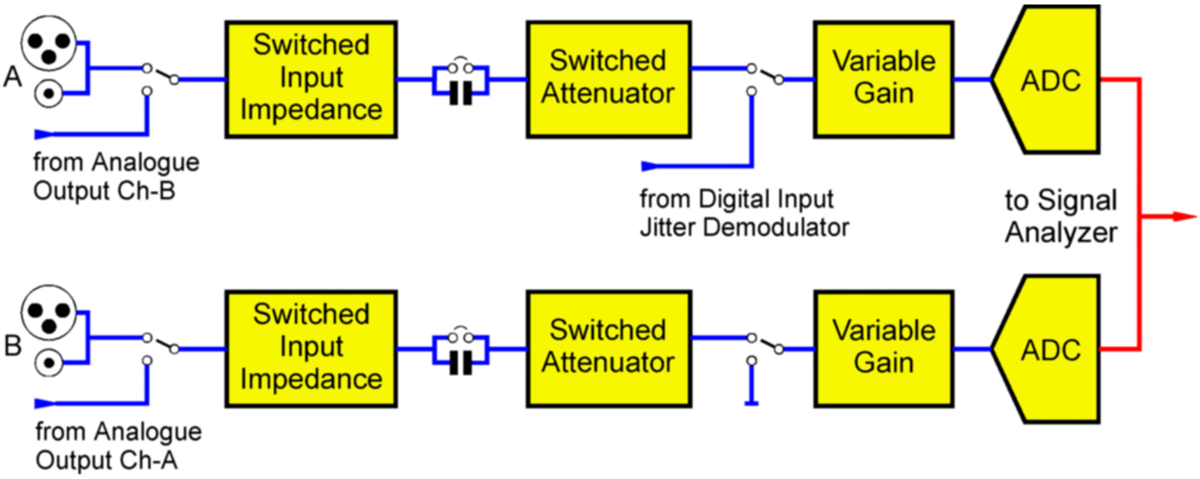
The figure above illustrates the functionality of the dScope's Analogue Output conditioning circuits.

The appropriate gain and attenuator settings are selected automatically according to the amplitude setting of the analogue signal generators, as set in the Signal Generator dialogue box. The remaining settings (mute, common-mode test, unbalanced/balanced and output impedance) are controlled through the Analogue Outputs dialogue box. Refer to these sections for more details.

NB: **The outer conductors of the Analogue Output BNC connectors are connected to the inverting leg of the balanced output circuit (i.e. to pin 3 of the associated XLR) and not to chassis or signal ground (although the Analogue Outputs can be switched into this mode). See the Unbalanced operation and grounding section for more information.**

The Analogue Output sample rate can be switched between 96kHz and 192kHz (unless the hardware is not 192kHz capable, in which case the sample rate is fixed at 96kHz). In the current software versions, the Analogue Input ADC and the Analogue Output DAC share the same sample clock, so switching the rate of the ADC also controls the rate of the DAC. For more information, see the Analogue I/O sample rate section.

5.2.6 Analogue Input architecture



The figure above illustrates the functionality of the dScope's Analogue Input conditioning circuits. Most of the settings are available in the Analogue Inputs dialogue box .

The Analogue Inputs can accept balanced or unbalanced inputs up to +46dBu (154.5V RMS). The dScope software sets the input range by adjusting the switched attenuator and variable gain stage in each channel. Normally this is done automatically, but the operator can fix the input range manually if desired.

The input impedance can be selected from 100kR, 600R, or a low impedance which is either 150R or 200R according to a PCB jumper setting. The dScope software is aware of the jumper setting and only offers the jumpered setting in the user-interface. The dScope software may over-ride the input impedance to 100kR if the input amplitude is high enough to damage the termination resistor. The termination resistors are also protected by a fuse in each channel.

The Analogue Inputs are normally DC-blocked by a capacitor, but this may be bypassed by a PCB jumper setting if DC-coupled analysis is needed.

The Analogue Input circuit is normally sourced from the XLR and coaxial input connectors (in parallel) but may be switched to be driven from the analogue Generator outputs if required. This allows direct analysis of the Generator outputs without re-plugging. Alternatively, since the Generator feed is from the opposite channel, the two Analyzer channels can be switched to measure the input and output of a device under test simultaneously, one channel at a time, so that input-to-output transfer measurements can be made.

The demodulated jitter signal from the selected Digital Input can also be switched to over-ride the Analogue Inputs if desired. This allows the incoming jitter to be displayed as a Scope Trace, or its spectrum as an FFT. In this case, the second Analyzer channel is disabled.

NB: **The outer conductors of the Analogue Input BNC connectors are connected to the inverting leg of the balanced input circuit (i.e. to pin 3 of the associated XLR) and not to chassis or signal ground. See the Unbalanced operation and grounding section for more information.**

The Analogue Input sample rate can be switched between 96kHz and 192kHz (unless the hardware is not 192kHz capable, in which case the sample rate is fixed at 96kHz). In the current software versions, the Analogue Input ADC and the Analogue Output DAC share the same sample clock, so switching the rate of the ADC also controls the rate of the DAC. For more information, see the Analogue I/O sample rate section.

5.2.7 Analogue I/O sample rate

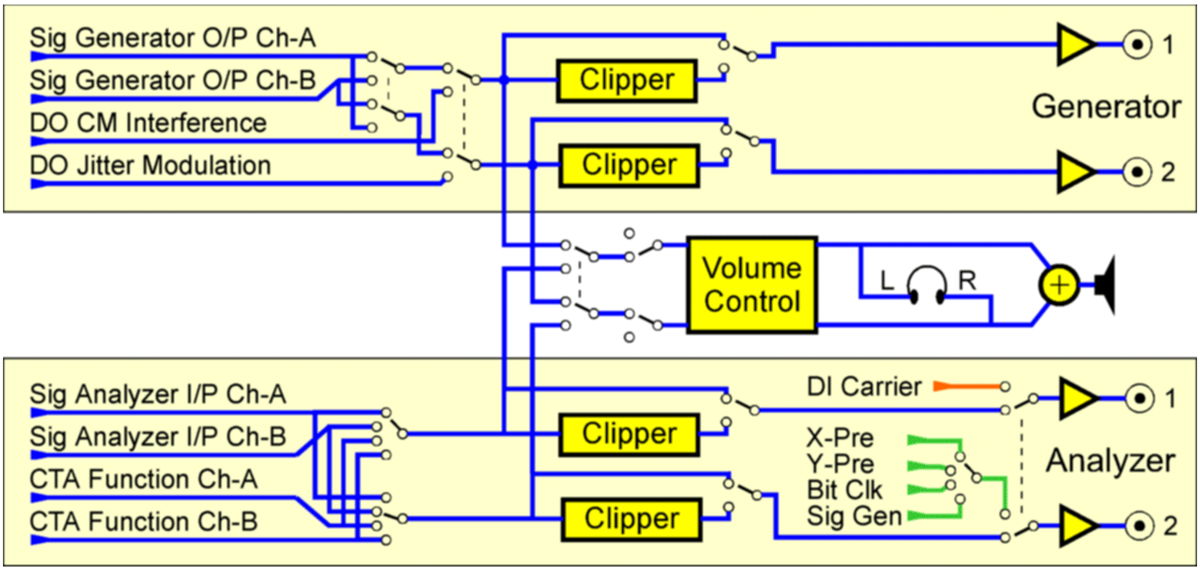
The Analogue Input sample rate can be switched between 48kHz, 96kHz and 192kHz. In the current software versions, the Analogue Input A/D converter and the Analogue Output D/A converter share the same sample clock, so switching the rate of the A/D converter also controls the rate of the D/A converter. The analogue sample clock is derived directly from the internal TCXO, and cannot be externally synchronized, or locked to digital I/O. The analogue sample rate is controlled from the Analogue Inputs dialogue box.

In the case of the Analogue Inputs, there is little performance difference between the rates, and selection of the different rates merely extends the upper band-edge (-3dB) frequency from 23.5kHz (fs=48kHz) to 47kHz (fs=96kHz) and 95kHz (fs=192kHz). See the Specifications section for more details. The lower sample rates can be beneficial in extending the low-frequency resolution of FFTs.

In the case of the Analogue Outputs, there is a similar extension of upper band-edge frequency from 22.75kHz (fs=48kHz) to 45.5kHz (fs=96kHz) and 91kHz (fs=192kHz). However, there is also a performance tradeoff, in that residual performance and flatness is slightly worse at fs=192kHz, and the maximum generated amplitude is reduced by 0.5dB at that rate, to +27.5dBu (balanced), and +21.5dBu (unbalanced). See the Specifications section for more details.

Note that the Signal Generator cannot be operated in 'split' mode at sample rates above 96kHz. If either the Analogue or Digital Outputs are operating above 96kHz, the A–channel of the Generator feeds both output channels, although they can be independently muted. An exception to this rule occurs if a table-based function is being generated; this allows multi-tone testing for both analogue channels at fs=192kHz. See the Signal Generator dialogue box section for more details.

5.2.8 Monitor Output architecture



The figure above illustrates the functionality of the dScope's Monitor Outputs.

The Monitor Outputs comprise two BNC connectors for each of the dScope's Signal Generator and Analyzer, plus a stereo headphone output and integral loudspeaker.

The BNCs have a 75R output impedance and are generally intended for connection to an oscilloscope or external audio monitoring system. They can output various audio, carrier or trigger signals as selected by the operator. Audio signals are usually automatically ranged to a nominal level between 2Vp–p and 4Vp–p (unterminated), although the gain can be manually set if required. Carrier signals are atenuated to half their input amplitude. In 'pulse' mode, audio signals can be sliced at their zero-crossings to provide TTL output pulses.

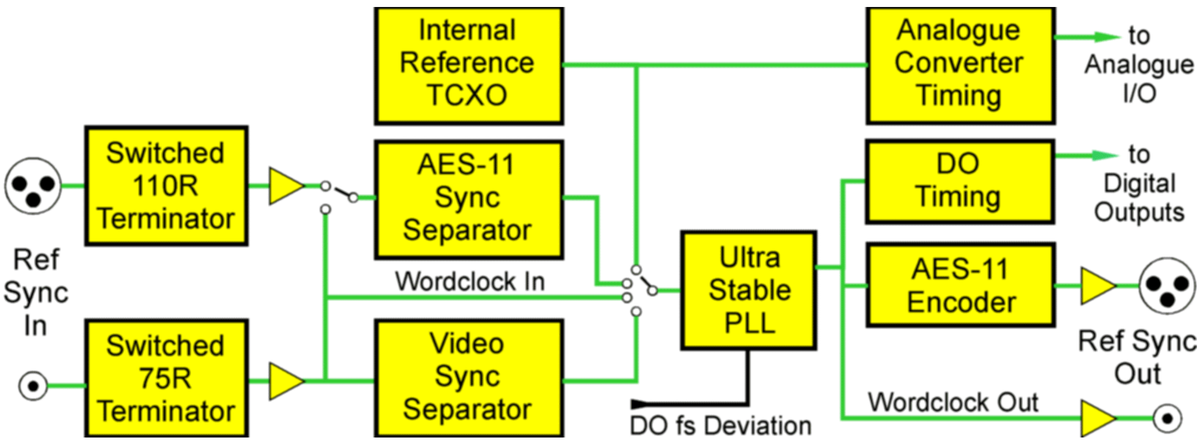
Audio signals selected to the BNC Monitor Outputs can also be monitored in mono or stereo at the headphone socket, or in mono on the integral loudspeaker, which share a volume control on the dScope front panel.

Operation of the monitor system is through the Monitor Outputs dialogue box. Refer to this section for more details.

5.2.9 Reference Sync architecture

NB:

This section may not be relevant, depending on the dScope model number.



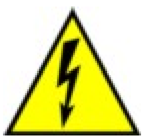
The figure above illustrates the functionality of the dScope's Reference Sync circuits. The primary function of the Reference Sync is to provide a synchronization source for the digital audio outputs, and so it is controlled through the Digital Outputs dialogue box.

The digital audio outputs are synchronized to an ultra-stable PLL (phase-locked loop), which generates all required output timing signals for the Digital Outputs, as well as the AES11 and WCK Ref Sync outputs which appear on the rear of the dScope. The PLL may be locked to either the internal TCXO (temperature compensated crystal oscillator) or to a variety of external references. The PLL has good jitter rejection capabilities down to very low frequencies, so that any clock jitter present on an external reference is not transferred to the Digital Output. The TCXO is accurate to within ±1ppm and, as such, is more accurate than most calibration equipment. It should generally not require recalibration, but any recalibration of the TCXO should be to an off-air reference standard. The PLL is able to fix the rate of the Digital Outputs with arbitrary frequency relationship to the selected reference. This allows, for example, Digital Outputs to run at 48kHz from a 44.1kHz reference, or to exhibit a designated frequency offset in either direction between 1 and 1500ppm.

The XLR Reference Sync input has a switchable 110R termination, and can only carry a AES11 (DARS) reference.

The BNC Reference Sync input has a switchable 75R termination, and can carry WCK (Wordclock), Video (PAL, SECAM, NTSC 30 or 29.97 fps), AES3–id or, using a supplied RCA (phono) adapter, S/PDIF. When a video reference is used, the relationship between the video frame rate of the reference and the audio frame rate of the Digital Output can be complex, depending on the video standard applied and the specified sample rate. This is described in detail in the Digital Output Synchronization panel section.

5.3 PCB jumper options



TO PREVENT SHOCK HAZARD, THE DSCOPE SHOULD ONLY BE OPENED BY QUALIFIED PERSONNEL. REMOVE THE POWER LEAD FROM THE DSCOPE BEFORE REMOVING THE TOP COVER.

Whilst dScope's operating modes are generally switched automatically under software control, some unusual modes of operation must be selected through PCB jumpers if required. These are:

Increasing the 150R Analogue I/O impedance options to 200R

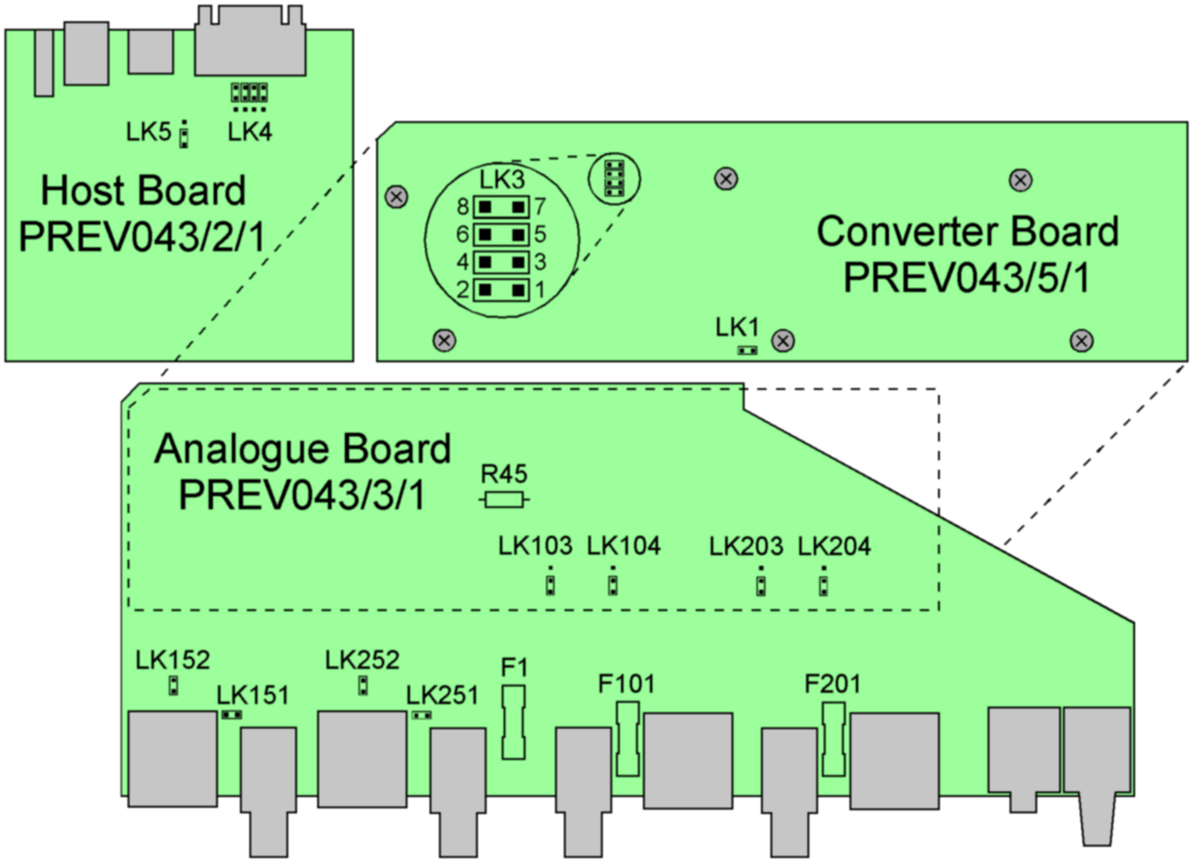
DC-coupling the analogue Analyzer inputs

Isolating the Analogue Input and Output signal grounds

Switching the dS–NET interface from RS–232 to RS–485

The top cover of the dScope must be removed to access the PCBs, which are shown in the diagram below. Jumpers should be removed, fitted or relocated using small snipe-nosed pliers. Removed jumpers should be carefully retained for future use.

The jumpers in the diagram are shown in the default positions.



Increasing the 150R Analogue I/O impedance options to 200R

Some measurement standards require the use of 200R impedances instead of the usual 150R. This can be done by changing the jumpers as detailed below.

Board	Jumper	150R (default)	200R
Analogue	LK151,152,251,252	fitted	removed
Converter	LK3 pins 1-2	fitted	removed

In the 200R state, the '150R' options in the Analogue Input and Analogue Output dialogue boxes are replaced by '200R' options.

DC-coupling the analogue Analyzer inputs

To change the DC-coupling of the analogue inputs, it is necessary to remove the Converter Board from its position on top of the Analogue Board. To do this, remove the six Converter Board screws shown in the diagram, and carefully lift the Converter Board off the Analogue Board. By tilting the Converter Board backwards after removal, it is possible to reach the Analogue Board jumpers without unplugging the Converter Board ribbon cable. When replacing the Converter Board, be careful to ensure that it is correctly located and pushed home before replacing the six screws.

Board	Jumper	AC-coupled (default)	DC-coupled
Analogue	LK103,104,203,204	1-2	2-3
Converter	LK3 pins 3-4	fitted	removed

In the DC-coupled state, the 'DC-block' high-pass filter option is replaced by 'off' for Analogue Inputs.

Isolating the Analogue Input and Output signal grounds

dScopes are shipped from the factory with their Analogue Input and Output signal grounds connected together, although neither is connected directly to the chassis, which is connected to mains earth. In general, this arrangement gives the best performance in the widest range of circumstances. However, in unusual circumstances it may be necessary to isolate the input and output signal grounds from each other. The method depends on whether the Analogue/Converter Board pair are at Rev B or Rev C. This can be determined from the 'About dScope Hardware' dialogue box, or by examining the boards themselves.

For Rev C boards, the grounds are isolated by simply removing jumper LK1 on the Converter Board.

For Rev B boards, LK1 is not present on the Converter Board, and the grounds are linked by a 0R resistor (R45) on the Analogue Board. To reach this resistor, it is necessary to remove the Converter Board from its position on top of the Analogue Board. To do this, remove the six Converter Board screws shown in the diagram, and carefully lift the Converter Board off the Analogue Board. By tilting the Converter Board backwards after removal, it is possible to reach the Analogue Board jumpers without unplugging the Converter Board ribbon cable. With the Converter Board removed, it is possible to clip out or unsolder R45. When replacing the Converter Board, be careful to ensure that it is correctly located and pushed home before replacing the six screws.

For more information, see the Unbalanced operation and grounding section.

Switching the dS–NET interface from RS–232 to RS–485

NB: **This section may not be relevant, depending on the dScope model number.**

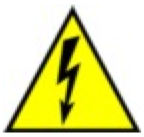
Although dS–NET is standardised on an RS–232 layer, it is also possible to operate it on an RS–485 instead, **although for reasons of compatibility this is not recommended or supported.**

Board	Jumper	RS–232 (default)	RS–485
Host	LK4 (all four jumpers)	Towards dSub	Away from dSub
Host	LK5	Away from dSub	Towards dSub

Note that in RS–485 mode, different dS–NET cables must be used, and the last dS–NET peripheral must be fitted with a termination. For more details, see the About dS–NET section.

The dScope software has no knowledge of whether RS–232 or RS–485 mode is selected.

5.4 Fuses and ratings



TO PREVENT SHOCK HAZARD, THE DSCOPE SHOULD ONLY BE OPENED BY QUALIFIED PERSONNEL. REMOVE THE POWER LEAD FROM THE DSCOPE BEFORE REMOVING THE TOP COVER.

Fuse locations and ratings are as follows:

FUNCTION	LOCATION	TYPE
Mains	Mains inlet IEC block (external)	2A(T) 20mm, CER
Analogue Input A termination	Analogue board, F101 (internal)	200mA(T) 20mm, GLS
Analogue Input B termination	Analogue board, F201 (internal)	200mA(T) 20mm, GLS
Analogue Outputs chassis link	Analogue board, F1 (internal)	12A(T) 20mm, GLS

Note that the locations of internal fuses are shown in the PCB diagrams in the PCB jumper options section.

For instructions on changing the mains voltage or fuse, see the Changing the mains voltage or fuse section.

Part 6

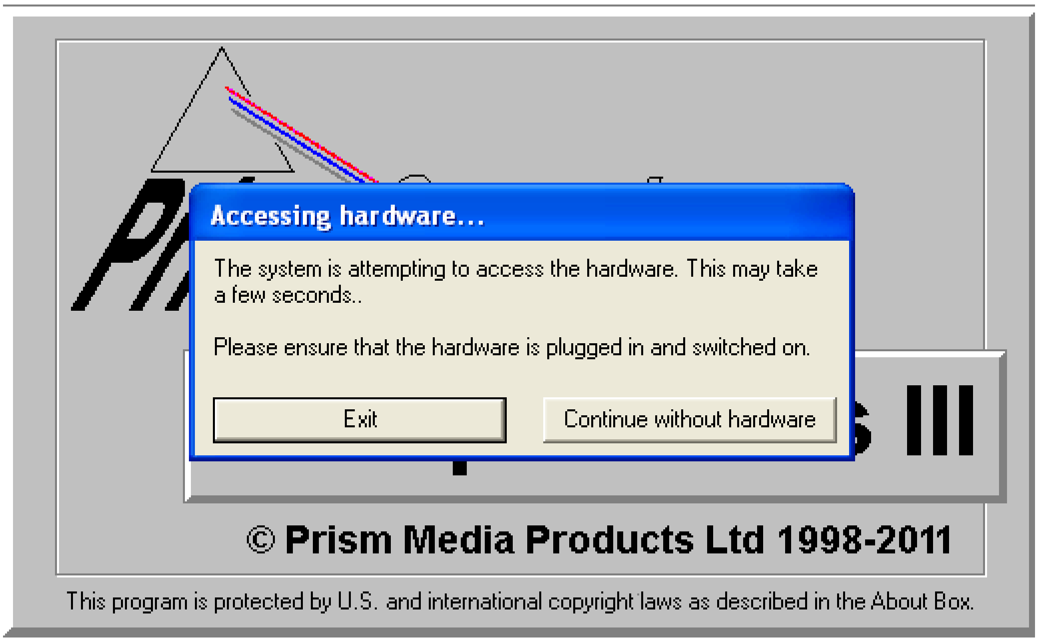
Service and Troubleshooting

6 Service and Troubleshooting

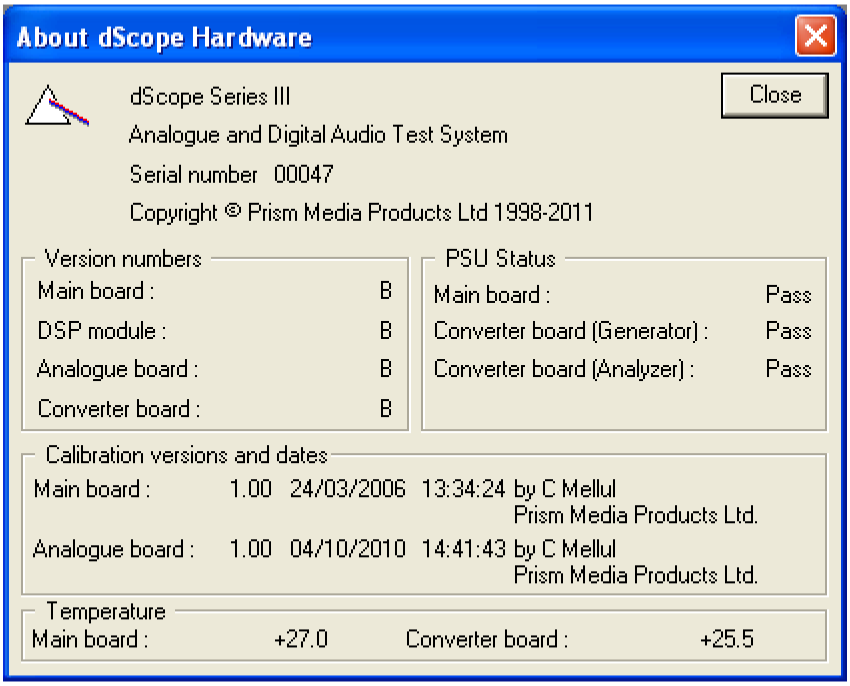
The following list outlines a series of checks that will verify that the dScope Series III system is operating correctly, and will help to troubleshoot any problems.

Installation and Connectivity

- Ensure that the dScope software has been correctly installed.
- To check which version of software you have installed on your host PC, consult the section on 'Software Version' below.
- Check that the mains voltage selector has been set correctly, and check the condition of the fuses at the dScope mains inlet and in the AC power cable.
- Connect AC mains power to the dScope hardware, and turn the mains power switch on. The front panel LED should illuminate red. If the front panel LED does not illuminate, connect the dScope hardware to the host PC using the USB cable supplied.
- If you are connecting the dScope hardware to the host PC for the first time, you will be prompted to install **two** Windows drivers before you can use the dScope system. Ensure that **both** drivers install successfully.
- Launch the dScope software from the host PC. You may see the following screen briefly:



- If the above screen is displayed for more than 10 seconds whilst the hardware is connected to the host PC and is powered on (front panel LED is red), please contact your Prism Sound representative for advice.
- If the hardware is correctly powered and connected to the host PC, the software should now launch, the hardware should switch from standby mode into active mode, and the front panel LED should switch from red to green. The fan on the rear of the instrument should also activate.
- If any error messages are displayed on launching the dScope software, consult your local Prism Sound representative for advice, quoting the error message.
- When the software has launched, select 'About dScope Hardware' from the 'Help' menu. The following window will be displayed:



Check that the PSU status is 'Pass' for each of the three PSUs. If any PSUs show '*Fail*', please return the dScope hardware to your local Prism Sound representative for further diagnostics.

- Check the Power Management options on the computer.
  Windows can turn off USB hubs on the PC to save power, but it sometimes does this when the port is in use, and this can cause problems with dScope operation.
  To turn off the power management:
  1. Open the Device Manager (either using the System icon in the Control Panel, or by right-clicking on the My Computer icon on the Desktop and selecting "Manage").
  2. In the Device Manager, select 'Universal Serial Bus Controllers" - this will probably be the last entry in the list.
  3. For EVERY "USB Root Hub" entry in this section of the tree, do the following:
     - Right-click on the entry and select "Properties"
     - If the Properties dialogue box has a "Power management" tab, select it and UNCHECK the box labelled "Allow the computer to turn off this device to save power".

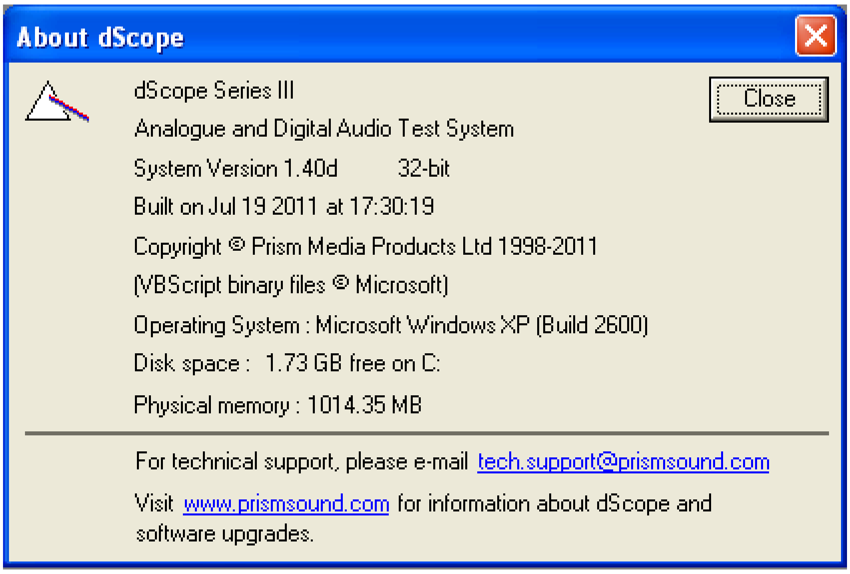
Self-Test Utility

If the above checks all indicate that the system is working, but you are experiencing problems making any measurements, there are a number of checks that can be undertaken to ensure that the analogue and digital signal generator and signal analyzer are working correctly.

Please contact your Prism Sound representative to obtain the latest version of Prism Sound's self-test software application. This will allow you to run a full functional test on your dScope system, and an HTML report will be generated. If the self-test report indicates a hardware failure, a calibration or repair may be needed. Please contact your Prism Sound representative and provide a copy of the self-test report.

Software Version

You can obtain the software version number currently installed on your PC by launching the dScope software (the hardware does not need to be connected to the host PC), and selecting 'About dScope' from the 'Help' menu. The following panel will be displayed:



In order to ensure that you benefit from the latest features, Prism Sound advises that you periodically check for software updates at www.prismsound.com; the latest software version will be available by clicking 'Support' -> 'Downloads'.

Technical Support

For help with using your dScope, or to report a problem, please contact the Prism Sound technical support team on tech.support@prismsound.com, or on 00 44 1353 648888. When communicating with Prism Sound or your Prism Sound representative about any problems with your system, please quote the version of the software that you are using, and the version of the Windows Operating System (this can usually be obtained by right-clicking on "My Computer" and selecting "Properties"). Please also include as much detail as possible about the problem, including steps to reproduce the problem if possible.

Calibration

Your Prism Sound dScope Series III system is a precision measurement instrument, and is supplied calibrated to traceable standards. The instrument is shipped complete with a calibration certificate, which is valid for 12 months from the date of manufacture.

In critical applications, it is recommended that your instrument is calibrated on a regular basis to ensure that its absolute amplitude and frequency accuracy remains within the manufacturer's quoted specifications.

If you wish to calibrate your instrument, and obtain a fresh 12 month calibration certificate, please contact your local Prism Sound representative.

Repairs

We recommend that all repairs are undertaken by Prism Sound or your local Prism Sound representative to ensure that your instrument's performance is not degraded. If a repair is needed, please contact your Prism Sound representative for instructions. You will be advised what further information may be needed to assist with diagnostics. Please *do not* send your dScope hardware for repair without first consulting Prism Sound's technical support team.

Part 7

Glossary

7 Glossary

A

AES3 - A two-channel digital audio interface standard, as provided at the dScope's Digital Output and Input. Also known as AES/EBU this format is used in professional applications usually with balanced XLR connections. It carries audio wordlengths up to 24 bits, plus Valid bit, User bit, Channel Status and Parity bit per channel.

AES11 - An AES3 carrier used as a Reference Sync rather than to carry an audio signal. Also known as 'DARS' (digital audio reference signal).

AES17 - An AES Standard describing measurement methods for digital audio systems.

AES17 low-pass filter - A low-pass filter specified by AES17 with an upper band-edge frequency of 20kHz and a stop-band attenuation of at least -60dB by 24kHz. This filter is useful in eliminating switching noise from class-D amplifier measurements and also out-of-band noise in measurements of noise-shaped systems.

ASIO - (Audio Stream Input/Output) - A digital audio device driver protocol for Windows computers, created by Steinberg, that bypasses the Windows audio processing, allowing audio to be passed unaltered and with low latency between different applications and devices.

B

Balanced - A method of transmitting an analogue audio signal or digital audio carriers, where two wires are used each carrying a representation of the signal or carrier in opposite polarity. Receiving equipment extracts the signal by subtracting one 'leg' from the other, thus rejecting any signals common to both wires. In this way, interference from mains, radio communications etc. is rejected, assuming the CMRR of the receiving equipment is adequate at the appropriate frequency. See also Unbalanced. dScope's Analogue Inputs and Outputs can work in balanced or unbalanced modes.

Bin - A single point in the graphical result of an FFT, corresponding to a small range of frequencies. The amplitude of each bin corresponds to the spectral content of the input signal within that frequency range.

Bin Centres (Generator Function) - A noise-like waveform which comprises a repeated sequence of 2^n samples, and which therefore contains a range of discrete sinusoidal tones. These tones are arranged to occur in equal measure at every frequency which exactly repeats within 2^n samples, excluding DC and half the sample rate. After capturing 2^n output samples from the stimulated EUT, subsequent synchronous FFT analysis consequently places a recovered tone at the centre of every FFT bin, excluding the first and last bins, thus rendering the frequency response of the EUT with great precision. This method is very fast and accurate, since it requires no sequential sweeping, and no averaging (as would be required with a noise stimulus which has randomly varying energy in each bin on successive acquisitions). This waveform is also useful in impulse response analysis.

BP/BR filter - A filter in the Continuous-Time Detector and FFT Detectors of the dScope's Signal Analyzer which can be set to band pass (BP) to make frequency-selective measurements, or to band reject (BR) to make residual measurements (e.g. THD+N).

Brick wall filter - An high-pass or low-pass filter with a very narrow transition band. dScope's FFT Detectors have idealised brick wall filters with perfectly flat pass bands, infinitely attenuative stop-bands, and transition bands of only a single one bin width.

C

Carrier - Usually used within dScope to refer to an AES3 or S/PDIF carrier, the digital interface signal carrying a sequence of binary data bits. These carriers are degraded in the real world by factors such as cable losses, which may cause jitter or other problems. The dScope's Digital Output Generator can simulate such degradations, and its Digital Input Analyzer can measure them.

Carrier Display - A graphical representation of a section of the Digital Input Carrier displayed by the dScope.

Channel Array - A predefined array of crosspoints made up of one or more dS-NET switchers, such as I/O Switchers. A Channel Array can be controlled as a single entity, removing the need for the user to control multiple switchers directly.

Channel Check - A special mode of operation of the dScope's Digital Output and Digital Input used to verify data integrity of audio samples. This is achieved using a PRBS (pseudo-random bit sequence), which can be generated at the Digital Output and verified at the Digital Input. Input and output need not be synchronised, and so may be separated in distance (e.g. satellite link) or time (e.g. digital recorder). The Prism Sound DSA–1 hand-held tester can generate and verify the same sequence.

Channel Status - status information embedded in an AES3 or S/PDIF digital interface, one bit per channel per sample-period, which accumulate into a 192–bit frame for each channel every 192 sample-periods. The frame is arbitrarily split into many fields of various lengths, with diverse functions as described in the appropriate interface standard document. The definitions of the fields and their meanings are different for 'Consumer' Channel Status (where the first bit of the frame is 0, used in S/PDIF) and 'Professional' Channel Status (where the first bit is 1, used in AES3). Originally conceived to add functionality to digital equipment interconnects, the proliferation of outputs with sloppy Channel Status implementation and inputs which mute if any unexpected Channel Status is received has led, like the Babel Fish, to much entirely unnecessary conflict.

Chirp - see swept sine.

CMRR - Common-mode rejection ratio. A measurement of the ability of a balanced input circuit to reject an undesired signal that is common to both input terminals. The dScope Analogue Outputs have a CMRR test mode where the output signal is presented as common-mode instead of differential.

Continuous-Time Analyzer (CTA), Detector (CTD) - The dScope's Signal Analyzer contains two discrete Analyzers. The Continuous-Time Analyzer runs continuously, so it does not miss momentary transients, but is limited in its choice of Detector functions, filters etc. Only one two-channel Continuous-Time Detector is available. See also FFT Analyzer.

Current Trace - The currently-selected Trace; many adjustments of Trace settings on the dScope's Trace window must be made by first making the desired Trace 'current' by clicking the left mouse button on the Trace, or by selecting it from the Quick legend.

Cursor - A Cursor can be positioned on a dScope Trace, causing its X and Y position to be displayed in the Cursor Toolbar. A Cursor can be 'relative', wherein the X and Y difference between a pair of Cursors is displayed.

D

D/A line-up - Digital/Analogue line-up. The dScope allows generation and analysis of signals in both the analogue and digital domains, and allows specification of amplitudes in the units of either domain. For example, a digital signal's generated amplitude can be entered in dBu, or an analogue signal can be measured in dBFS. In order for this to occur, the dScope has a D/A line-up setting, which simply allows the user to specify what level of analogue signal corresponds to a full-scale digital signal (0dBFS).

DARS - see AES11.

Data jitter - A type of interface jitter. Data jitter is that part of the interface jitter which is caused by variations in the duty cycle of the AES3 carrier acting with high-frequency losses in the transmission medium (e.g. cable capacitance) such that edge timing in the carrier is modulated by the activity of the data bits. This is distinct from fs jitter, which is inherent in the carrier source. dScope can measure data jitter and fs jitter independently, so that the cause of jitter problems can be identified. Also referred to as 'inter-symbol interference'.

dBFS - decibels with respect to digital full scale. 0dBFS is defined as the RMS amplitude of a sine wave whose peak reaches a positive full scale sample value (0x7FFFFF Hex).

dBm - decibels relative to an amplitude of 1.000 milliwatt. Because this is a power measurement, it requires knowledge of the impedance, as set using the Generator or Analyzer's Reference Impedance.

dBr - decibels relative to a reference amplitude. The reference amplitude must be specified for measurements in dBr to be meaningful. In the dScope, dBr measures with respect to the Reference Amplitude as specified on the Signal Generator or Signal Analyzer panels.

dBu - decibels relative to an amplitude of 0.7746 Volts (1mW in 600R).

dBV - decibels relative to an amplitude of 1.000 Volts.

dBSPL - an acoustic amplitude unit ("decibels, sound pressure level"): decibels relative to a the threshold of hearing (0dBSPL). 94dBSPL is equivalent to a sound pressure level of 1 Pa (Pascal), and is a common amplitude at which to state the calibrated output voltage of a measurement microphone.

DC-coupled, DC-blocking - An analogue audio input or output is said to be DC-coupled if it does not remove DC content from transmitted signals. If not, it is said to be DC-blocking. The dScope Analogue Outputs are DC-coupled. The Analogue Inputs are DC-blocking by default, but can be DC-coupled if required.

Detector - A rectifying voltmeter with a particular dynamic response, used to measure the amplitude of audio signals or residuals. dScope's Continuous-Time Detector and FFT Detectors offer two alternative versatile types of Detector.

Dither - low amplitude noise, added to a signal before quantization, or re-quantization, to linearize the loss of precision. In the dScope, dither is applied by default to Digital Outputs. Best linearization is achieved by TPDF dither.

dS–NET - A proprietary serial interface protocol used to connect peripherals such as I/O Switchers to the dScope.

E

EUT - 'Equipment Under Test' – the device being tested by the dScope.

Event - A causal occurrence for the dScope Event Manager or an Event-driven VBScript. Examples might be breaching of a Limit, or a change in received Channel Status.

Event-driven - A dScope VBScript is said to be Event-driven if its main body has finished running, and the only code that subsequently runs is triggered by an Event occurring (for example, a Limit Line being breached).

Event Manager - A dScope feature which allows the user to set links between various causes and effects. Thus a range of interesting occurrences in the EUT can be pre-armed to trigger responses such as audible or visible warnings, entries in log files, or even running of VBScripts.

Eye-diagram - The AES3 standard defines acceptable carrier degradation in terms of amplitude and edge-timing using an eye-diagram, which shows the minimum acceptable differential carrier amplitude over a defined period within 1 UI of the carrier. This can be verified on the dScope using the Carrier Display feature.

Eye-narrowing - The dScope can measure the worst-case narrowing of the eye of an AES3 carrier. This is essentially a measurement of data jitter, and can be referred to the eye-diagram in the AES3 standard.

F

FFT (Fast Fourier Transform) - A Discrete Fourier Transform (DFT) calculates the spectrum of a sampled signal, i.e. it transforms the time domain signal (e.g. the dScope's FFT buffer, as shown by the Scope Trace) into the frequency domain. A Fast Fourier Transform (FFT) allows a DFT to be calculated more efficiently, i.e. faster, assuming that the length of the data set (sample buffer) is 2^n samples. See also Window function.

FFT Analyzer (FFTA), Detector (FFTD) - The dScope's Signal Analyzer contains two discrete Analyzers. The FFT Analyzer runs intermittently, initiated by a scope-like trigger, so it can miss momentary transients, but it has a wider range of Detector functions, filters etc. Up to 40 two-channel FFT Detectors can be active simultaneously. See also Continuous-Time Analyzer.

Frame rate - Usually applied to video or digital audio carriers. The frame rate of an AES3 (or AES11) digital audio carrier is generally the same as its sample rate, i.e. the rate of transmission of a pair of two-channel samples. However, if the interface is operating in Split96 mode, the frame rate is half the associated sample rate.

fs - Abbreviation for sample rate.

fs jitter - A type of interface jitter. fs jitter is that part of the interface jitter which is inherent in the equipment which is the source of the AES3 carrier. This is distinct from data jitter, which is caused by variations in the duty cycle of the AES3 carrier acting with high-frequency losses in the transmission medium (e.g. cable capacitance) such that edge timing in the carrier is modulated by the activity of the data bits. dScope can measure data jitter and fs jitter independently, so that the cause of jitter problems can be identified.

FS - Abbreviation for digital full-scale, e.g. dBFS

G

H

Hex (hexadecimal) - A convention for conveniently representing binary data such as digital audio samples. Each four-bit 'nibble' of the binary word is represented by a character indicating its value: 0..9,A..F. For example, the 24–bit binary value 000000010010110111101111 would be represented in hex as 012DEF. Hex is available as an amplitude unit throughout dScope in order to facilitate some digital measurements.

High-pass filter - A filter in the Continuous-Time Detector and FFT Detectors of the dScope's Signal Analyzer which eliminates frequencies below a pre-set limit from the measurement.

I

IMD - Intermodulation Distortion. When a signal consists of more than one frequency, a non-linear device under test will produce the original frequencies plus an infinite number of IMD products, given by

$$(a * F1) + (b * F2) + (c * F3) + ...$$

where (a, b, c) etc. are all possible integer numbers, and (F1, F2, F3) etc. are the frequencies of the original tones.

IMD difference-tone measurement - An IMD measurement method (e.g. CCIF) wherein two tones, of equal amplitude, close together in frequency (e.g. 19kHz and 20kHz, or 14kHz and 15kHz for band-limited systems) are applied to the EUT. The amplitude of the distortion component at the difference frequency (e.g. 1kHz) is measured, usually relative to one of the original tones.

IMD side-band measurement - An IMD measurement method, typically using a low-frequency high-amplitude tone, and a high-frequency tone at 1/4 the amplitude (the SMPTE standard uses 60Hz and 7kHz). The intermodulation distortion appears as side-bands around the high frequency tone, although historically it has been measured after demodulation to the base-band.

Impulse response analysis - An analysis technique for measuring EUT parameters such as temporal dispersion and frequency response by comparing the EUT's output with its input. Impulse response analysis is most often used in acoustic testing, to characterise rooms or loudspeakers; however it can also be useful in testing ordinary analogue or digital EUTs. The technique is general, and requires no special stimulus, although the stimulus should ideally cover the entire band of interest at reasonable amplitude. In the dScope, the method of comparison is such that the stimulus must repeat exactly over a 2^n sample period - for these reasons, 'swept sine' and 'bin centres' stimuli are most often used. The comparison of output and input yields the 'impulse response' of the EUT, i.e. the output which it would produce if a perfect narrow impulse were applied to its input. The impulse response itself is indicative of such factors as delay and reverberance of the EUT. The FFT of the EUT reveals the EUT's frequency response. By excluding all noise before and after the impulse prior to calculating the FFT, a loudspeaker can be measured as if in anechoic conditions. For more information, see the Principles of impulse response analysis section.

Interface jitter - Jitter present on a digital audio carrier or reference sync. Interface jitter usually comprises fs jitter and data jitter components. The dScope can generate and measure interface jitter directly, and incoming interface jitter can also be demodulated for analysis by the Signal Analyzer. Interface jitter is often blamed for sonic degradation in A/D and D/A converters, but this is usually due to sampling jitter within the conversion equipment resulting from the equipment failing adequately to remove incoming interface jitter from the conversion clock. Where good quality converters are used, interface jitter is not usually problematic until it reaches very high levels, when data loss can result. The AES3 standard defines a jitter tolerance template (jitter vs frequency) for correct receipt of data.

I/O Switcher - A dS-NET peripheral, comprising a 16x2 relay crosspoint switcher, a stereo load switcher, DC bus meter and output balance switcher.

J

Jitter - Variation in edge-timing of a clock. In audio systems, manifestations are interface jitter and sampling jitter.

Jitter Time Analyzer (JTA) - An element of the dScope's hardware which analyzes the incoming digital audio carrier. It performs time-domain analysis (jitter and amplitude measurement) of the carrier, and collects data for the Carrier Display.

K

L

Limit - A Limit can be applied to a Reading, so that an Event is triggered if the Limit is breached. This might cause an audible or visual warning, logging of the Event in a log file, or even the automatic running of a VBScript. See also Event Manager.

Limit Line - A Limit Line can be applied to a Trace, so that an Event is triggered if the Limit Line is breached. This might cause an audible or visual warning, logging of the Event in a log file, or even the automatic running of a VBScript. See also Event Manager.

Live Trace - A Scope, FFT or Sweep Trace in the dScope's Trace window. These Traces are subject to 'Live update' as opposed to Copy Traces, Filter Traces etc.

Log chirp - see swept sine.

Low-pass filter - A filter in the Continuous-Time Detector and FFT Detectors of the dScope's Signal Analyzer which eliminates frequencies above a pre-set limit from the measurement.

M

Mark - One of a number of annotations on a dScope Trace. These can be placed either manually, using the Cursor, or automatically as in the case of the 'Mark harmonics' function. The X & Y values of the Marks, along with a comment or annotation for each, can be appended to graphical prints or exports if desired.

Memo - A yellow note which can be appended anywhere on the dScope desktop providing an annotation, reminder etc. Memos can be hidden and restored individually or collectively.

Multi-tone testing - A method of testing where a large number of discrete tones are used to stimulate the EUT simultaneously. By capturing a single data set of the output of the EUT, many measurements can be calculated simultaneously. These can include scalar Results such as noise, distortion etc. as well as graphical plots against frequency, such as frequency response, distortion spectrum etc. This method is much faster than traditional methods, which would require stimuli to be changed for each scalar measurement and stepped through many frequencies for each plot. The dScope has a uniquely user-friendly way of setting up multi-tone tests.

N

Nested Sweep - A Sweep which is repeated for each state of a second varying 'outer' source. A 'two-dimensional' Sweep.

O

OLE - Object Linking and Embedding. It is the mechanism by which dScope can be VBScripted or remotely controlled.

P

Pixel - A single dot on the screen. If the screen resolution is set up to be 800 x 600, this means that there are 800 pixels across the screen, and 600 down.

Print legend - An expanded version of the Quick legend, used to control the appearance of individual Traces during graphical printing or export. The Print legend allows inclusion of separate comments, line-styles etc. for each Trace.

Q

Q–Peak - 'Quasi-peak' response. CCIR 468–2 specifies a fast-attack, slow-decay 'Q–Peak' detector which is intended to produce a measure of noise signals which corresponds to subjective level. It is usually used in conjunction with a special Weighting filter also specified in CCIR 468–2.

Quick legend - A list of all the Traces currently displayed on the Trace window. The Quick legend is a dockable Toolbar which is normally docked on the right-hand side of the Trace window, but can be dragged off and floated over the desktop if required. Traces can be turned on and off, have their display colours changed etc. from the Quick legend or, by right-clicking on the list entries, all settings of individual Traces can be edited. The Print legend allows adjustment of the appearance of each Trace for graphical printout or export.

R

Reading - A dScope Result can be converted to a Reading by dragging it off its home dialogue box. Readings have many additional functions over native Results: for example, they can be resized, user-coloured, and can have Limits and bar graphs attached to them.

Reference Sync - A signal passed between digital audio equipment for the purpose of defining the sampling clock. It is usually in AES11, Wordclock or video format. The dScope can accept a Reference Sync for its Digital Outputs in any of these formats.

Regulation - A process by which an Analyzer Result is brought to a target value by varying a defined Generator parameter.

Result - Any numerical output in a dScope dialogue box. Results can be converted to Readings by dragging them off the dialogue box in order to give them additional functionality.

S

S/PDIF - A two-channel digital audio interface standard, as provided at the dScope's Digital Output and Input. This format is used in consumer applications usually with unbalanced RCA (phono) or optical (TOSLINK) connections. It carries audio wordlengths up to 24 bits, plus Valid bit, User bit, Channel Status and Parity bit per channel.

Sample-rate - The rate at which a digital audio signal has been sampled. Standard sample rates include 32kHz, 44.1kHz, 48kHz, 88.2kHz, 96kHz, 176.4kHz and 192kHz.

Sampling jitter - Caused by jitter present on the sampling clock of an A/D or D/A converter (or a sample-rate converter). Sampling jitter results in distortion of the converted audio, which is worse at higher frequencies. In practice, sampling jitter often occurs in conversion equipment which does not adequately remove interface jitter from its reference sync. Sampling jitter is usually measured by passing a high-frequency tone through the converter under test, and applying jitter of varying frequency to its reference sync. Sampling jitter is manifest by side-bands on the converted tone, and the variation in amplitude of these with varying jitter frequency enables the jitter rejection characteristic of the conversion equipment to be measured.

Script - see VBScript.

Split96 - Split96 is a mode of digital interfacing (usually of AES3) whereby a two channel interface carries a single audio channel sampled at twice the frame rate of the interface. Each frame contains the data for two successive samples of the same channel, rather than a sample from the left channel and a sample from the right channel. Also known as 'two-wire' interfacing.

Status bar - An area at the bottom of the dScope screen which displays various information about the current state of the system. Progress bars are provided for FFT triggering / acquisition / averaging and Sweep progress. Warnings are displayed of attempts to generate signals at the Analogue and Digital Outputs which may be prevented by other settings, for example attempts to generate frequencies beyond those allowed by the selected sample rate. Errors or anomalous Channel Status on Digital Inputs selected for analysis are also shown. The right-hand end of the Status bar contains the Page selector tabs.

Sweep - A sequence of individual measurements made whilst varying a parameter of the stimulus. For example a frequency response Sweep would be made by measuring the gain of an EUT whilst varying the frequency of the stimulus. dScope provides a versatile sweeping capability, wherein many different Generator parameters can be varied whilst plotting up to four simultaneous Results. As well as being progressive, Sweeps can be table-based or sensed. In unusual circumstances which cannot be addressed within the sweep system, VBScripting allows automatic collection of sequential Results interspersed with any desired setting changes. Many tests which have traditionally been frequency-swept are now better performed using multi-tone techniques, which are much faster.

Swept sine - a Signal Generator function often used as a stimulus in impulse response and acoustic testing (also known as a 'chirp') comprising a sine wave swept smoothly across the audio band. The rate of progress may be linear or logarithmic (a 'log chirp'). The entire sweep is usually arranged to repeat exactly over the FFT Analyzer buffer acquisition period to allow synchronous (windowless) analysis and contiguous averaging, and often contains a period of silence between repetitions in order to prevent reverberance spilling between acquisitions.

T

Title bar - A window's title bar is the bar at the top of the window, containing the window name, and usually buttons to minimize, maximize and close the window.

Toolbar - A bar in the dScope user-interface containing a variety of 'icons' which can be clicked as shortcuts to various functions. The 'Main' Toolbar at the top of the dScope screen can be customised from a range of available icons. dScope Toolbars are 'dockable' – they can be dragged from their usual position and floated over the desktop if desired.

TPDF - Triangular Probability Distribution Function – A noise function, where a graph of probability vs amplitude is triangular. This type of noise is often used for dithering re-quantizations in digital audio, since it produces a precisely linear transfer function.

Trace - One of the graphical plots displayed on the dScope's Trace window. Traces can be of various types, e.g. Scope, FFT, Sweep, Limit Line, Filter, Window function etc. The user can select which Traces are to be displayed at any time, and in what colours etc. In two-channel mode, Traces for both Analyzer channels an be displayed simultaneously, either on the same or separate axes.

Trigger - The dScope's FFT Analyzer is activated by a scope-like trigger. The trigger is sensitive to a user-defined threshold and transition polarity, and can be set to be iterative or single-shot. The trigger can also be over-ridden for continuous or manual operation if required. The Continuous-Time Analyzer runs continuously, independently of the FFT Analyzer's trigger.

U

UI - (Unit Interval). A UI of an AES3 carrier is 1/128 of the frame period, the duration of a single biphase-mark 'cell', or half a bit period.

Unbalanced - A method of transmitting an analogue audio signal or digital audio carriers, where a single wire carries the signal with respect to a ground, or screen conductor. This method is more common in consumer equipment and is more prone to interference than the balanced method commonly used in studios. dScope's Analogue Inputs and Outputs can work in balanced or unbalanced modes. See the Unbalanced operation and grounding section for more details.

User bits - A per-channel, per-sample status bit in the AES3 interface. There are many different User bit implementations in use, some pseudo-standard (such as CD and DAT sub-codes and AES18 data transmission) and others which are entirely proprietary.

V

Valid bit - A per-channel flag bit carrier in the AES3 interface. The meaning of the Valid bit has changed slightly since the AES3 standard was originated, so unfortunately its implementation sometimes differs between equipment. In general, it indicates (when 0) that the channel is suitable for conversion to analogue, and most receiving equipment mutes if it detects the flag set to 1. However, in some instances it has been used to indicate that error correction or concealment has taken place, the effort of which may have been wasted if receiving equipment mutes as a result of seeing the flag. The dScope can set the Valid bits in its Digital Outputs and monitor them at its Digital Inputs.

VBScript - A script or program in the VBScript language which customises an element of the dScope's operation. This may be an Automation script, which allows the dScope to perform a pre-defined series of operations, or it may define various mathematical functions such as Weighting filters or Window functions.

VSIO Adapter - A dS-NET peripheral which adapts the dScope's AES3 outputs and inputs for connection to an EUT which has I2S-like serial audio multiplex interfaces. The multiplex is programmable to interface with a wide range of component-level devices.

W

Weighting filter - A filter in the Continuous-Time Detector and FFT Detectors of the dScope's Signal Analyzer which controls the emphasis of certain parts of the audio spectrum in the measurement. These filters are usually 'standard responses' used to provide compliance with specific measurement standards, although FFT Detector Weighting filters can be user-defined with a VBScript.

Window function - When performing an FFT, a finite-length buffer of sample values is used. Since this buffer is effectively infinitely repeated over time, the discontinuity where the end of the buffer wraps to the beginning produces high skirts on the resulting FFT frequency components. This usually seriously limits the usefulness of the FFT, so a Window function is applied to the buffer prior to calculating the FFT. A Window function is a bell-shaped envelope by which the data buffer is multiplied, effectively emphasizing the middle part of the buffer at the expense of the edges, where the discontinuity occurs. This lowers the skirts on the FFT components, to a greater or lesser degree depending on the quality of the Window function. Although good dynamic range is retained by this method, energy from individual frequencies is spread between a few adjacent bins, compromising frequency resolution. Where test stimuli can be controlled, the optimum FFT resolution is derived by ensuring that the test waveform repeats precisely over the period of the buffer, thus eliminating edge discontinuities and allowing a rectangular window (i.e. no window at all) to be used. This technique is used in the dScope's synchronous multi-tone testing feature.

Wordclock - A Reference Sync signal in the form of an unbalanced square clock at the sample rate. It is nominally at TTL level with 75R impedance, on a BNC connector.

X

Y

Z

Index

- ~ -

- 2 -

- 6 -

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- T -

- U -

- V -

- W -

- Z -