



XC-8752A

DC - 1 GHz (1.3 GHz) Vector Network Analyzer

Operating Manual

Rev.1.3

Certification

XERXES electronics GmbH certifies that this product met the published specifications at the time of shipment from the factory.

Warranty

This product is warranted against defects in material and workmanship for a period of 1 year from the date of delivery. During this warranty period, XERXES will, at its own option, either repair or replace products which prove to be defective. For warranty service or repair, this product must be returned to the XERXES electronics factory. The buyer shall pay shipping charges to XERXES and XERXES shall pay shipping charges to return the product to the Buyer. However, the Buyer shall pay all shipping charges, duties and taxes for products returned to XERXES from another country. XERXES warrants that its software and firmware designated by XERXES for use with the instrument will execute its programming instructions when properly installed on the PC. XERXES does not warrant that the operation of the instrument, or software, or firmware will be uninterrupted or error free. The foregoing warranty shall not apply to defects resulting from improper or inadequate maintenance by the Buyer, Buyer supplied software or interfacing, unauthorized modification or misuse, operation outside the environmental specifications for the product, or improper site preparation or maintenance.

No other warranty is expressed or implied. XERXES specifically disclaims the implied warranties of merchantability and fitness for a particular purpose. The remedies provided herein are the Buyer's sole and exclusive remedies. XERXES shall not be liable for any direct, indirect, special, incidental, or consequential damages, whether based on contract, tort, or any legal theory.

Safety

SAFETY INSTRUCTIONS

Carefully read through the following safety instructions before switching the Analyzer to operation. Observe all the precautions and warnings provided in this Manual for all the phases of operation, service and repair of the Analyzer. The VNA must be used only by skilled and specialized staff or well trained personnel with the necessary skills and knowledge of safety precautions. The Analyzer is a IEC-61010 CAT I device, there for no measurements shall be done involving an CAT II, III or IV environment. Operators must not remove the covers or parts of the housing. The Analyzer must not be repaired by the operator. Component replacement or internal adjustment shall only be performed by qualified maintenance personnel only. Do not use excessive force on connecting cables to any connector of the instrument. Keep away liquids and inflammable gases. Electrostatic discharge can damage the Analyzer when connected or disconnected from the DUT(device under test). Static charge can build up on the body and damage the sensitive circuits of internal components of both the Analyzer and the DUT. To avoid damage from electric discharge, observe the following:

- Always use a desktop anti static mat under the DUT.
- Always wear a grounding wrist strap connected to the desktop anti static mat via daisy-chained 1 MΩ resistor.
- Connect the PC and the body of the DUT to protective grounding before you start operation.

Contents

1	Introduction to the XC-8752A vector network analyzer	8
1.1	Introduction.....	8
1.2	System requirements.	8
1.3	Packaging List.	9
1.4	Unpacking and first use	10
1.5	Technical data.	14
1.6	Network analyzer operation.....	16
1.6.1	Stimulus.....	16
1.6.2	Averaging.....	17
1.6.3	Receiver section.	17
1.6.4	Connection and sweep time.....	18
1.6.5	Selecting the port number.	18
1.6.6	Measurement panel	19
1.6.7	Simple De-Embedding.	19
1.6.8	Utility menu.....	20
1.7	Principle of operation.....	21
2	VNA Calibration.	23
2.1	Introduction.....	23
2.2	Calibration standards.	24
2.3	Calibration Procedure.	27
2.3.1	Spur calibration	28
2.3.2	Calibration Bandwidth and Average.....	28
2.3.3	Calibration procedure	29
2.3.4	3.5 mm to SMA compensation.....	29
2.4	Loading or Saving calibration coefficients.....	29
3	Measurements and Diagrams	30
3.1	Creating a diagram	30
3.1.1	The Equation Editor.....	36

3.2	The Diagram Dialogs.....	38
3.2.1	Using Mouse and Keys on diagram dialogs.	40
3.2.2	Diagram Traces.....	43
3.2.3	Diagram Marker	45
4	The Main Dialog Menu	53
4.1	File / Load / Save Calibration.....	53
4.2	File / Load / Save Setup	53
4.3	File / Save S-Parameter	53
4.4	File / Select Home Path	54
4.5	System / System Test	54
4.6	System / Unlock Calibration	55
4.7	System / Change password.	55
4.8	System / Service	55
4.9	Measurement / Edit Memory Slots.....	56
4.10	Measurement / De-Embedding (full)	57
4.11	Measurement / Show Limits	58
4.12	Limits definition in the trace edit dialog	58
5	Utilities	61
5.1	Spectrum Analyzer Utility.....	61
5.1.1	Spectrum Analyzer	61
5.1.2	Phase Noise Analyzer	64
5.1.3	Logarithmic spectrum analyzer	66
5.1.4	EMI Analyzer.....	68
5.2	Oscilloscope utility.....	72
5.2.1	Oscilloscope Trigger	74
5.2.2	Measurements	79
5.2.3	Protocol analyzer.....	81
5.2.4	Signal generator	83
5.3	Signal Generator Utility	83
5.4	Meters Utility.....	85
5.4.1	Broadband Voltmeter.....	85
5.5	THD meter	87
5.6	Power sweep	90
6	Recommendations & Supplemental Information.	92

6.1	Connector wear	92
6.2	Differential & Common Mode Measurements.	92
6.2.1	Differential mode in oscilloscope mode:.....	93
6.2.2	Differential mode in all other measurements:.....	93
6.3	Time domain measurements.	94
6.3.1	Application: Fault location.....	95
6.3.2	Application: Impedance variation	95
6.3.3	Application: Filter tuning.....	95
6.4	VNA direct component measurements.....	96
6.5	Loaded Q measurement.....	97
6.6	Calibration standards File Format	98
6.7	VNA gain measurement function	100
6.7.1	G_{vf} Matched forward voltage gain.....	100
6.7.2	G_{vr} Matched reverse voltage gain.	101
6.7.3	G_{if} Matched forward current gain.	101
6.7.4	G_{ir} Matched reverse current gain.....	101
6.7.5	k Stability Factor	102
6.7.6	G_{ms} Maximum stable gain	102
6.7.7	G_{max} Maximum available gain	102
6.7.8	G_{vfo} Forward voltage gain with no load.	103
6.7.9	G_{ifs} Forward current gain with output short circuited.....	103
6.7.10	G_{vfa} Forward voltage gain with arbitrary load.	104
6.7.11	G_{ifa} Forward current gain with arbitrary load.....	104
6.7.12	G_{tp} Forward transducer gain with arbitrary load.....	105
6.8	Antenna Factor and EMI measurement correction	106
7	Performance data.....	108
7.1	Vector network analyzer.	108
7.1.1	Port isolation	108
7.1.2	Trace noise	109
7.1.3	Measurement speed	110
7.1.4	Signal source data.....	111
7.1.5	Raw directivity source match and load match.	111
7.1.6	Measurement Uncertainty	113
7.2	Utility Specification.....	115

8	System Calibration.....	116
8.1	Spectrum analyzer calibration.....	116
8.2	Oscilloscope calibration.....	117
8.3	Signal Generator Calibration.	119

1 Introduction to the XC-8752A vector network analyzer

1.1 Introduction

A vector network analyzer measures the magnitude and phase characteristics of networks and components such as filters, amplifiers, attenuators or antennas. While the main function of the XC-8752A is the measurement of 2-port networks, additional utilities integrated into the instrument will ease the electronics design process:

- Signal generator DC - 1 GHz with 1 Hz resolution.
- Two channel 60 MHz digital oscilloscope with 125 Ms/s and 12 bit resolution.
- DC - 1 GHz spectrum analyzer, with an unambiguous frequency range of DC - 60 MHz and an active sideband suppression for spurious-free signal measurements up to 1 GHz (under certain conditions).
- EMI analyzer with quasi-peak and average measurements.
- Power sweep measurements.
- Wideband Power/Voltage measurements.
- THD analyzer.
- Phase noise analyzer.
- General noise and bode measurements.

1.2 System requirements.

The XC-8752A system software requires an Microsoft Windows(R) 7 (SP1) 64 Bit or upwards compatible computer with an USB 2.0 interface, CD-ROM drive, about 200 MB free disk space, an Intel Core I3 (R) processor with 3 GHz clock speed (or equivalent) and 4 GB of system memory as minimum recommendation. A high resolution screen of better than 1280x1024 minimum is required, 4k resolution recommended for multi windows view simultaneous measurements.

1.3 Packaging List.

When opening the XC-8752A wooden box please check if following items are included in the package:

- XC-8752A Vector network analyzer.
- 1.8 m long USB cable with type A and type B connector
- 12 V / 2 A power supply adaptor.
- Software Installation CD.
- 8 x SMA calibration standards:
 - 1 x SMA-Male OPEN
 - 1 x SMA-Male SHORT
 - 1 x SMA-Male TERMINATION
 - 1 x SMA-Female OPEN
 - 1 x SMA-Female SHORT
 - 1 x SMA-Female TERMINATION
 - 1 x SMA-Female to SMA-Female THROUGH adaptor
 - 1 x SMA-Male to SMA-Male THROUGH adaptor
- 2 x SMA-Male TERMINATION.

1.4 Unpacking and first use.

When opening the wooden box, unpack the instrument first and get familiar with its connectors and display. Figure 1.1 shows the front side of the instrument.

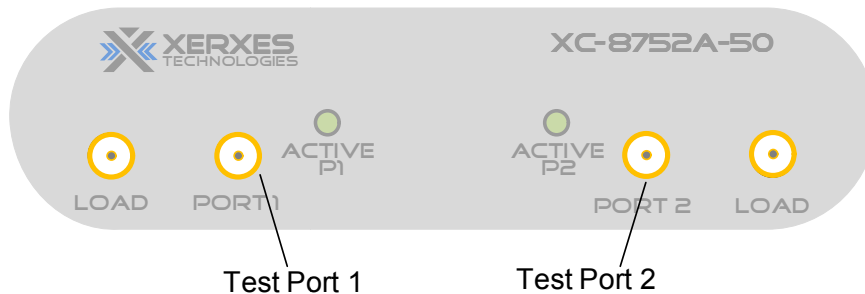


Figure 1.1: Front panel of the vector network analyzer.

There are four female SMA connectors visible, Port 1 and Port 2 are used for the measurement of the unknown device under test (DUT). There are two more connectors labeled with "LOAD". Please use any two of the delivered three SMA-Male terminations to connect to these connectors. These terminations are used as reference impedance for the built in active SWR-bridges. Figure 1.2 shows one of the SMA-Male 50 Ohm terminations.



Figure 1.2: SMA-Male termination



The SMA termination at the two LOAD ports shall remain on the instrument at all time. They have only internal signal processing purposes, and need not to be touched in normal operation or calibration.



Do not use excessive torque on the analyzers connectors, the connector or the internal PCB may be damaged!

Figure 1.3 shows the SMA Termination screwed on the vector network analyzer (VNA)'s "LOAD" ports.

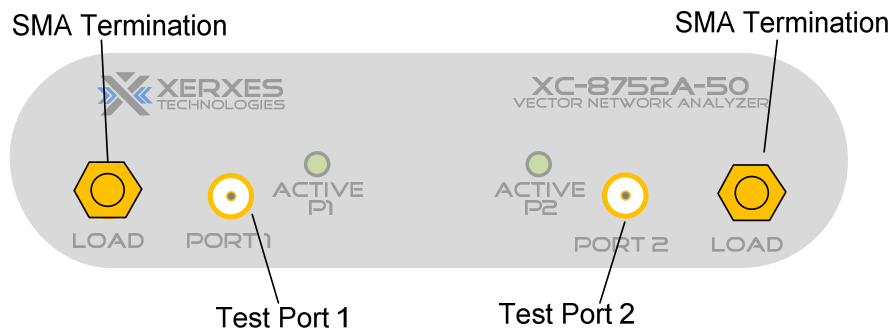


Figure 1.3: Front panel of the vector network analyzer (VNA) with the terminations mounted.



Figure 1.4: Backside of the instrument (VNA)

On the backside of the instrument are another two connectors. One labeled as "Power" connecting to the +12V / 2 A power adaptor and another to connect the USB cable.



Figure 1.5: Supplied wall power adaptor.



Figure 1.6: USB-B cable.

The next step is to install the system-software from the supplied CD-ROM. Insert the CD-ROM into the computers CD-ROM drive and display the folders on the CD-ROM. Check your operating-systems manual, if you don't know how to open the disks folders view. You will also need to install a recent version Adobe Acrobat Reader(R) to view the supplied Install_Instructions.pdf file. Open, read and follow the Install-Instructions. After the installation of the system software and driver, the instrument can be powered on.

First connect the wall adapters barrel connector to the instrument, then connect the wall adaptor to the line-power socket. The wall adaptor is a universal voltage type and can work within a line voltage range from 90 VAC to 240 VAC, 50 Hz or 60 Hz. Next connect the instrument to the computer using the USB cable. Check the device manager, if the VNA shows up within the section "Ports (COM & LPT)".

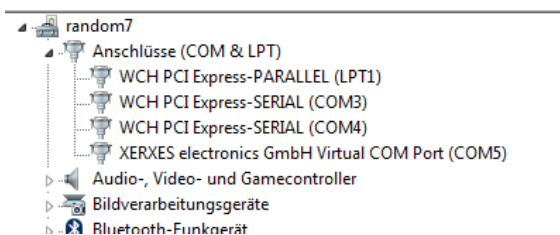


Figure 1.7: Device manager showing instrument.

The instrument shall be shown as: XERXES electronics GmbH Virtual COM Port (ComX).

If the instrument does not show up, check the power supply and USB connection, check for any errors reported by the computer operating-system.

If the device is properly enumerated in the device manager, start the main software XC-8752A-50 in the program folder "xerxes electronics gmbh".

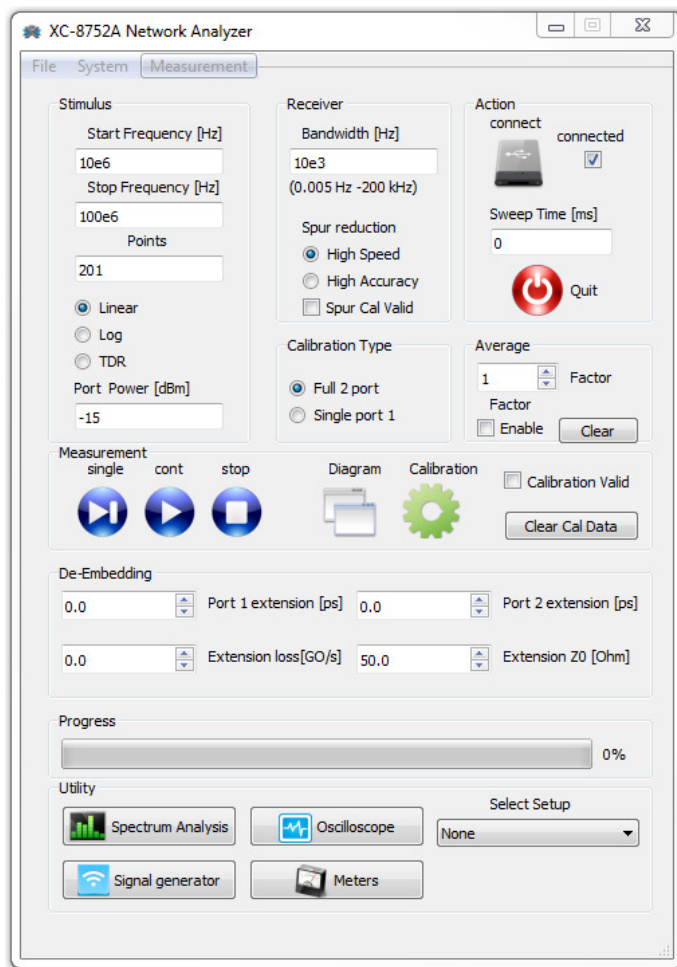


Figure 1.8: XC-8752A main window.

After starting the software, the main window shall appear. The "connected" checkbox should be checked if the instrument is connected to the computer. It is possible to try a reconnect by pressing the "connect" button on the left side of the checkbox "connected". It is also possible to disconnect and reconnect the USB cable during the instrument is powered on and then pressing connect to try to connect the main software to the device. Sometimes it can be necessary, if the software seems to be stuck, to unplug the power supply from the device and reconnect it, further pressing the connect button to re-establish the USB connection.

To exit the software, the red "Quit" button can be used to terminate all operation.

1.5 Technical data.

VNA specifications

Measured Parameters	S_{11} , S_{12} , S_{21} , S_{22}
Measurement Channels	Four parallel receiver chains
Data Traces	Arbitrary number of traces and diagrams
Memory traces	4 full S-parameter memory slots
Data display formats	Rectangular and smith diagram, over 50 trace functions, including time domain and group delay.
Sweep type	Linear and Logarithmic
Measured points per sweep	2 - 2048
Power Settings	-15 dBm to -80 dBm in 0.1 dB steps
Sweep Trigger	Continuous, Single, Hold
Trace Math	Normalization, Magnitude, Phase, log, Real, Imag, Complex, Delay
De-Embedding	Port Extension with loss, delay and Z_0 adjustment, full de-embedding.
Calibration	SOLT (short, open, load, through), normalization
Calibration Types	1-port or 2-port
Measurement bandwidth	0.1 Hz - 200 kHz adjustable
Frequency range	0.25 Hz - 1000 MHz
Frequency resolution	0.25 Hz to 500 MHz 0.5 Hz to 1 GHz
Setups	Arbitrary number of predefined setups
Output signal amplitude accuracy (typ.)	+/- 1 dB @ -15 dBm to -25 dBm
	+/- 2 dB @ -25 dBm to -40 dBm
	+/- 3 dB @ -40 dBm to -80 dBm
Trace noise (typ.)	3 mdB
Measurement speed (typ.)	1.5 ms / frequency point
Frequency accuracy	+/- 25 ppm
Operating temperature	0°C - 55°C ambient
Operating humidity	0% to 80% rel. humidity
Power consumption	15 Watt max.
Power requirements	+12V / 2A
Connection	USB 2.0, Full-Speed

Table 1.1 VNA specifications

Spectrum analyzer, EMI analyzer and Phase Noise analyzer utility

Frequency range	10 Hz - 1000 MHz
Resolution bandwidth	10 Hz - 200 kHz
Frequency resolution	0.25 Hz to 500 MHz 0.5 Hz to 1 GHz
Frequency accuracy	+/-25 ppm
Amplitude accuracy	+/- 1.5 dB typ.
Unambiguous frequency range	10 Hz - 60 MHz
Low spurs technology (sideband suppression)	Multi frequency sampling
Frequency points	Arbitrary
Display functions	RMS, Minimum, Maximum and Average
Maximum linear input power	19 dBm
Phase noise (low noise mode) @ 300 MHz	< -90 dBc @ 100 Hz offset
	< -115 dBc @ 1kHz offset
	< -115 dBc @ 10kHz offset
	< -115 dBc @ 100kHz offset
	< -125 dBc @ 1MHz
Quasi Peak	Following CISPR 16-1-1 (9 kHz - 1 GHz)
Input noise voltage	< 30 nV/Sqrt(Hz) @ f > 10 kHz

Table 1.2 Spectrum Analyzer specifications

Signal Generator utility

Frequency range	1 Hz - 1GHz
Frequency resolution	0.25 Hz to 500 MHz 0.5 Hz to 1 GHz
Output power range	-15 dBm to -75 dBm (1Hz to 500 MHz)
	-15 dBm to < -90 dBm (500 MHz to 1GHz)
Output signal amplitude accuracy (typ.)	+/- 2 dB @ -15 dBm to -25 dBm
	+/-2.5 dB @ -25 dBm to -40 dBm
	+/-3 dB @ -40 dBm to <-80 dBm
Phase noise (low noise mode) @ 300 MHz	<-90 dBc @ 100 Hz offset
	< -115 dBc @ 1kHz offset
	< -115 dBc @ 10kHz offset
	< -115 dBc @ 100kHz offset
	< -125 dBc @ 1MHz

Table 1.3 Signal Generator specifications

Resolution	12 Bit (up to 16Bits with CIC Filter)
Input range	Max. +/- 3V
Memory	Max. 2048 points
Lowpass Filter	CIC type, adjustable
Sampling range (real)	15 S/s - 125 MS/s
Sampling range (Sin(x)/x)	250 MS/s - 4 GS/s
Bandwidth	60 MHz (Nyquist), 500 MHz (real)
Protocol analyzer	SPI, I2C, RS232
Measurement functions	24 measurement functions like RMS, period..
Trigger Modes	Edge Trigger, Pulse Trigger, Manual, Auto, A->B
Trigger Delay	0 - 508 samples
Input	50 Ohms single ended, 100 Ohms differential
Special	Trigger aperture and HF Suppression filters

Table 1.4 Oscilloscope specifications

1.6 Network analyzer operation.

1.6.1 Stimulus.

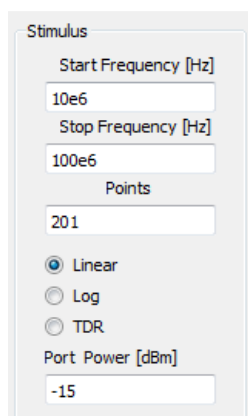


Figure 1.9 Stimulus button group

The stimulus panel is used to control the signal source of the analyzer. Start and Stop frequency need to be typed in Hz. The maximum frequency range is 1 Hz to 1 GHz. Any band within this frequency range can be used for measurement with a frequency resolution of 1 Hz. The field "points" represents the number of discrete measurements in the specified frequency band. The distribution of the measurements can be selected to be linear, logarithmic or TDR.

A linear distribution has equidistant frequency distribution like 1 kHz, 2 kHz, 3 kHz, 4 kHz etc., while a logarithmic distribution like 10 Hz, 100 Hz, 1 kHz, 10 kHz has increasing frequency distances between

each measurement point. Logarithmic frequency distributions are very useful for multi-octave measurements.

The button "TDR" is used for low-pass time domain measurements. It will produce a linear distribution like "linear", but the start frequency needs to be 0 Hz. As the analyzer cannot make a measurement at 0 Hz, which is required for the time domain transformation, the 0 Hz measurement point will be replaced by a fitting low frequency measurement. The "TDR" option is ONLY used for low-pass time domain measurements.

The Port Power reflects the calibrated output power of the active port during measurement. It can be set between -7 dBm to -80 dBm. A port level of -15 dBm is recommended. A lower output power will increase noise while a higher one may not be possible in some frequency bands but will lower the noise floor.



Please note that changing start-frequency, stop-frequency, points or distribution, will render an active calibration invalid, since the calibration is made for every frequency point. However changing the output power may reduce accuracy of the measurement, but will not invalidate a calibration.

1.6.2 Averaging

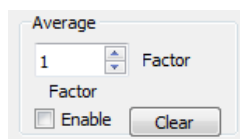


Figure 1.9b Averaging section

If averaging is enabled the analyzer starts S-parameter coherent averaging. So measurement accuracy and noise floor can be improved by cost of measurement time. Select the number of measurements to be used and if necessary the averaging can be restarted by pressing the "Clear" button. The averaging will create perform a gliding mean filter.

1.6.3 Receiver section.

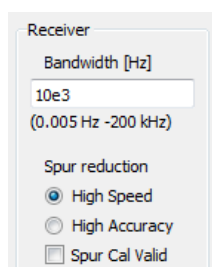


Figure 1.10 Receiver section

The network analyzer uses four isolated receiver chains to measure the 2-port S-parameters. The bandwidth of the receivers can be adjusted to be within about 0.005 Hz and 200 kHz. A lower bandwidth produces less noise and so a better signal to noise ratio, however the measurement time

will also increase with a lower bandwidth. For low frequency measurements the system will choose automatically the necessary bandwidth, and so the entered value can be interpreted as upper boundary of the used bandwidth. Although the bandwidth can be entered in arbitrary steps, the effective bandwidth will follow 2^n steps, like 1 Hz, 2 Hz, 4 Hz, 8 Hz etc. The bandwidth can be changed anytime without invalidating the active calibration.

The VNA has also a built-in spur reduction. This spur reduction will eliminate effectively discrete frequency spurious signals produced by the signal generators harmonics and higher order mixing products. The "High Accuracy" mode will eliminate any possible spurious peak, but will increase measurement time. The reduction is accomplished by a slight variation of the internal sampling clocks. The spur reduction can also be calibrated in the calibration menu. If calibrated the necessary sweeps are reduced from three to two. Further In calibrated measurements a spur pre-calibration will significantly increase accuracy.

1.6.4 Connection and sweep time

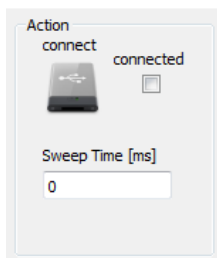


Figure 1.11 Connection section

If the device is connected to the PC on starting the analyzer software, the software will automatically attempt to establish a connection. If the connection is made the "connected" checkbox will be checked. If the device is connected to the PC at a later time, the device symbol can be click to try a connection. If the "connected" checkbox is not activated, check the power cable, USB cable and the device manager.

If the system should fail to make a measurement and reports a problem, try to reconnect the device (power and USB) and press the "connect" button.

Sweep Time in milliseconds will introduce a delay after each sweep. If a slower point to point measurement is desired, reduce the bandwidth in the receiver section.

1.6.5 Selecting the port number.

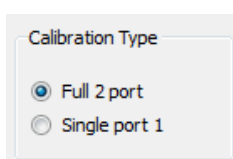


Figure 1.12 Port selection

The device can measure in two different modes:

Single Port Mode on Port 1:

This setup allow the analyzer to make a reflection measurement on Port 1. This will also change the type of calibration, now using only "Short", "Open" and "Termination" standard.

Full two port mode:

This setup will require a full two port calibration and will allow to measure all S-Parameters of a two port device.



Changing the Calibration type will invalidate the current calibration.

1.6.6 Measurement panel

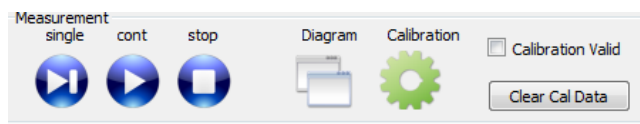


Figure 1.13 The measurement panel.

The three buttons "single", "cont" and "stop" control the measurement process. "Single" will initiate a single measurement, while "cont" will perform a continuous sweep operation. After each sweep any diagrams and traces will be updated. The button "stop" is used to interrupt a continuous measurement.

The diagram button handles all diagram related lists, diagrams and traces, and will be discussed in a later chapter.

The button "Calibration" integrates all VNA calibration related submenus and will also discussed in detail in a later chapter.

The Checkbox "Calibration valid" shows if there is currently an active calibration used. Precision measurements can only be made with an active calibration. "Clear Cal Data" will invalidate a current active calibration.

1.6.7 Simple De-Embedding.

Although the analyzer implements a full S-Parameter de-embedding function, there is an additional simple line extension feature, which can be controlled during measurement.

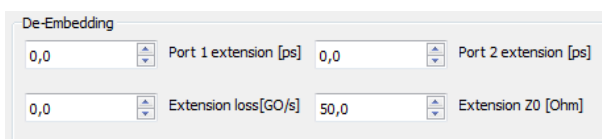


Figure 1.14 Simple Port Extension De-Embedding.

This feature is used to extend (or shrink) a virtual transmission line attached to each ports calibration plane. By careful adjustment a physically attached transmission line can be eliminated in measurements by using negative delay values. The transmission line length must be entered in picoseconds, further loss and the characteristic impedance of the transmission line can also be adjusted.

1.6.8 Utility menu.

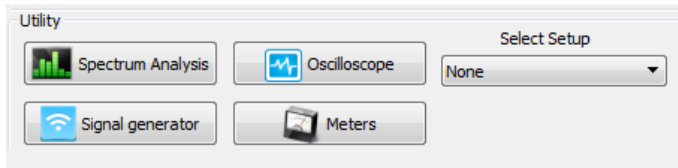


Figure 1.15 Utility menu buttons.

The analyzer features many utilities, which can be selected by the corresponding button. This utilities will be explained in detail in a later chapter.

The "Setup Selector" is used to load quickly any predefined setups. These setups contain measurement setups, calibration data, diagrams and traces.

1.7 Principle of operation.

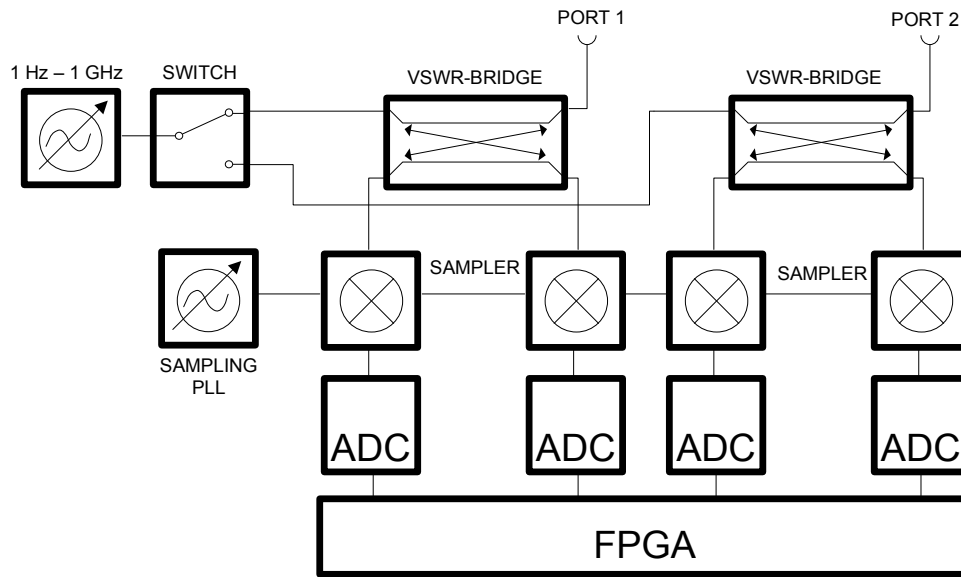


Figure 1.16 Instrument Block Diagram.

The VNA consists of two VSWR-bridges which operate like directional couplers. Each Port can be fed by the signal generator over a RF switch. This allows forward and reverse measurements, and so full S-Parameter capture. The output of the directional bridges is sampled with a separate adjustable sampling clock. The sampling mixes the up to 1 GHz signal frequency down to a lower frequency band below 70 MHz. The analog to digital converters capture the low frequency band and the digital signals are further processed in an attached FPGA. The adjustable sampling clock prevents down conversion to DC, and keeps the baseband signal larger than 1 MHz.

The analyzer measures four complex voltages: a_1 , b_1 , a_2 , b_2 .

The number denotes the port number the 'a' stand for the forward signal and the 'b' for the reflected signal.


By calculating the complex quotients b_1/a_1 , b_2/a_1 , b_1/a_2 , b_2/a_2 voltage independent vectors can be derived which describe the scattering parameters S_{11} , S_{12} , S_{21} , S_{22} .

These vectors are not directly representations of the desired S-Parameters, since they are influenced by the limited accuracy of the VSWR-bridges, switches and cables. There for a calibration procedure is required to eliminate these errors.

One calibration method called "**SOLT**" is implemented in the analyzers software. This calibration procedure uses four precision known line-elements, to be connected to the analyzer one after another. These standards consist of three single port standards: **Short**, **Open**, 50 Ohms **Load** and one two port element called "**Through**".

After these standards have been connected to the VNA and been measured, the analyzer will calculate 10 complex calibration coefficients for each frequency point, to correct for all the errors.

A simpler but much quicker calibration method can be normalization of the measured quotients, which can also give good results, if carefully done. Normalization can be defined in the trace-edit window.

 Please note that a specific calibration will measure a one-port or two-port device at/between the reference plane(s). This virtual plane is shown in Figure 1.17. Any cable between will add to the measurement, and needs to be de-embedded.

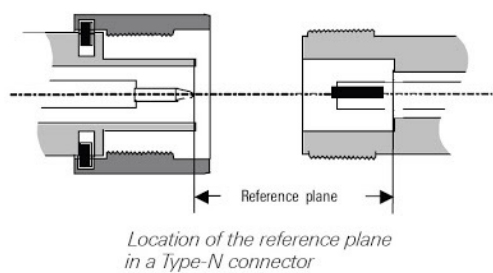


Figure 1.17 Definition of the reference plane.

2 VNA Calibration.

2.1 Introduction.

The accuracy of a network analyzer directly depends on the quality of its calibration. Every frequency range, resolution and cable setup needs a specific calibration procedure. This calibration is performed by using the delivered precision calibration standards:

Open (male and female)

Short (male and female)

Load (male and female)

Through(male and female)

A single port calibration uses only the Open, Short and Load standard, while a full two-port calibration utilizes all four standards.

The Open standards represent open transmission lines, while the Short standards are realized by shorted transmission lines. The Load standard is a 50 Ohms termination. The through standards are pieces of 50 Ohms transmission lines. Each of these standards have very accurate measured length, impedance and parasitic components.

In the XC-8752A there are two ways of defining calibration standards:

- Formula driven.
- S-Parameter driven.

The formula driven standards are represented by parameters like transmission line delay, loss, impedance etc. while the S-Parameter definitions simply use supplied S-Parameter files in Touchstone(c) .s2p format.

2.2 Calibration standards.

Before being able to perform a calibration, calibration standard definitions must be loaded or entered in the standard definition window.

First press the "calibration" button in the main window.



The XC-8752A must be connected to the USB and power to be able to open the calibration window.

In the calibration window press the "Define Standards" button to open the standards dialog. A window as shown below shall appear:

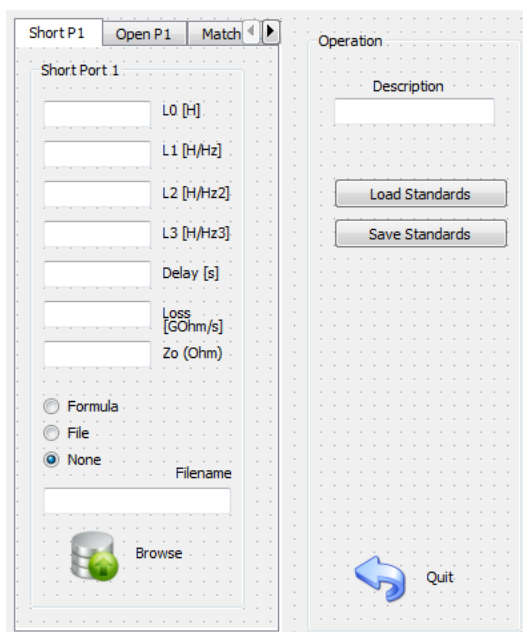
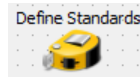


Figure 2.1 Standards Definition dialog.

On the right side of the dialog it is possible to load or save standard definition files. The description text field is used to give some better description of the standards. On the left side of the dialog it is possible to choose the desired standard by a tab selector. For each standard it can be chosen if either a formula description or S-Parameter file is used. If it is desired to use other standards than the supplied ones, it is possible to enter the description by hand and save the data as new standard definition file. The format of the parameters used to describe the standards are a common way used in the industry by many other VNA manufacturers.



However care must be taken by the magnitude if values are given in picoseconds (ps) and need to be entered in seconds (s) or pico-Henries (pH) or femto-Farads (fF) and need to be entered in Henries (H) or Farad(F).

Short Standard Parameter	Meaning
L0, L1, L2, L3	Inductance polynom coefficients of the non ideal short circuit. in [H], [H/Hz]...
Delay	Delay in seconds of the offset transmission line from the reference plane to the short circuit. [s]
Loss	Loss of the offset transmission line from the reference plane to the short circuit. [GOhm/s]
Zo	Characteristic impedance of the offset transmission line from the reference plane to the short circuit. [Ohm]

Table 2.1 Short equation parameters

Open Standard Parameter	Meaning
C0, C1, C2, C3	Capacitance polynom coefficients of the non ideal open circuit. in [F], [F/Hz]...
Delay	Delay in seconds of the offset transmission line from the reference plane to the open circuit.[s]
Loss	Loss of the offset transmission line from the reference plane to the open circuit. [GOhm/s]
Zo	Characteristic impedance of the offset transmission line from the reference plane to the open circuit. [Ohm]

Table 2.2 Open equation parameters

Load Standard Parameter	Meaning
Zo	Characteristic impedance of the offset transmission line from the reference plane to the open circuit. [Ohm]

Table 2.3 Load equation parameters

Through Standard Parameter	Meaning
Delay	Delay in seconds of the transmission line from reference plane to reference plane. [s]
Loss	Loss of the transmission line from reference plane to reference plane. [GOhm/s]
Zo	Characteristic impedance of the transmission line from reference plane to reference plane. [Ohm]

Table 2.4 Through equation parameters



Figure 2.2 The XC-87610A 6 GHz calibration kit

The delivered XC-87610A calibration kit consists of:

- 1 x XC-87601A Male Short Standard.
- 1 x XC-87602A Male Open Standard.
- 1 x XC-87603A Female Short Standard.
- 1 x XC-87604A Female Open Standard.
- 1 x XC-87605A Male 50 Ohms Broadband Load.
- 1 x XC-87608A Female 50 Ohms Broadband Load.
- 1 x XC-87606A Male - Male Through Standard.
- 1 x XC-87607A Female - Female Through Standard.

The definition .xsf definition file of these standards is saved on the CD-ROM or USB Stick and will be installed in the program sub-folder standards. Ready available to the "Load Standards" button.

2.3 Calibration Procedure.

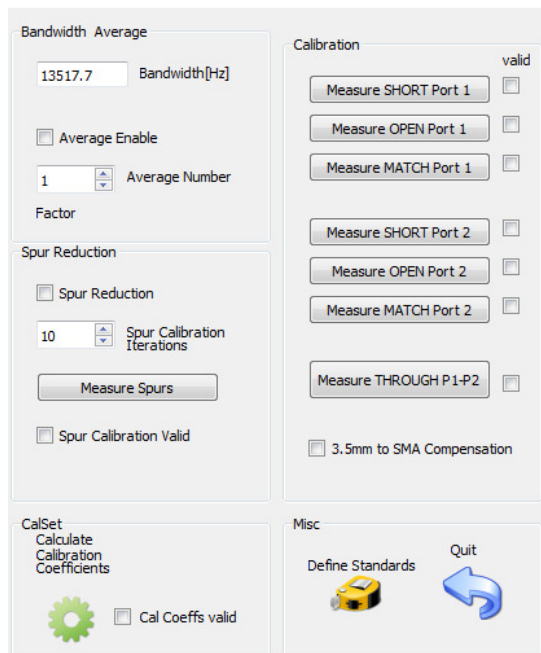


Figure 2.3 Calibration Window

After a suitable standards definition file has been loaded, the actual calibration procedure can be started. Prior to calibration it is necessary to select the port number in the main window.

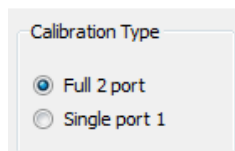


Figure 2.4 Port number selection.

If only a single port calibration is desired, less calibration steps are required. The not needed calibration steps will be grayed-out in the calibration window. (Fig. 2.3). Select the frequency range and the number of frequency points in the main window:

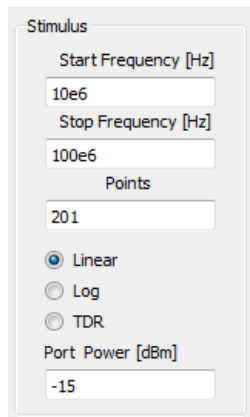



Figure 2.4 Stimulus setup group in main-window.

Further a linear or logarithmic frequency slope can be selected. After the setup in the main window,

the  "Calibration" button need to be clicked to reach the calibration window (Fig. 2.3).

2.3.1 Spur calibration

Since signal generator harmonics and no harmonics related spurs can create slight measurement deviation at certain discrete frequencies, it is recommended to use "Spur Reduction" calibration for high accuracy measurements. To enable "Spur Reduction" for calibration, check the "Spur Reduction" check-box. Select the number of Spur Calibration iterations to further refine spur reduction. Press the "Measure Spurs" button to start spur reduction measurement. It is necessary to connect the "OPEN" standard to Port 1 to proceed. If the spur reduction calibration is finished, the "Spur Calibration Valid" check box will be checked automatically. The spur-cal-data will be saved with calibration data in the "File/ Save Calibration" menu of the main menu, if present.

2.3.2 Calibration Bandwidth and Average

To further refine the calibration data, it is possible to set the receiver bandwidth and the number of measurements for averaging for the calibration measurements.

2.3.3 Calibration procedure

The calibration standards need to be connected to the corresponding ports. After connecting a standard, the corresponding button needs to be clicked. For example after the "Short Male" has been connected to port 1, press the "Measure SHORT Port 1" button and wait until the "valid" check-box is checked. Measure one standard after another in any sequence until all necessary "valid" check-boxes are checked. Finally press the "Calculate Calibration Coefficients" button to finalize the calibration and wait until the "Cal coeffs valid" check-box becomes checked. Now the VNA is calibrated and ready to make accurate measurements.



Please note that changing the frequency range or the sweep type will draw a calibration invalid. Always press "Calculate Calibration Coefficients" to render a calibration measurement valid.

2.3.4 3.5 mm to SMA compensation.

If 3.5 mm standards are used instead of the supplied SMA standards, it is necessary to compensate for the change of the inner connector and outer connector diameter step between these two connector types. This step creates a frequency dependent shunt capacitance which will be compensated by checking the "3.5mm to SMA compensation" check-box.

2.4 Loading or Saving calibration coefficients.

After a successful calibration procedure it is recommended to save the calibration coefficients to disk. This can be accomplished in the main-windows menu. Choose "File" from the main menu and "Save Calibration" from the sub-menu. The calibration coefficients along with the frequency setup and spur data will be saved to disk. Further choosing "Load Calibration" will allow loading calibration setups for instant use.



Please note that "Save Setup" will as well save the calibration data along with diagrams, traces and other configuration data.

3 Measurements and Diagrams

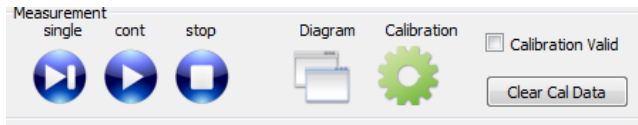


Figure 3.1: Measurement button group on the main window.

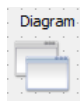
It is possible to perform measurements with the VNA with or without a valid calibration by pressing one of the blue "Measurement" buttons. the button "single" will initiate a single measurement, while "cont" will start continuous measurements. The button "stop" can be used to stop continuous measurements.



Uncalibrated measurements will capture raw data and can be used for a simple normalization calibration.

To display some measurements on the screen it is necessary to create diagrams and traces.

3.1 Creating a diagram



By pressing the "Diagram" button on the main window the Diagram/Trace dialog will pop up.

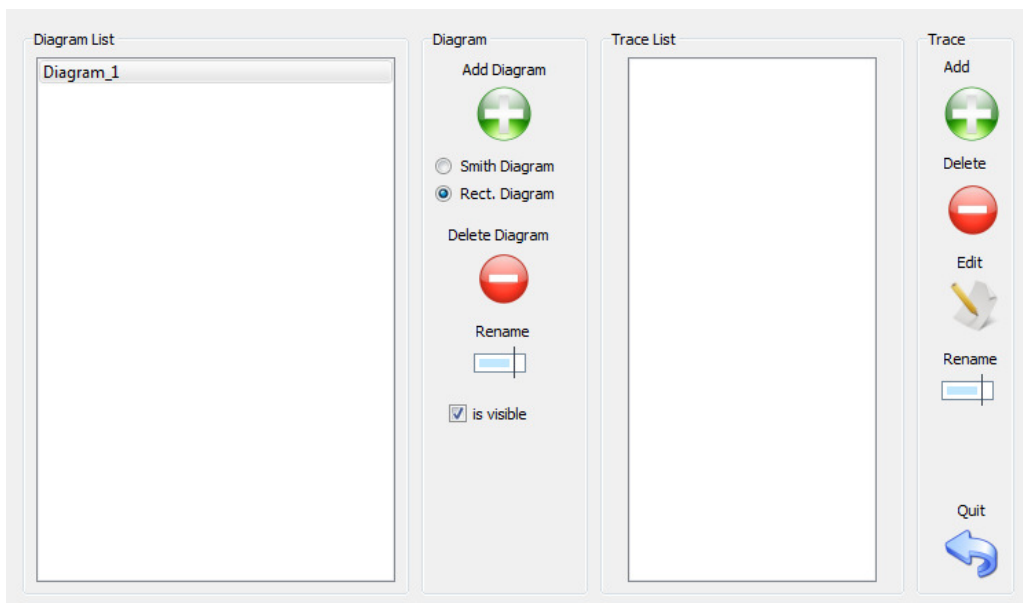


Figure 3.2. Diagram/Trace Dialog

The first decision to make is if a rectangular diagram or a smith diagram is desired. After selecting the type of diagram by the radio-buttons, the button "Add Diagram" can be pressed to add the diagram to the diagram list. The diagram will be instantly displayed on the screen.

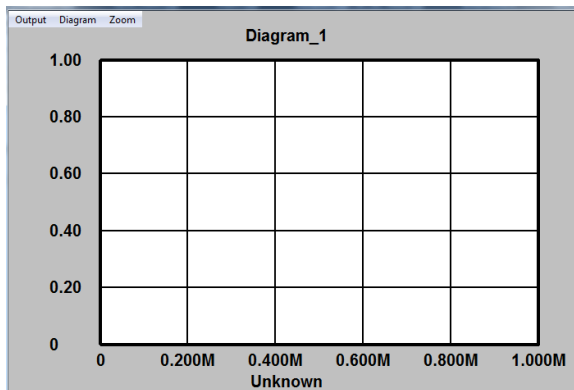


Figure 3.3. New rectangular diagram.

Now the new diagram in the diagram list can be clicked to select it, or double-clicked to add a measurement trace.

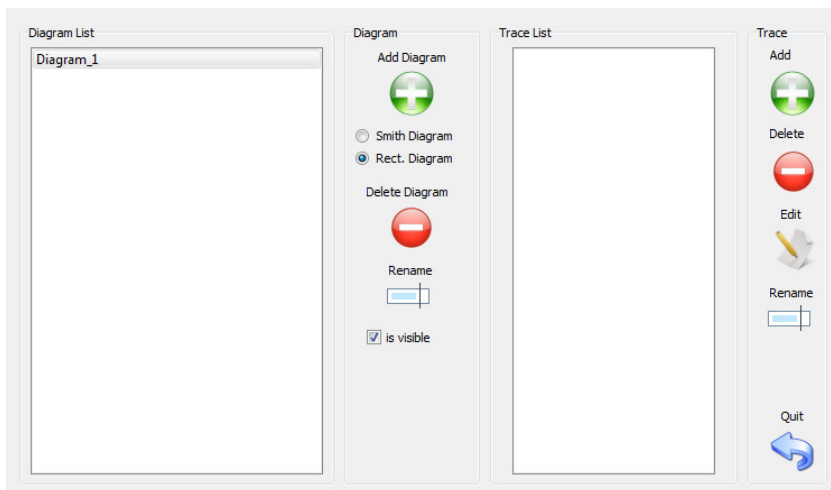


Figure 3.4. Diagram List

Further it is possible to change the name of the diagram by selecting the diagram's name in the "Diagram List" and then pressing the "Rename" button.

First select a diagram by single left click on the item in the Diagram List. Then select Rename or Delete Diagram.



A selected diagram type of "rectangular" or "smith" cannot be changed after creating the diagram.

The right side of the window holds the "Trace List" for the selected diagram.

The corresponding trace list buttons will now allow to create, delete or edit traces for the selected diagram.

Press the "Add" button to add a new trace.

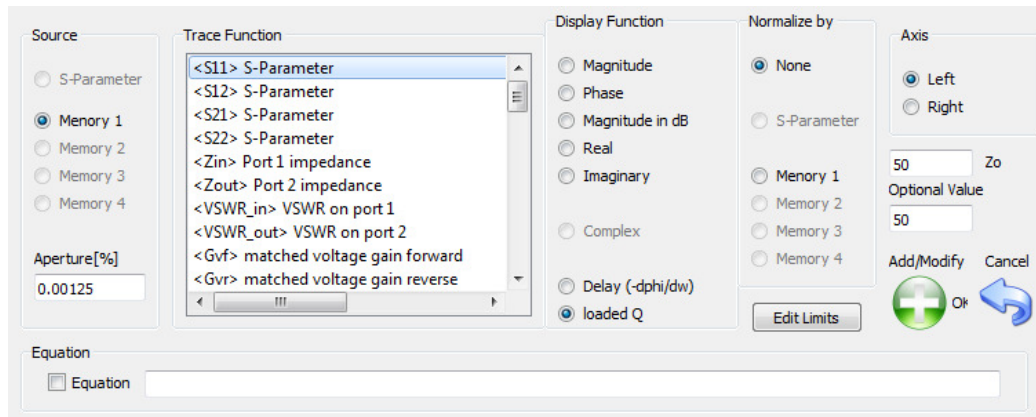




Figure 3.6. Trace Edit/Add dialog

 Closing a diagram will make the diagram invisible, although all data is retained. Select a diagram in the diagram list and click the "is visible" checkbox to make the window re-appear.

The trace definition process starts from the left window side and ends at the right end of the dialog.

1. Select a source for the trace. this can be the actual measured S-Parameter or one of four memory slots, which can be loaded by the main-windows menu. Not available sources will be grayed-out.

 It is necessary to make at least one measurement or load some S-Parameters to a memory slot to create a trace.

2. Select a Trace Function. There are a number of trace function available to the source parameters.

Trace Function	Description
<S11> S-Parameter	S_{11} scattering parameter
<S12> S-Parameter	S_{12} scattering parameter
<S21> S-Parameter	S_{21} scattering parameter
<S22> S-Parameter	S_{22} scattering parameter
<Zin> Port 1 impedance	Complex impedance at Port 1 with termination on Port 2
<Zout> Port 2 impedance	Complex impedance at Port 2 with termination on Port 1
<VSWR_in> VSWR on port 1	Voltage standing wave at Port 1 with termination on Port 2
<VSWR_out> VSWR on port 2	Voltage standing wave at Port 2 with termination on Port 1
<Gvf> matched voltage gain forward	Complex V_2/V_1 voltage ratio with termination on Port 2
<Gvr> matched voltage gain reverse	Complex V_1/V_2 voltage ratio with termination on Port 1
<Gif> matched current gain forward	Complex I_2/I_1 current ratio with termination on Port 2
<Gir> matched current gain reverse	Complex I_1/I_2 current ratio with termination on Port 1
<k> stability factor	Stability of a two port if $k > 1$
<Gms> maximum stable gain	Maximum stable gain of a two port
<Gmax> maximum available gain	Maximum available gain of a two port
<Gvfo> open output voltage gain forward	Complex V_2/V_1 voltage ratio with open Port 2
<Gifs> short output current gain forward	Complex I_1/I_2 current ratio with open Port 2
<Gvfa> arbitrary load output voltage gain forward (use optional value)	Complex V_2/V_1 voltage ratio with resistance on Port 2 given by the optional input field.
<Gifa> arbitrary load output current gain forward (use optional value)	Complex I_2/I_1 current ratio with resistance on Port 2 given by the optional input field.
<series RC> R-C in series equivalent, Real = R, Imag = C	Calculates R-C equivalent series circuit from S_{11} . Resistance in Ohm is the real-value, capacitance in farad is the imaginary value.
<parallel RC> R-C in parallel equivalent, Real = R, Imag = C	Calculates R-C equivalent parallel circuit from S_{11} . Resistance in Ohm is the real-value, capacitance in farad is the imaginary value.
<series LR> R-L in series equivalent, Real = R, Imag = L	Calculates R-L equivalent series circuit from S_{11} . Resistance in Ohm is the real-value, Inductance in Henry is the imaginary value.
<parallel LR> R-L in parallel equivalent, Real = R, Imag = L	Calculates R-L equivalent parallel circuit from S_{11} . Resistance in Ohm is the real-value, Inductance in Henry is the imaginary value.
<Z11> Z-Parameter	Z_{11} impedance parameter
<Z12> Z-Parameter	Z_{12} impedance parameter
<Z21> Z-Parameter	Z_{21} impedance parameter
<Z22> Z-Parameter	Z_{22} impedance parameter
<A> ABCD-Parameter	A - ABCD parameter.
 ABCD-Parameter	B - ABCD parameter.
<C> ABCD-Parameter	C - ABCD parameter.
<D> ABCD-Parameter	D - ABCD parameter.

Table 3.1 Available Measurement function

Trace Function	Description
<TDR_S11_LP_RW>	Time Domain converted S_{11} parameter, using a low pass characteristics, DC - fmax conversion using no window function.
<TDR_S11_LP_HW>	Time Domain converted S_{11} parameter, using a low pass characteristics, DC - fmax conversion using a Hamming window function.
<TDR_S11_BP_RW>	Time Domain converted S_{11} parameter, using a band pass characteristics, fmin - fmax conversion using no window function.
<TDR_S11_BP_HW>	Time Domain converted S_{11} parameter, using a band pass characteristics, fmin - fmax conversion using Hamming window function.
<TDR_S21_LP_RW>	Time Domain converted S_{21} parameter, using a low pass characteristics, DC - fmax conversion using no window function.
<TDR_S21_LP_HW>	Time Domain converted S_{21} parameter, using a low pass characteristics, DC - fmax conversion using a Hamming window function.
<TDR_S21_BP_RW>	Time Domain converted S_{21} parameter, using a band pass characteristics, fmin - fmax conversion using no window function.
<TDR_S21_BP_HW>	Time Domain converted S_{21} parameter, using a band pass characteristics, fmin - fmax conversion using Hamming window function.
<TDR_S12_LP_RW>	Time Domain converted S_{12} parameter, using a low pass characteristics, DC - fmax conversion using no window function.
<TDR_S12_LP_HW>	Time Domain converted S_{12} parameter, using a low pass characteristics, DC - fmax conversion using a Hamming window function.
<TDR_S12_BP_RW>	Time Domain converted S_{12} parameter, using a band pass characteristics, fmin - fmax conversion using no window function.
<TDR_S12_BP_HW>	Time Domain converted S_{12} parameter, using a band pass characteristics, fmin - fmax conversion using Hamming window function.
<TDR_S22_LP_RW>	Time Domain converted S_{22} parameter, using a low pass characteristics, DC - fmax conversion using no window function.
<TDR_S22_LP_HW>	Time Domain converted S_{22} parameter, using a low pass characteristics, DC - fmax conversion using a Hamming window function.
<TDR_S22_BP_RW>	Time Domain converted S_{22} parameter, using a band pass characteristics, fmin - fmax conversion using no window function.
<TDR_S22_BP_HW>	Time Domain converted S_{22} parameter, using a band pass characteristics, fmin - fmax conversion using Hamming window function.
<Gtp> Transducer Gain	Calculates the two port transducer gain using an arbitrary output Port 2 load defined by the optional value
<Q> Quality factor	Calculates the Quality factor from the S_{11} parameter. (Divides imaginary impedance by real impedance)

Table 3.1 Cont. Available Measurement functions

3. Select a Display Function. The Display function will be applied on the Trace-Function (TF).

Display Function	Description
Magnitude	Equals to $ TF $ or $\text{abs}(TF)$
Phase	Equals to $\arg(TF)$ in Degrees
Magnitude in dB	Equals to $10 \cdot \log(TF)$ or $10 \cdot \log(\text{abs}(TF))$
Real	Equals to $\text{real}(TF)$
Imaginary	Equals to $\text{imag}(TF)$
Complex	Only used in Smith diagrams and equals to TF
Delay	Equals to $-d\phi/d\omega$, the derivation of phase over frequency
Loaded Q	Equals to $-d\phi/d\omega \cdot \omega/2$

Table 3.2 Available Display Functions

4. Select a source to normalize measurement. This can be used for simple calibration, or other relative measurements.
5. Select axis for rectangular diagram. Since rectangular diagrams may have a right or left Y-axis the function can be assigned to either of them.
6. Set options: For Delay and loaded Q Measurement the Aperture can be set, to get a less noisy result. Default is an aperture size of 0.125% of the total frequency range. Some trace function need an optional value, this value can be set in the optional field. Further the system Z_0 can be set to a specific value. All subsequent calculations will use this characteristic impedance Z_0 as a basis.

3.1.1 The Equation Editor.

Unlike defining an output trace by selecting all 6 steps like in the last chapter it is possible to define an output equation, which will be used to generate the display trace. The equation can be entered in the text field, and activated by checking the "Equation" check box.

The equation editor supports following standard operators, with declining binding priority:

Operator	Description
\wedge	Power function
$*$	Multiply
$/$	Divide
$+$	Add
$-$	Subtract

Table 3.3 Equation Standard Operator

Further the equation editor supports standard brackets () and floating-point numbers and following built in constants and variables

Constant/Variable	Description
FREQ	Frequency
W	Frequency * 2 * PI
PI	3.14159...
I or J	Complex imaginary unit = sqrt(-1)

Table 3.4 Equation built in constants or variables

Following source variables are available

Source Data	Description
S	Measured S-Parameters
M1	S-Parameter Memory slot 1
M2	S-Parameter Memory slot 2
M3	S-Parameter Memory slot 3
M4	S-Parameter Memory slot 4

Table 3.5 Available source data identifiers

Every source variable consists of 4 parameters which can be accessed by a dot and following the parameter kind as:

Parameter kind	Description
S	S-Parameters
T	T-Parameters
Z	Z-Parameters
Y	Y-Parameters
A	ABCD-Parameters

Table 3.6 Available source data parameters

The parameter kind follows, without any spaces the parameter identifier:

Parameter identifier	Description
11	X_{11} Parameter
12	X_{12} Parameter
21	X_{21} Parameter
22	X_{22} Parameter

Table 3.7 Available parameter identifier

So for example: S.Z11 identifies the Z11 parameter of the Measured S-Parameters, M1.T21 identifies the T21 parameter of the memory slot 1.

Further there are some built in functions $f(x)$ which can be used in conjunction with above:

Function	Description
REAL()	Real part of complex number
IMAG()	Imaginary part of complex number
ABS()	Absolute value of complex number
ARG()	Phase of complex number in radians
DB10()	$10 \cdot \log_{10}(x)$
DB20()	$20 \cdot \log_{10}(x)$
LOG()	$\log_{10}(x)$ logarithm to the base of 10
LN()	$\ln(x)$ logarithm to the base of e
EXP()	e to the power of x
SQRT()	Square root
SIN()	Sine
COS()	Cosine
TAN()	Tangens
COT()	Cotangens
ASIN()	Arcussine
ACOS()	Arcuscosine
ATAN()	Arcustangens
ACOT()	Arcuscotangens
SINH()	Hyperbolic sine
COSH()	Hyperbolic cosine
TANH()	Hyperbolic tangens
ASINH()	Arcus hyperbolic sine
ACOSH()	Arcus hyperbolic cosine
ATANH()	Arcus hyperbolic tangens
TO_IMAG()	Convert from real to imaginary
TO_Z()	Convert from reflection to impedance
TO_Y()	Convert from reflection to admittance
TO_VSWR()	Convert from reflection to VSWR
CONJ()	Make conjugate complex

Table 3.8 Available functions



The equation editor will immediate display any syntax errors on pressing <return>. The Source variables are only accessible if they exist, otherwise there will be an error on entry.

3.2 The Diagram Dialogs.

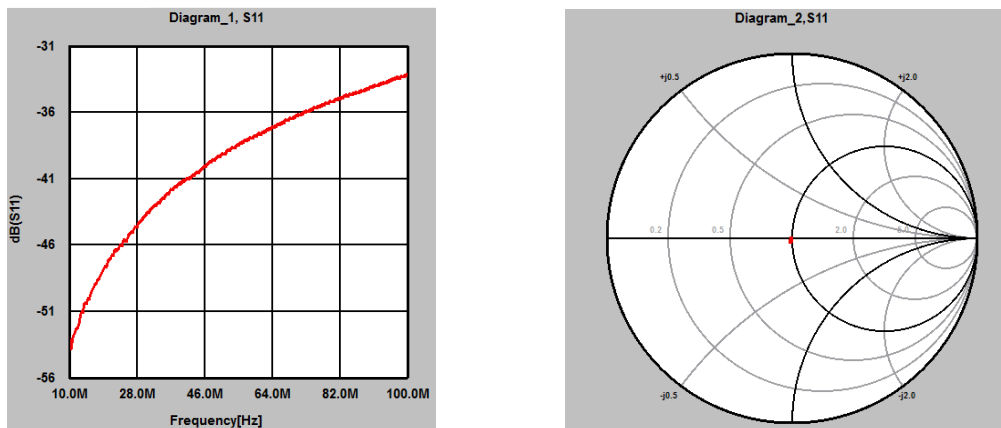


Figure 3.7: Rectangular (left) and Smith (right) diagram examples.

Figure 3.7 shows the two different types of diagrams. The rectangular diagram can be used to display any real valued function over frequency or time. The Smith diagram is useful in displaying complex valued functions with an absolute value (magnitude) of less than one. Both diagram types are automatically updated on a new measurement sweep. To setup a diagram there are three ways:

- Select Diagram->Properties from the dialog menu.
- Double-Click on an empty (trace free) region on the diagram.
- Right click on an empty (trace free) region on the diagram to get the context menu and select "Edit diagram properties".

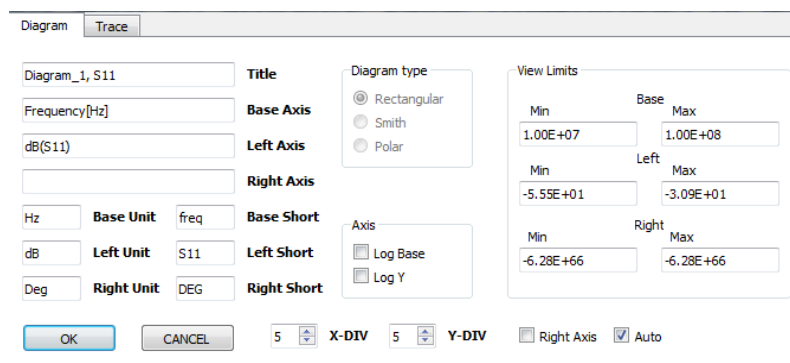


Figure 3.8: Diagram properties dialog.

Following items can be edited:

- **Title:** Represents the title of the diagram displayed on top of the dialog.
- **Base Axis:** Represents the description of the diagram x-axis.
- **Left Axis:** Represents the description of the diagrams left y-axis.
- **Right Axis:** Represents the description of the diagrams right y-axis (if present).
- **Base Unit:** The unit of the x-axis like [Hz] or [s].
- **Base Short:** Short description of the x-axis for marker display.
- **Left Unit:** The unit of the left y-axis like [V] or [dBm].
- **Left Short:** Short description of the left y-axis for marker display.
- **Right Unit:** The unit of the right y-axis like [V] or [dBm].
- **Right Short:** Short description of the right y-axis for marker display.
- **Log Base:** This checkbox allows to display the x-axis in logarithmic scale.
- **Log Y:** This checkbox allows to display the y-axis in logarithmic scale.
- **Right Axis:** This checkbox allows to make the right y axis appear.
- **Auto:** This checkbox allows to override the manual Y scaling with automatic scaling.
- **Base Min/Max:** Represents the minimum and maximum value of the x-axis.
- **Left Min/Max:** Represents the minimum and maximum value of the left y-axis.
- **Right Min/Max:** Represents the minimum and maximum value of the right y-axis.

To show the diagram in the VNA main window "Diagram" dialog, Right click on an empty (trace free) region on the diagram to get the context menu and select "Edit diagram source". The diagram dialog from Fig. 3.2 will appear if applicable.

3.2.1 Using Mouse and Keys on diagram dialogs.

3.2.1.1 X-Axis Scaling, Horizontal Zoom, Horizontal Scrolling.

To horizontally zoom into a diagram it is possible to use the mouse wheel. Scrolling down the mouse wheel will zoom-in and scrolling up will zoom-out. The point to stay fixed can be selected by moving the mouse pointer in a certain region of the diagram.

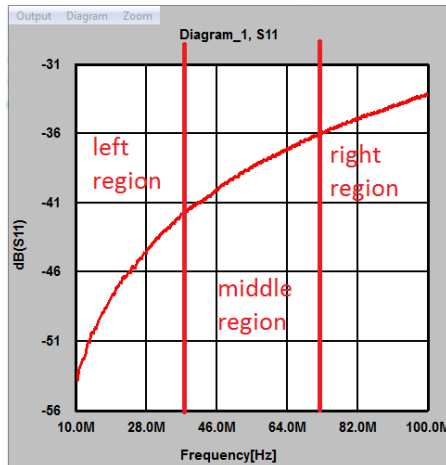


Figure 3.9: Zoom pivot regions for horizontal zooming.

Figure 3.9 Shows the three different regions for horizontal scrolling. If the mouse pointer is in the left region during scrolling the mouse wheel, the end frequency will stay the original value while the start frequency will increase or decrease. If the mouse pointer is located in the middle region, the center frequency remains constant during scrolling, while in the right region the start frequency stays fixed.

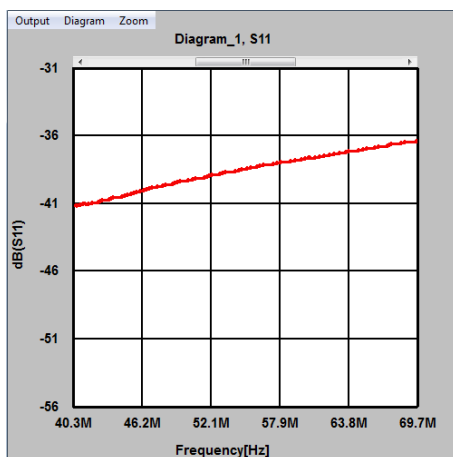


Figure 3.10: Diagram with horizontal scrollbar on top.

Figure 3.10 shows a diagram, where horizontal zoom has been applied. Please note that on top a scrollbar will appear, which can be used to pan the view left or right. Further it is possible to click into the diagram, hold the left mouse button pressed, and move left or right to pan the view.

Another option is use the diagram dialogs main menu and select Zoom->Zoom in horizontal or Zoom->Zoom out horizontal.

3.2.1.2 Y-Axis Scaling, Vertical Zoom, Vertical Scrolling.

To vertically zoom into a diagram it is also possible to use the mouse wheel. Scrolling down the mouse wheel while holding the "CTRL/STRG" key will zoom-in and scrolling up while holding the "CTRL/STRG" key will zoom-out. The point to stay fixed can be selected by moving the mouse pointer in a certain region of the diagram.



Make sure to have disabled the "Auto" check-box in the diagram properties to be able to zoom vertically.

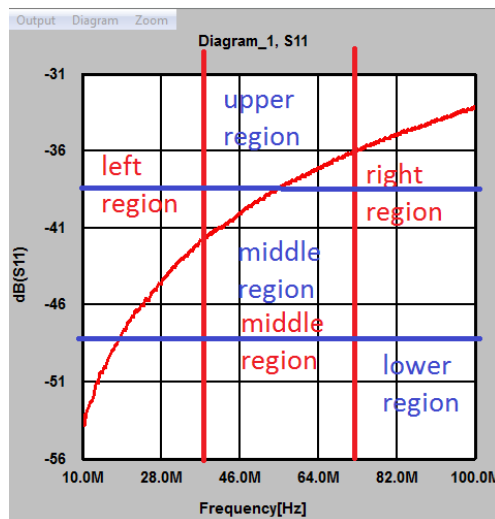


Figure 3.11: Zoom pivot regions for vertical zooming.

Figure 3.11 Shows the three different regions for vertical scrolling. If the mouse pointer is in the upper region during scrolling the mouse wheel, the bottom value will stay the original value while the upper value will increase or decrease. If the mouse pointer is located in the middle region, the center value remains constant during scrolling, while in the upper region the start value stays fixed. Further it depends if the mouse pointer is in the left region, when only the left Y axis will zoom, in the middle region where the left and right Y axis will zoom or in the right region where only the right Y axis will zoom (if present).

Alternatively Zoom->Zoom in left axis or Zoom->Zoom in right axis etc. can be selected from the diagram dialogs main menu.

To perform vertical block panning, move the mouse either in the left, middle or right region of Fig. 3.11 to select the left, both or right axis, press the left mouse button and keep it pressed while moving up and down with the mouse.

To perform vertical fine panning, move the mouse either in the left, middle or right region of Fig. 3.11 to select the left, both or right axis, press the left mouse button and the "Shift" key simultaneously and keep them pressed while moving up and down with the mouse.

3.2.1.3 Printing and copy to clipboard diagrams.

It is possible to print the content of a diagram by selecting Output->Print from the diagram dialogs main menu. For a copy to clipboard select Output->Copy to clipboard from the menu.

3.2.2 Diagram Traces.

Each trace in a diagram can be selected by a single left mouse button click on any part of the trace. Once it is highlighted, a context menu appears on clicking the right mouse button. following options are available:

- Center trace
- Edit trace properties
- Edit trace source
- Delete trace

"Edit trace properties" will open the Diagram properties with the trace-tab activated and the highlighted trace selected in the trace list.

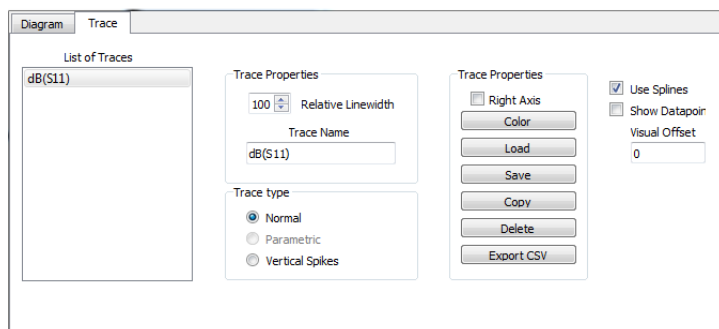


Figure 3.12: Trace properties windows.

The "List of Trace" List shows all available traces for the specific window. The relative line-width in percent can be changed to create thinner or thicker trace curves. The trace name can also be changed. The Trace type radio buttons allow to choose between:

- Normal: Connected dots
- Parametric: For the smith diagram
- Vertical Spikes: Will draw a line from the corresponding point down to the bottom of the diagram.

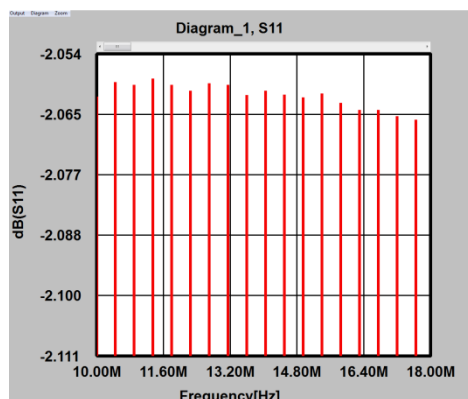


Figure 3.13: Vertical spikes trace.

The color button will allow to change the trace color, while the load and save buttons provide the possibility to load or save the trace data to disk. A saved trace can be loaded multiple times, and may be used for comparison. The copy function will duplicate the current trace, while the delete button will delete the current trace. Another function allows to save trace data to a comma separated text file for import to a spreadsheet program.

The "Use Splines" check-box allows to activate smooth spline curve fitting through the data points, while without splines the data points will be connected by simple lines.



Please note that some measurements with peaks and step changes in values will not be displayed correctly when splines are activated.

The "Show data points" check box will allow to mark the discrete measured data-points on a trace function. The interpolation between the data-points becomes visible.

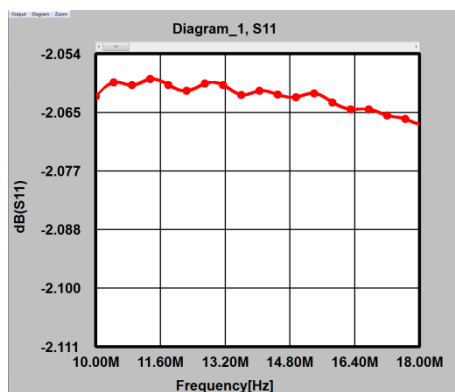


Figure 3.14: Discrete data-points marked on the trace.

The "Visual Offset" line edit element allows to shift the trace vertically in the diagram. Please note that a visual offset does not influence marker measurements, but the axis grid will not be longer valid for a shifted trace. This feature is useful if multiple traces need to be shown in a single diagram, while they would all overlay. This can also be compared to a channel offset function in oscilloscopes.

Now the other points of the trace context menu shall be described:

- Center trace
- Edit trace source
- Delete trace

Center trace will set the "Visual Offset" to zero, while "Edit trace source" will open the trace edit window from Figure 3.6. The "Delete trace" menu item will delete the currently selected trace.

The trace properties window may also be reached by a double-click on a trace.

3.2.3 Diagram Marker

Diagram markers are important measurement functions to accurately determine measured values at certain frequencies or times. The XC-8752A system software will provide the following markers:

- Single Markers
- Delta Markers
- Horizontal Markers (+ Delta)
- Vertical Markers (+ Delta)

Markers can be created by using the diagram's "Marker" menu or for simple markers by pressing "CTRL + M"

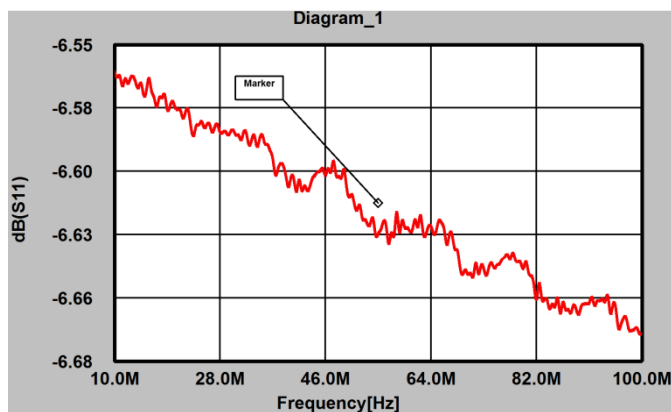


Figure 3.15: Simple Marker added to diagram.

When a fresh marker is created, it needs first to be assigned to a trace, to be useful. Click at the little rhomb and drag the rhomb close to a trace, while pressing the left mouse button. The rhomb shall get red colored when selected.

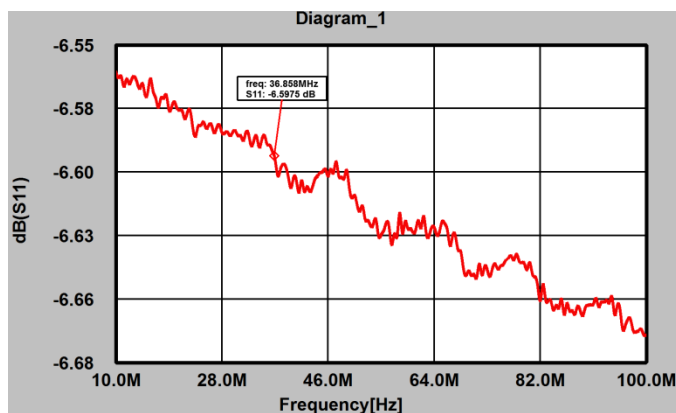


Figure 3.16: Simple Marker assigned to a trace.

Clicking into the little rhomb or the text box will select the corresponding marker. Now it is possible to drag the little rhomb along the trace to find the desired spot. Further it is possible to use the Cursor left and cursor right keys to move a selected marker. Modifier keys like <SHIFT> or <CTRL> in combination with the cursor keys will change the marker step width. The text box can also be selected by a left-click and dragging it to a convenient place.

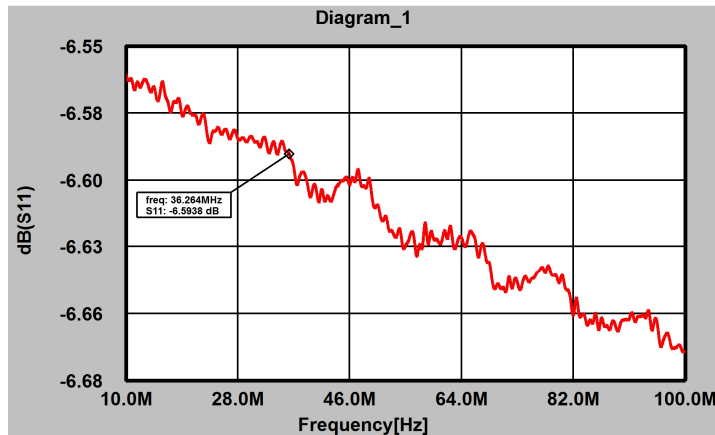


Figure 3.17: Moving the markers text box.

If a marker is selected a specific context-menu can be opened by a right click. Following options are available with the simple marker:

- Set Base Value
- Find Marker Max
- Find Marker Min
- Marker to Center
- Find Marker Next Peak Right
- Find Marker Next Peak Left
- Find Marker Next Valley Right
- Find Marker Next Valley Left
- Find Y Value
- Track Y Value
- Track Max Value
- Track Min Value
- Find Next Y Value Left
- Find Next Y Value Right
- Delete

3.2.3.1 Set Base Value

To accurately set a marker to a certain frequency of point of time, select this option. A little dialog with the current markers base value will appear. Now you can enter the desired value and press return the marker will automatically jump to the set value.

3.2.3.2 Find Marker Max/Min

This function will automatically set the marker to the trace's maximum or minimum value.

3.2.3.3 Marker to Center

This option will move the base (x-axis) of the diagram so that the marker will be in the center of the diagram. This is a useful function to center a trace.

3.2.3.4 Find marker next peak/valley right/left

This function will find the next peak/valley close to the current value.

3.2.3.5 Find Y value

This function will search the trace for a certain Y-value in the trace and set the marker to it.

3.2.3.6 Track Y Value

This function will let the currently selected Marker track the current Y-value in subsequent sweeps. This function is also available with delta-markers. Selecting this function twice will disable it. An active tracking function is marked with a check symbol in the context menu.

3.2.3.7 Track Max Value

This function will let the currently selected Marker track the traces maximum value in subsequent sweeps. This function is also available with delta-markers. Selecting this function twice will disable it. An active tracking function is marked with a check symbol in the context menu.

3.2.3.8 Track Min Value

This function will let the currently selected Marker track the traces minimum value in subsequent sweeps. This function is also available with delta-markers. Selecting this function twice will disable it. An active tracking function is marked with a check symbol in the context menu.

3.2.3.9 Find Next Y Value Right/Left

Since a trace may have several base (X-axis) values with the same Y-value this function will allow to jump to the next Y-value same as the current Y value left or right from the current position.

3.2.3.10 Delete

This option will delete the currently selected marker.

3.2.3.11 Delta Markers.

Delta Markers are like simple markers, but showing the difference in the Y-value and base (X-axis) to another markers position. These kind of markers are useful for measuring bandwidth and other relative measurements. A Delta-Marker can be created by selecting "Marker/Add Delta Marker" from the diagram menu or by pressing <CTRL> D. The reference of the delta marker is the last simple non delta marker defined. If the reference marker is deleted the delta marker will become a simple marker.



The delta marker will show their values relative to the last define simple marker. Multiple delta marker can be defined referencing the same simple marker.

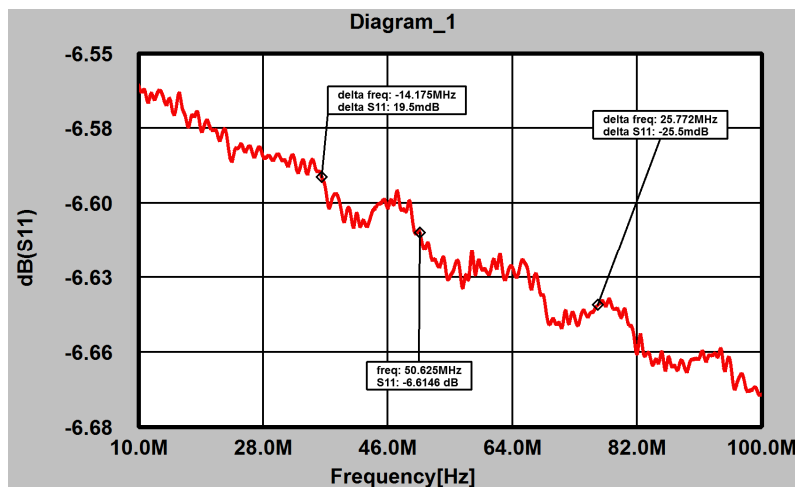


Figure 3.18: Multiple Delta Marker

3.2.3.12 VLine Marker

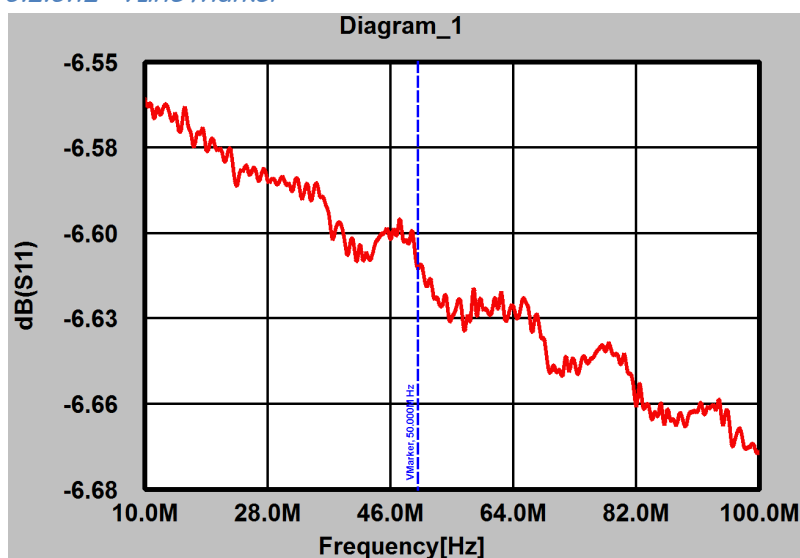


Figure 3.19: VLine Marker

The VLine (Vertical Line) marker is an infinite vertical line. It is possible to drag the line with the mouse and/or use the cursor keys with the <SHIFT> or <CTRL> modifier keys. It is also possible to create and Delta-VLine marker, which will behave like the Delta marker already discussed. A VLine Marker can be created by selecting "Marker/Add Marker VLine" from the diagram menu.

3.2.3.13 HLine Marker

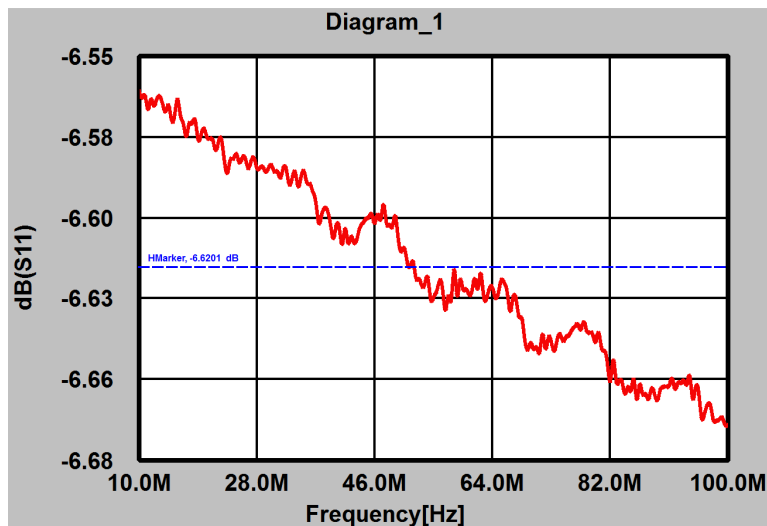


Figure 3.20 HLine Marker

The HLine marker or horizontal-line marker is the complement to the VLine marker. It is also possible to drag the line with the mouse and/or use the cursor keys with the <SHIFT> or <CTRL> modifier keys. It is also possible to create and Delta-HLine marker, which will behave like the Delta marker already discussed. One difference is that it is possible to create a HLine marker referenced to the left or right Y-axis. A HLine Marker can be created by selecting "Marker/Add Marker HLine" from the diagram menu.

3.2.3.14 Legend

The legend is an explanatory text field, which can be placed anywhere in a diagram.

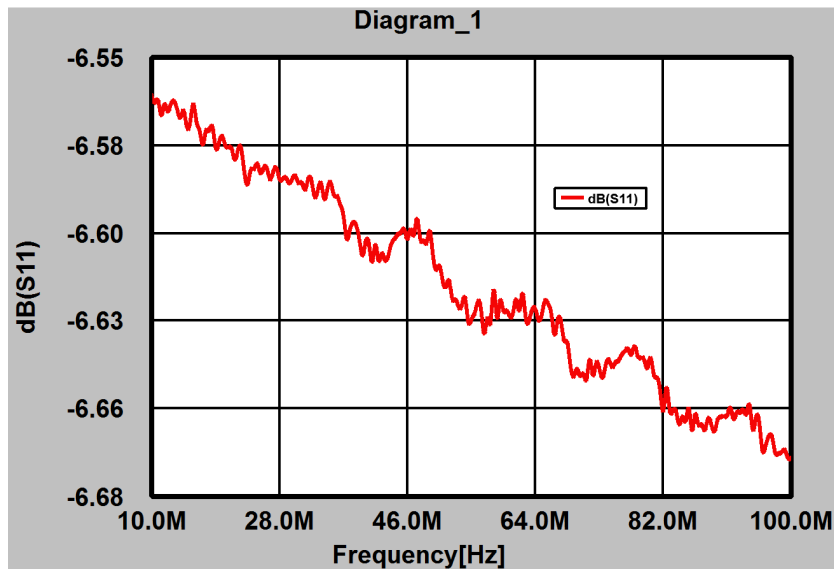


Figure 3.21 Legend added to a diagram.

A legend can be created by selecting the "Diagram/Add Legend" menu or by pressing <CTRL + L>. The legend will update automatically if new traces are added or trace names will change. It is possible to add multiple legends in a diagram.

3.2.3.15 Marker properties.

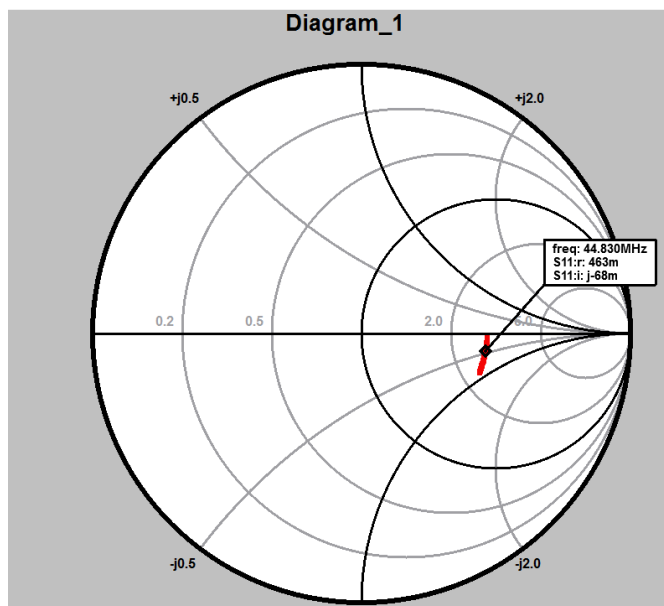


Figure 3.22 Simple marker

If a simple marker is used in a smith diagram, the values are represented in real and imaginary values of the reflection coefficient. If it is desired to show e.g. impedance values, it is possible to select the marker and select "Marker Properties" from the context menu. A dialog will appear, and several display options are presented:

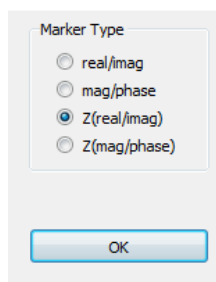


Figure 3.23 Marker properties.

4 The Main Dialog Menu

4.1 File / Load / Save Calibration

This menu items are used to load a new calibration set or save the current calibration.

4.2 File / Load / Save Setup

The XC-8752A setup system is a very convenient way to preserve a prior system state. Once all diagrams, traces and calibration is defined, the save setup command will save all the data to disk. Later on it is possible to continue measurement where it was saved. Since it is possible to define an arbitrary number of setups, a switch to a different workbench is a matter of a click. Further there is a quick setup select box in the lower right corner of the main dialog.

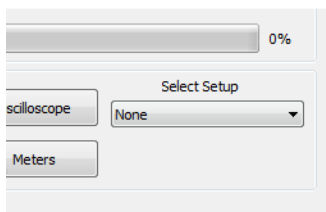


Figure 4.1 Setup Select box.

All setups saved in the "Setups" folder are listed in the setup select box. A single drop down select will switch to different setups.

4.3 File / Save S-Parameter

The Save S-parameter menu item will save the current measured s-parameter in the popular Touchstone .s2p file format.

4.4 File / Select Home Path

The select home path command will allow to change to a different home path than the <USERS>\<CURRENT_USER>\XERXES_electronics_GmbH\XC-8752A folder.

The default folder uses following sub-directory structure to save data:

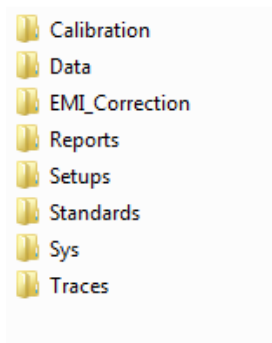


Figure 4.2 XC-8752A user directories.

It is recommended to keep this folder structure, since the different system functions use this default names. Once a new home path has been chosen, the setup is changed in the:

<Program Data>\XERXES_electronics_GmbH\XC-8752A

subfolder.

4.5 System / System Test



Figure 4.3 XC-8752A system test.

The system test will reveal any electronic malfunction and also report Software and Hardware Revision.

4.6 System / Unlock Calibration

System Update and Calibration is locked by a password. In order to enable these menus, it is necessary to unlock these functions first. After installation no password is set, so by unlocking the system functions it is sufficient just to press enter when the password request dialog appears. However it is highly recommended to define a password to protect the device from an unintended calibration attempt or software update.

4.7 System / Change password.

First enter the current password, then the new password.



Select a safe password to protect the XC-8752A!

It is further recommended to make the

<Program Data>\XERXES_electronics_GmbH\XC-8752A\app_config.cfg

file write protected and ownership granted only to the system administrator!

4.8 System / Service

This is a password protected menu item. Following dialog will appear:

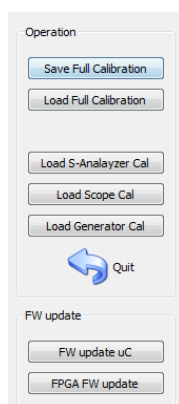


Figure 4.4 System Service dialog.

It is possible to save or load the full units calibration data. it is recommended to store a backup of the calibration data in a safe place. Further it is possible to restore only partial calibration data from the backup. With the FW update it is possible to update the XC-8752A microcontroller and FPGA Flash.



Do Not unplug the XC-8752A during a Firmware update or Calibration load process! After FW update, wait another 10 seconds after a power cycle!

4.9 Measurement / Edit Memory Slots

The memory slots are used to store S-Parameter as a reference used in Trace measurement functions. S-Parameter files can be either loaded from disk or copied for the current measured S-parameters.

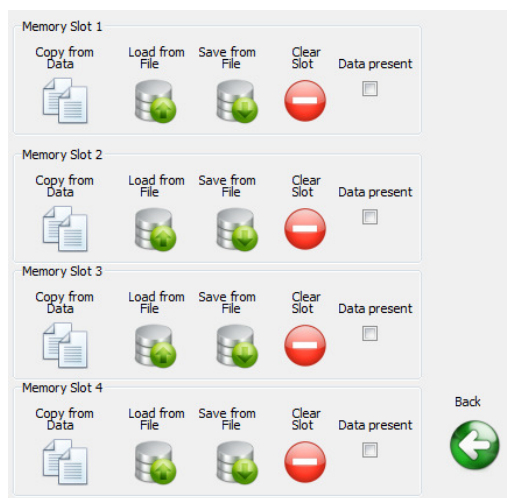


Figure 4.5 Memory slot dialog.

As can be seen in Figure 4.5, every memory slot can be edited and slot data can be even saved to disk. (Touchstone format). The "Clear Slot" button will delete all slot data.

4.10 Measurement / De-Embedding (full)

This command allows the definition of a electrical network for each port. During measurement this "thought" network will be extracted from the measurement. It is sufficient to have the S-parameter description of the parasitic network in Touchstone-Format to perform this task.

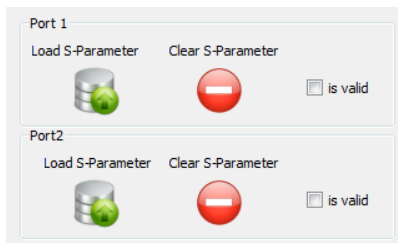


Figure 4.6 De-Embedding dialog.

Unlike the interactive simple de-embedding possibilities from the main dialog:

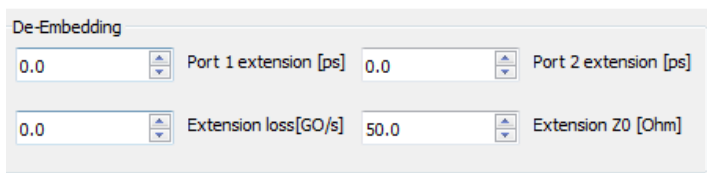


Figure 4.7 Simple De-Embedding.

which can only simulate a lossy transmission line, the full de-embedding allows the removal of any complex two port network.

The simple de-embedding is very useful, when coaxial lines are used to connect the DUT with the analyzer and the reference planes are at the connection of these coaxial lines and the plane of calibration. As the coaxial lines will introduce extra phase shift and loss to the calibration it is useful to remove these lines from measurement. This can be done during measurement. Short circuit the end of these lines and adjust the parameters in figure 4.7 until you can read a clear short circuit in the smith-diagram of S11 and S22. So you can shift the reference plane with little errors to the end of the test lines.

4.11 Measurement / Show Limits

	Diagram Name	Trace Name	f_min	f_max	a_min	a_max	Condition	PASS/FAIL
1	Diagram_1	dB(S11)	1e+07	1e+08	-5		>= Minimum	PASS
2								
3								
4								
5								
6								
7								
8								
9								
10								
11								

Result

PASS

Refresh Quit

Figure 4.8 Limits Test Window.

The limits test window will show all the limits defined in the trace dialogs, and perform the defined tests on the current measurements. A Pass/Fail evaluation for every test is shown in each line. The system will then perform an evaluation of each test to generate the total output result. This feature is helpful in production, where a set of complex measurement conditions must be evaluated.

4.12 Limits definition in the trace edit dialog

Source

☒ S-Parameter

☐ Memory 1

☐ Memory 2

☐ Memory 3

☐ Memory 4

Aperture[%]

1.25e-05

Trace Function

<S11> S-Parameter

<S12> S-Parameter

<S21> S-Parameter

<S22> S-Parameter

<Zin> Port 1 impedance

<Zout> Port 2 impedance

<VSWR_in> VSWR on port 1

<VSWR_out> VSWR on port 2

<Gvf> matched voltage gain forward

<Gvr> matched voltage gain reverse

Display Function

☐ Magnitude

☐ Phase

☒ Magnitude in dB

☐ Real

☐ Imaginary

☐ Complex

☐ Delay (-dphi/dw)

Normalize by

☒ None

☐ S-Parameter

☐ Memory 1

☐ Memory 2

☐ Memory 3

☐ Memory 4

Axis

☒ Left

☐ Right

50 Zo

Optional Value

50

Add/Modify Cancel

Ok

Equation

☐ Equation

Edit Limits

Figure 4.9 Trace Edit Dialog

Once a trace has been defined, several test limits can be defined. This can be done by opening the trace edit dialog and pressing the "Edit Limits" dialog.



Edit limits is only available on already existing trace-functions. If the trace add function has been executed, it is necessary to leave the dialog then edit (modify) the trace function.

Frequency Range
f_min[Hz]
f_max[Hz]

Level
Minimum Level
Maximum Level

Condition
GREATER THAN

Add Delete

☐ Display Limits in Diagram

Quit

	f_min	f_max	a_min	a_max	Condition
1	1e+07	1e+08	-5	0	Greater than
2					
3					
4					
5					
6					
7					
8					
9					
10					

Figure 4.10 Trace Limits Dialog

In the trace limits dialog it is possible to create up to 20 limit definition. Each limit definition consists of a valid frequency range, a limit condition and min/max values.

Following condition can be defined :

- Greater than minimum value
- Less that maximum value
- Within min/max value
- Outside the min/max value band

The "Display Limits in diagram" check box will allow to display the limits in a diagram.



Limits can only be displayed in rectangular diagrams.

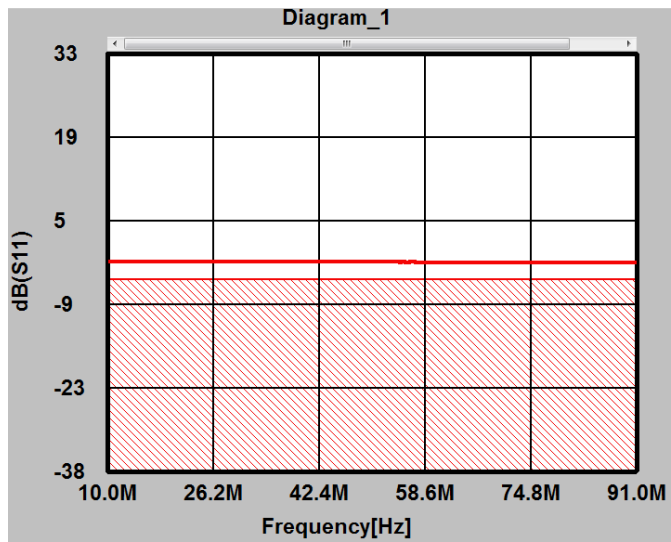


Figure 4.11 Lower limit displayed in diagram

5 Utilities

5.1 Spectrum Analyzer Utility

The spectrum analyzer utility contains several useful sub function which can perform very helpful tasks to support the electronic design process:

- Genuine Spectrum Analyzer
- Phase Noise Analyzer
- Logarithmic Spectrum Analyzer
- EMI Analyzer

5.1.1 Spectrum Analyzer

The Spectrum analyzer will automatically open a single trace window displayed like:

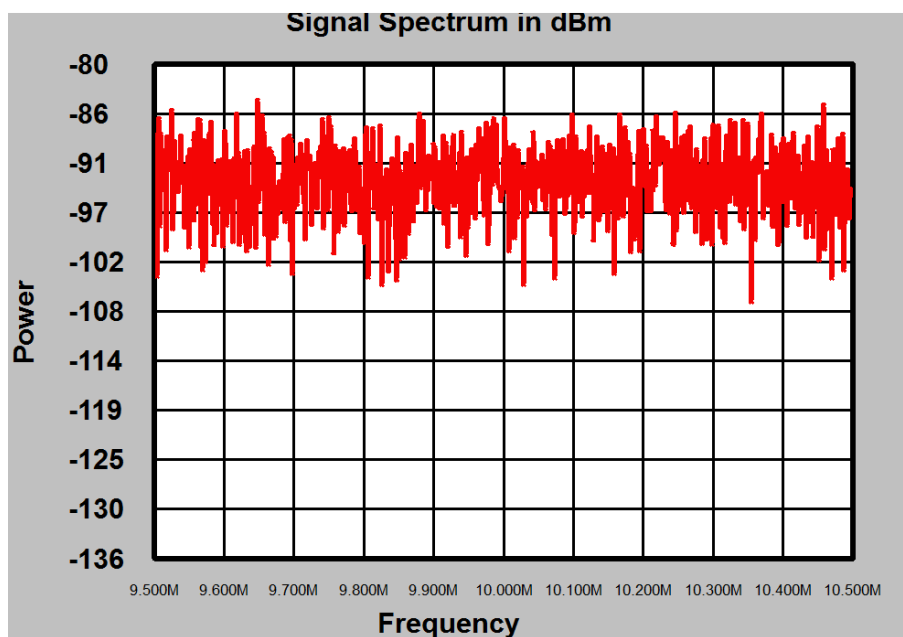


Figure 5.1 Spectrum Analyzer data window.

Please note that auto-scaling will be enabled.

Next to the spectrum analyzer window the spectrum analyzer control window will appear.

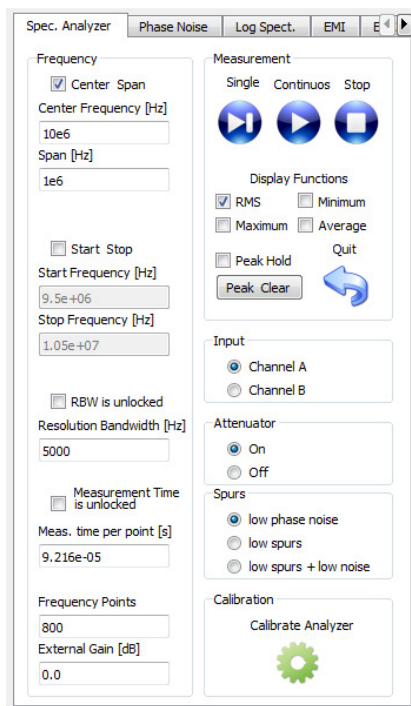


Figure 5.2 Spectrum Analyzer control window.

The spectrum analyzer shows two frequency range control options:

- Center and Span
- Start- and Stop Frequency.

Once the required frequency range has been selected, resolution bandwidth "measurement time per point" and Frequency points are set automatically. It is possible to unlock the resolution bandwidth and "measurement time per point". Measurement time per point corresponds to the video bandwidth often used in analog spectrum analyzer, it will average several magnitude values. Further it is possible to define an external gain in case an external amplifier is used to interface the XC-8752A. This will allow to extend the dynamic range and perform ultra low noise or differential measurements. The resolution bandwidth can be set in a range of 10 Hz - 200 kHz, with no granularity.

Several measurement functions are calculated simultaneously and can be displayed in the single spectrum analyzer diagram window:

- RMS = root mean square or "real" power
- Maximum value (maximum peak)
- Minimum value (minimum peak)
- Average.

A peak hold function is also available. Check the "Peak Hold" Checkbox to start to memorize peak values on each sweep. The "Peak Clear" button can be used to clear the current maximum values, and so reset the function.

It is possible to use either channel A or B as input channel and so to easily switch between two measurement setups.

The built in 20 dB attenuator is used to switch between a maximum input level of 13 dBm or -7 dBm.



Do not exceed the +15 dBm input level limit on either input! Excessive input power can damage the instrument.

5.1.1.1 Spur reduction and signal ambiguity.

Since the analyzer is using an about 125 MHz ADC clocking system, the unambiguous frequency range is to about 60 MHz. This mode is selected by the "low phase noise" mode. However, by using several slightly different sampling frequencies, the unambiguous frequency range can be extended to 1.3 GHz for (multiple)single narrowband carrier signals. This mode takes about twice the measurement time of the "low phase noise" mode. When measuring broadband signals like OFDM or CDMA modulated carriers exceeding 10 MHz or more, the spectrum might show spurious signals. This mode is called "low spurs" mode. In low spurs mode the carrier SSB noise is worse than in low phase noise mode. However the combined "low spurs + low noise" mode will compensate for this, by taking a longer measurement time (about 3 times low noise mode).



In low spurs mode, out of band spurious signals can be seen when measuring broad band modulated signals.

The spectrum analyzer can be calibrated by using a calibrated signal source. The Calibration button is password protected.



The built in signal generator can be activated by the "Signal Generator" button under the "Phase Noise" tab. It can be used during spectrum analyzer measurements. However values cannot be altered during measurements.

5.1.2 Phase Noise Analyzer

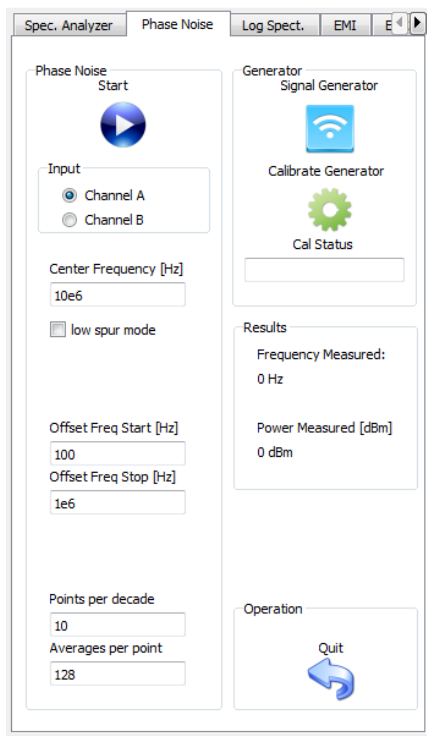


Figure 5.3 Phase Noise Analyzer control window.

The phase noise analyzer allows the measurement of the single sideband phase noise of an unmodulated signal carrier from 10 Hz to 1290 MHz. (SSB noise).

First the corresponding input channel has to be selected. Then choose the carrier frequency (within +/- 50 kHz). Choose low spur mode when measuring at frequencies > 60 MHz.

The next input values are the offset frequency range from carrier. This can be in the range of 100 Hz - 10 MHz). Select the number of measurement point per offset frequency decade. The number of averages are used to smooth the noise signal. Increase the values to get a smoother result.

Once an carrier has been fed to one of the input ports the phase noise measurement can start.

The first task of the instrument is to measure the accurate level and frequency of the carrier. These values will be shown in the "Results" box of the window. If no carrier is found, an error message will appear.

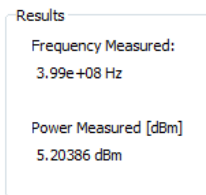


Figure 5.4 Result of the measurement of the internal signal generator, using 399 MHz.

If a carrier has been found the phase noise diagram will appear in the left upper corner of the screen.

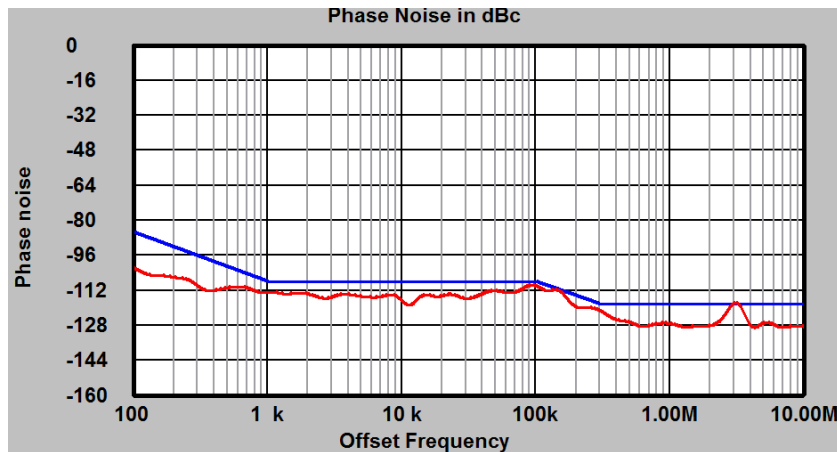


Figure 5.6. 399 MHz internal carrier signal measurement.

The blue limit line will show the lower measurement limit. Whenever the measured signal comes within 3 dB range of the lower limit. The measured phase noise could be lower than the system's limit, and can not accurately be measured. The result is presented in dBc (from measured carrier level) in a 1 Hz bandwidth.

5.1.3 Logarithmic spectrum analyzer

The logarithmic low frequency spectrum analyzer is useful for measuring broadband signals in the range of 10 Hz to 60 MHz. In this mode the resolution bandwidth is set as a fraction of the measurement frequency and frequency step distance to create a logarithmic equidistant frequency sweep. The signal power is measured in dBuV.

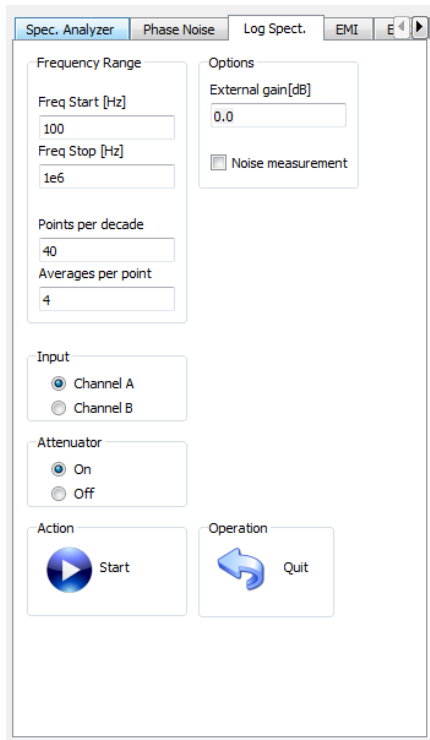


Figure 5.7 Logarithmic Spectrum Analyzer control window.

Besides the start- and stop-frequency it is possible to set the measurement points per decade and the number of averaged measurement per point. Increasing the averages will flatten the measurement with low level signals. Further the input port can be selected between Port A and Port B. The built in 20 dB attenuator is used to switch between a maximum input level of 13 dBm or -7 dBm.



Do not exceed the +15 dBm input level limit on either input! Excessive input power can damage the instrument.

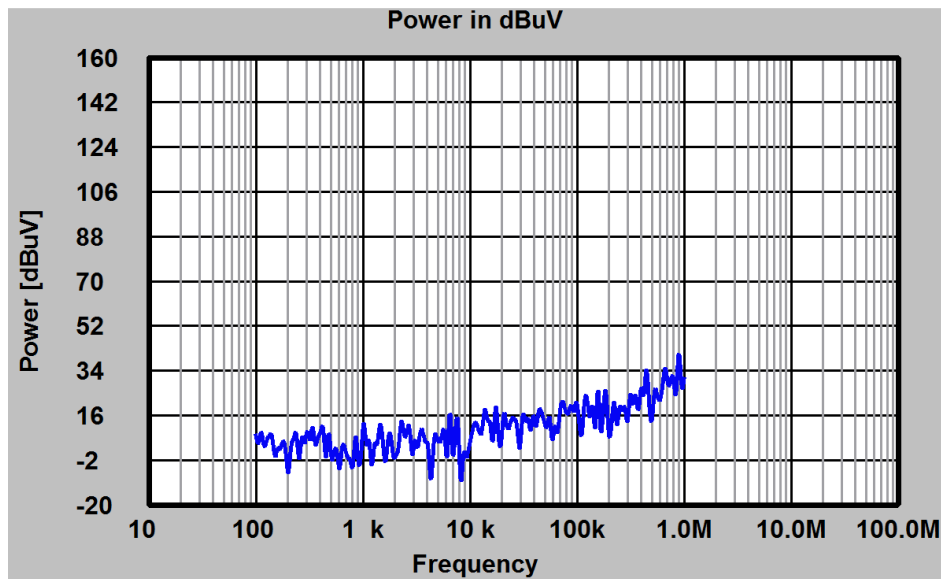


Figure 5.8 Logarithmic Spectrum Analyzer control window.

The external gain option allows the usage of an external amplifier or attenuator. A constant flat gain is assumed.

A further option is the noise analyzer, which can be activated by checking the "Noise measurement" check box. The output will show the equivalent noise voltage in $\text{nV}/\sqrt{\text{Hz}}$. The measurement bandwidth is 1 Hz. It is recommended to increase the number of averages per point to get a clear result. Figure 5.9 shows an open port noise measurement with 500 averages. The noise analysis will open a separate window.

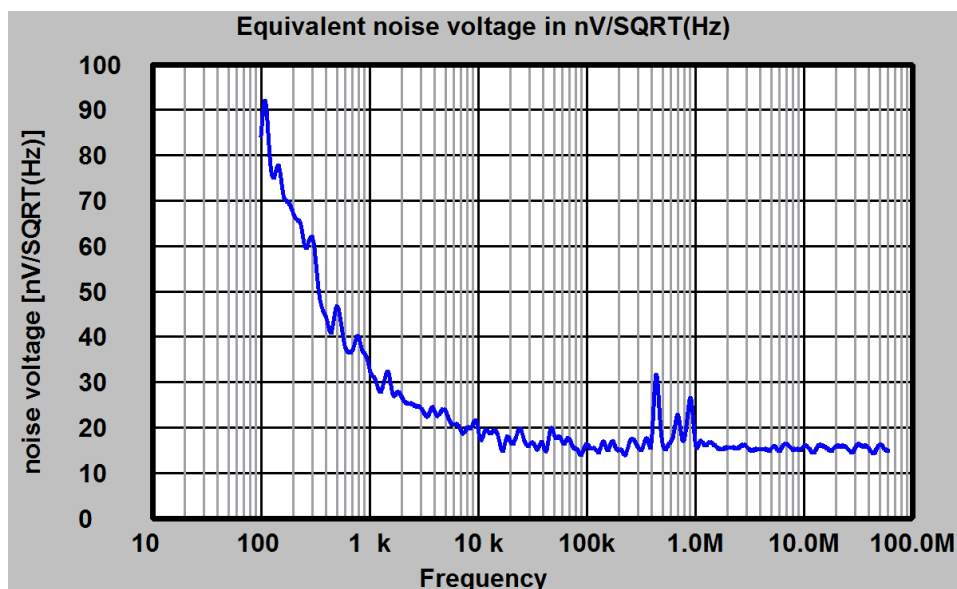


Figure 5.9 Noise level measurement

5.1.4 EMI Analyzer

The EMI analyzer implements a EMI receiver functionality according to CISPR 16-1 with following exceptions:

- The instrument has no frequency preselector.
- The measurement range from 60 MHz to 1 GHz may be suffering from spurious signals already discussed in the spectrum analyzer section.
- 1 GHz maximum frequency range.
- The built in quasi-peak and average detector have not been certified.

The EMI analyzer setup requires two tabs in the spectrum analyzer utility window. One for the regulation selection, the other for measurement setup.

The screenshot displays the EMI analyzer setup and control interface. It features several sections for configuration and operation:

- Frequency:** Start Frequency [Hz] (150e3) and Stop Frequency [Hz] (30e6).
- Mode:** Radio buttons for low phase noise, low spurs (selected), and low.s. low.n.
- Channel:** Radio buttons for Channel A (selected) and Channel B.
- Attenuator:** Radio buttons for on and off (selected).
- Samples per point:** A numeric input field set to 1000.
- Step size in BW/2 multiples:** A numeric input field set to 1.00.
- Current Status:** A text area for displaying the current status.
- Operation:** Start, Stop, and Quit buttons.
- Correction:** Load Correction File button, a text field for Filename, and a checkbox for Use Correction.
- Options:** Attenuation [dB] (0) and QP/AV meas. threshold [dB] (10).
- Export:** Export Report to .csv button.

Figure 5.10 EMI analyzer setup and control

The first window allows the selection of the required regulation. By selecting the regulation all limit values and frequency ranges are automatically setup. However it is possible to change the frequency range after selecting a regulation.

Currently following regulations are implemented:

Regulation	Specific Regulation	Specifics
EN 55022	EN 55022 A	conducted
EN 55022	EN 55022 B	conducted
EN 55022	EN 55022 A	radiated
EN 55022	EN 55022 B	radiated
EN 55022	EN 55022 A	conducted signal current
EN 55022	EN 55022 B	conducted signal current
EN 55022	EN 55022 A	conducted signal voltage
EN 55022	EN 55022 B	conducted signal voltage
EN 55011	EN 55011 A	conducted AC, group 1
EN 55011	EN 55011 B	conducted AC, group 1
EN 55011	EN 55011 A	conducted AC, group 2
EN 55011	EN 55011 B	conducted AC, group 2
EN 55011	EN 55011 A	conducted DC, group 1
EN 55011	EN 55011 B	conducted DC, group 1
EN 55015	EN 55015	conducted mains
EN 55015	EN 55015	conducted load
EN 55015	EN 55015	conducted control
EN 55015	EN 55015	radiated magnetic field
EN 55015	EN 55015	radiated electric field
EN 55032	EN 55032 A	conducted AC
EN 55032	EN 55032 B	conducted AC
EN 55032	EN 55032 A	conducted wired
EN 55032	EN 55032 B	conducted wired
EN 55032	EN 55032 A	conducted radiated
EN 55032	EN 55032 B	conducted radiated
EN 55014	EN 55014	conducted
EN 55014	EN 55014	disturbance
EN 55014	EN 55014	radiated

Table 5.1: Regulations covered by the instrument.

Further options are:

Attenuation:

This option allows the usage of an external attenuator or amplifier.

QP/AV measuring threshold:

In the first sweep the instrument will perform a simple peak and average measurement frequency sweep. After this sweep the instrument will check if any measured value comes close to the allowed limit. This internal limit value is calculated by regulation limit minus the "QP/AV meas. threshold" level field. So a value of 10 dB means that if the peak or average value comes within 10 dB of the regulation limit, the system will perform a CISPR 16-1-1 compliant Quasi-Peak and Average measurement. This procedure will save a lot of measurement time, since the QP/AV measurement takes considerably more time than a peak measurement.

Quasi Peak and Average Detector.

The built in quasi-peak and average detector have been implemented according to CISPR 16-1-1 regulation. Following bands have been implemented:

Frequency Band	6 dB bandwidth	QP charge time	QP discharge time	Instrument tau
CISPR A (9 kHz - 150 kHz)	200 Hz	45 ms	500 ms	160 ms
CISPR B (150 kHz - 30 MHz)	9 kHz	1 ms	160 ms	160 ms
CISPR C (30 MHz - 300 MHz)	120 kHz	1 ms	550 ms	100 ms
CISPR D (300 MHz - 1 GHz)	120 kHz	1 ms	550 ms	100 ms

Table 5.2: CISPR-16-1-1 quasi-peak detector specification as implemented.

The right side in figure 5.10 shows further setup possibilities:

It is possible to use either input ports for signal measurement, however it is important not to exceed the maximum input power:



Do not exceed the +15 dBm input level limit on either input! Excessive input power can damage the instrument.

The built in 20 dB attenuator is used to switch between a maximum input level of 13 dBm or -7 dBm. Please note that exceeding the above values will saturate the input amplifier and will lead to erroneous measurements.

The "Samples per point" spin-box allows to change the measurement time per frequency point. Increase the number of samples if it is necessary to catch relatively rare signal spikes.

The "Step size in BW/2 multiples" will allow denser or wider frequency step sizes.

The field "current status" will deliver status information on the measurement process.

For radiated measurements it will become necessary to use correction information to transform the measured signal power to a calibrated field strength. There for it is possible to load a correction file in textual form with following syntax:

<frequency value> <n x SPACE> <correction value in dB/m>

for example this could look like:

1000000.0 -15.2

2000000.0 -18.3

3000000.0 -22.0

The first column is the frequency row in Hz (floating point). The second column separated by any number of white-spaces is the floating point correction value in dB.

The correction value will be **added** to the measured voltage level in dBuV.

For example for the EN 55022 B radiated measurement the output level is in dBuV/m. So it is necessary to have a correction value which will transform the **dBuV to dBuV/m**.

For example for the EN 55015 radiated magnetic field the correction value shall transform the dBuV to dBuA/m.

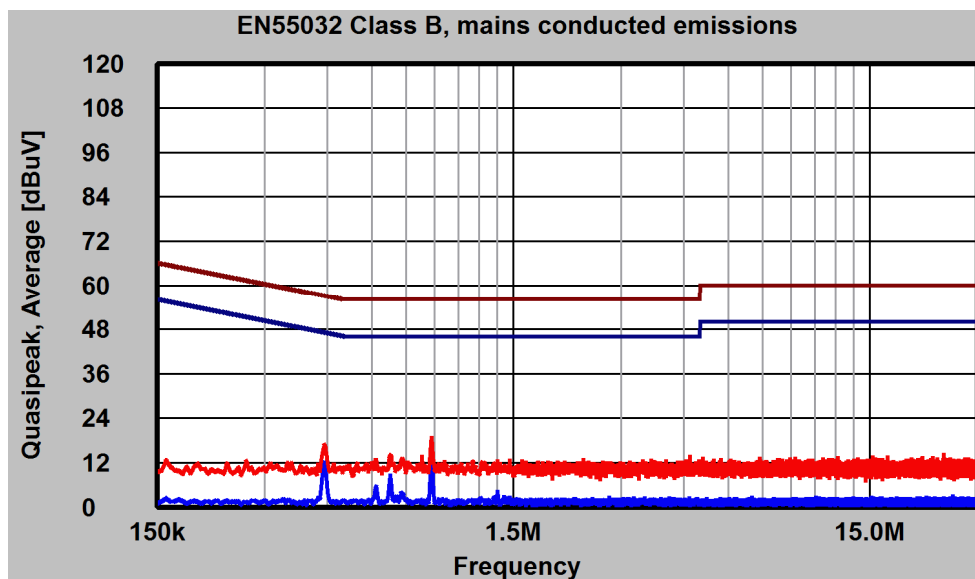


Figure 5.11: EN 55032 measurement with inputs open.

Once a complete measurement has been performed it is possible to export a textual report in .csv format. This report look like:

! XC-8752A EMI Report

EN55032 Class B, mains conducted emissions

! =====

Frequency[Hz]	QP_limit[dBx]	QP Measurement[dBx]	QP Margin[dB]	AV_limit[dBx]	AV Measurement[dBx]	AV Margin[dB]
---------------	---------------	---------------------	---------------	---------------	---------------------	---------------

If there is any or any close violation of a regulation limit, it will be reported in the spreadsheet file.

5.2 Oscilloscope utility

The oscilloscope implements a two channel 125 MSMP/s digital storage oscilloscope with many features.

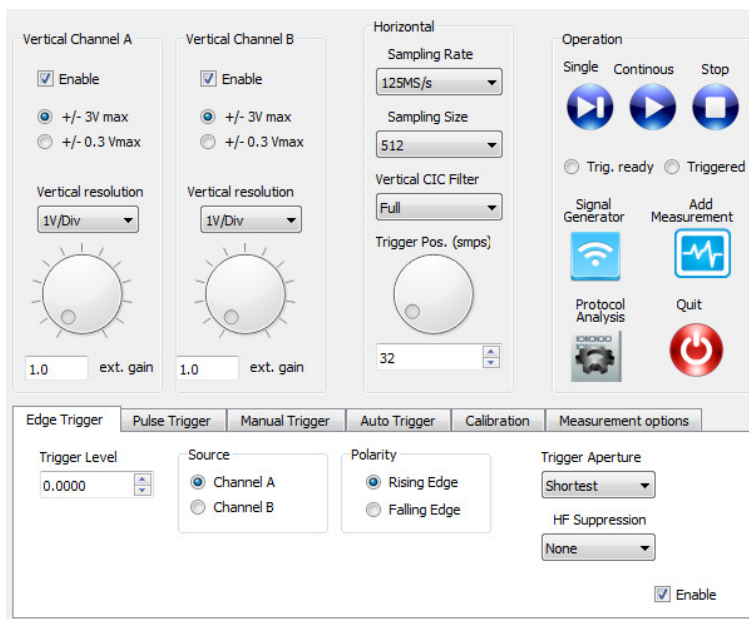


Figure 5.12: oscilloscope control window.

Both channels can be individually enabled or disabled by checking the "Enable" checkbox.

The input range may be selected to be $\pm 3.0\text{V}$ or $\pm 0.3\text{V}$ the latter one with better noise performance. The list box "Vertical resolution" will change the diagrams vertical settings of V/Divisions. The range is 100 uV / Div to 1 V / Div .



Do not exceed the $\pm 3\text{V}$ input level limit on either input! Excessive input power can damage the instrument.

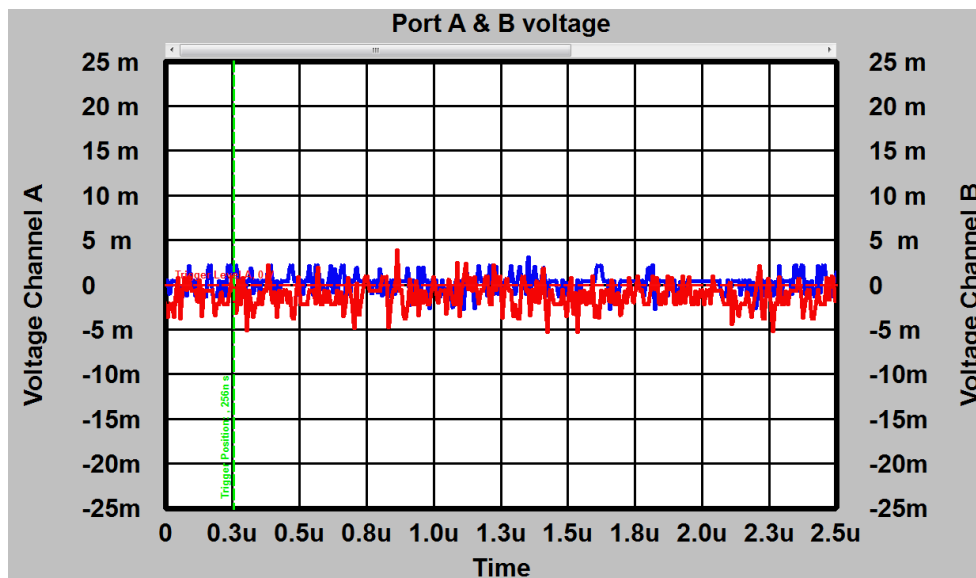


Figure 5.13: oscilloscope data window.

Figure 5.13 shows the oscilloscope output in 5mV/DIV resolution and $\pm 3.0\text{V}$ range. Please note that the blue trace corresponds to the right Y axis and the red line to the left axis. Both axis may have different resolutions. The resolution may also be changed by the wheel inputs for better comfort. Each channel also implements an external gain selector to adjust for an optional external amplifier or impedance converter.

The horizontal control allows to change the Sampling rate from 15 S/s to 4 GS/s . However sampling rates of greater than 125 MS/s are $\sin(x)/x$ interpolated. This interpolation allows correct signal interpretation up to 60 MHz .

The number of sample can also be adjusted from 2 to 2048. Further a vertical CIC (cascaded intergrated comb filter) allows for low noise at low sampling rates.

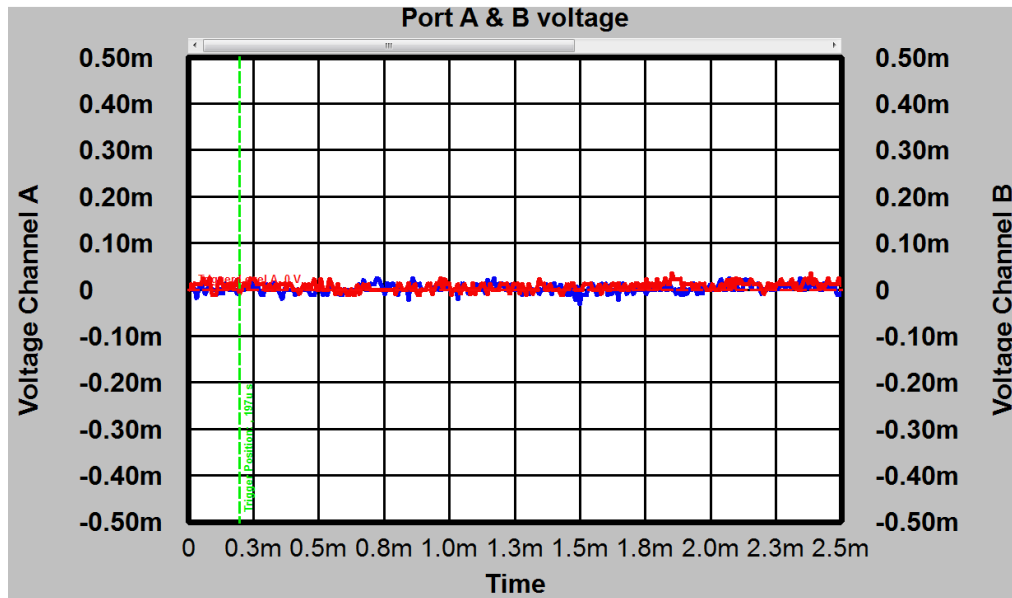


Figure 5.14: oscilloscope data window with 122 kS/s and 54 kHz bandwidth.

In figure 5.14 shows the noise voltage at 122 kS/s sampling rate and 54 kHz bandwidth. It is less than $\pm 25\mu\text{V}$ peak to peak.

The next wheel called "Trigger Position" can be used to adjust the trigger point in time. The trigger point is shown in the oscilloscope diagram as vertical green line and can be adjusted from 0 to 512 samples. This feature is used to view events before the trigger event.

5.2.1 Oscilloscope Trigger

5.2.1.1 Edge trigger

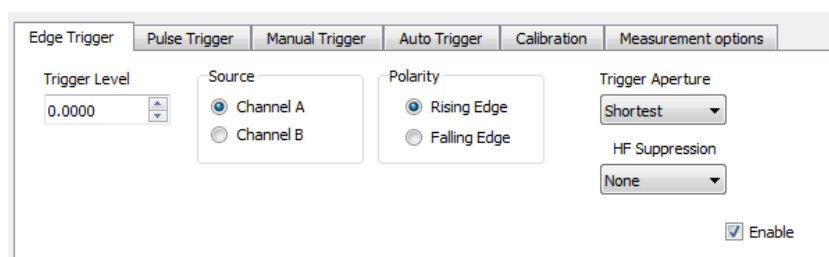


Figure 5.15: Edge trigger tab

To enable the edge trigger it is necessary to check the "Enable" check box. The edge trigger will trigger oscilloscope sampling if the input signal on the selected channel will raise above or fall under

a selectable trigger level. The trigger level can be adjusted in the spin box "Trigger level" or by dragging the red horizontal trigger line in the diagram.

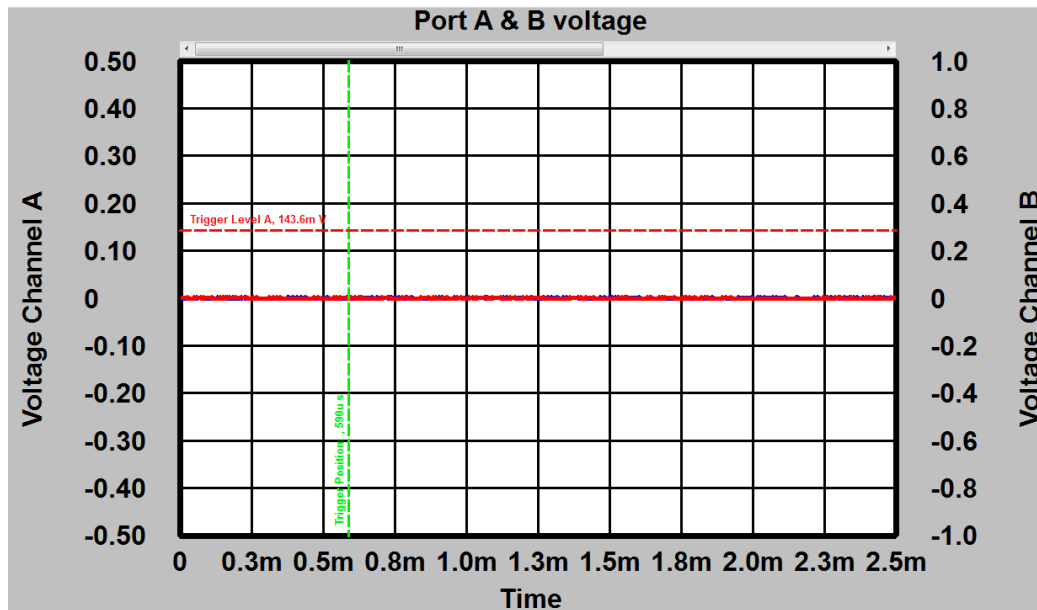


Figure 5.16: oscilloscope data window trigger level of 143.6 mV

The source channel of the edge trigger can be selected in the "Source" group box. The polarity of the edge trigger in the "Polarity" group box.

There are two other list-boxes named as "Trigger Aperture" and "HF suppression".

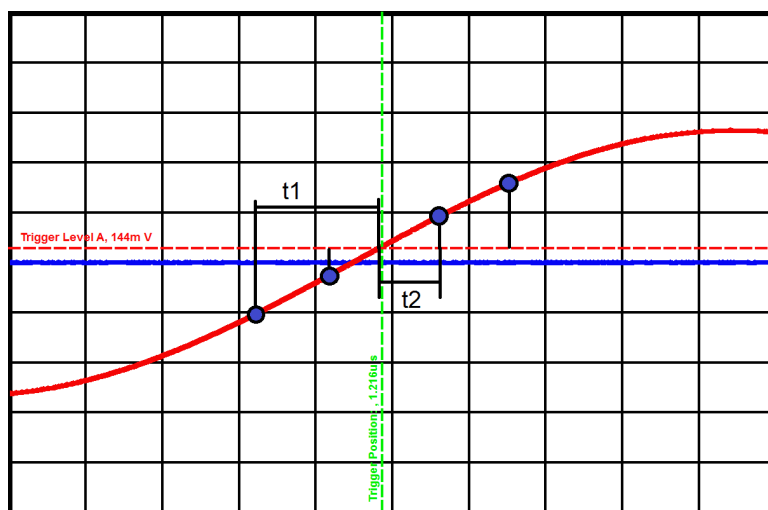


Figure 5.17: Trigger aperture time.

The trigger aperture defines the time window in which the edge detection takes place. In figure 5.17 two different aperture windows are shown. The short window $2 \times t_2$ and the longer window $2 \times t_1$. Assume that the signal in Figure 5.17 is distorted with high frequency noise, and the edge trigger is set to falling edge detection. There is a high possibility that the trigger event occurs within the time window $2 \times t_2$ since the signal is close to the trigger level it might cross the trigger line multiple times within the trigger time window. This is shown in figure 5.18.

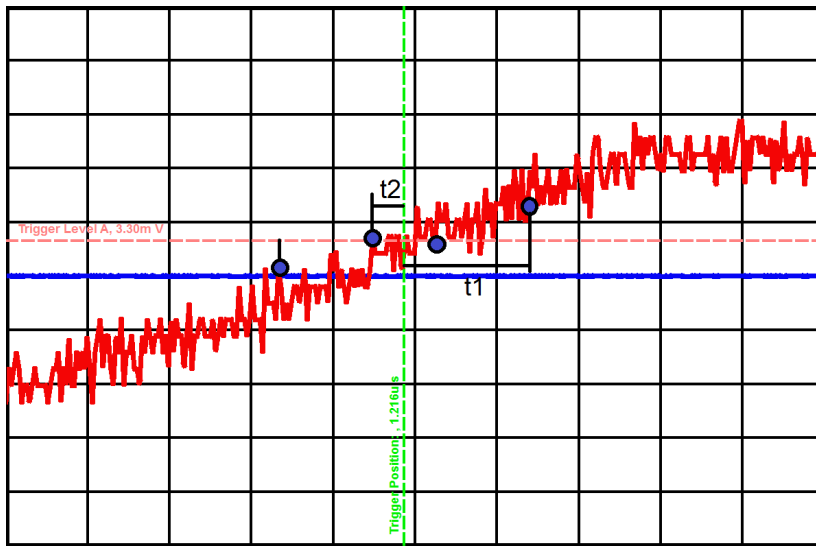


Figure 5.18: Trigger aperture time in noise environment.

Although the low frequency signal is raising, the falling edge has been triggered when using the $2 \times t_2$ trigger aperture time. If a trigger aperture time of t_1 would have been use, the falling edge trigger would not have been falsely set. However, the broadening of the trigger aperture window causes increased jitter, so additionally the list box "HF suppression" has been added. This HF suppression mode will cause an logical "AND" on multiple aperture window. If it is assumed that the falling edge event must be true for both trigger aperture windows (t_1 & t_2) the trigger accuracy will remain. One drawback of this method is that the trigger could miss some trigger event due to HF noise.



Please note that Trigger Aperture and HF Suppression settings are also valid for the pulse trigger modes.

5.2.1.2 Pulse Trigger

Edge Trigger	Pulse Trigger	Manual Trigger	Auto Trigger	Calibration	Measurement options
Trigger Level A	0.0000 [V]		Trigger Level B	0.0000 [V]	
minimum length	0.000 [us]		maximum length	0.000 [us]	
First source	First polarity		Second Source	Second polarity	
<input checked="" type="radio"/> Channel A	<input checked="" type="radio"/> Rising Edge		<input checked="" type="radio"/> Channel A	<input checked="" type="radio"/> Rising Edge	
<input type="radio"/> Channel B	<input type="radio"/> Falling Edge		<input type="radio"/> Channel B	<input type="radio"/> Falling Edge	
					<input type="checkbox"/> Enable

Figure 5.19: Pulse trigger tab

The pulse trigger modes allows triggering on certain pulse width events within or across channels. It can be enable by checking the "Enable" check-box

It is possible to adjust a minimum and maximum pulse width and the triggering will be valid between those timings. To accomplish this kind of triggering it is necessary to have two events. One start event, a valid time windows between the second event and of course the second event.

So it is possible to trigger on the rising edge of channel A as first event and on the falling edge of channel A as second event. Let's assume the minimum length is t_1 and the maximum length t_2 . The system will trigger on all positive pulses with a length between t_1 and t_2 on Channel A.

Otherwise it is possible to set the first source to Channel A the first polarity to rising edge and the second source to Channel B and the second polarity also to raising edge. Further let the minimum length be 0 us and the maximum length 1000000 us. Now the system will trigger on a rising edge on Channel A followed by a rising edge on Channel B within a time distance of maximum 1 second.

Channel A and B are equipped with separate trigger levels.

5.2.1.3 Manual Trigger

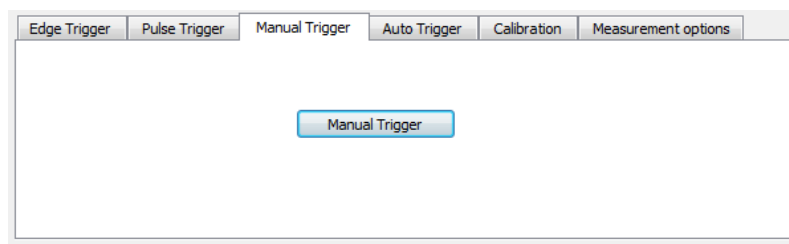


Figure 5.20: Manual trigger tab

The manual trigger can be used to trigger the oscilloscope sampling by pressing the "Manual Trigger" button. This is also useful if the trigger level has been changed while the oscilloscope is waiting for triggering. In order to have the trigger level changed, the oscilloscope needs to leave the "wait on event" state. This can be done by pressing the "Manual Trigger" button.

5.2.1.4 Auto Trigger

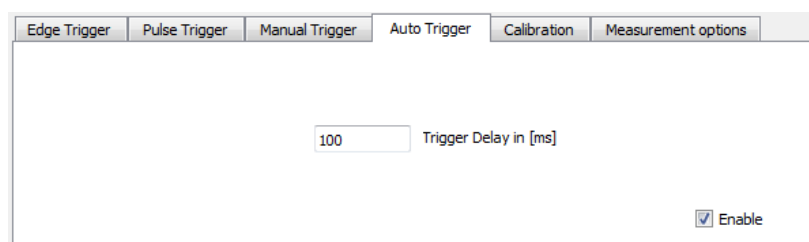


Figure 5.21: Auto trigger tab

The auto-trigger mode can be enabled by checking the "Enable" checkbox. The oscilloscope will then start a manual trigger if no trigger event occurs within the selectable time period.

5.2.1.5 Calibration

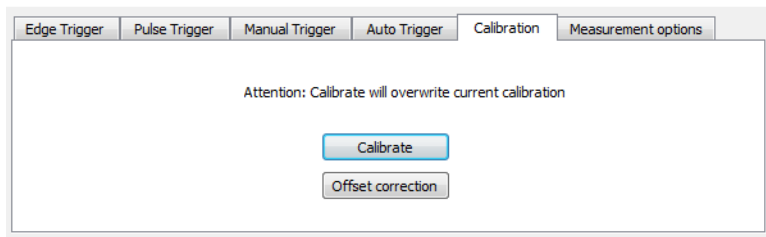


Figure 5.22: Calibration tab.

The Calibrate command is locked by a password. Once the calibration is unlocked. The calibration procedure will ask to apply different DC voltages to port A and B to calibrate the oscilloscope. This procedure needs a precision DC source. It is also possible to use the XERXES XC-7112A low voltage calibrator to perform an auto-calibration.

The button "Offset correction" will correct for any DC offsets in the amplifier chain. Leave both inputs open or terminate with 50 Ohms to perform the offset correction. This offset correction will be very useful in measuring low voltage signals in the μV range.

5.2.1.6 Measurement options.

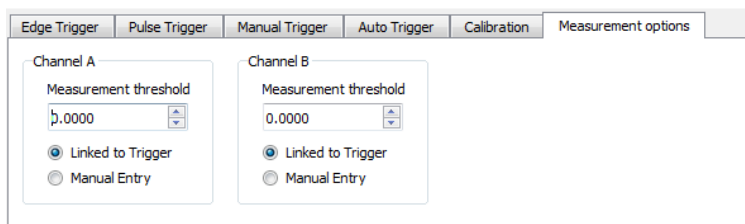


Figure 5.23: Measurement options tab.

The measurement options tab is used to select a reference voltage for the built in measurement functions. When measuring the frequency of a sinusoidal signal with an offset it is necessary to have an threshold for defining zero transition. With a symmetrical signal this might be zero volts, but with an applied DC-offset this threshold may change. These thresholds are just mathematical values for the measurement evaluation. Normally they are linked to the corresponding trigger level, but in some cases it might be helpful to set them by hand.

5.2.2 Measurements

Since measurements within oscilloscope diagrams a laborious job, the oscilloscope provides a set of built in measurement functions. These measurements can be added to a diagram by pressing the "Add Measurement" button.

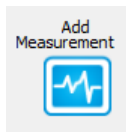


Figure 5.24: Add measurement button.

The measurement selection dialog will appear:

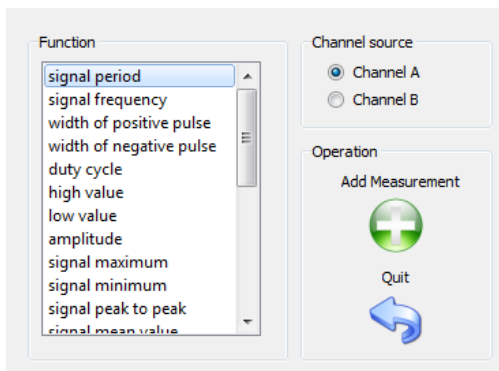


Figure 5.25: oscilloscope measurements.

Following measurements are included:

Function	Description
Signal period	Period time of a repetitive signal in seconds
Signal frequency	Frequency of a repetitive signal in Hertz
Width of positive pulse	Length of a positive pulse in seconds
Width of negative pulse	Length of a negative pulse in seconds
Duty cycle	Duty cycle in %
High value	Top value of a rectangular/digital signal without overshoot
Low value	Bottom value of a rectangular/digital signal without undershoot
Amplitude	Zero to average peak value
Signal maximum	Positive peak value
Signal minimum	Minimum peak value
Signal peak to peak	Peak to Peak value
Signal mean value	Mean value of captured data
Signal RMS value	Root mean square value of captured data, effective value
Signal standard deviation	Standard deviation from mean value
Auto trigger amplitude	Calculates the peak to peak amplitude with auto threshold level selection.
Rise time	Signal 10% - 90% rise time in seconds
Fall time	Signal 10% - 90% fall time in seconds
Overshoot positive	Amount of positive overshoot in rectangular/digital signal
Overshoot negative	Amount of negative overshoot in rectangular/digital signal
Positive delay A to B	Rising edge delay from channel A to B in seconds
Negative delay A to B	Falling edge delay from channel A to B in seconds
Phase A to B	Phase lag between channel A to channel B in degrees
Jitter rms	RMS Jitter in seconds
Jitter peak	Peak jitter in seconds

Table 5.3: Oscilloscope measurement functions.

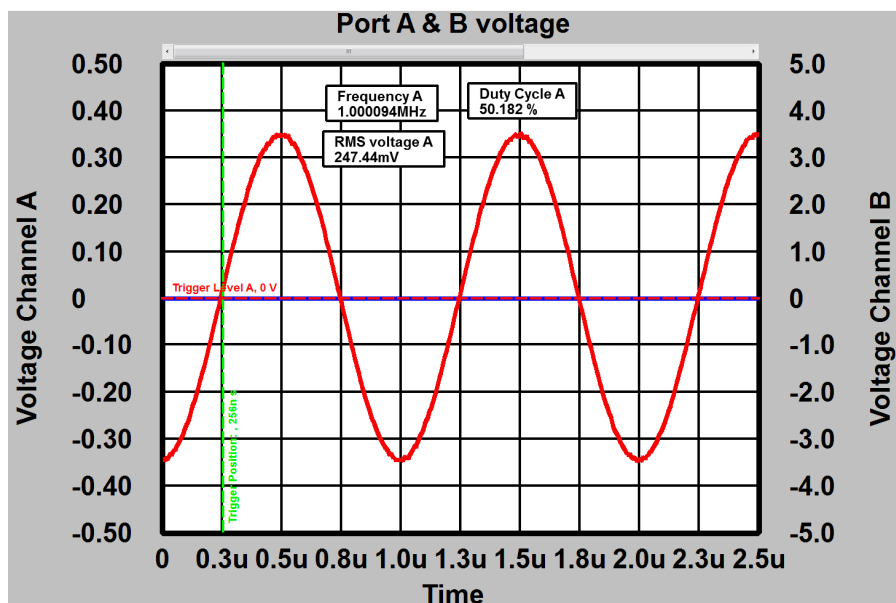


Figure 5.26: oscilloscope shot with multiple measurements

Figure 5.26 shows multiple measurements defined in an oscilloscope diagram. They will update as the oscilloscope is retriggered. It is possible to delete a measurement like a marker, by a right click and selecting Delete from the context menu or by selecting the measurement and pressing the delete key.

5.2.3 Protocol analyzer

The oscilloscope implements a simple protocol analyzer for frequently used digital communication protocols.

- I2C protocol.
- RS232 protocol.
- SPI protocol.

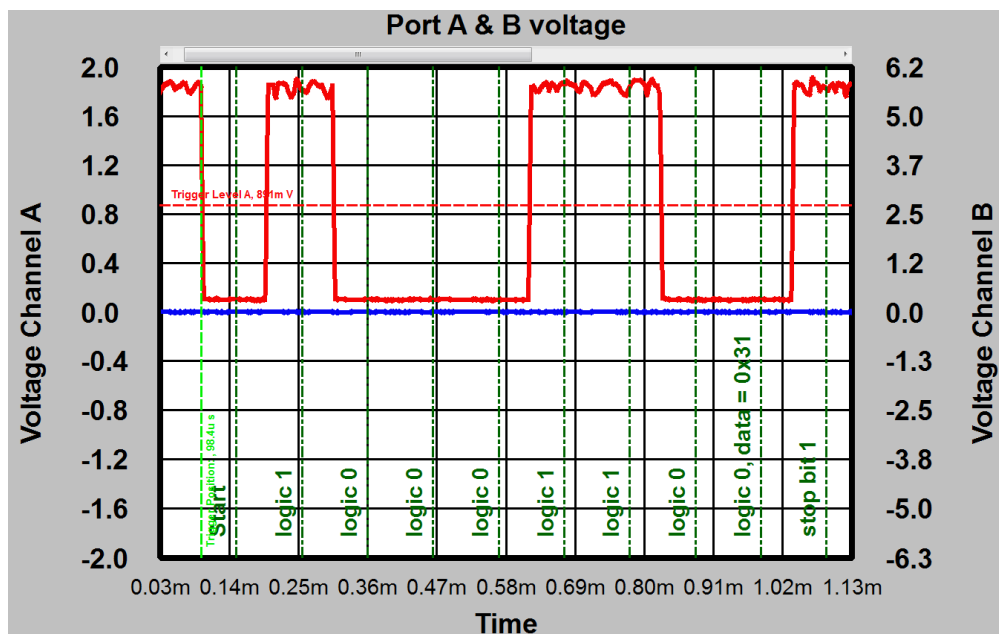


Figure 5.27: oscilloscope shot with RS232 protocol analysis.

The protocol analyzer can be applied to a scope shot after pressing the "Protocol Analysis" button:

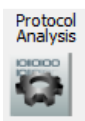


Figure 5.27b: Protocol analyzer button.

5.2.3.1 RS232 analysis

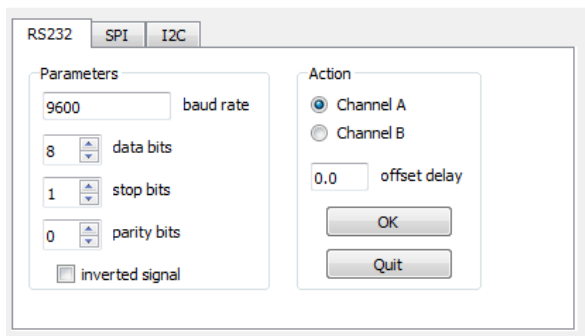


Figure 5.28: RS232 protocol analysis tab.

The protocol analyzer supports 1- 64 bits data bits, 1-2 stop bits and 0-1 parity bits. The signal polarity can be inverted and an offset delay allows protocol analysis after a certain dead time.

As seen in figure 5.27 each bit is detected and evaluated, after all data bits are detected, the output data is also shown in the diagram.

Press "OK" to start analysis.

5.2.3.2 SPI analysis

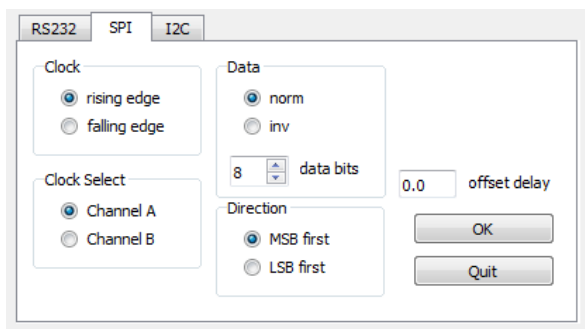


Figure 5.29: SPI protocol analysis tab.

The implemented SPI analysis is using only two wires. The "clock select" group-box allows to decided which channel is used as clock source. Further it is possible to select whether the rising- or falling-edge is used for data sampling. The data may be processed with normal or inverted polarity. The data bits may be in the range between 1 and 64 bits. The "Direction" group-box allows to select bit direction. MSB for most significant first, LSB for least significant bit first and an "offset delay" allows protocol analysis after a certain dead time

Press "OK" to start analysis.

5.2.3.3 I2C analysis

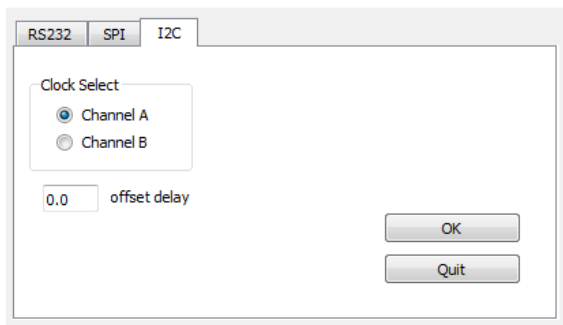


Figure 5.30: I2C protocol analysis tab.

The I2C protocol analysis tab allows selection of the clock input channel and an "offset delay" allows protocol analysis after a certain dead time.

Press "OK" to start analysis.

5.2.4 Signal generator

By pressing the "Signal Generator" button:



Figure 5.31: Signal generator button.

After pressing the button the signal generator window will appear. The signal generator can be used during oscilloscope measurements. The generator will be further discussed in a later chapter of this document.

5.3 Signal Generator Utility

The signal generator implements a synthesized sinusoidal signal source from 0.25 Hz to 1 GHz with 0.25 Hz resolution to 500 MHz and 0.5 Hz to 1 GHz. Figure 5.32 shows the signal generator's control panel:

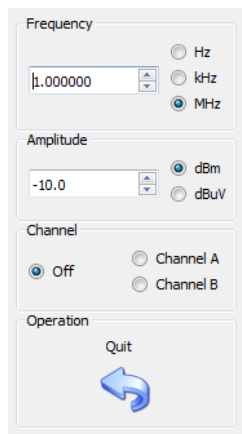


Figure 5.32: Signal generator control window.

The first spin box named "Frequency" is used to enter the desired signal frequency. The amplitude level can be entered in dBm or dBuV. The range is -7 dBm to -80 dBm or 100 dBuV to 17 dBuV respectively. The signal generator can be shut off by selecting the "Off" radio button, or switched on by selecting the Channel A or Channel B radio-button. An external amplifier can be used to increase the output level.

5.4 Meters Utility

5.4.1 Broadband Voltmeter

The broadband voltmeter is an instrument to measure the RMS signal power in a very broad frequency band. In the case of the XC-8752A the -3 dB bandwidth is about DC to 700 MHz.

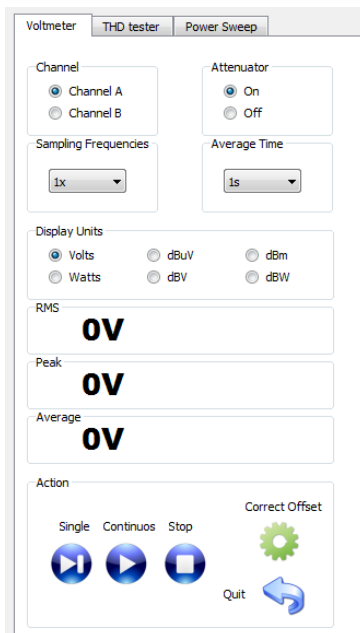


Figure 5.33: Broadband voltmeter control window.

The first selection is the corresponding input port. The input range may be selected to be $\pm 3.0V$ with attenuator and $\pm 0.3V$ without.



Do not exceed the $\pm 3.0 V$ input level limit on either input! Excessive input power can damage the instrument.

The number of different sampling frequencies will eliminate any possible blind frequency spots. Choose between 1x to 16x sampling frequencies. The update time will increase proportionally with the number of sampling frequencies.

The Average-Time can be adjusted between 1 ms to 4 s. The total voltmeter update time will result from sampling frequency times average time. A longer average time will give more accurate results and lower noise.

The measured values can be presented in following units:

- Volts
- Watts (into 50 Ohms)
- dBuV
- dBV
- dBm (into 50 Ohms)
- dBW (into 50 ohms)

The output will be shown as:

- RMS = root mean square, effective value
- Peak value
- Time average value

It is highly recommended to start a Offset correction before performing measurements. This can be

Correct Offset



done by clicking the "Correct Offset" button:

During offset correction it is recommended to terminate the desired input with a 50 Ohms load or to leave the inputs open.

It is possible to make single or continuous measurements. The stop button is used to stop a continuous measurement.

5.5 THD meter

The THD (Total Harmonic Distortion) meter is used to measure the harmonic power of a carrier signal or the distortion of a nonlinear two port in a frequency range of 40 Hz to 499 MHz.

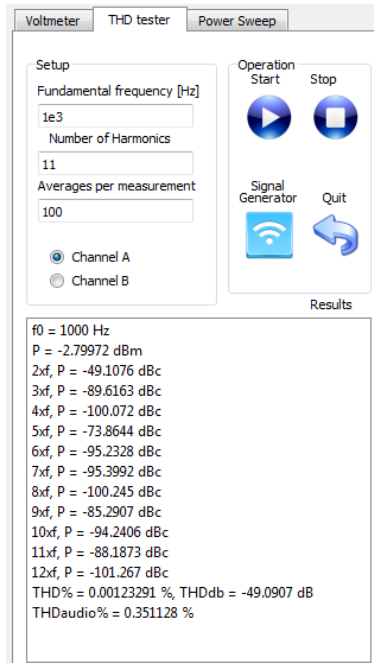


Figure 5.34: THD meter control window.

The basic parameter of a THD measurement is the fundamental frequency of the carrier of which the THD shall be measured. Next step is the definition of the number of harmonics used in the calculations. An increase of the averages per measurement will allow to measure with higher accuracy at lower harmonic levels. Further the input port can be chosen to be port A or port B. Be sure that the maximum harmonic frequency is lower than the 1 GHz maximum input bandwidth.

If the quality of the internal signal generator is not good enough, an external low harmonic source shall be used. Be sure that the sources frequency is stable within 2% of the carrier frequency during measurements.

The internal signal generator window can be opened by clicking the "Signal Generator" button:



Now the combination of internal signal generator and THD meter can be used to measure any two port nonlinear behavior.

After starting the measurement the instrument will determine the carrier power and frequency. The result will be shown in the "Result" test box. The measurement will also open a new diagram and present all harmonic and fundamental signal powers in dBm.

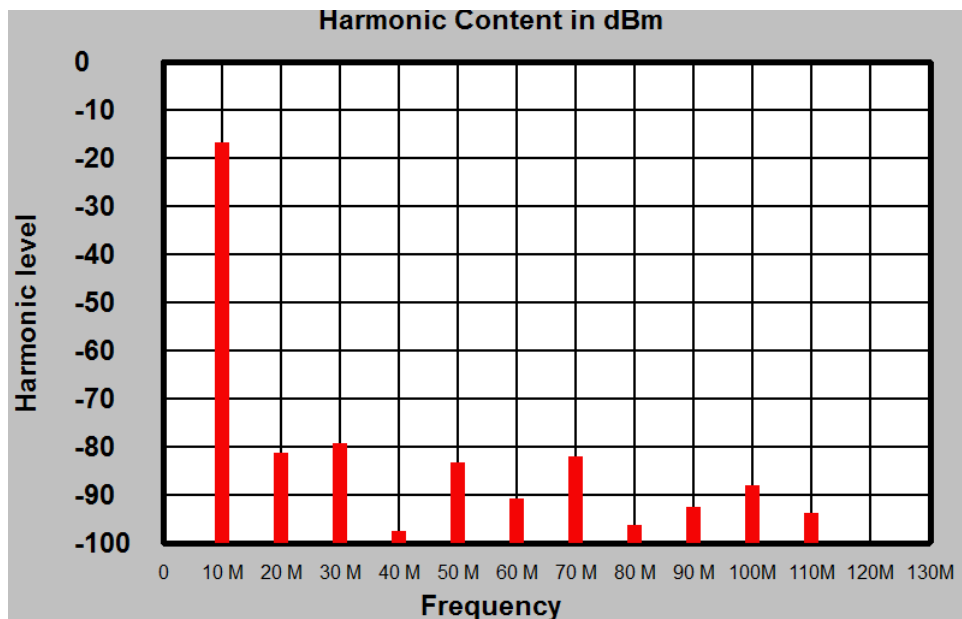


Figure 5.35: THD meter harmonics diagram.

Further the text output will show the in detail measured harmonic values in dBc and also calculate different THD metrics.

```
f0 = 1e+07 Hz
P = -18.4449 dBm
2xf, P = -64.5494 dBc
3xf, P = -63.2448 dBc
4xf, P = -80.0309 dBc
5xf, P = -66.9793 dBc
6xf, P = -73.375 dBc
7xf, P = -66.4954 dBc
8xf, P = -74.8193 dBc
9xf, P = -75.5169 dBc
10xf, P = -68.9661 dBc
11xf, P = -79.4007 dBc
12xf, P = -78.8912 dBc
THD% = 0.000151732 %, THDdb = -58.1892 dB
THDAudio% = 0.12318 %
```

Figure 5.36: Example THD measurement result.

It is recommended to perform a non distorted or through measurement to verify the systems performance and signal source quality before making measurements with a nonlinear two port. The signal generators harmonic performance worsens at frequencies below 100 kHz through the non linear behavior of the solid state switches used. Values of around -40 dBc for the first harmonic and -60 dBc for the second harmonic can be assumed.

The THD value in percent is calculated by the sum of the harmonic powers in relation to the fundamental power.

$$THD_{\%} = \frac{P_2 + P_3 + P_4 + P_5 + \dots}{P_1}$$

with P1 the fundamental power and P1 - Pn the harmonic power values. The THD value in dB is just:

$$THD_{dB} = 10 \cdot \log \left(\frac{P_2 + P_3 + P_4 + P_5 + \dots}{P_1} \right)$$

And for audio measurements the THDAudio is defined as:

$$THD_{audio} = \frac{\sqrt{V_2^2 + V_3^2 + V_4^2 + V_5^2 + \dots}}{V_1}$$

the effective harmonic voltage relative to the fundamental voltage.

5.6 Power sweep

The power sweep is a method to observe the linearity of a two port device at a certain frequency, but a swept input power. While the input power is raised from a minimum value to a maximum value the output is measured in a diagram.

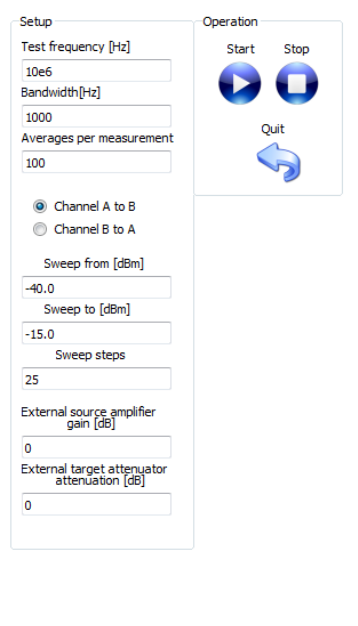


Figure 5.36: Power sweep window.

The test frequency can be entered in a range of 10 Hz - 1GHz. The resolution bandwidth will determine noise floor and measurement speed. A small bandwidth will allow accurate measurements at low input power. The averages per measurement will increase or decrease the statistical flatness of the measurement spot. Further the direction of measurement can be chosen. Whether Port A to B with Port A as the signal source or vice versa is selected.

The sweep range need to be entered, while the minimum power shall be greater or equal to -80 dBm the maximum output power needs to stay below -14 dBm to stay in the linear range of the signal generator. The number of steps will give the resolution of the power sweep. It is possible to define a an amplification or attenuation before and after the DUT (device under test).

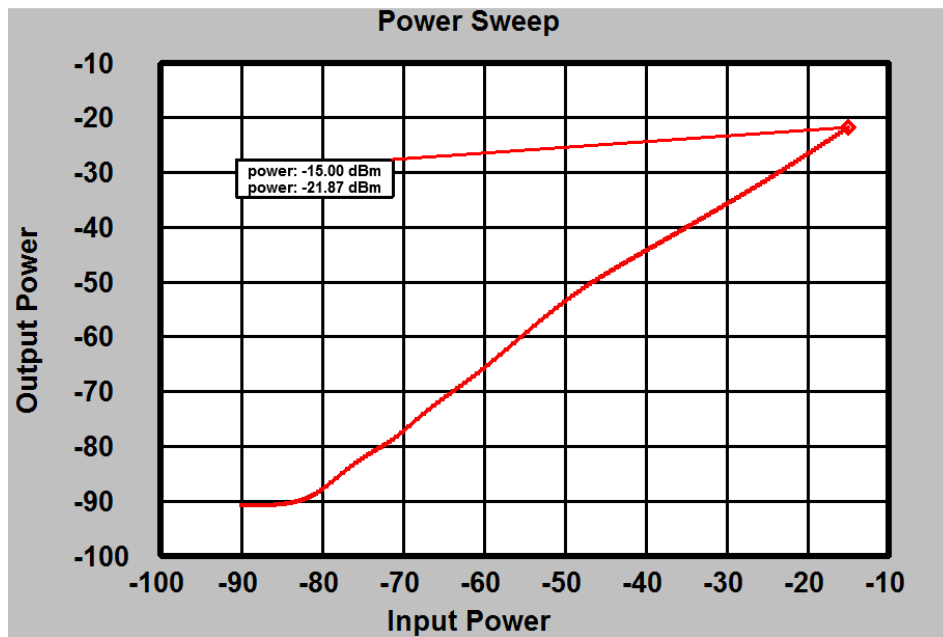


Figure 5.37: Power sweep data window.

The generated diagram will show input power over output power in dBm. In the case of Figure 5.37 a gain of -6.87 dB at -15 dBm input power can be assumed.

6 Recommendations & Supplemental Information.

6.1 Connector wear

to protect the instruments SMA connectors from excessive wear it is highly recommended to use female/male SMA adapters of high quality to prolong lifetime. Such connectors may be ordered with the product number: XC-CON-ADP-SMA-FM-001-002.



Figure 6.1: XC-CON-ADP-SMA-FM-001-002 set of adapters.



Do not use excessive torque on the analyzers connectors, the connector or the internal PCB may be damaged!

6.2 Differential & Common Mode Measurements.

Although the instrument has no designated differential input ports, the input ports can be used as differential inputs using the "LOAD" connection.



Figure 6.2: Differential configuration.

Remove the load-terminations to access the fully differential input ports,

The differential input impedance is 100 Ohms with 50 Ohms single ended. The common mode suppression is better 40 dB to 100 MHz (60 dB typ.) and better 20 dB beyond 1GHz (25 dB typ.).

In this configuration the function of the input attenuators changes:

6.2.1 Differential mode in oscilloscope mode:

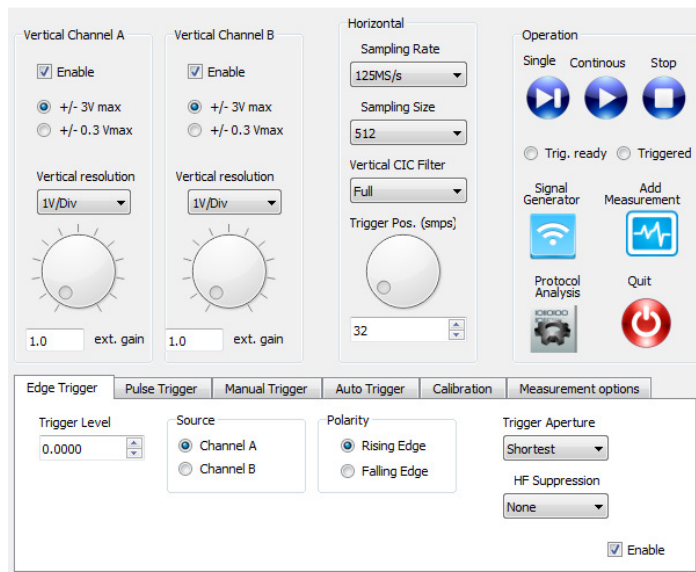


Figure 6.3: oscilloscope control window.

The range selector (+/- 3V, +/- 0.3V) function changes to:

- +/- 0.3V range: Differential input measurement 0.3 Vpp max.
Scaling is 1:1
- +/- 3V: Common mode input measurement: 1.5V Vpp max.
Scaling 1:2. This will not be compensated in measurements. The ext. gain field needs to be manually set to a value of 0.5.

6.2.2 Differential mode in all other measurements:

With attenuator: ON the measurement will be a common mode measurement with an internal gain of 2. This needs to be compensated by setting the external gain to 0.5 if possible.

With attenuator: OFF the measurement will be a differential measurement with an internal gain of 1. No compensation is needed.

6.3 Time domain measurements.

Time domain analysis is a useful tool for measuring impedance values along transmission lines or for evaluating a discontinuity in time and distance. The time domain display provides a much more understandable way of the DUT (device under test) behavior than a frequency domain plot does. Broadband performance can be observed by identifying each discontinuity by time or distance. Although the VNA provides a TDR display, there are differences between a traditional TDR meter using discrete pulses and the VNA frequency sweep method. The VNA time domain modes can be separated in two major methods:

- Low Pass mode
- Band Pass mode

Low Pass mode will use a frequency band starting with DC (or close DC) and a certain stop frequency. The VNA will then extrapolate the DC and assume that the negative frequency values are the conjugate complex of the positive frequency values. An inverse FFT will calculate the time domain behavior. Each frequency step from DC to stop frequency is equidistant.

In Band Pass mode the analyzer will use the Start and Stop frequency and a number of equidistant frequency points in between. Through the low noise receiver of the VNA the noise performance is superior to traditional TDR instruments. The band pass mode is very useful when measuring with in an IF range or filter range, where sweeping down to DC is impossible.

The XC-8752A provides four times four different time domain functions:

TDR_Sxx_LP_RW : Sxx parameter low pass mode time domain transform with rectangular window.

TDR_Sxx_LP_HW : Sxx parameter low pass mode time domain transform with Hamming window.

TDR_Sxx_BP_RW : Sxx parameter band pass mode time domain transform with rectangular window.

TDR_Sxx_BP_HW : Sxx parameter band pass mode time domain transform with Hamming window.

Each mode can be applied to any of the four S-parameters.

When using the low-pass mode it is recommended to select TDR as frequency sweep mode:

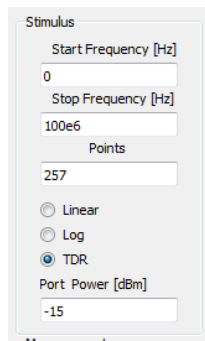


Figure 6.4: VNA sweep modes.

Now the start frequency can be set to zero. The number of points must fulfill the formula 2^{n+1} . This means for example 257, 513, 1025 or 2049 points.

For the bandpass mode, choose linear as frequency sweep type and select a proper start- and stop frequency.

6.3.1 Application: Fault location

When viewing a filter's transmission response, ripple may appear due to reflection. However it is impossible to tell where in the filter setup the problem occurs. What is seen is the total reflection of cables, connectors, transmission lines and the filter structure. With the time domain analysis tool the discontinuities are separated in time/distance and any inductive or capacitive mismatch can be easily identified. This gives an intuitive insight into fault location.

6.3.2 Application: Impedance variation

To determine the discontinuity in transmission lines the TDR function can be used to analyze the S11 parameter. S11 is a measurement of how different the DUT's impedance is from the system's characteristic impedance Z_0 . Once the time domain function has been applied, the impulse response can be used to analyze the position and magnitude of the impedance changes.

6.3.3 Application: Filter tuning

Time domain measurements can show the individual responses of resonators and coupling apertures, each filter can be tuned individually. A clear representation is not easily possible in the frequency domain since the interactive nature of coupled-resonator structures makes it difficult to decide which filter element needs to be tuned. The TDR measurements can significantly reduce tuning time in production.

Low Pass Mode	Band Pass Mode
Emulates traditional TDR	Like narrow band time domain
DC value extrapolated	General purpose mode
Equidistant span from DC to stop frequency	Any arbitrary frequency range
Twice the resolution than the bandpass mode	
Good for seeing small responses in devices that passes low frequencies like cable	Best for measuring band limited devices such as filters. Also useful for fault location, but cannot determine the type of discontinuity

Table 6.1 time domain mode features.

For deeper understanding of the time domain analysis, please consult the available literature on this topic.

6.4 VNA direct component measurements

The VNA implements following measurement functions for the measurement of electronic components:

<series RC> R-C in series equivalent, Real = R, Imag = C	Calculates R-C equivalent series circuit from S_{11} . Resistance in Ohm is the real-value, capacitance in farad is the imaginary value.
<parallel RC> R-C in parallel equivalent, Real = R, Imag = C	Calculates R-C equivalent parallel circuit from S_{11} . Resistance in Ohm is the real-value, capacitance in farad is the imaginary value.
<series LR> R-L in series equivalent, Real = R, Imag = L	Calculates R-L equivalent series circuit from S_{11} . Resistance in Ohm is the real-value, Inductance in Henry is the imaginary value.
<parallel LR> R-L in parallel equivalent, Real = R, Imag = L	Calculates R-L equivalent parallel circuit from S_{11} . Resistance in Ohm is the real-value, Inductance in Henry is the imaginary value.

Table 6.2 Component measurement functions.

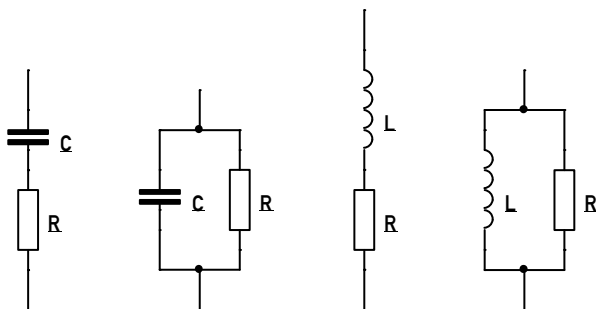


Figure 6.5: Four different component representations measured by the four functions. From left to right: R-C in series, R-C in parallel, R-L in series, R-L in parallel.

The component values will be calculated from a S_{11} or S_{22} measurement. The real value of the function will represent the ohmic part R in Ohms while the imaginary part will represent L in Henries or C in Farads. The measurement functions will assume one of the four circuit representations in Figure 6.5.

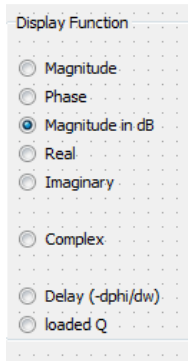


Figure 6.6 Display Function in trace edit window.

Choose "Real" or "imaginary" as display function in the trace edit window to set a trace to the Ohmic or imaginary component value over frequency.

6.5 Loaded Q measurement

The calculation of loaded Q has been added to the display functions section of the trace edit window. Loaded Q is an important function in filter and resonator characterization. The loaded Q value is calculated by the function:

$$Q_l = -\frac{d\phi}{d\omega} \frac{\omega}{2} = \frac{t_{gr}\omega}{2}$$

and can be applied on all Sxx parameters. It represents the quality factor of a DUT under loaded conditions.

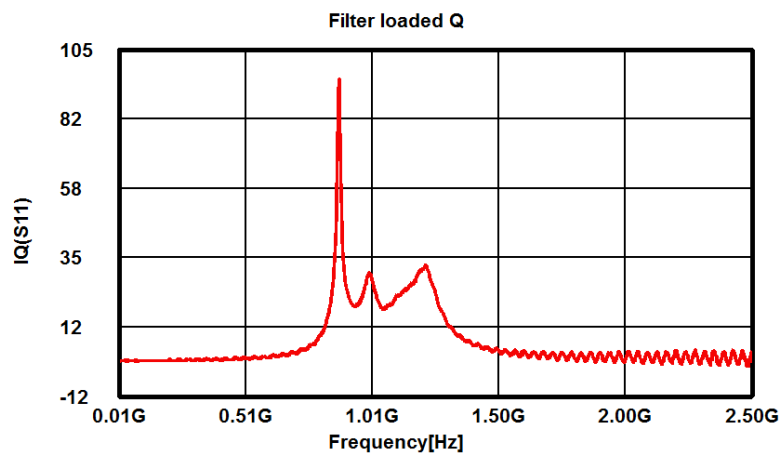


Figure 6.7 Filter loaded Q example plot

6.6 Calibration standards File Format

The calibration standards file format is explained in order to be able to manually enter standard calibration data to the XERXES .xsf format

The "standard" file is a text file which can be edited with any available text editor. Following lines can be in a standard file:

Comments:

Syntax: ! <comment>

Comments shall begin with the character "!" followed by a white space. There can be any number of character lines in a file.

Description:

Syntax: .NAME <description>

The description text is later used to be displayed in the calibration menu and has only informational purposes.

Calibration type:

Syntax: .CALMODE <type = P_2 | P_1 >

sets the type of standards, if the file is a two port standard definition "P_2" or a one port definition "P_1". A one port definition will only contain open/short and load definition for port 1.

Standards:

Short standard Port 1/2:

Syntax: .STANDARD_SHORT_P1_P <L0> <L1> <L2> <L3> <delay> <loss> <Z0>

Syntax: .STANDARD_SHORT_P2_P <L0> <L1> <L2> <L3> <delay> <loss> <Z0>

Parameter	Meaning
L0	Polynomial coefficient for inductor in [H]
L1	Polynomial coefficient for inductor in [H/Hz]
L2	Polynomial coefficient for inductor in [H/Hz ²]
L3	Polynomial coefficient for inductor in [H/Hz ³]
delay	Transmission line delay from reference plane to short in [s]
loss	Transmission line loss in [GOhm/s]
Z ₀	Transmission line characteristic impedance

Table 6.2 standard definition for SHORT standard

Open standard Port 1/2:

Syntax: .STANDARD_OPEN_P1_P <C0> <C1> <C2> <C3> <delay> <loss> <Z0>

Syntax: .STANDARD_OPEN_P2_P <C0> <C1> <C2> <C3> <delay> <loss> <Z0>

Parameter	Meaning
C0	Polynomial coefficient for capacitor in [H]
C1	Polynomial coefficient for capacitor in [H/Hz]
C2	Polynomial coefficient for capacitor in [H/Hz ²]
C3	Polynomial coefficient for capacitor in [H/Hz ³]
delay	Transmission line delay from reference plane to open in [s]
loss	Transmission line loss in [GOhm/s]
Z ₀	Transmission line characteristic impedance

Table 6.3 standard definition for SHORT standard

Load standard Port 1/2:

Syntax: .STANDARD_LOAD_P1_P <Z0> <delay>

Syntax: .STANDARD_LOAD_P2_P <Z0> <delay>

Parameter	Meaning
delay	Transmission line delay from reference plane to open in [s]
Z ₀	Transmission line characteristic impedance

Table 6.4 standard definition for LOAD standard

Through Standard:

Syntax: .STANDARD_THRU_P <delay> <loss> <Z0>

Parameter	Meaning
delay	Transmission line electrical length in [s]
loss	Transmission line loss in [GOhm/s]
Z ₀	Transmission line characteristic impedance

Table 6.5 standard definition for THRU standard

6.7 VNA gain measurement function

The instrument provides multiple gain function to characterize amplifier two ports:

<Gvf> matched voltage gain forward	Complex V_2/V_1 voltage ratio with termination on Port 2
<Gvr> matched voltage gain reverse	Complex V_1/V_2 voltage ratio with termination on Port 1
<Gif> matched current gain forward	Complex I_2/I_1 current ratio with termination on Port 2
<Gir> matched current gain reverse	Complex I_1/I_2 current ratio with termination on Port 1
<k> stability factor	Stability of a two port if $k > 1$
<Gms> maximum stable gain	Maximum stable gain of a two port
<Gmax> maximum available gain	Maximum available gain of a two port
<Gvfo> open output voltage gain forward	Complex V_2/V_1 voltage ratio with open Port 2
<Gifs> short output current gain forward	Complex I_1/I_2 current ratio with open Port 2
<Gvfa> arbitrary load output voltage gain forward (use optional value)	Complex V_2/V_1 voltage ratio with resistance on Port 2 given by the optional input field.
<Gifa> arbitrary load output current gain forward (use optional value)	Complex I_2/I_1 current ratio with resistance on Port 2 given by the optional input field.
<Gtp> Transducer Gain	Calculates the two port transducer gain using an arbitrary output Port 2 load defined by the optional value

Table 6.6 gain related functions.

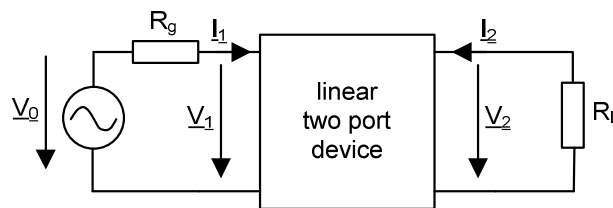


Figure 6.8 Definition of a linear two port device under test.

Figure 6.8 shows a linear two port device under test. V_0 with R_G denote the source port, with $R_G = Z_0 = 50 \text{ Ohm}$. The resistor R_L denote the load port, with $R_L = Z_0$. V_1 , V_2 are the port voltages and I_1 , I_2 the port currents.

6.7.1 G_{vf} Matched forward voltage gain.

The measurement of G_{vf} follows the schematics on Figure 6.8. This functions calculates the complex voltage gain in forward direction with the assumption that $R_G = R_L = Z_0$.

$$G_{vf} = \left. \frac{V_2}{V_1} \right|_{R_G=R_L=Z_0}$$

6.7.2 G_{vr} Matched reverse voltage gain.

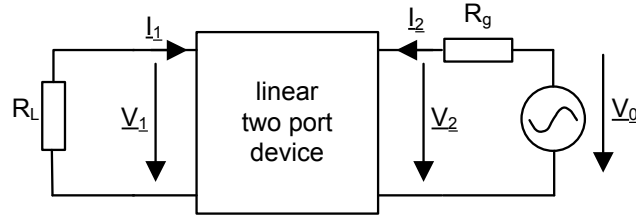


Figure 6.9 Definition of a linear two port device under test in reverse direction

The measurement of G_{vr} follows the schematics on Figure 6.9. This functions calculates the complex voltage gain in forward direction with the assumption that $R_G = R_L = Z_0$.

$$G_{vr} = \left. \frac{V_1}{V_2} \right|_{R_G=R_L=Z_0}$$

6.7.3 G_{if} Matched forward current gain.

The measurement of G_{if} follows the schematics in Figure 6.8. This functions calculates the complex current gain in forward direction with the assumption that $R_G = R_L = Z_0$.

$$G_{if} = \left. \frac{I_2}{I_1} \right|_{R_G=R_L=Z_0}$$

6.7.4 G_{ir} Matched reverse current gain.

The measurement of G_{ir} follows the schematics in Figure 6.9. This functions calculates the complex current gain in forward direction with the assumption that $R_G = R_L = Z_0$.

$$G_{if} = \left. \frac{I_2}{I_1} \right|_{R_G=R_L=Z_0}$$

6.7.5 k Stability Factor

The stability factor k is used in amplifier design as a stability criteria. A k factor of greater than 1 guarantees unconditional stability (under all source and load conditions).

$$k = \frac{1 - |S_{11}|^2 - |S_{22}|^2 + |\Delta|^2}{2|S_{21}S_{12}|}$$

with $\Delta = S_{11}S_{22} - S_{12}S_{21}$

6.7.6 G_{ms} Maximum stable gain

The maximum stable gain G_{ms} of a device is defined when maximum available gain is undefined ($k < 1$). It is the ratio of $\text{mag}(S_{21})/\text{mag}(S_{12})$. It is not advised to get more than this amount of gain from a conditionally stable device. It is recommended to try to stabilize the device using resistive components, until k is greater than 1, then optimize the matching networks for the required gain.

6.7.7 G_{max} Maximum available gain

The maximum available gain G_{max} of a device is only defined where k is greater than one. Algebraically, this is because the term under the square-root becomes negative for values of k less than 1. Another way to look at it is that maximum available gain is infinite. Infinite gain means oscillations.

$$G_{max} = \left(k - \sqrt{k^2 - 1}\right) \left| \frac{S_{21}}{S_{12}} \right|$$

6.7.8 G_{vfo} Forward voltage gain with no load.

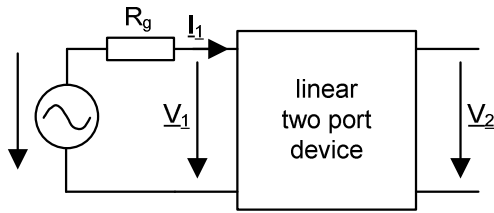


Figure 6.10 Definition of a linear two port device with open output

For this gain it is assumed that the load resistor $R_L = \infty$, although the measurement will be done like shown in Figure 6.8.

$$G_{vfo} = \left. \frac{V_2}{V_1} \right|_{R_G=Z_0, R_L=\infty}$$

6.7.9 G_{ifs} Forward current gain with output short circuited.

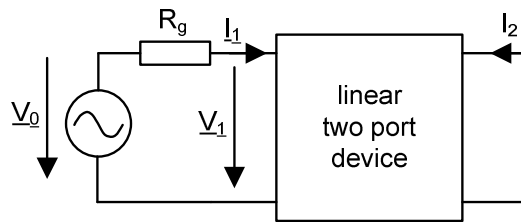


Figure 6.11 Definition of a linear two port device with short circuit output port

For this gain it is assumed that the load resistor $R_L = 0$, although the measurement will be done like shown in Figure 6.8.

$$G_{ifs} = \left. \frac{I_2}{I_1} \right|_{R_G=Z_0, R_L=0}$$

6.7.10 G_{vfa} Forward voltage gain with arbitrary load.

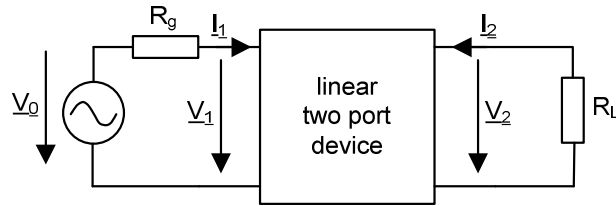


Figure 6.12 Definition of a linear two port device under test.

For this gain it is assumed that the load resistor R_L is set by the option value in the trace edit field, although the measurement will be done like shown in Figure 6.12.

$$G_{vfa} = \left. \frac{V_2}{V_1} \right|_{R_G=Z_0, R_L=arbitrary}$$

6.7.11 G_{ifa} Forward current gain with arbitrary load.

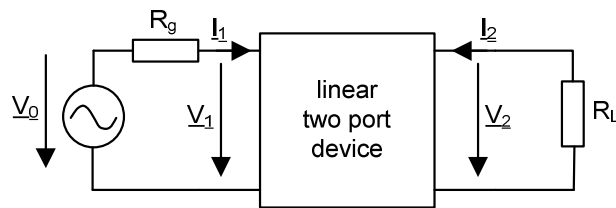


Figure 6.13 Definition of a linear two port device under test.

For this gain it is assumed that the load resistor R_L is set by the option value in the trace edit field, although the measurement will be done like shown in Figure 6.13.

$$G_{ifa} = \left. \frac{I_2}{I_1} \right|_{R_G=Z_0, R_L=arbitrary}$$

6.7.12 G_{ip} Forward transducer gain with arbitrary load.

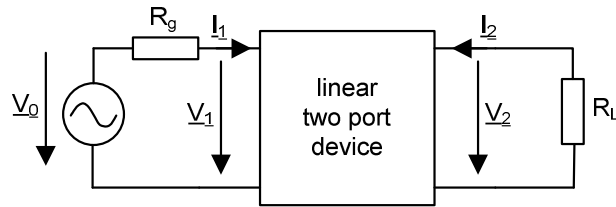


Figure 6.14 Definition of a linear two port device under test.

This gain is a power gain and it is assumed that the load resistor R_L is set by the option value in the trace edit field, although the measurement will be done like shown in Figure 6.14

$$G_{TP} = \frac{|S_{21}|^2 (1 - |\Gamma_L|^2)}{|1 - S_{22}\Gamma_L|^2} \text{ under the condition that } R_G = Z_0.$$

6.8 Antenna Factor and EMI measurement correction

In order to perform accurate measurements of the electrical field strength, it is necessary to correct the measured signal from the antenna. One important parameter is the so called Antenna Factor. Antenna factor is defined as the ratio of the electromagnetic field incident on the antenna to the voltage output at its output terminal (connector). This is a linear definition expressed in volts per meter (V/m). So, in linear terms, the field strength is equal to the voltage output multiplied by the antenna factor. In practice, a logarithmic expression for the ratio is much simpler to apply, and is more commonly used. The advantage of using log terms is that instead of multiplying the two numbers, you simply add. So, the antenna factor normally has a dimension of dB/m, and is added to the voltage measured at the antenna output terminal (in dBuV) to determine the actual E-field level in dBuV/m.

For example, let's say an antenna is present in an electromagnetic field equal to 60 dBuV/m. If the voltage measured at its output connector is 40 dBuV. Then its antenna factor, logarithmically, would be equal to the difference between two, which in this case is 20 dB/m. It is important to note that the factors vary with respect to frequency, so the appropriate factor must be applied, based on the frequency of the signal being measured.

To determine the actual radiated emission level, you simply add the antenna factor (corresponding to frequency being measured) to the measured voltage level at the antenna terminal. So, if 40 dBuV is measured at the antenna output connector, and the antenna factor at the frequency of measurement is 12 dB/m, the value of the E-field is determined as follows:

$$E = 40 \text{ dBuV} + 12 \text{ dB/m} = 52 \text{ dBuV/m}.$$

Generally, the actual measurement system will include other components along with the antenna, which also must be accounted for in determining the actual E-field level. In order to achieve the overall sensitivity of the measurement system, a preamplifier is typically used to boost the signal level at the output of the antenna, so the gain of this amplifier needs to be subtracted from the measured level. Also, the system components will typically be connected to each other via two or more 50-ohm coaxial cables. One or more of these cables is usually quite long, as the measurement equipment is usually located some distance away from the test site, so as to not interfere with the measurements. The cabling may have significant attenuation (known as cable loss).

So, the output at the antenna connector is amplified by the preamplifier and attenuated by the cables before reaching the receiver. Therefore, the E-field value is calculated by subtracting the amplifier gain and adding the cable loss (along with the antenna factor) to the receiver reading. In the above example, if the cable loss is 2 dB and the amplifier gain is 28 dB, the actual value indicated on the receiver would be 54 dBuV. This corresponds to the actual value at the antenna output (40 dBuV), plus the preamp gain (28 dB), minus the cable loss (2 dB), minus the antenna factor (12 dB/m).

To use the above formulas to create an antenna correction file for the EMI receiver the process must be reversed:

Correction value = Antenna Factor + cable loss - pre-amplifier gain

The correction file format is already discussed in an earlier chapter:

<frequency value> <n x SPACE> <correction value in dB/m>

for example this could look like (using the example values in the text, for 100 MHz, 200 MHz and 300 MHz) :

100000000.0 -14.0

200000000.0 -14.0

300000000.0 -14.0

The analyzer will use an interpolation to calculate correction values in between.

7 Performance data.

7.1 Vector network analyzer.

7.1.1 Port isolation

Settings:

- Port Power: - 10 dBm
- Bandwidth: 100 Hz
- Average: 50
- Start Frequency: 1 MHz
- Stop Frequency: 1.3 GHz
- Linear Sweep
- No spur reduction
- Full two Port sweep
- No de-embedding.
- Calibration: Normalization with reference to Port 1 - Port 2 cable.
- All ports terminated with 50 Ohms.

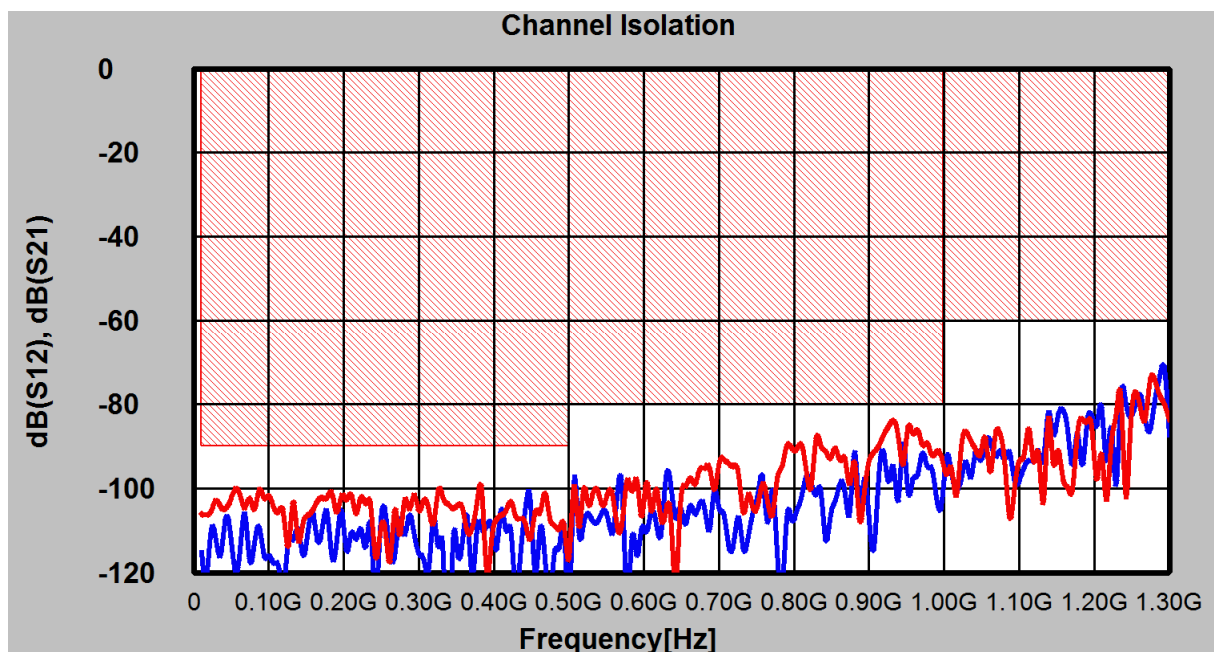


Figure 7.1: Typical channel isolation and limits.

The S12/S21 port isolation specification is:

Frequency range	Limit	Typical
10 MHz - 500 MHz	-90 dB	-100 dB
500 MHz - 1 GHz	-80 dB	-90 dB
1 GHz - 1.3 GHz	-60 dB (over range limit not tested)	-80 dB

Table 7.1 Port Isolation limits.

7.1.2 Trace noise

7.1.2.1 Trace Noise as function of Port Power and Bandwidth

Table 7.2 shows typical results for trace noise measurements. The dB values in the table are mean values of 5 samples of peak to peak measurements. The measurement has been done at 100 MHz.

	10 Hz	100 Hz	1 kHz	10 kHz	100 kHz	200 kHz
-10 dBm	0.15 mdB	0.38 mdB	0.57 mdB	1.5 mdB	4.5 mdB	6.5 mdB
-15 dBm	0.22 mdB	0.62 mdB	0.8 mdB	2.57 mdB	7.2 mdB	11 mdB
-20 dBm	0.33 mdB	0.97 mdB	1.54 mdB	3.68 mdB	11 mdB	19 mdB
-25 dBm	0.40 mdB	1.33 mdB	2.5 mdB	6 mdB	-19 mdB	36 mdB
-30 dBm	0.65 mdB	2.5 mdB	4.1mdB	10 mdB	37 mdB	54 mdB

Table 7.2 Typical Peak to Peak mean Trace noise as function of Port-Power and Measurement bandwidth.

7.1.2.2 Trace noise as function of frequency

Table 7.3 shows typical peak to peak trace noise as a function of frequency. Peak power was -15 dBm and measurement bandwidth 10 kHz.

Frequency	Trace Noise
100 kHz	2.6 mdB
1 MHz	2.6 mdB
10 MHz	2.4 mdB
100 MHz	2.6 mdB
250 MHz	3 mdB
400 MHz	3.6 mdB
500 MHz	5.6 mdB
650 MHz	3.9 mdB
800 MHz	4.7 mdB
1 GHz	7.2 mdB
1.15 GHz	25 mdB
1.3 GHz	75 mdB

Table 7.3 Peak to Peak Trace noise as function of frequency. Port Power = -15 dBm, bandwidth = 10 kHz.

7.1.3 Measurement speed

Bandwidth	time/point
0.01Hz	80 s/point
0.1 Hz	9 s/point
1 Hz	1.2 s/point
10 Hz	75 ms/point
100 Hz	9 ms/point
1 kHz	3.1 ms/point
10 kHz	1.25 ms/point
100 kHz	240 us/point
200 kHz	220 us/point

Table 7.4 Measurement speed including data transfer time over measurement bandwidth.

7.1.4 Signal source data

Frequency range	0.25 Hz - 1GHz
Frequency resolution	0.25 Hz to 500 MHz 0.5 Hz to 1 GHz
Output power range	-7 dBm to -75 dBm (1Hz to 500 MHz)
	-15 dBm to < -90 dBm (500 MHz to 1GHz)
Output signal amplitude accuracy (typ.)	+/- 1 dB @ -15 dBm to -25 dBm
	+/-2.5 dB @ -25 dBm to -40 dBm
	+/-3 dB @ -40 dBm to <-80 dBm

Table 7.5 Signal Generator specifications

7.1.5 Raw directivity source match and load match.

Raw directivity is measured by terminating Load and Port connector with a HP-902D precision load. Raw directivity is measured relative to an open port. Figure 7.2 shows limits and typical values.

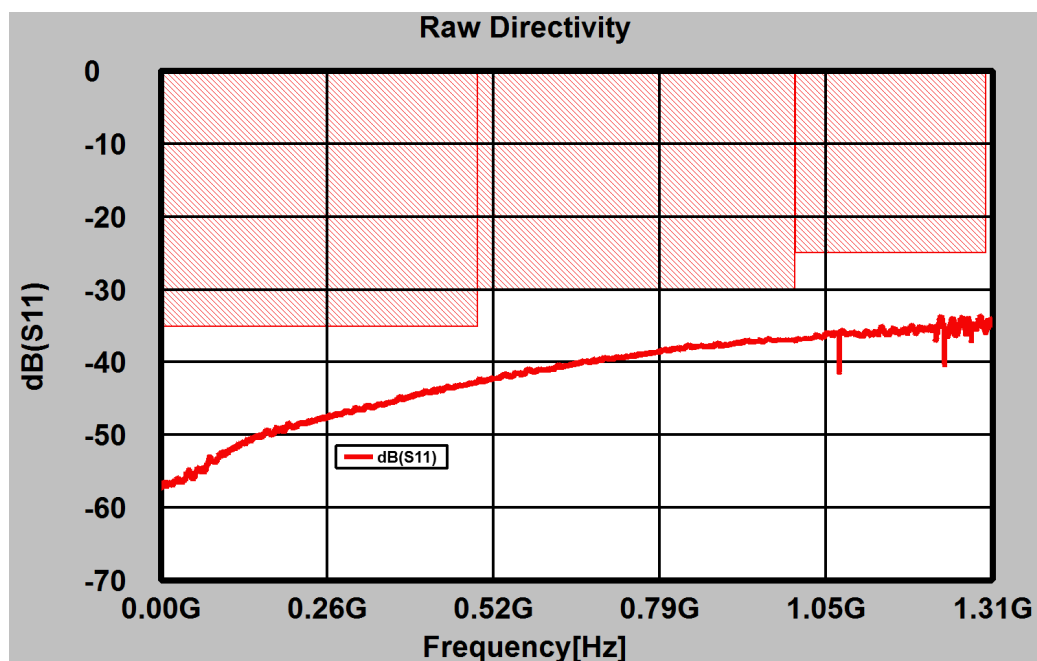


Figure 7.2: Raw directivity limits and typical values.

Frequency range	Limit	Typical
DC - 500 MHz	-35 dB	-40 dB
500 MHz - 1 GHz	-30 dB	-38 dB
1 GHz - 1.3 GHz	-25 dB (over range limit not tested)	-35 dB

Table 7.6 Raw Directivity values over frequency.

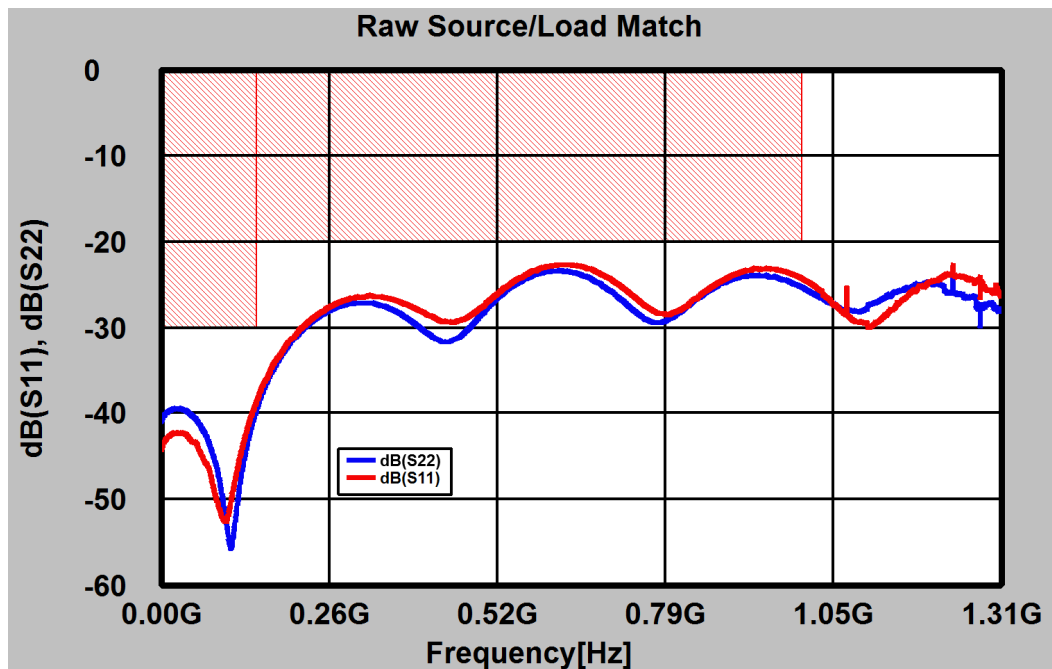


Figure 7.3: Raw Source/Load Port matching limits including typical values.

Frequency range	Limit	Typical
DC - 150 MHz	-30 dB	-40 dB
150 MHz - 1 GHz	-20 dB	-25 dB

Table 7.6 Raw Source/Load Port matching.

7.1.6 Measurement Uncertainty

7.1.6.1 Measurement Uncertainty Reflection Measurement

Frequency range: 10 MHz - 1 GHz.

Bandwidth 100 Hz.

Port Power: -10 dBm

Reflection coefficient (Lin)	Worst case uncertainty (Lin)
1.0	0.03
0.25	0.01
0.1	0.006
0.01	0.005
0.001	0.005
0.00025	0.005

Table 7.7 Linear Reflection Measurement uncertainty, using XC-87610A calibration kit

Reflection coefficient (Lin)	Worst case uncertainty (Degree)
1.0	2°
0.25	2°
0.1	2°
0.01	2°
0.001	3°
0.00025	7°

Table 7.7 Phase Reflection Measurement uncertainty, using XC-87610A calibration kit

7.1.6.2 Measurement Uncertainty Transmission Measurement

The uncertainty includes residual systematic errors, system dynamic accuracy, SMA repeatability, noise and switch repeatability.

Frequency range: 0.01 MHz - 500 MHz.

Bandwidth 100 Hz.

Port Power: -10 dBm

7.1.6.2.1 Reflection Measurement Uncertainty

Measurement Level from reference (dB)	Typ. uncertainty (dB)
0 dB	0.01 dB
-6 dB	0.02 dB
-20 dB	0.03 dB
-40 dB	0.05 dB
-60 dB	0.3 dB
-70 dB	0.7 dB

Table 7.7 Linear Reflection Measurement uncertainty, using XC-87610A calibration kit

Measurement Level from reference (dB)	Typ. uncertainty (Degree)
0 dB	0.05°
-6 dB	0.05°
-20 dB	0.05°
-40 dB	0.4°
-60 dB	1.6°
-70 dB	7°

Table 7.7 Phase Reflection Measurement uncertainty, using XC-87610A calibration kit

7.1.6.2.2 Transmission Measurement Uncertainty

Frequency range: 500 MHz - 1000 MHz.

Bandwidth 100 Hz.

Port Power: -10 dBm

Measurement Level from reference (dB)	Typ. uncertainty (dB)
0 dB	0.01 dB
-6 dB	0.02 dB
-20 dB	0.03 dB
-40 dB	0.05 dB
-60 dB	0.8 dB
-70 dB	1.6 dB

Table 7.7 Linear Reflection Measurement uncertainty, using XC-87610A calibration kit

Measurement Level from reference (dB)	Typ. uncertainty (Degree)
0 dB	0.05°
-6 dB	0.05°
-20 dB	0.05°
-40 dB	1°
-60 dB	3°
-70 dB	15°

Table 7.7 Phase Reflection Measurement uncertainty, using XC-87610A calibration kit

7.2 Utility Specification

Since the oscilloscope, spectrum analyzer and signal generator are declared as utility, please check Chapter 1.5 for Technical specifications.

8 System Calibration.

8.1 Spectrum analyzer calibration.



Calibration is password protected and needs to be unlocked first. A wrong calibration will invalidate all measurements using the spectrum analyzer function. Always perform calibration with the two XC-8752A "LOAD" ports properly terminated.

To accurately calibrate the spectrum analyzer following equipment is needed.

- An accurate signal source, like Keysight N5171B or HP 8664B or 8665B from Hewlett Packard.
- A low loss RG402 SMA-m to SMA-m semi-rigid connection cable with a length of less or equal to 300 mm.

Connect the signal source to Port A of the XC-8752A using the RG402 cable. on the spectrum analyzer tab press the button "Calibrate" analyzer, as shown in Figure 8.1.

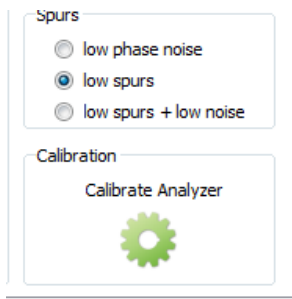


Figure 8.1: Calibrate Analyzer button.

A window will open, asking if calibration is required.

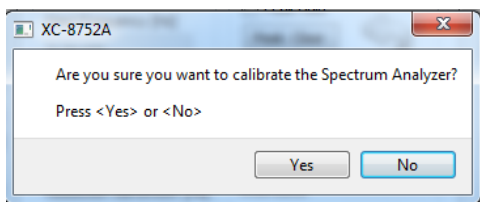


Figure 8.2: Calibration question window.

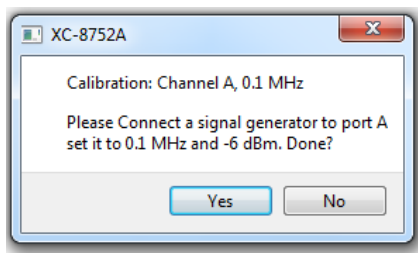


Figure 8.3: Calibration action window.

A new window will appear, requiring to apply 100 kHz, -6 dBm on Port A.

Then a series such windows will requiring different frequencies:

100 kHz, 1 MHz, 10 MHz, 100 MHz, 200 MHz, 300 MHz, 400 MHz, 500 MHz, 600 MHz, 700 MHz, 800 MHz, 900 MHz, 1 GHz, 1.1 GHz, 1.2 GHz and 1.3 GHz.

After finishing this frequency series the window will request changing the Port to Port B and applying 100 kHz with -6 dBm. Disconnect the RG402 cable from port A and connect it to the XC-8752A port B.

Now the frequency series repeats. After finishing the second series, the analyzer will save the calibration data to the internal flash memory.

After a window reports that the calibration data has been successfully saved, disconnect the RG402 cable. The spectrum analyzer is now calibrated.

8.2 Oscilloscope calibration



Calibration is password protected and needs to be unlocked first. A wrong calibration will invalidate all measurements using the oscilloscope function. Always perform calibration with the two XC-8752A "LOAD" ports properly terminated.

To accurately calibrate the oscilloscope following equipment is needed.

- An accurate DC source, like a voltage calibrator for 4 1/2 digit multimeters or the XERXES XC-7112A low voltage calibrator.
- A low loss RG402 SMA-m to SMA-m semi-rigid connection cable with a length of less or equal to 300 mm.
- Possibly a BNC to SMA-f adapter and a banana type to BNC cable.

Please note that the inputs of the XC-8752A have a 50 Ohms input impedance, and this need to be considered if the voltage calibrator has an output impedance other than virtually zero. The calibration voltages are considered at the port.



Do not exceed maximum voltage ratings.

To start calibration press the "Calibrate" button on the "Calibration" tab on the oscilloscope window.

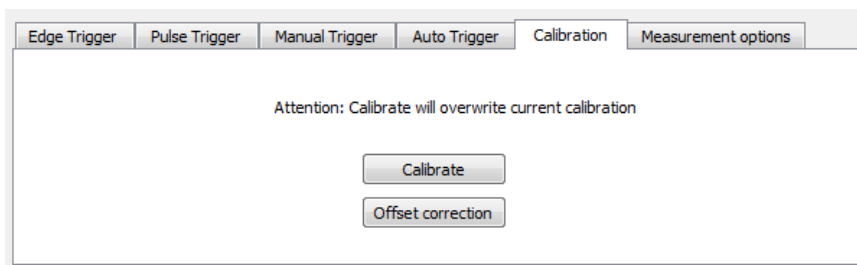


Figure 8.4: Calibrate oscilloscope button.

A window series will appear if calibration needs to be executed.

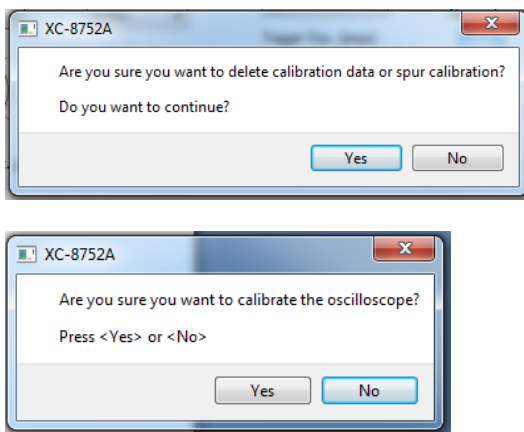


Figure 8.5: Calibration question window.

Now a series of voltage requests for Port A will follow

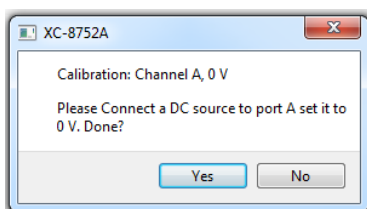


Figure 8.6: Calibration voltage request.

Following voltages will be requested:

0V, 0.25V, 2V.

Adjust the voltage calibrator to the request values, and make sure the real port voltages do reflect the settings.

After finishing Channel A, Channel B follows. After finishing Channel B, another round of Channel A and B voltages requests follow.

After finishing the four series, the instrument will save the calibration data to the internal flash memory.

After a window reports that the calibration data has been successfully saved, disconnect the RG402 cable. The oscilloscope is now calibrated.

8.3 Signal Generator Calibration.



Calibration is password protected and needs to be unlocked first. A wrong calibration will invalidate all measurements using the signal generator function. Always perform calibration with the two XC-8752A "LOAD" ports properly terminated.

To accurately calibrate the oscilloscope following equipment and procedures are required.

- A valid spectrum analyzer calibration.
- A low loss RG402 SMA-m to SMA-m semi-rigid connection cable with a length of less or equal of 150 mm.

To calibrate the signal analyzer press the "Calibrate Generator" button on the spectrum analyzer utility and the phase-noise tab.

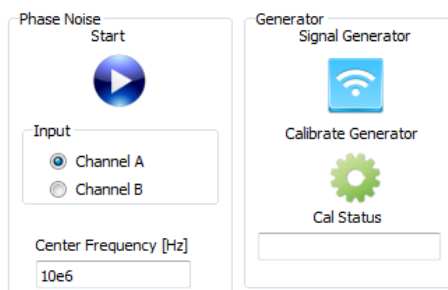


Figure 8.7: Calibrate signal generator button.

After pressing the "Calibrate Generator" button the calibration question window appears.

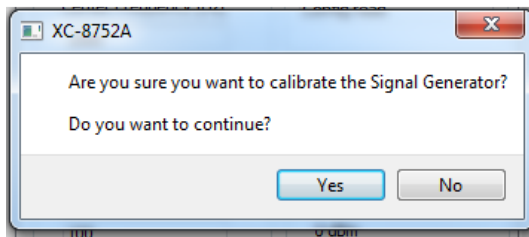


Figure 8.8: Calibration question window.

Press "Yes" to continue.

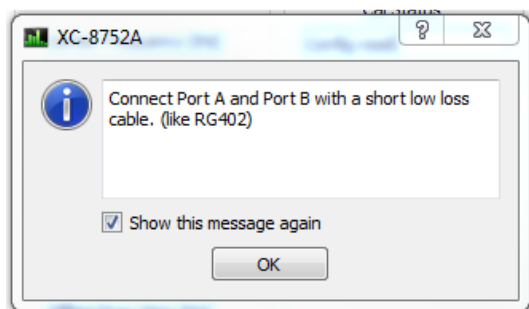


Figure 8.9: Connection request window.

Now connect Port A and Port B with the low loss RG402 SMA cable and press OK.

The analyzer will start auto-calibration. Follow the "Cal Status" field in Figure 8.7. The calibration may take several minutes. After the calibration has finished, remove the connection cable.