Disclaimer

The VNWA software is an experimental software targeted for educational purposes and for experimenters, e.g. radio amateurs.
I will take no responsibility for any damage that may result from its usage.

The VNWA software is still in beta-state . This means, that it may contain bugs and is subject to changes. If you find bugs, please report them.
▸▸▸ **Note:** Also file formats are subject to change, so **do not attempt to use older calibration files and ini-files on newer software** versions!

My home page

www.mydarc.de/dg8saq/

SDR-Kits home page

http://www.sdr-kits.net/

VNWA Web Forum

You are encouraged to post questions and suggestions regarding this software and the related hardware in the VNWA forum:

http://groups.yahoo.com/group/VNWA/

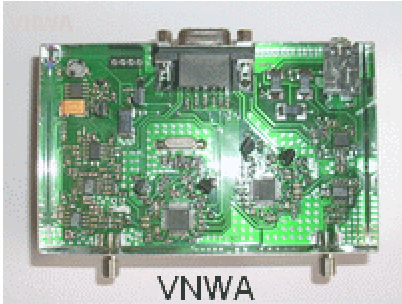
E-mail

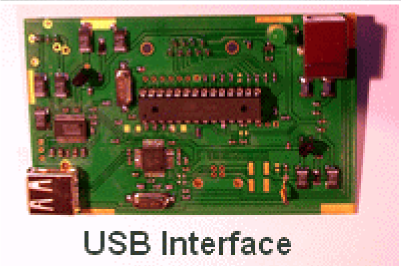
dg8saq@darc.de

Please read this help file carefully and consult the Yahoo VNWA forum before mailing me questions.

January 4th. 2011
Tom Baier DG8SAQ

Prerequesites





This software was originally written to control a **DG8SAQ vector network analyzer (VNWA)** by means of a standard PC running Microsoft **Windows2000®** or **Windows XP®** via the **parallel printer port (LPT)**.
New software builds also support the **DG8SAQ USB_VNWA interface** and **Windows Vista / Windows 7** 32bit and 64bit. Special thanks to Guido PE1NNZ for providing the 64bit WinUSB driver interface!

And sorry, it is not possible to compile this software for Linux or MacOS, as I make abundant use of Windows functions. There is a chance to run the software under Wine if someone manages to USB access in Wine. Apparently, there are patches available. Please report, if you find a solution.

The software can also control an N2PK VNA via LPT or via the USB interface of Dave Roberts G8KBB. Thanks to Paul Kiciak N2PK, Ivan Makarov VE3iVM, Andreas Zimmermann DH7AZ, Eric Hecker, Dave Roberts G8KBB and Roderick Wall VK3YC for supporting this part of the project. See here how to activate N2PK support.

Also thanks to all the beta-testers and users who have provided valuable feedback to optimize the software and this help file.

The software can also be run without hardware connected for data display and analysis purposes. If no VNWA is connected, select the LPT interface, otherwise a warning is issued (no VNWA detected) at every program start.

Software requirements:
-Windows2000, WindowsXP, Windows Vista 32bit or Windows7 32bit

Hardware requirements:
-a Standard PC. Memory is not an issue, computational power is no longer an issue eiuther, as the software has been optimized on CPU usage. The software has been verified to run properly on a 233 MHz PII in LPT mode and USB mode (with USB1.0 interface!).
If you still hit 100% CPU load during sweeping, your machine is too slow or your settings are unsuitable. See How to reduce CPU load in such a case.
-a sound capture device with stereo line input, which may also be connected externally via USB (integrated in the DG8SAQ VNWA_USB interface already).
-a parallel printer port interface or the DG8SAQ USB_VNWA interface.

ATTENTION: An USB to LPT adapter will not work due to critical timings! The DG8SAQ USB_VNWA interface takes over all the timing tasks from the PC.

Features of the VNWA2.* hardware in combination with the VNWA software

- Coverage from below 1 kHz to 500 MHz with dynamic range of up to 90dB – Useful performance up to 1.3 GHz with reduced accuracy.

- Vector networkanalyzer mode and Spectrum analyzer mode

- Possibility to measure frequency converters and mixers in spectrum analyzer mode

- Low power consumption allows power supply out of a USB terminal.

- 2 port S-parameters S11, S12, S21 & S22, VSWR

- 3 port S-parameters and differential / common mode device analysis

- Component Measurements – Resistance, Admittance Capacitance, Inductance & Quality Factor (Q)

- Time domain reflectometry & gating in time domain + FFT

- Linear, Logarithmic and Listed sweep

- Matching tool and Complex Calculator

- Crystal analyzer tool for extracting equivalent circuit parameters

- User defined S-parameter calculus

- Optional S-parameter test set

- Optional USB-interface allows to run the VNWA with a single (USB) cable connection to the PC for highest mobility.

Technical Data

- unidirectional vector network analyzer

- frequency span 1 kHz...1.3 GHz with 12 MHz crystal and overclocking (see warning!)
▶▶▶ **Note:** High accuracy results can only be expected up to 500 MHz, as this is the specification limit of the used mixers and the DDSes show increasing spurs beyond 500 MHz.

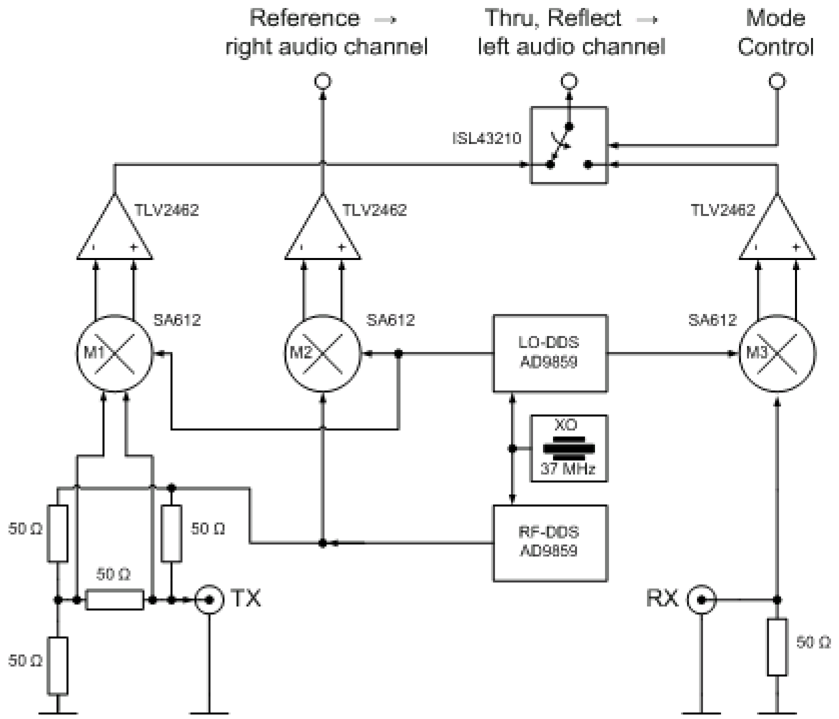
- power supply: 4.5...5.5V DC @400mA max (with switching regulator mounted) or 4.5...5.5V DC @300mA and 8.25V...12V DC @ 15mA (without switching regulator but with appropriate shunts mounted)

- controlled via PC parallel printer port or via DG8SAQ USB_VNWA interface
▶▶▶ **Warning:** You can damage your PC if you generate a short circuit on the printer port. I will not be responsible if you kill your PC!

- can be powered from a self powered PC's USB interface.
▶▶▶ **Warning:** Some PCs and USB-hubs switch off the USB power supply, if the connected device hasn't negotiated for currents > 100mA, as might be the case here.
▶▶▶ **Warning:** You can damage your PC if you generate a short circuit on the USB interface. I will not be responsible if you kill your PC!

- board size: 60mm x 100mm

Function of the VNWA



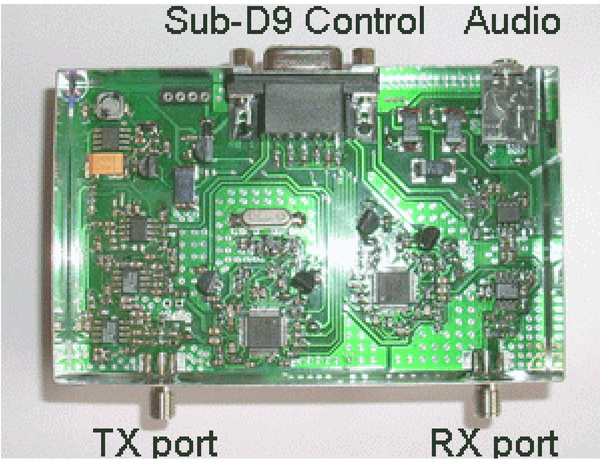
The above schematic shows the fundamental design of the new VNWA2.*. It consists of two digitally tunable Direct Digital Synthesizer (DDS) oscillators, realized by two fast Analog Devices AD9859 chips. The clock generation could be realized in a simple way, as the AD9859s contain an on-chip clock multiplier PLL circuit. Both DDSes are clocked from one and the same crystal oscillator, realized with a low cost standard e.g. 12.3 MHz crystal, which is oscillating on the third overtone at about 37 MHz. The exact crystal frequency is of no importance, as it can be accounted for in the VNWA software. It is crucial, that the two DDS cores are clocked with DIFFERENT frequencies, if one wants to omit anti-aliasing filters and make use of higher order alias frequencies. This is simply achieved here by setting the clock multipliers of the two DDS chips to two different values, e.g. 20 and 19, leading to clock frequencies of 703 MHz and 740 MHz respectively. Note, that in this frequency scheme, the DDSes operate well beyond their specification limits of 400 MHz maximum core clock. Quite remarkably, all tested DDS chips (30 pieces by

betatesters so far), work nicely without getting hot under these conditions. This is a very experimental approach, but pushing the DDS clock frequency means pushing the usable fundamental frequency range of the VNWA, which amounts to 600 MHz under the selected operating conditions.
The RF-DDS output signal is fed into a VSWR-bridge formed by 50 Ohms resistors. The balanced bridge output signal is fed into the balanced inputs of the Gilbert cell mixer M1 (SMD type SA612), followed by an operational amplifier. The amplifier output signal is guided through a CMOS switch to the left channel of the PC sound card line-in. The CMOS switch multiplexes this Reflect signal with the Thru measurement signal originating from mixer M3 and its following amplifier. The multiplexing is necessary, as standard sound cards only have a stereo line-in channel, which can only capture two signals simultaneously. But a third signal, the Reference signal, is required to acquire the phase information. The Reference signal is obtained by mixing the LO- and RF-DDS signals in M2 and amplifying the output signal with the following operational amplifier. It is then fed into the right channel of the sound card line-in. Note that the VNWA works with an IF of about 1 kHz. The sound card is used as IF amplifier, the PC realizes a digital IF filter.
If a device under test (DUT) with two ports is placed between the TX- and RX-terminals of the VNWA, its scattering parameters S11 and S21 can be derived from the three measurement signals (Thru, Reflect, Reference). By manually inverting the device, also S12 and S22 can be measured. In the current design, both, the CMOS switch and the DDSes are controlled with the PC parallel printer port.

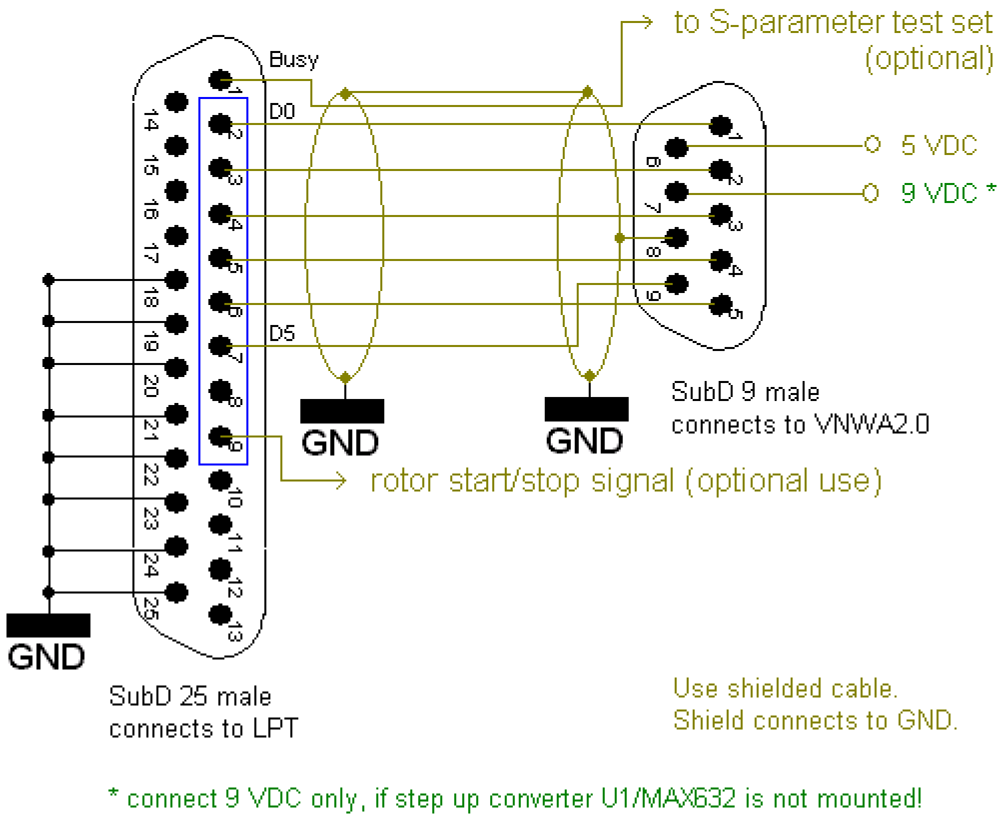
Warning

Using a 12 MHz crystal in 3rd overtone inside a VNWA2.* will generate a 36 MHz clock for the DDS chips. Beware that with clock multipliers of up to 20x this leads to **dramatically overclocking the DDS chips with up to 720 MHz** (This is well outside the DDS specs, which allows for 400 MHz max internal clock). I have personally tested 8 different DDS chips (both AD9951 and AD9859) and found that these worked flawlessly at room temperature under these conditions without even generating excessive heat. The chips should not get warmer than hand warm under these operating conditions.
Nevertheless, I clearly cannot give any guarantee that the VNWA will work under these conditions.
If you insist to run the DDS chips according to the Analog Devices specification, you should use a 20 MHz crystal and run it on the fundamental mode. This will limit the maximum frequency span of the VNWA to about 350 MHz for the first two aliase and to about 800 MHz for the higher ones.

Connecting the VNWA with the PC



As can be seen left, the VNWA2.* has four connectors. The front SMA connectors TX port and RX port are the test ports for connecting the test object or DUT (device under test). The rear connectors are the VNWA's interface to the PC. The audio-connector is a standard 3.5mm female one. Use a standard audio cable to connect the VNWA's audio terminal with the **stereo** line in of your PC's soundcard. **The microphone input is mono and thus won't work!**
The female Sub-D9 connector contains the control lines as well as the power supply lines. The connection scheme to the PC's parallel port (LPT) interface can be seen in the following:

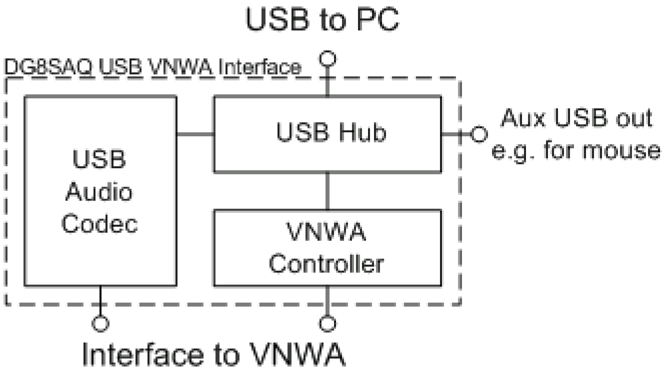


The above schematic shows the cable connecting the PC's parallel port with the VNWA Sub-D9 connector. **Don't apply the 9V DC power, if you have mounted the step-up regulator MAX632 on the VNWA board .** If you have not mounted it, you need to provide two supply voltages, 5V DC and 9V DC.
If you want to power the VNWA from a USB port, you must have the step-up regulator in place and you need to connect an additional cable to the Sub-D9 with a USB connector on the other end.
▸▸▸ **Hint:** In my instrument, I reroute the supply voltage through the parallel port cable and run the USB power cable out of the Sub-D25 LPT connector. This lets the LPT connector and the USB connector both end at the PC and leaves only one cable running to the VNWA (apart from the audio cable).

▸▸▸ **Hint:** In LPT mode, the software issues a high level on LPT port D7 on sweep start, which will return to low level on the end of the sweep. This signal is useful to start and stop an antenna rotor when performing antenna radiation pattern measurements.

▸▸▸ **Note:** I have observed extremely slow signal rise times on some new PCI LPT interface cards. This is a problem when controlling the VNWA. Should you encounter unreliable communtication between PC and VNWA in LPT mode, you might need a signal conditioner, which sharpens the digital pulses.

Functionality of the DG8SAQ USB_VNWA Interface



The above schematic shows the fundamental design of the new DG8SAQ USB_VNWA Interface. It connects to the host PC with a single USB cable and performs both control and data acquisition. At the same time it provides the power supply for the VNWA from the USB cable.
The host PC USB signal is fed into USB hub, which provides 3 USB interfaces. One of them is used to connect an on board USB sound codec, which captures the VNWA audio signals. The second USB hub interface connects to the VNWA controller, a simple microcontroller, which takes care of the sweep timing and DDS control, thus relieving the host PC from a lot of realtime action compared to the LPT modes. The third USB hub interface is not needed and thus available for custom use.

▸▸▸ **Warning: The free USB hub interface is only intended for low power devices like a mouse or a memory stick. You might damage your PC and / or your DG8SAQ USB_VNWA Interface by connecting high power USB devices, as no power detection / control is implemented inside the DG8SAQ USB_VNWA Interface!**

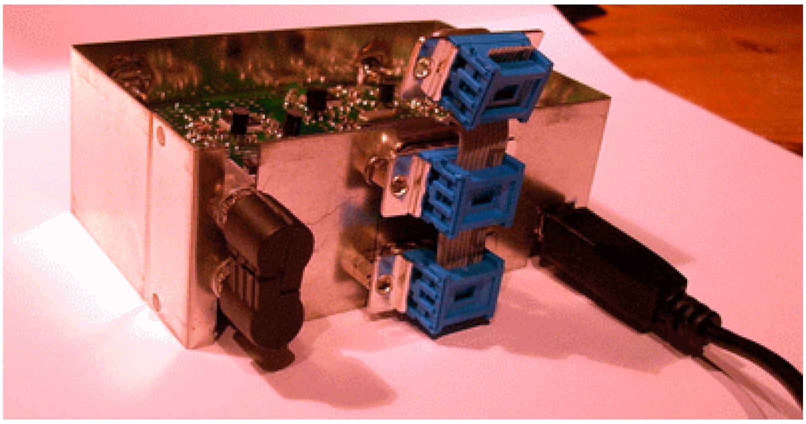
▸▸▸ **Note:**The free USB hub interface does not support full speed, thus transfer rates to memory sticks will be limited to about 1MByte/s.

Connecting the DG8SAQ USB_VNWA Interface to the VNWA and to the host PC

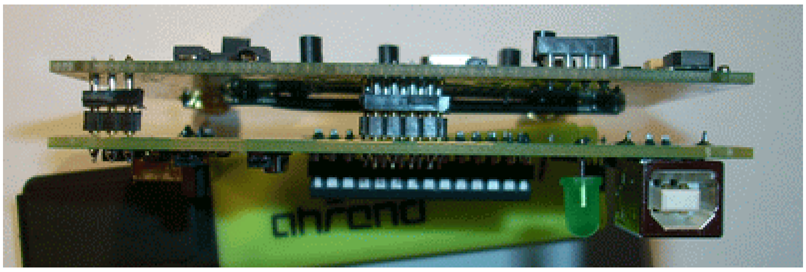
The DG8SAQ USB_VNWA Interface simply connects with a **single standard USB cable** to the host PC. The interface has been proven to work on PCs with USB1.0 and USB2.0 interfaces.

There are two possibilities to connect the DG8SAQ USB_VNWA Interface to the VNWA:

1. External connections



As can be seen in the upper picture, the audio and data connections can be made via the external 3.5mm connectors and the Sub-D9 connectors. This allows to upgrade an original LPT mode VNWA to USB support. Also, VNWA and USB-Interface may be housed in different boxes. Note, that the unused top Sub-D connector is for diagnostic purposes.



The late VNWA boards support directly plugging to the USB interface as seen above. The Sub-D connectors and audio connectors are replaced by appropriate inter-pcb connectors in this case. No additional external connections will be required other than the USB cable to the host PC.

Connecting an S-parameter test set

The control signal for the S-parameter test set is available on the USB_VNWA Interface at pin 3 / J26 or alternatively pin 7 / J5 (Sub-D9) provided the appropriate resistors are in place.

Steps to operate the DG8SAQ VNWA instrument

1) Preparing the VNWA hardware for connection
2) Software and driver installation
3) Software configuration
4) Adjusting sweep Settings
5) Instrument calibration
6) Performing a Measurement

▸▸▸ **Note:** It is good practice to **make regular backups of your data** on your PC, particularly before installing drivers. Installing the VNWA software and drivers is by no means any more dangerous than installing any other Windows software. **SDRKits and the author (DG8SAQ) will not be responsible for any damage or data loss, though.**

Todate the USB-version of the VNWA is supported by all Windows versions from Windows 98 to the latest Windows 7 64 bit. If the VNWA is to be controlled via an LPT port instead of USB, generally only 32 bit Windows systems are supported.
Please consult the "Driver Compatibility Table and Driver Installation".

The latest driver files and software updates are always available from the **VNWA Yahoo Newsgroup**

If you want to use the VNWA application without VNWA hardware, you may go straight on to **installing the VNWA application**.

USB support

Thanks to Fred PE0FKO there is a **unified signed LibUSB driver available for all Windows versions** from Windows98 to Windows7 64bit.
Note, that you **must use VNWA V33.0 or newer and firmware v4.6 or newer to use this signed driver**. For earlier firmware versions use the unsigned LibUSB driver (only installable on 32 bit systems).

▸▸▸ **Important for Vista / Windows 7 64 bit users: Only VNWA firmware v4.6 or newer and VNWA software v33.0 or newer are supported.**
If you have received your VNWA after April 2010, you have already received firmware v4.6 or higher. If you have a working VNWA installation, you can also check your firmware version with the VNWA software in "**setup-USB settings**" by pressing "**Test USB**". After the test is performed successfully, the firmware version is being displayed in the bottom status line. Consult chapter "Upgrading VNWA firmware to v4.6" for details..

Installation examples:
Example 1: Installing the LibUSB driver on Windows XP (English screenshots)
Example 2: Installing the LibUSB driver on Windows 7 (mostly English screenshots)
Example 3: Installing the LibUSB driver on Windows XP (German screenshots)
Example 4: Installing the LibUSB driver on Windows 7 (German screenshots)

LPT support

LPT support compatibility table

	Windows				
Windows Driver	98	2000	XP 32 bit+64 bit	Vista + W7 32 bit	Vista + W7
zlportio	no	☑	XP 32 bit only!	☑*	not insta

* only, if program started with administrator rights

▸▸▸ **Note:** In order to enable LPT support, you must copy the driver file **zlportio.sys** into your VNWA program directory.

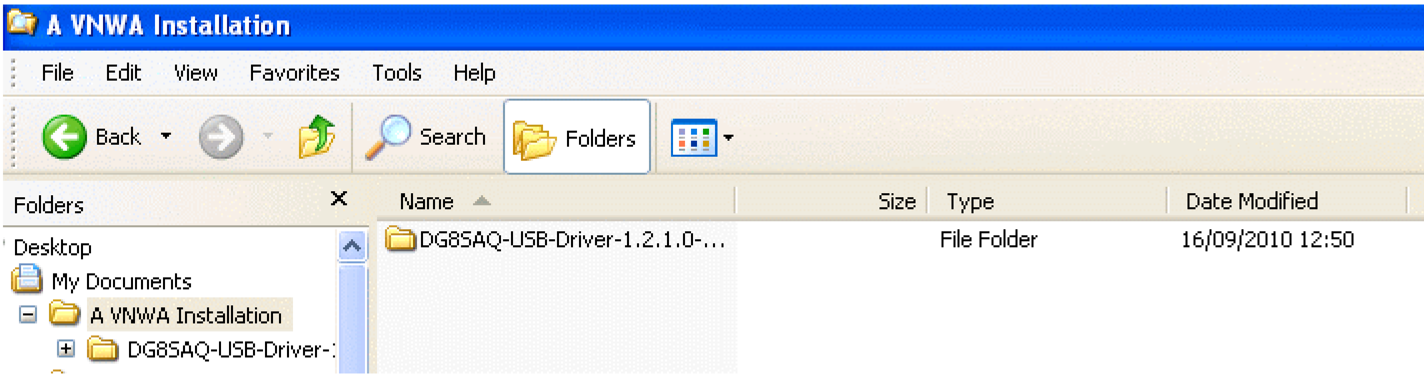
▸▸▸ **Note:** If you do not use the VNWA USB interface and you do not intend to install the LibUSB driver, you must manually provide the file **libusb0.dll**. You must copy the libusb0.dll version from the LibUSB driver package into your VNWA program directory in this case.

▸▸▸ **Note:** Since the parallel printer interface (LPT) is subject to extinction, LPT support for 64 bit OSes will not be implemented.

If you do not intend to install USB drivers, you can go straight on to installing the VNWA application software.

▶▶▶ **Note:** This step is only required if the **DG8SAQ USB_VNWA** Interface is used. **Skip this step if you control your VNWA via the parallel printer port.**

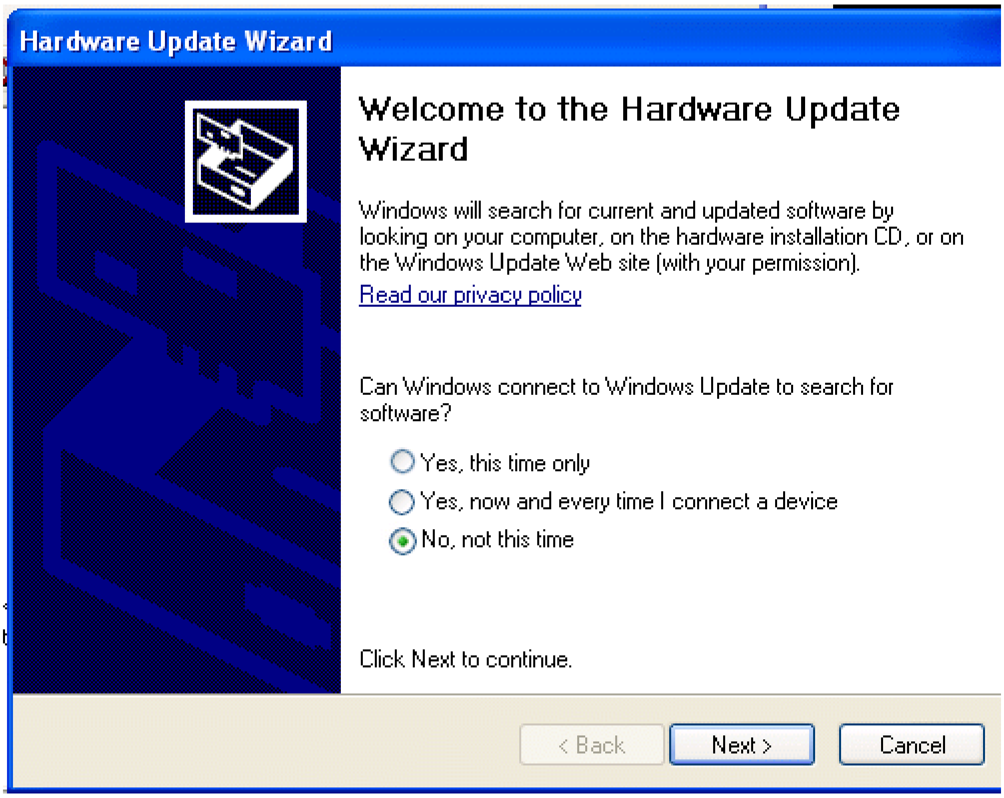
a) Download and unpack the LibUSB driver files to any empty directory:



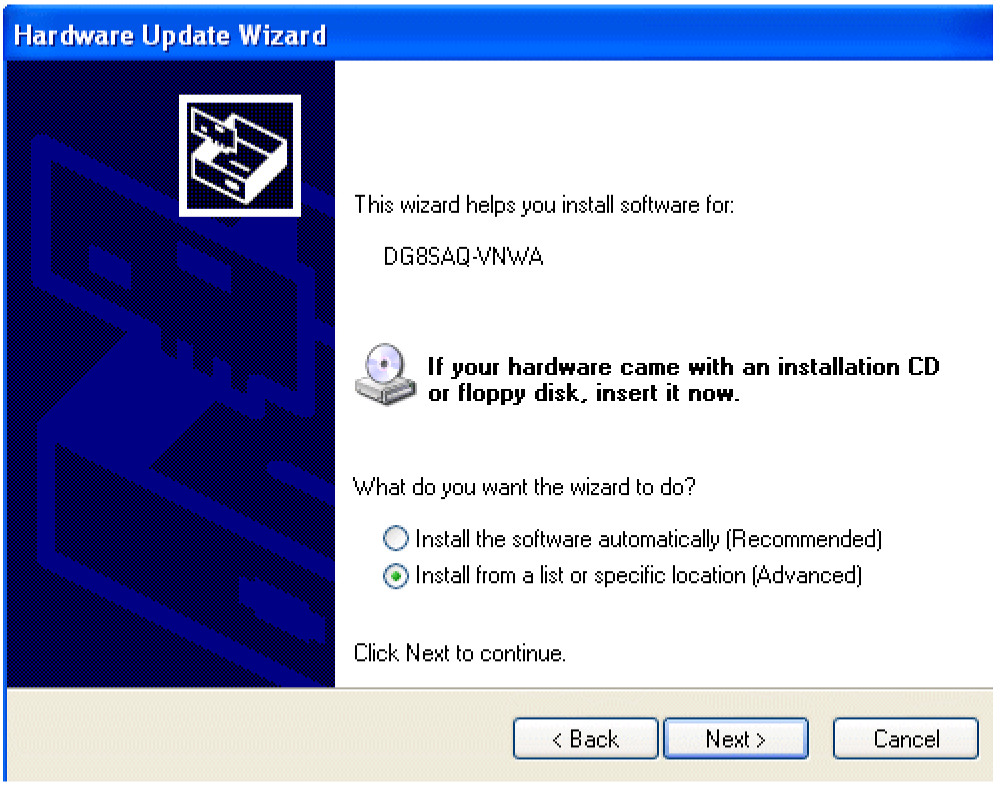
b) Connect the DG8SAQ USB_VNWA Interface with your PC. The VNWA board doesn't need to be connected to the interface board. Windows will notify "Found New Hardware" and will ask to install a driver.

▶▶▶ **Hint:** In case Windows does not detect your USB_VNWA interface, continue with the USB troubleshooting guide. In case Windows does not start driver installation automatically, proceed with manual driver installation via the Windows device manager as is described in Example 2: Installing the LibUSB driver on Windows 7 (mostly English screenshots).

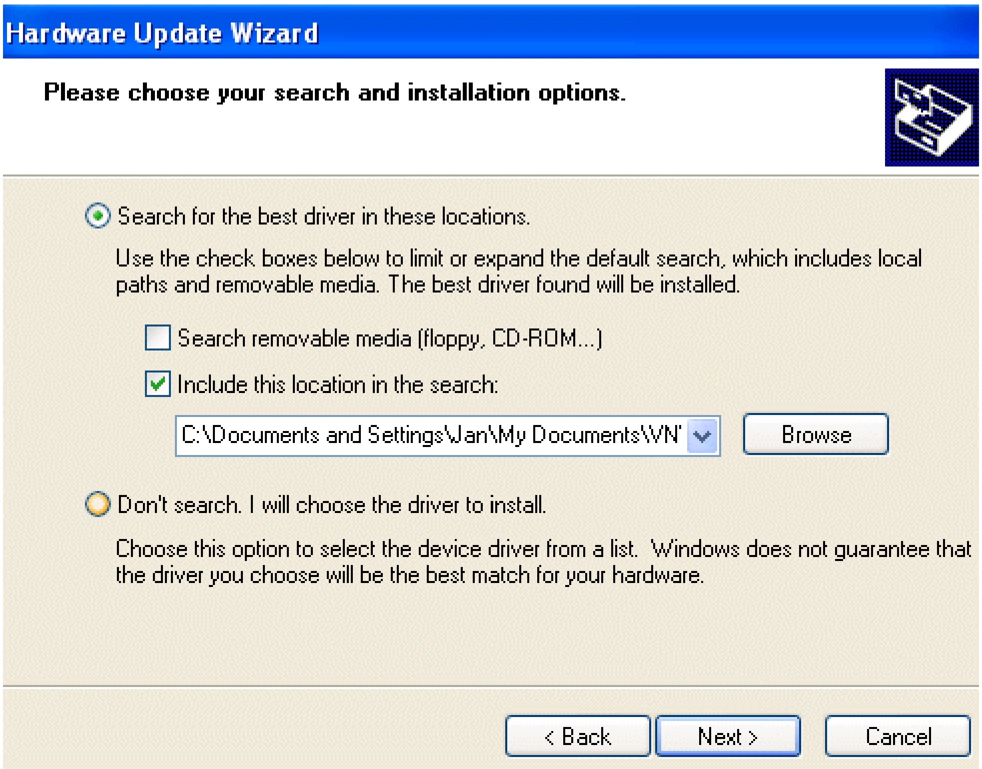
c) Click on "no, not this time" and press continue to prevent driver search on the web.



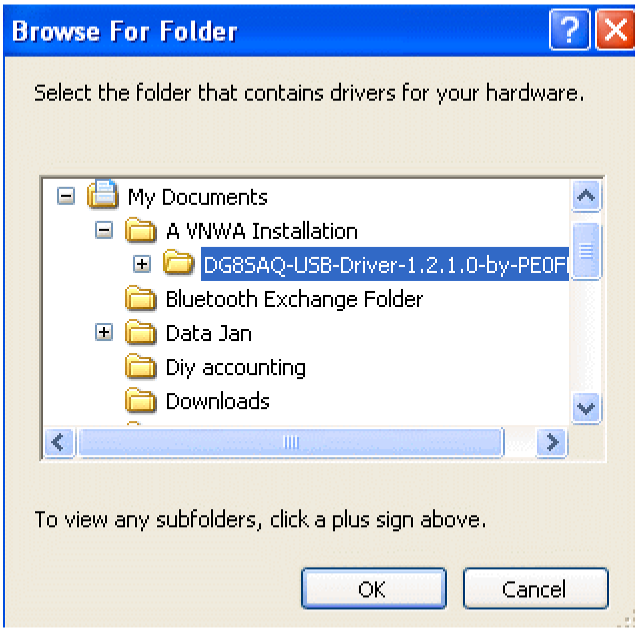
d) Click on "install from a list or specific location (Advanced)" and press continue to install the driver from your hard disk.



e) Browse for the path to your driver files ...

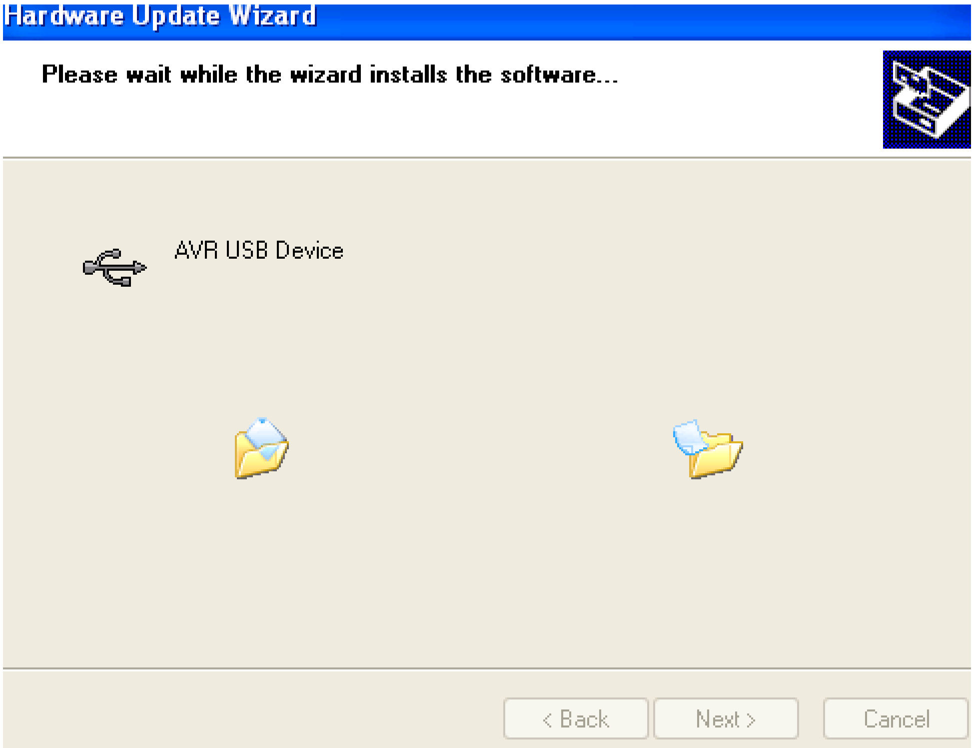


...

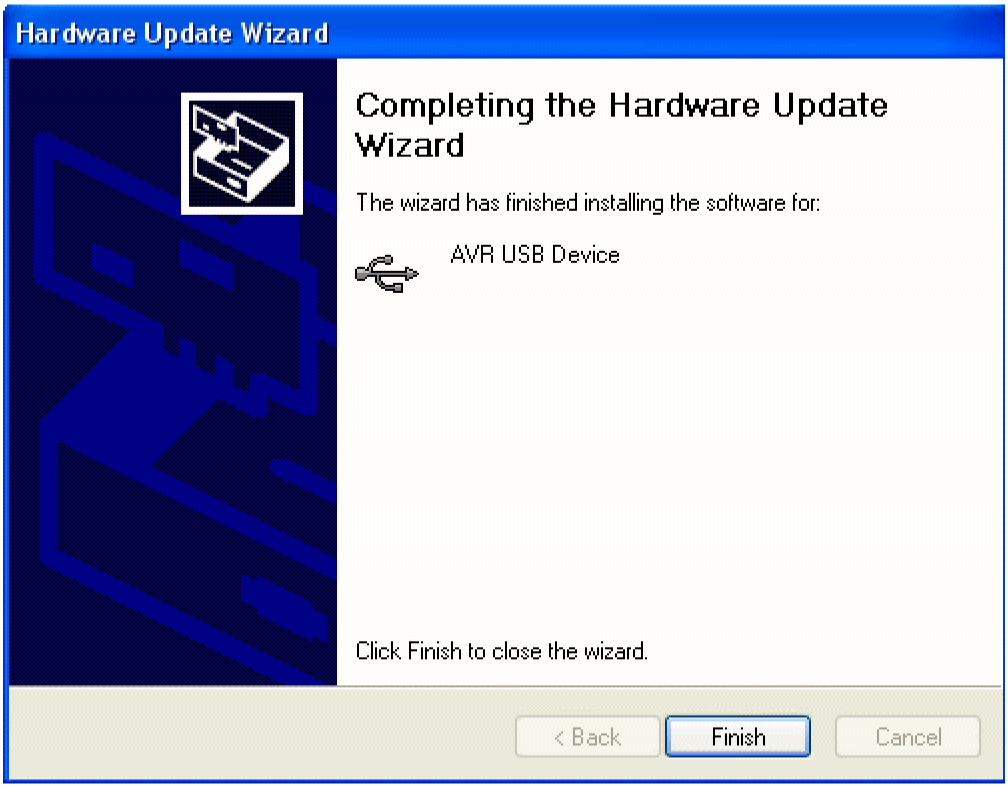


confirm path with ok ... and press >Next on the previous window.

f) The driver will be installed...



Finally you will be notified about the successful installation. Press "Finish" and you can use the USB_VNWA interface on your computer.



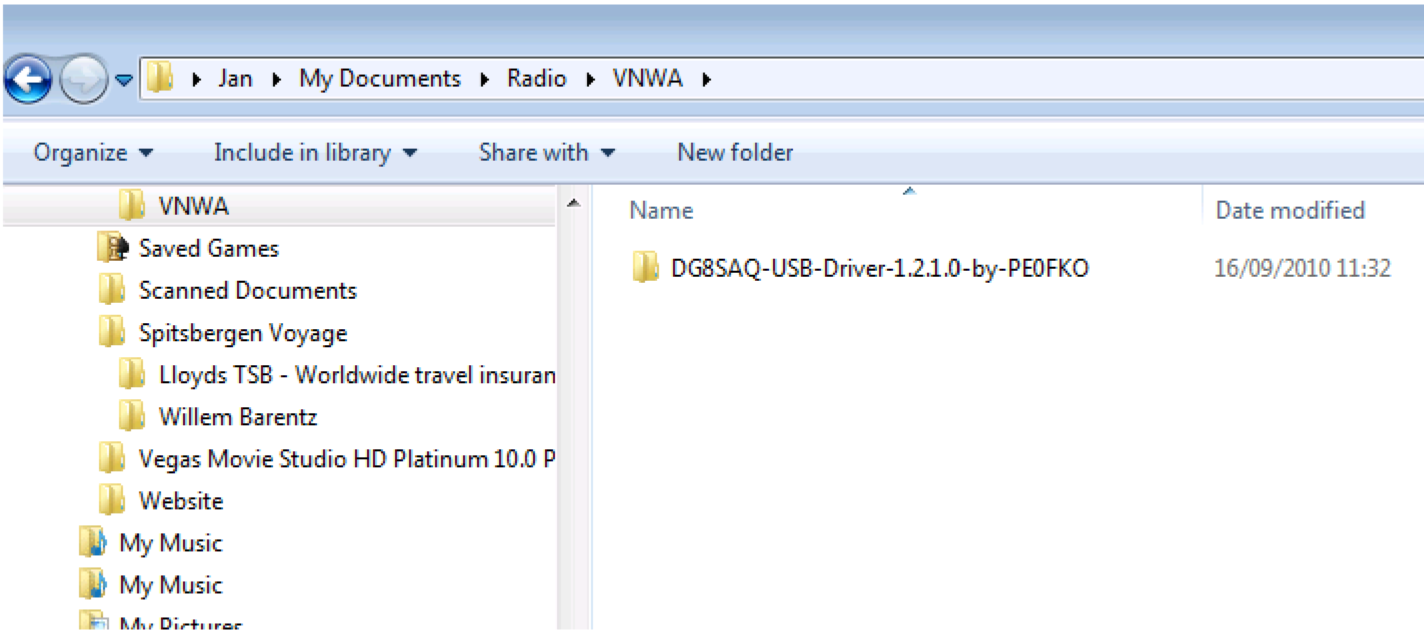
Now you are all set to install the VNWA application software.

▸▸▸ **Note:** This step is only required if the **DG8SAQ USB_VNWA** Interface is used. Skip this step if you control your VNWA via the parallel printer port.

▸▸▸ **Note: You need VNWA software version V33.0 or higher and firmware version v4.6 or newer to work with the signed LibUSB driver.**
See chapter **"Upgrading VNWA firmware to v4.6"** on how to upgrade your firmware.

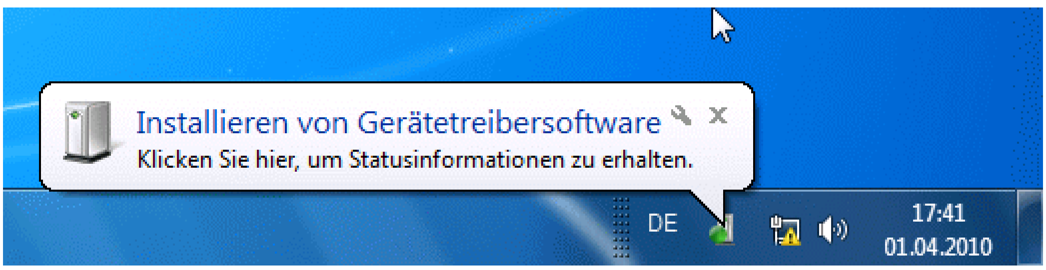
▸▸▸ **Note** The following describes a step by step installation procedure on a Windows 7 / Windows Vista machine, where no previous attempt has been made to install a VNWA driver. If you previously have tried to install different USB drivers for the VNWA **make sure to completely remove all previously installed driver relicts from your system**.

0. Download and unpack the LibUSB driver files to any empty directory your hard disk.

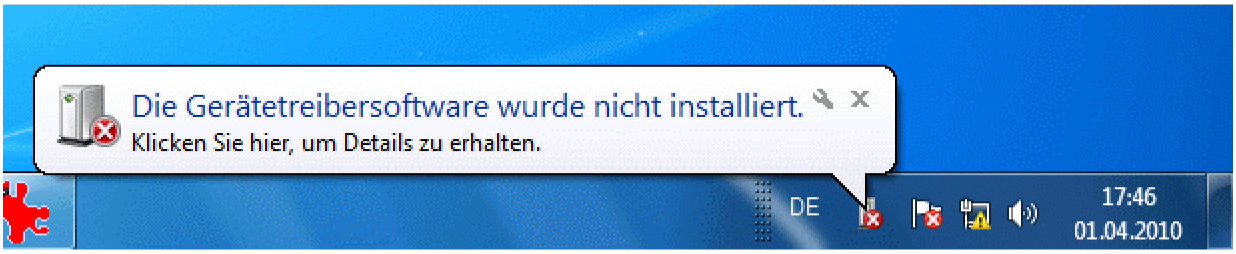


1. Connect your VNWA to a USB port of your computer

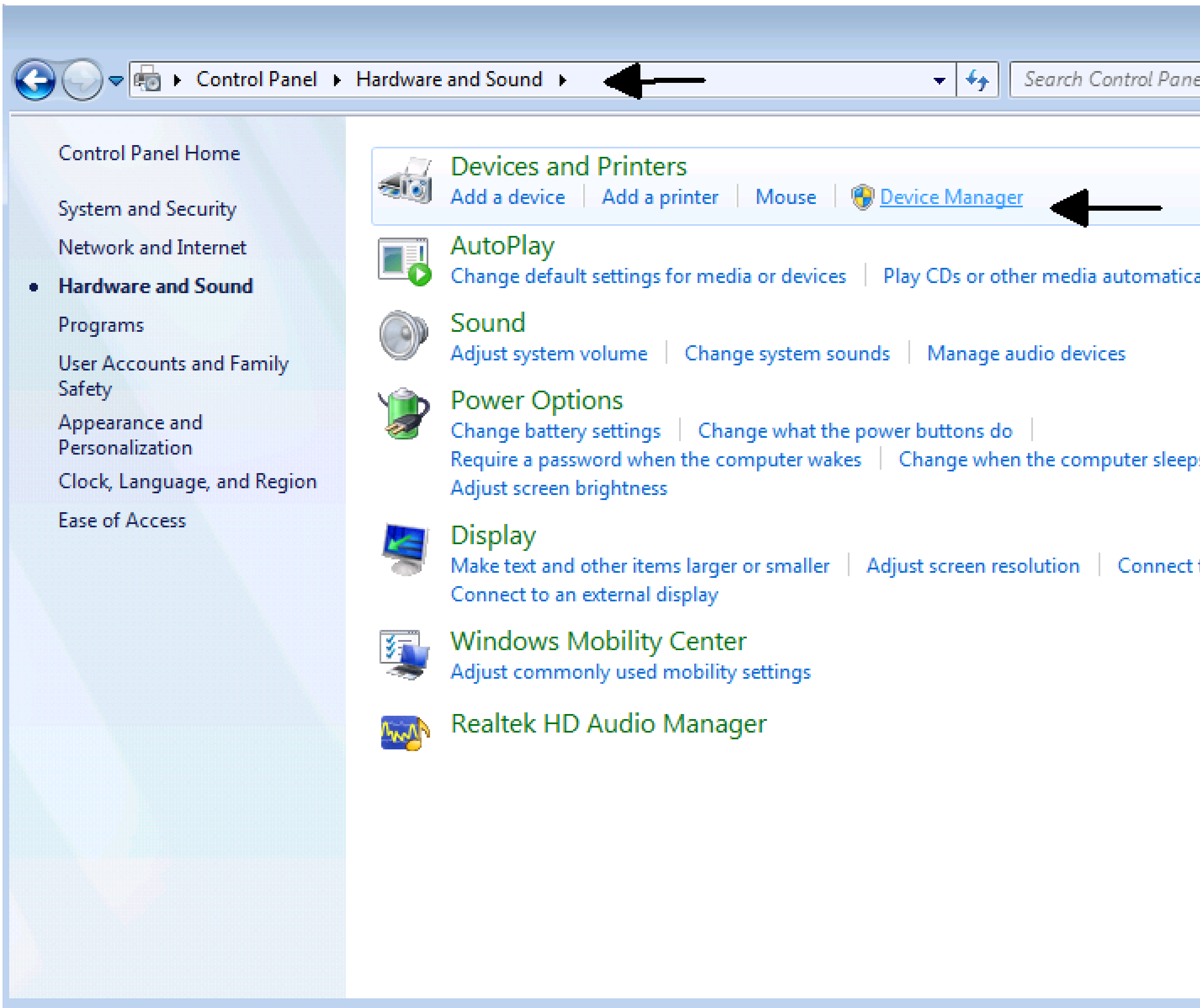
Windows 7 will attempt to install USB drivers for the VNWA ...



... but it will fail as it can't find any drivers.

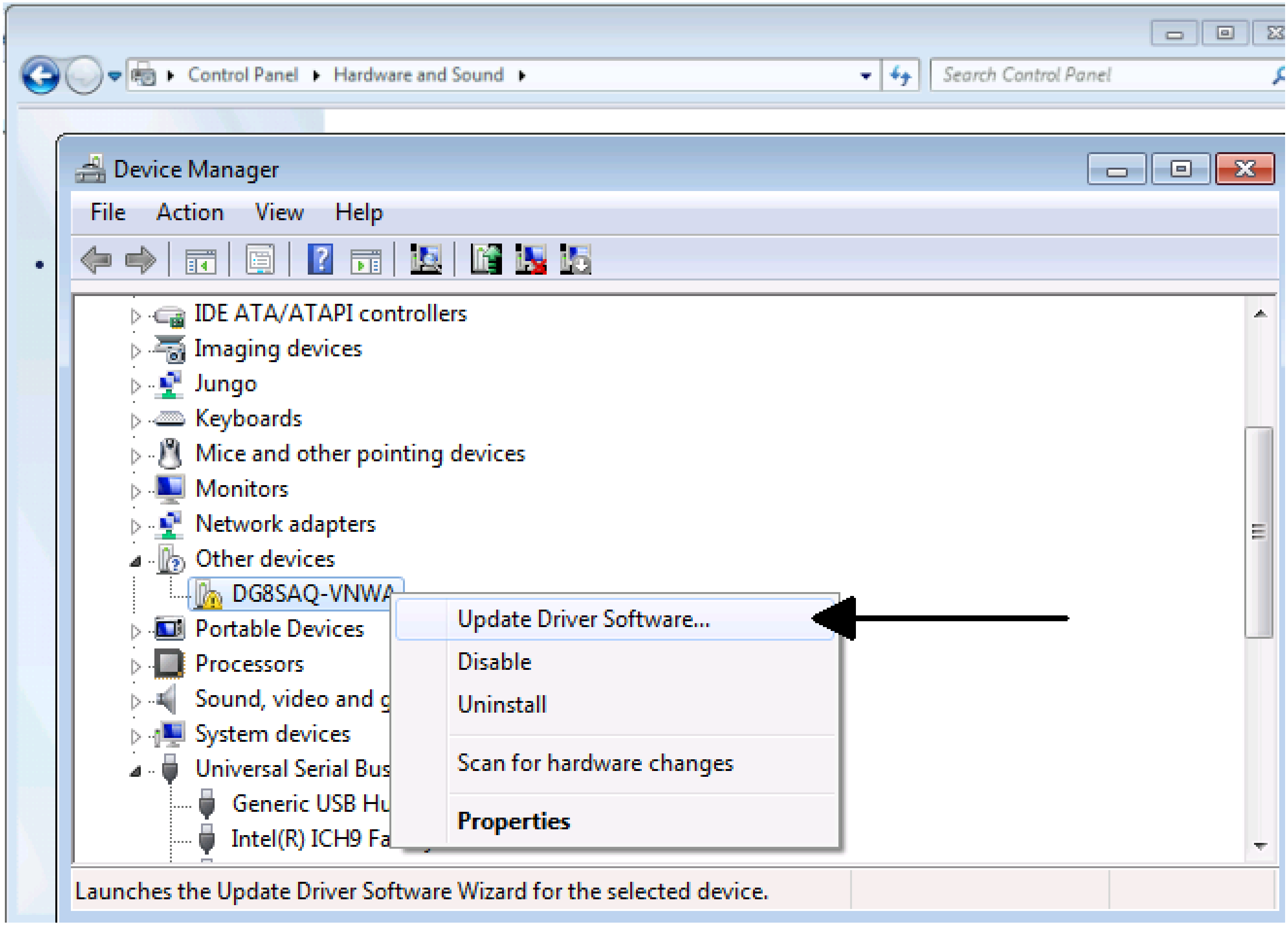


2. Open your device manager



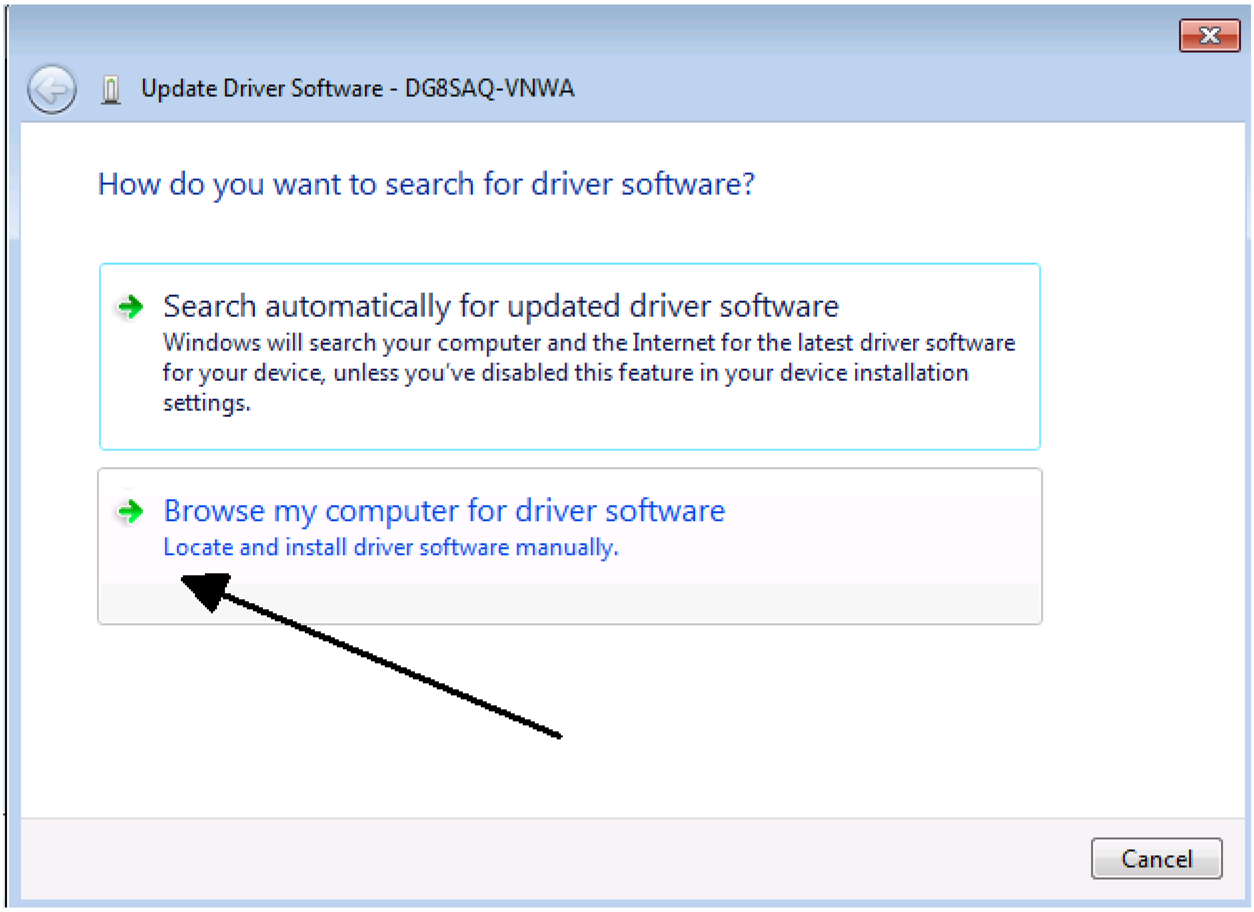
You will see the VNWA as up to now non functional DG8SAQ-VNWA device under "other devices":

3. Right-click DG8SAQ-VNWA and select "update driver software"

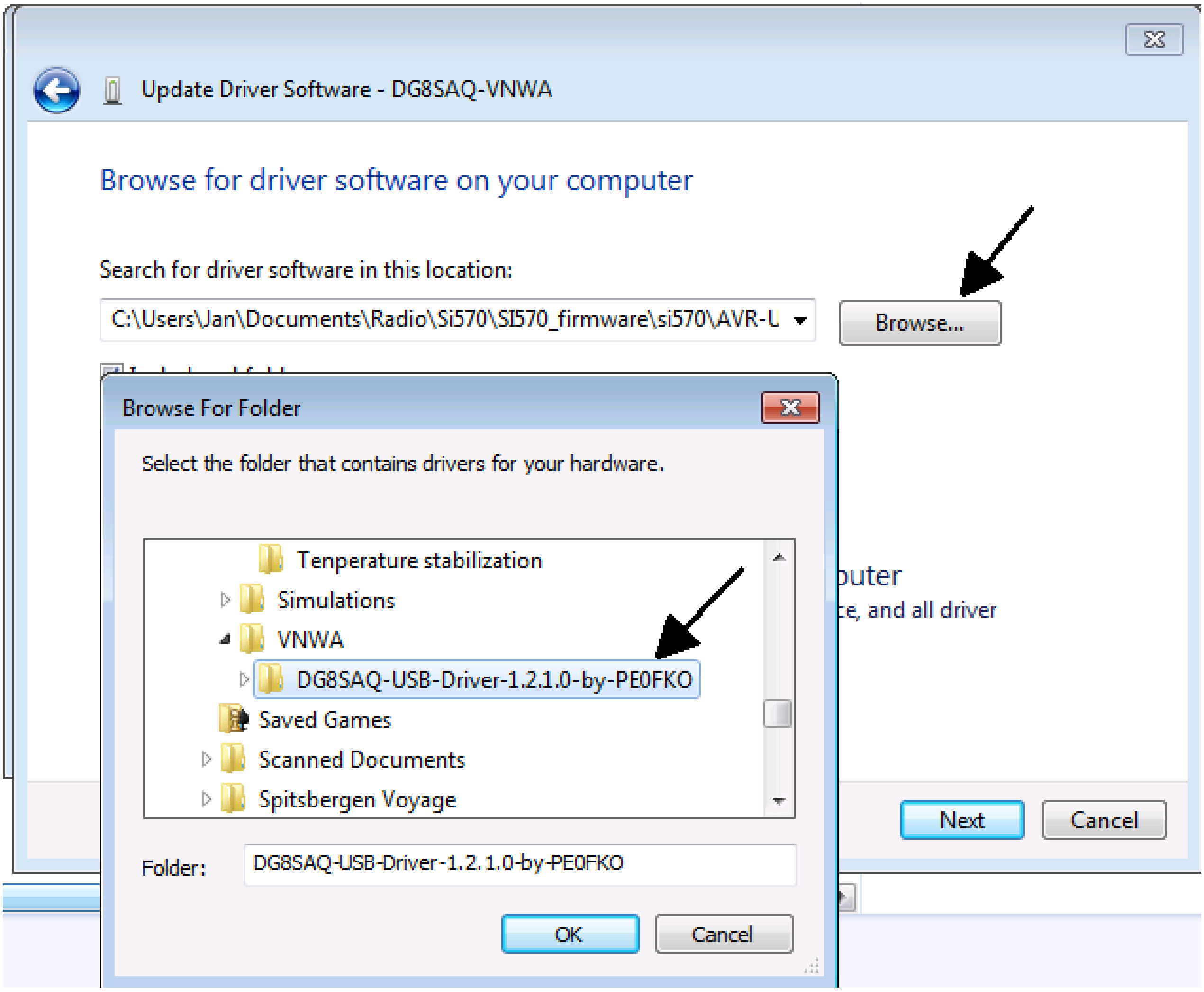


Windows 7 will ask you how you would like to search for driver files.

4. Select "Browse my computer for driver software"



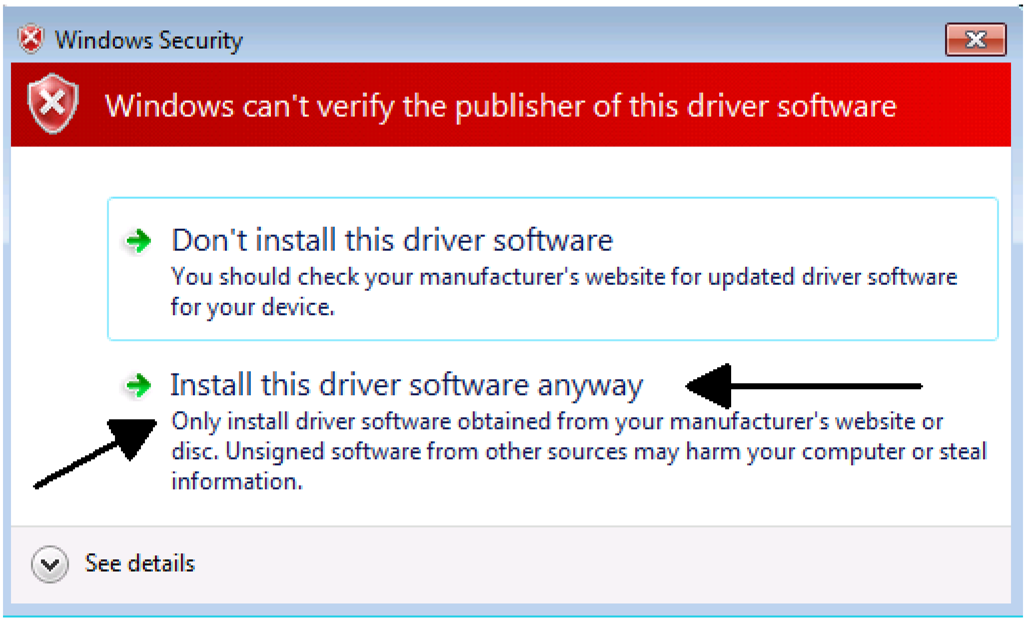
5. Select the unzipped directory which contains the drivers...



... and press Next.

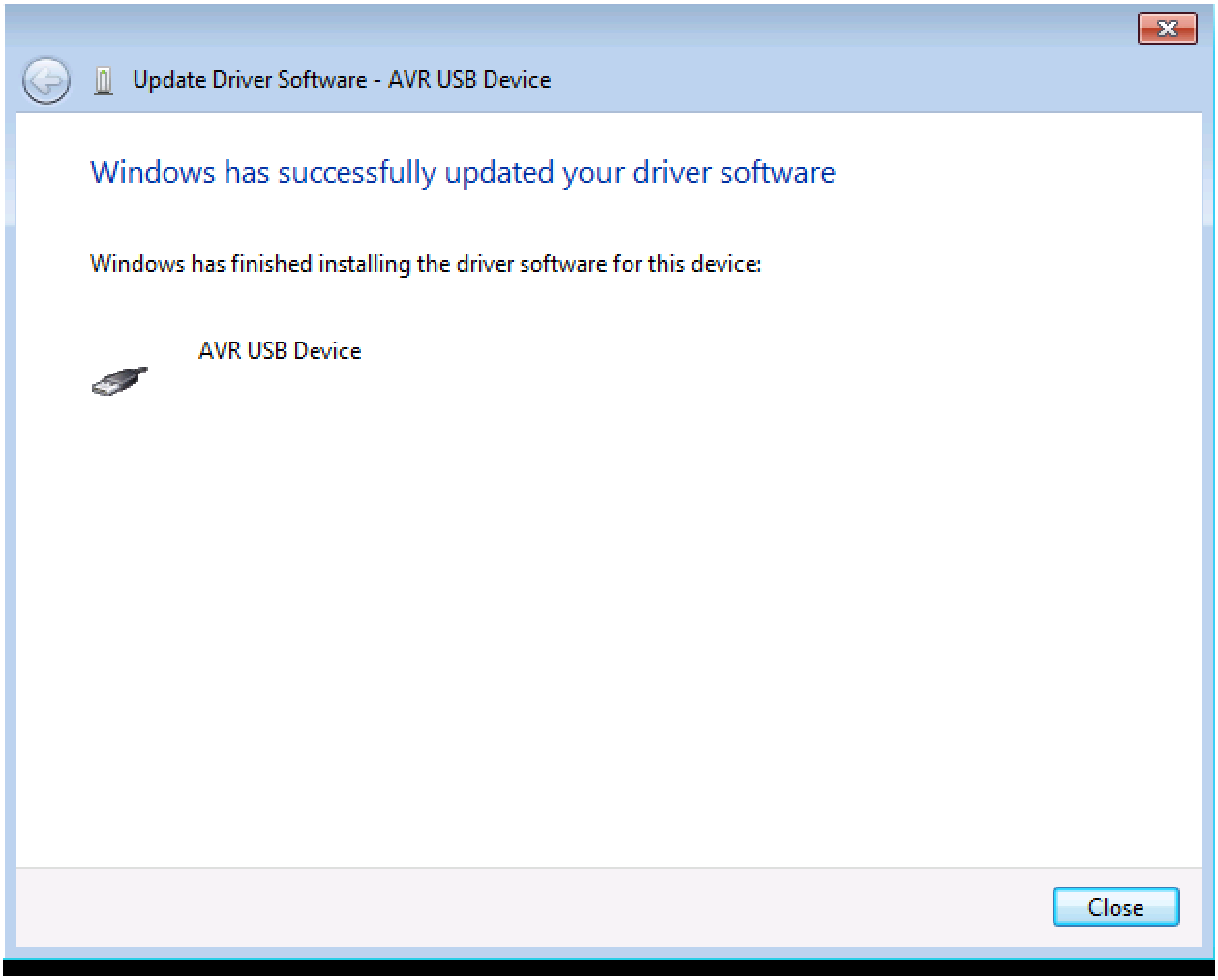
Windows 7 will issue a security warning "publisher of driver software could not be verified".

Even though the USB driver is signed, Windows 7 will issue a security warning "publisher of driver software could not be verified". This is ok and no problem.
If you desire, the warning can be avoided by installing the root certificate prior to driver installation, but this is not required for installing and running the driver.



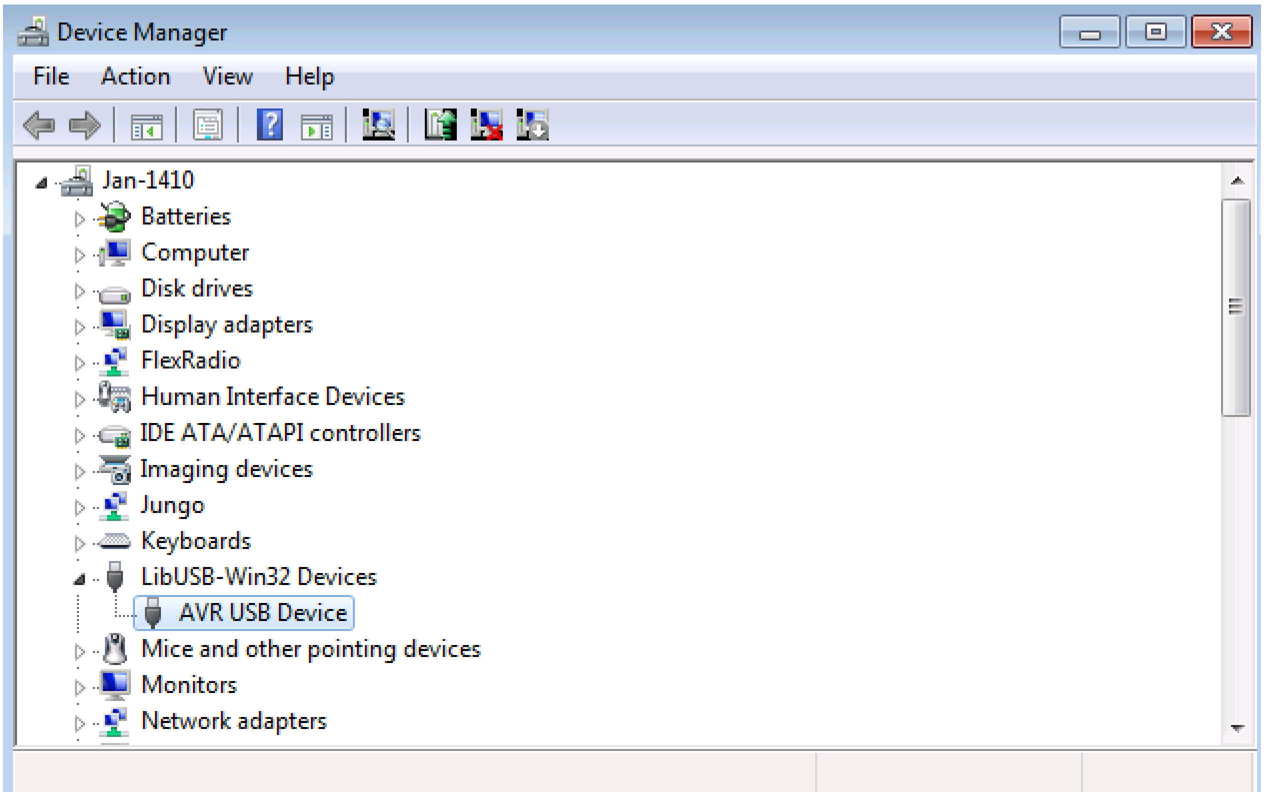
6. Select "install this driver software anyway ..."

... and Windows 7 will start installing the driver software. This might take a couple of minutes until done:



7. Press the close button

The USB driver is now installed and the VNWA device is operational as can be seen in the device manager, where the VNWA will show as AVR USB device:



Now you are all set to install the VNWA application software.

▸▸▸ **Note:** This step is only required if the **DG8SAQ USB_VNWA** Interface is used. **Skip this step if you control your VNWA via the parallel printer port.**

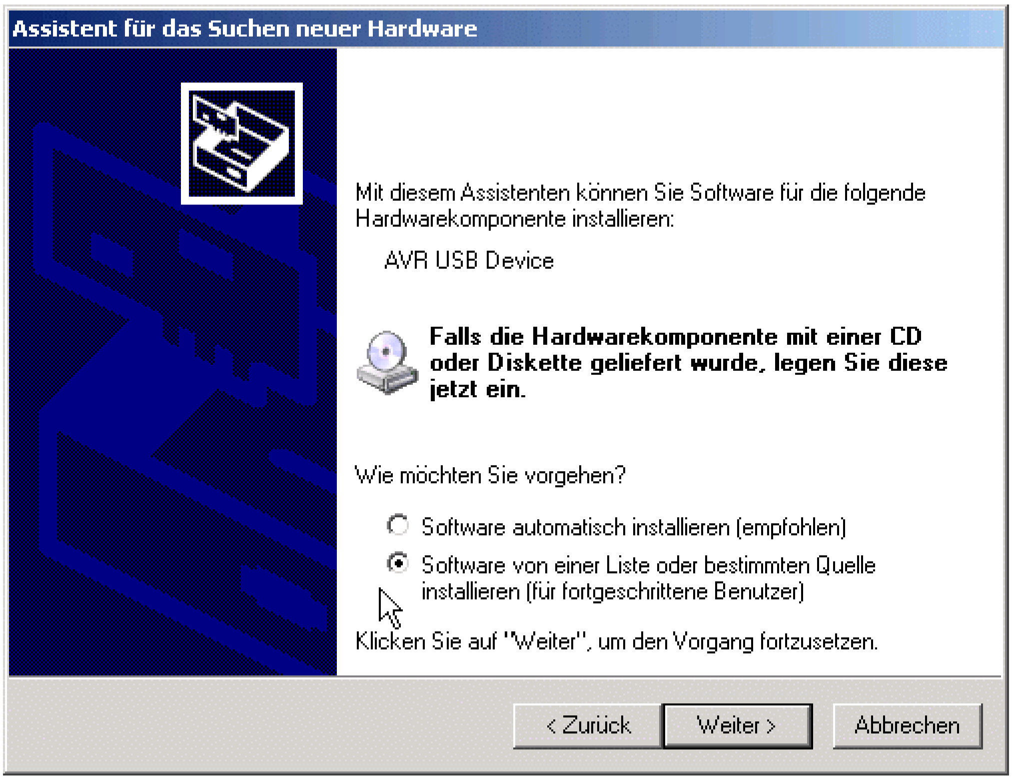
a) Download and unpack the LibUSB driver files to any empty directory:

b) Connect the DG8SAQ USB_VNWA Interface with your PC. The VNWA board doesn't need to be connected to the interface board. Windows will notify "new hardware found" and will ask to install a driver.

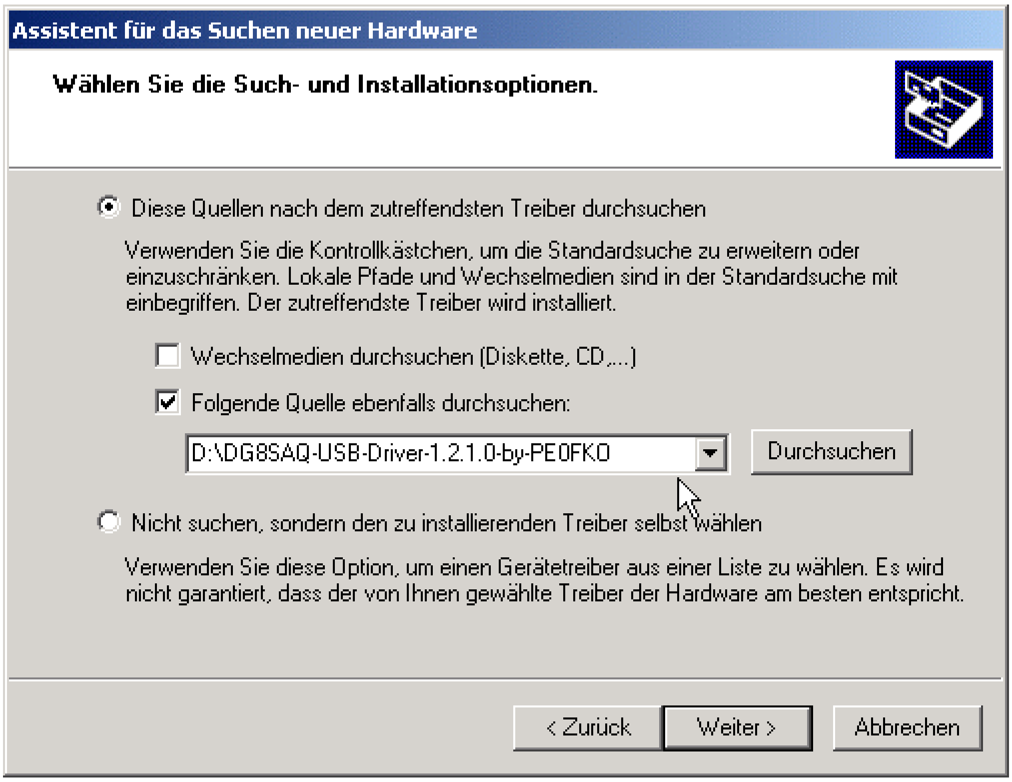
▸▸▸ **Hint:** In case Windows does not detect your USB_VNWA interface, continue with the USB troubleshooting guide. In case Windows does not start driver installation automatically, proceed with manual driver installation via Windows device manager as is described in Example 4: Installing the LibUSB driver on Windows 7 (German screenshots).

c) Click on "no, not this time" and press continue to prevent driver search on the web.

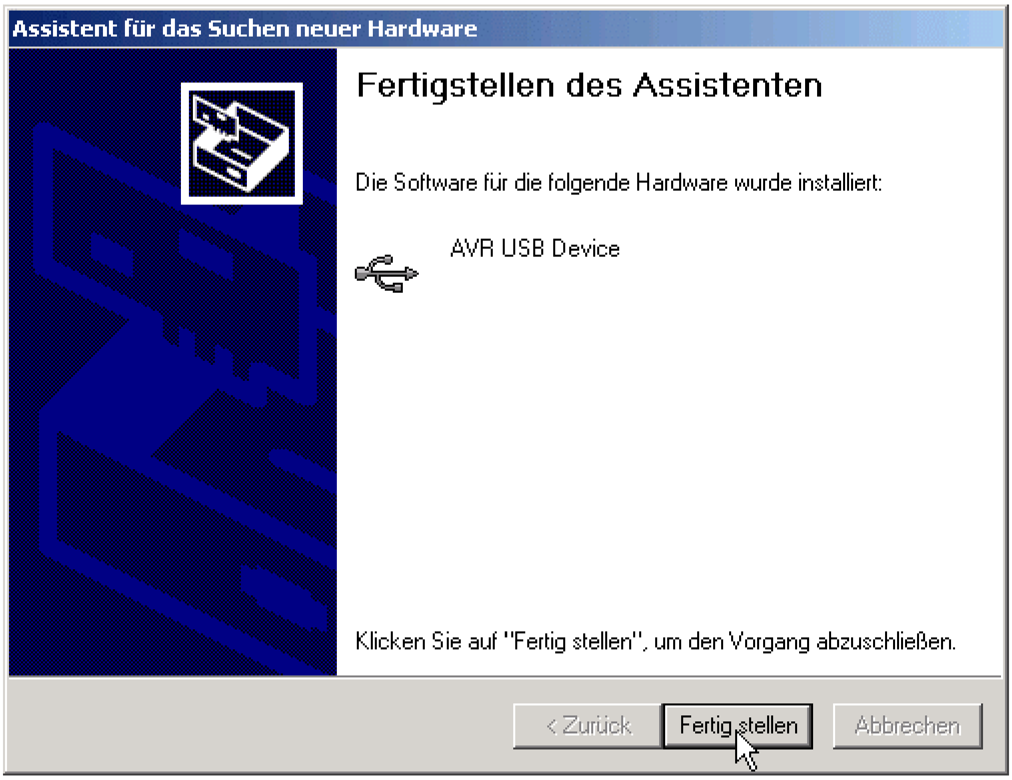
d) Click on "install from list..." and press continue to install the driver from your hard disk.



e) Enter the path to your driver files and press continue.



f) The driver will be installed. Finally you will be notified about the successful installation. Press "finish" and you can use the USB_VNWA interface on your computer.



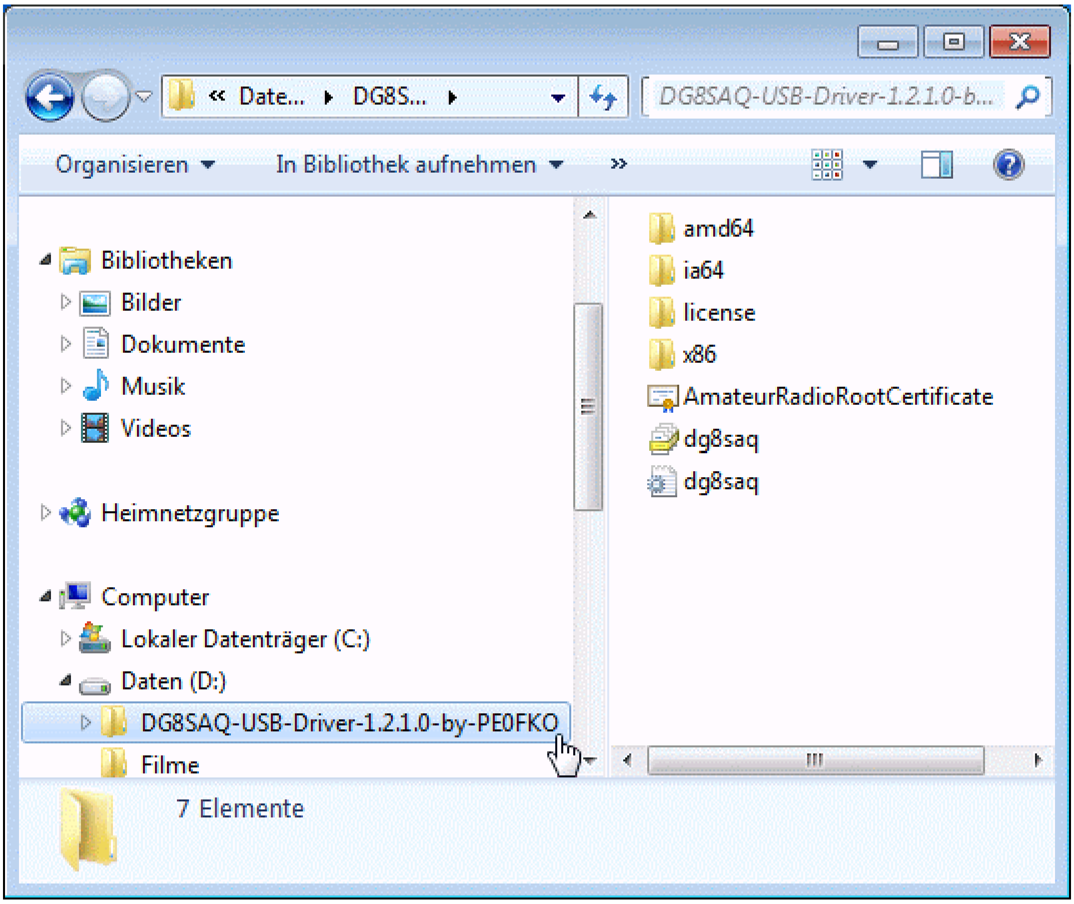
Now you are all set to install the VNWA application software.

▶▶▶ **Note:** This step is only required if the **DG8SAQ USB_VNWA** Interface is used. Skip this step if you control your VNWA via the parallel printer port.

▶▶▶ **Note: You need VNWA software version V33.0 or higher and firmware version v4.6 or newer to work with the signed LibUSB driver.**
See chapter **"Upgrading VNWA firmware to v4.6"** on how to upgrade your firmware.

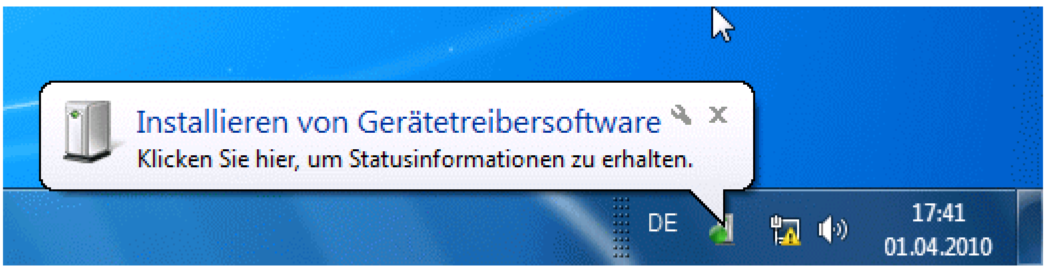
▶▶▶ **Note** The following describes a step by step installation procedure on a Windows 7 / Windows Vista machine, where no previous attempt has been made to install a VNWA driver. If you previously have tried to install different USB drivers for the VNWA **make sure to completely remove all previously installed driver relicts from your system**.

0. Download and unpack the LibUSB driver files to any empty directory your hard disk.

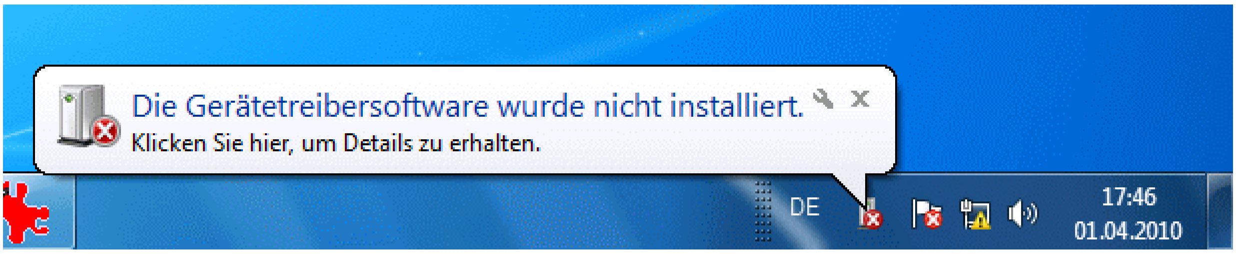


1. Connect your VNWA to a USB port of your computer

Windows 7 will attempt to install USB drivers for the VNWA ...

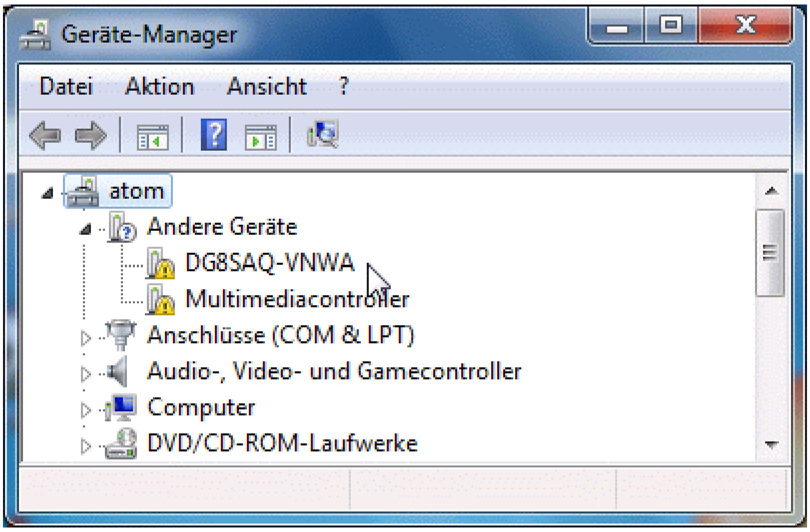


... but it will fail as it can't find any drivers.

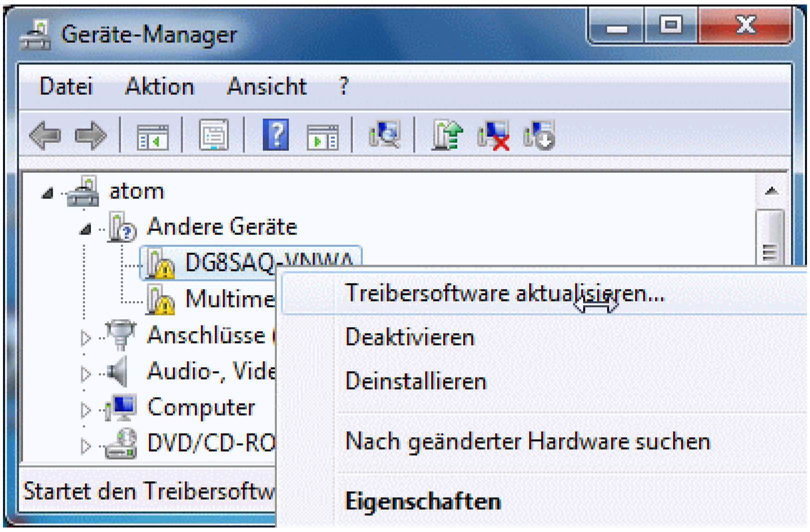


2. Open your hardware manager

You will see the VNWA as up to now non functional DG8SAQ-VNWA device under "other devices":



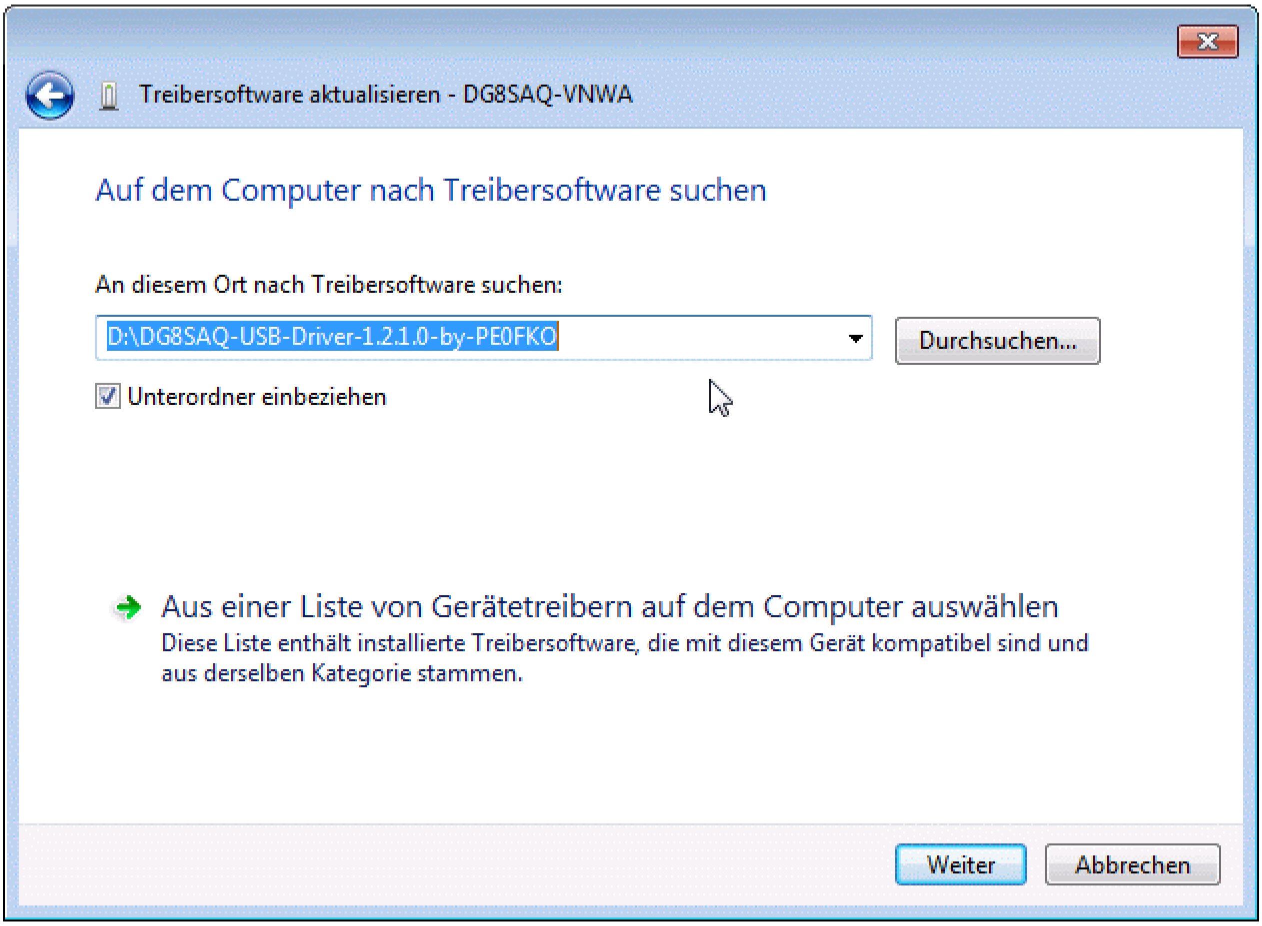
3. Right-click DG8SAQ-VNWA and select "update driver software"



Windows 7 will ask you how you would like to search for driver files.

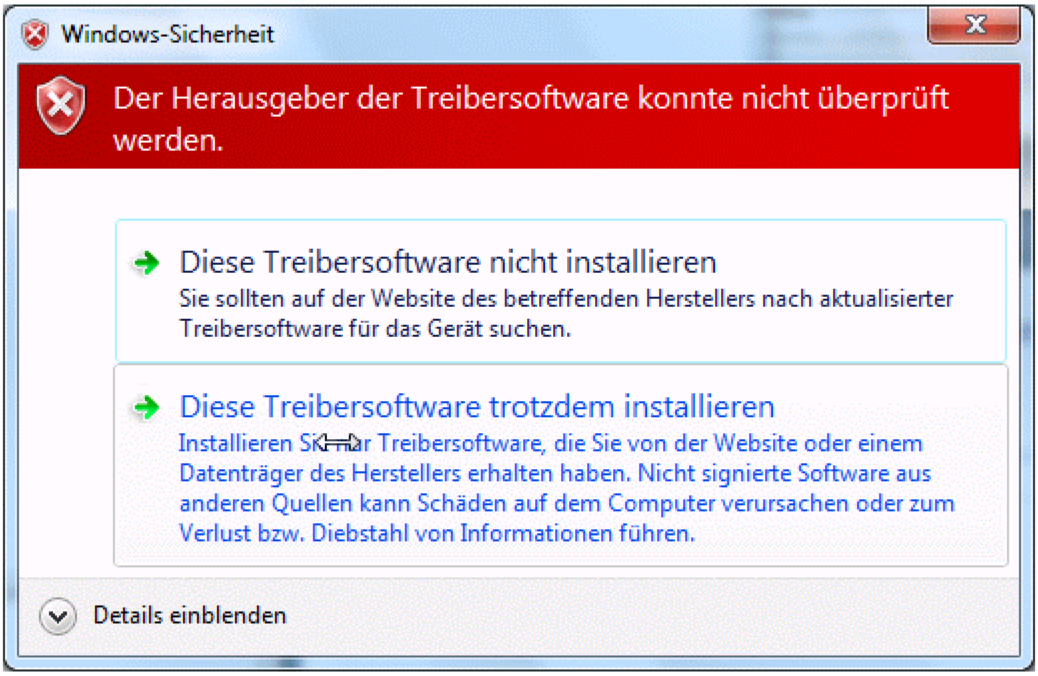
4. Select "search on computer for driver files"

5. Select the unzipped directory which contains the drivers...



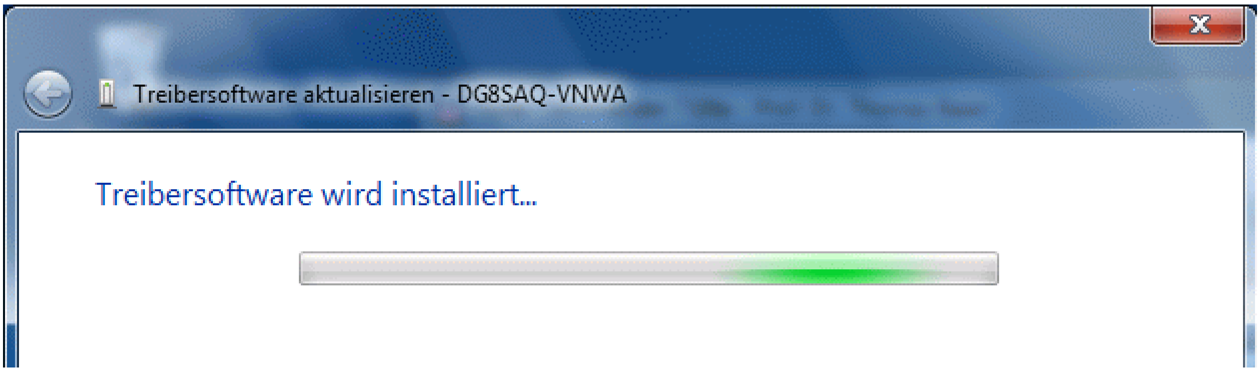
... and press continue.

Even though the USB driver is signed, Windows 7 will issue a security warning "publisher of driver software could not be verified". This is ok and no problem.
If you desire, the warning can be avoided by installing the root certificate prior to driver installation, but this is not required for installing and running the driver.

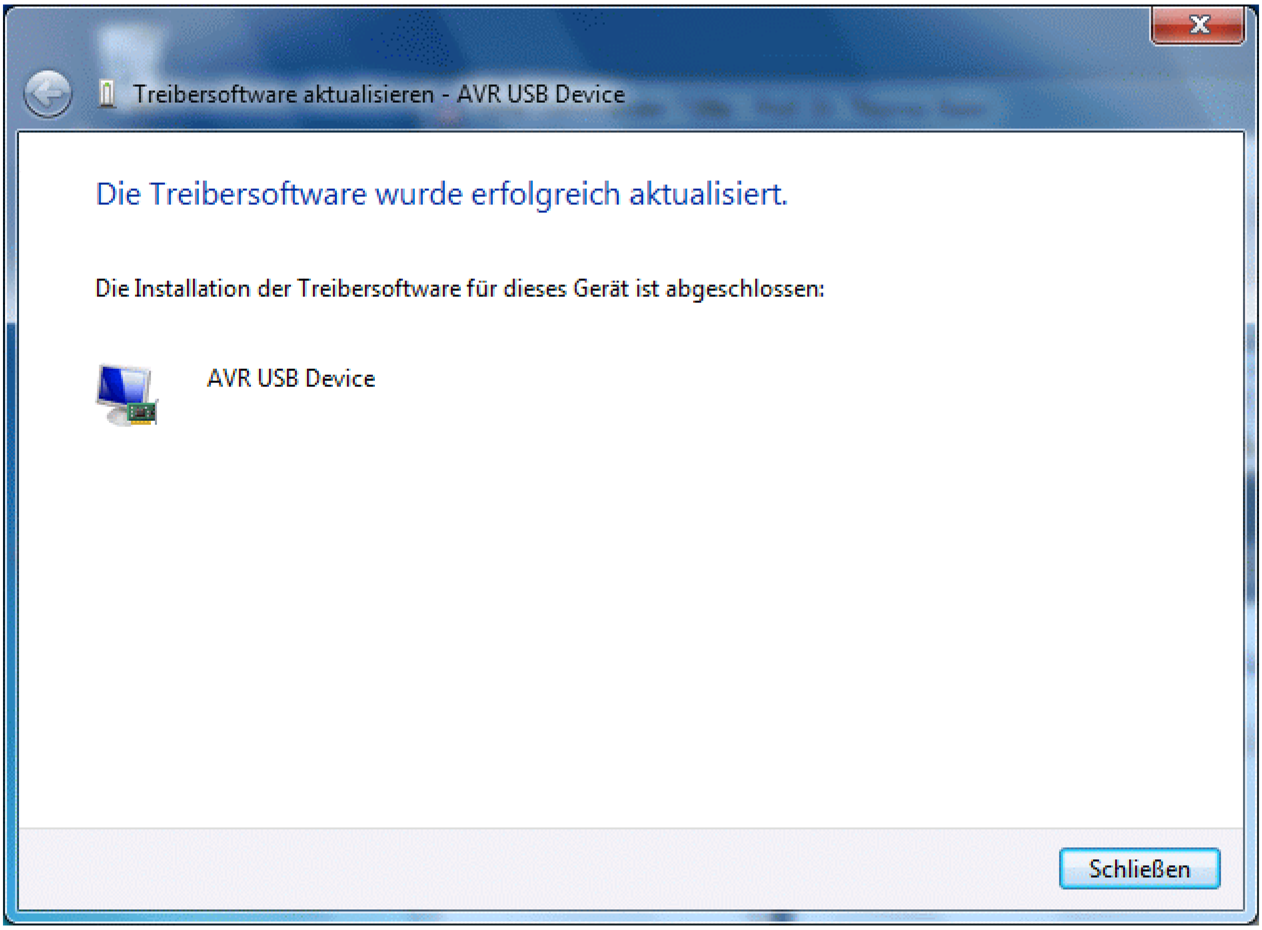


6. Select "install anyway ..."

... and Windows 7 will start installing the driver software.

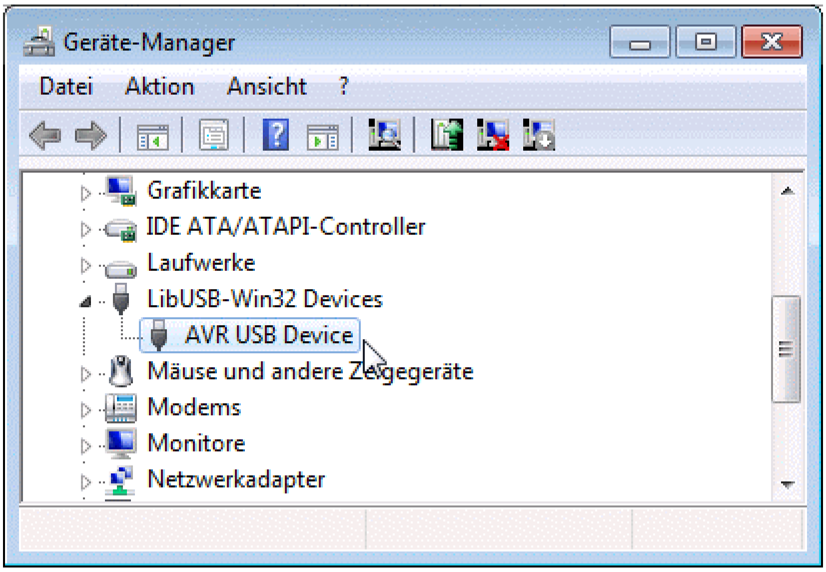


This might take a couple of minutes until done:



7. Press the close button

The USB driver is now installed and the VNWA device is operational as can be seen in the hardware manager, where the VNWA will show as AVR USB device:



Now you are all set to install the VNWA application software.

The VNWA hardware is controlled by the **VNWA application software**. Installing the software is a means of simply copying the file VNWA.exe to any directory you wish to use as program directory.
You can even install the VNWA application onto a memory stick which may remain attached to the extra VNWA USB port of your VNWA2. The advantage of doing so is that when moving the VNWA from one computer to another you also move all your settings and calibration files along.

▸▸▸ **Windows Vista / Windows 7 note:** On Windows Vista / Windows 7 the "program files" directory on the system partition is particularly write protected. Since the VNWA application writes many files (ini-files, calibration files,...) into the program folder, it is recommended to **NOT install the VNWA application into any system folder like "program files"** in order to avoid conflicts. Use any folder where you have normal write access instead.

Installation

You should start out with an empty program folder into which you unzip the VNWA application **VNWA.exe**, the LPT driver file **zlportio.sys** if LPT support is desired and the help file system **"VNWA.HLP"** and **"VNWA.cnt"**. No special software installation procedure is required as the files are ready to run. The VNWA application does not make use of the Windows registry.

▸▸▸ **Note:** If you have not installed a USB driver for the VNWA, then you must copy the file **libusb0.dll** from the LibUSB package into this directory.

▸▸▸ **Note:** You need to have **administrator privileges** in order to run the application in **LPT** mode. If you attempt to run it with lower privileges, the LPT driver will fail to load. The driver will also fail to load on systems that do not support the LPT driver zlportio.sys (e.g. Win98 and all 64 bit Windows versions).

▸▸▸ **Note:** You can **add an icon** to your desktop by right-clicking onto VNWA.exe and selecting *copy*. Then right-click onto your Windows desktop and select *add link*.

▸▸▸ **Windows Vista / Windows 7 note:** Please consult **"Configuring and running the VNWA application under Windows Vista or Windows7"** for special Vista / Windows 7 issues. Note, that Windows Vista and Windows 7 require that you set the VNWA audio device inside Windows to stereo operation and that you manually adjust the recording level.

Now you're ready to control your DG8SAQ VNWA. See the **configuration guide** for details

If you want to control an N2PK VNA with the VNWA application, then you need to create an additional file named **N2PK.ini** with arbitrary contents in the program folder. You can do so by e.g. typing *echo* > *n2pk.ini* on the console prompt. If you want to control your N2PK VNA via **Dave Robert's G8KBB USB interface**, you must provide the files **delphivna.dll** and **vnadll.dll** in your VNWA program directory. Configuration instructions for controling an N2PK VNA can be found here.

▸▸▸ **IMPORTANT:** For flashing, you **must use a VNWA version that supports the latest firmware version** to be flashed, e.g. only **VNWA33.q supports firmware v4,7 and newer**.

Generally, upgrading from one firmware version to another is very straight forward, see here for a detailed description.
Transition from firmware version v4.2 to v4.6 and from 4.6 to 4.8 is a bit more complicated, as some sections of the firmware need to be reprogrammed, that are by default write-protected. Therefore, several codes need to be uploaded in the correct order that make the MCU reprogram protected sections AFTER code upload. This not quite straight forward upgrade procedure is described in the following.

▸▸▸ **Note:** All **calibration files will remain valid** also after the firmware update! There is no reason not to do this firmware update.

First, a summary of the necessary steps:
- flash v4.3
- powercycle USB interface
- rescan USB bus
- flash v4.5
- close VNWA application
- powercycle USB interface
- install new USB driver
- flash v4.6
- powercycle USB interface
- rescan USB bus
- flash v4.7 (non functional but required for uploading v4.8)
- powercycle USB interface
- rescan USB bus
- flash v4.8
- powercycle USB interface
- rescan USB bus
- flash v4.9
- powercycle USB interface
- rescan USB bus

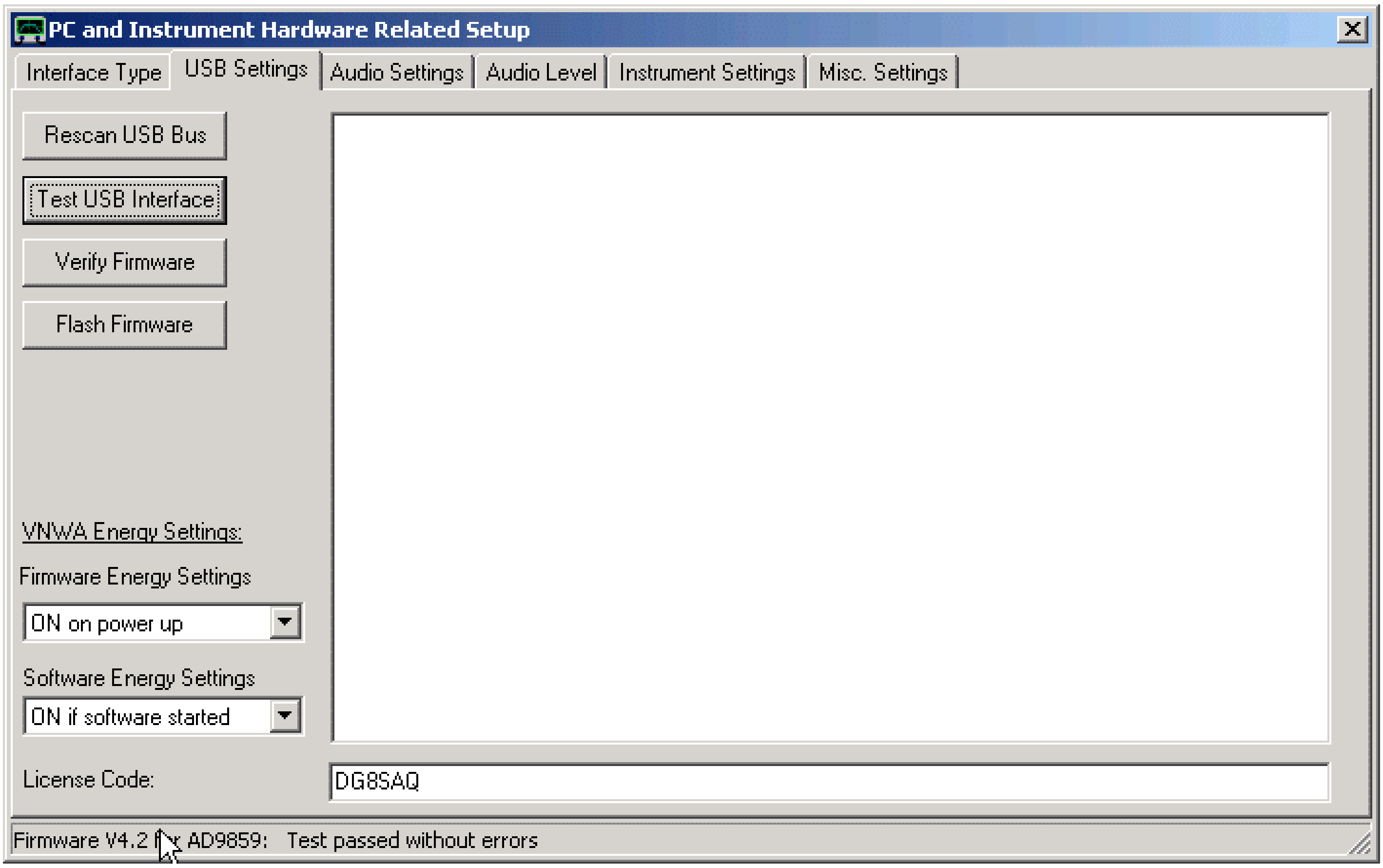
Note, that flashing of v4.4 may be skipped. We go straight from v4.3 to v4.5.

DETAILED DESCRIPTION, READ CAREFULLY:

▸▸▸ **Note: You need a working VNWA installation in order to do this firmware upgrade!**
▸▸▸ **Note: You need VNWA software version V33.0 or higher to perform this upgrade and to use firmware v4.5 or newer.**
▸▸▸ **Note: The upgrade from v4.2 to v4.6 cannot be undone with the USB flashing utility of VNWA. Nevertheless we recommend this upgrade for all 32 bit OS users, too, because ...**
▸▸▸ **Note: ... future firmware upgrades will require v4.6 in order to be loadable via USB.**
▸▸▸ **Note: Firmware V4.5 will require a reinstallation of the USB driver.**

1. Check what firmware version you are currently running.

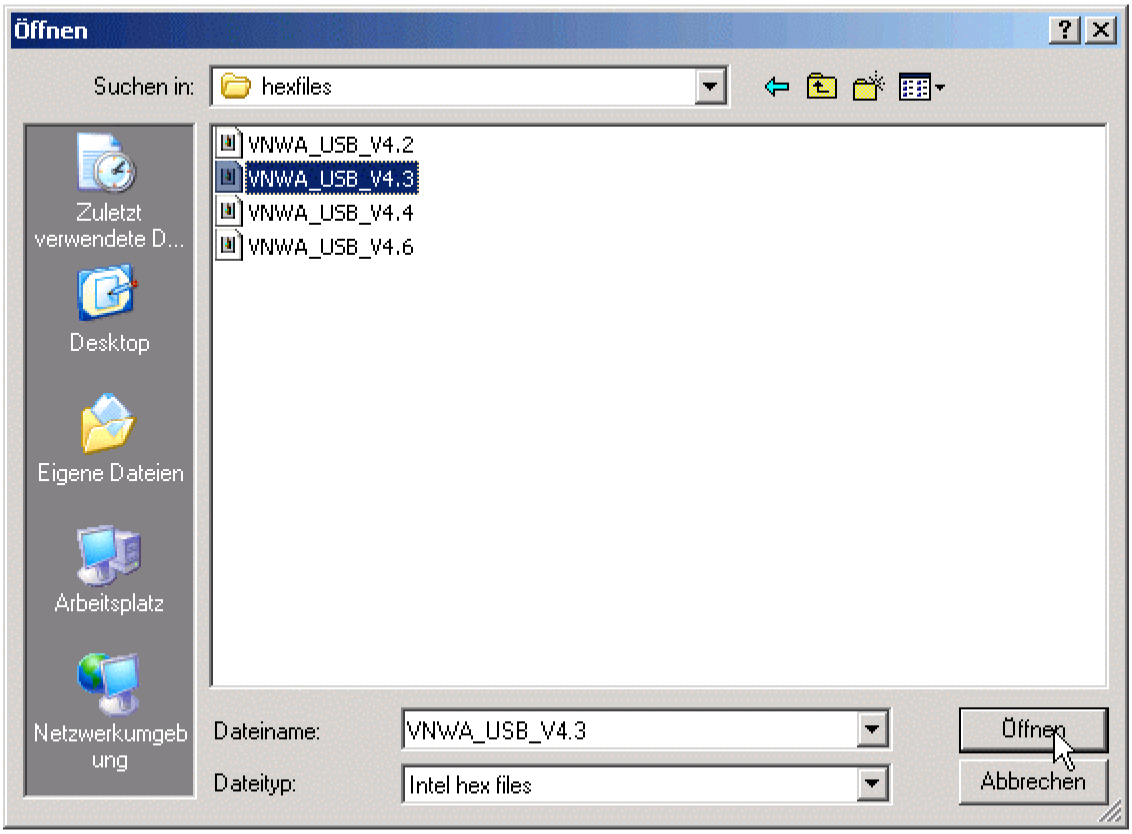
You do this by using the "**Test USB**" feature in Setup-USB Settings. The result will be displayed in the bottom status line:



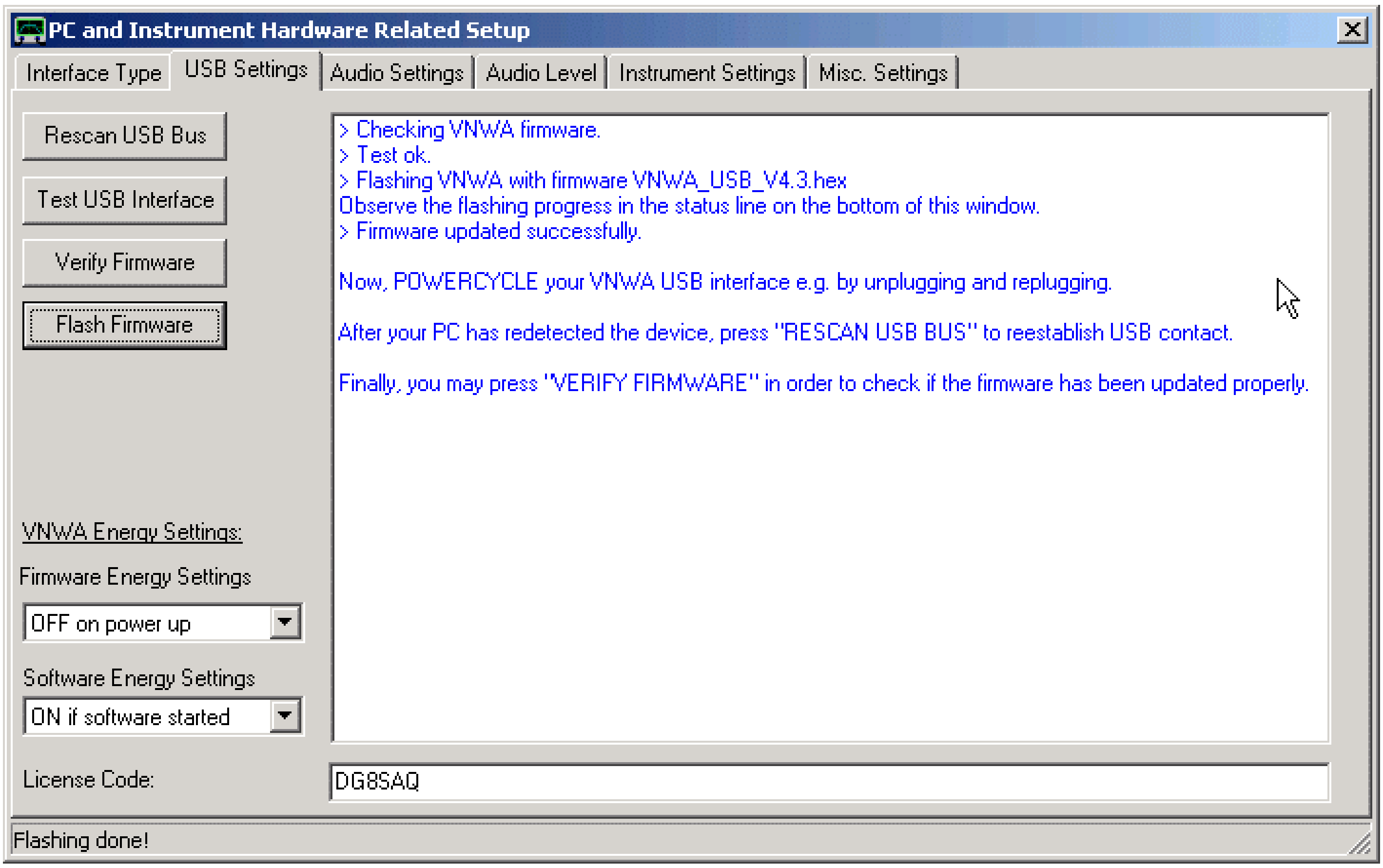
It can be seen in the bottom status line (see mouse pointer above), that firmware version v4.2 is running. If your firmware version is newer, then move on to the appropriate step.

2. Flash firmware v4.3.

Press the "**Flash Firmware**" button and load firmware v4.3:



After pressing the "**Open**" button, you can observe flashing progress in the bottom status line of the setup window. After the flashing process is over, you will see further instructions on the screen:



a) **Powercycle** the USB interface in order to generate a reset condition for the AVR controller. This is necessary to trigger some further internal reprogramming action of v4.3.
b) **Rescan USB Bus**. This is necessary to reestablish contact between the VNWA application and the USB interface after the hardware reset in a).

▶▶▶ **Note:** When you do "**Verify Firmware**" after powercycling and rescanning the USB bus, you will see an error message:

This is perfectly ok and proves that the firmware has modified itself after the power reset. Now it is time to

3. Flash firmware v4.5

This is done like described in step 2. Note, that we are skipping v4.4.
▶▶▶ **Note:** v4.5 will again modify some parts of the USB core routines after power reset. Therefore ...

4. Completely close the VNWA application and powercycle the USB interface e.g. by unplugging and replugging your VNWA.

This done, **Windows will recognize the VNWA as a new device** and will ask for a new driver to be installed.
As you are NOT doing this firmware upgrade on a Windows 7 64bit machine, you should reinstall the **LibUSB driver** now.
Consult the driver compatibility tables in order to gather information on how to install the LibUSB-driver on your system.

After successful driver installation, **start VNWA again**.
▶▶▶ **Note: You need VNWA software version V33.0 or higher to use firmware v4.5 or higher.**
Go to "Setup-USB Settings". If you try to verify firmware v4.5, it will result in an error message like for v4.3. Again, this is perfectly ok and proves that the firmware has again modified itself after the last power reset. Now you are set to ...

5. Finally flash firmware v4.6

This is the final firmware version that will no longer reprogram itself and allows the full VNWA functionality. This is also the **minimum firmware requirement to use your VNWA on the 64 bit versions** of Windows Vista or Windows 7.

6. Flash firmware v4.7, power-cycle VNWA and press "Rescan USB Bus"

A warning will pop up stating **"firmware non functional, flash with functional firmware"**. This is perfectly ok. V4.7 cannot operate the VNWA, but it is necessary to load v4.8.

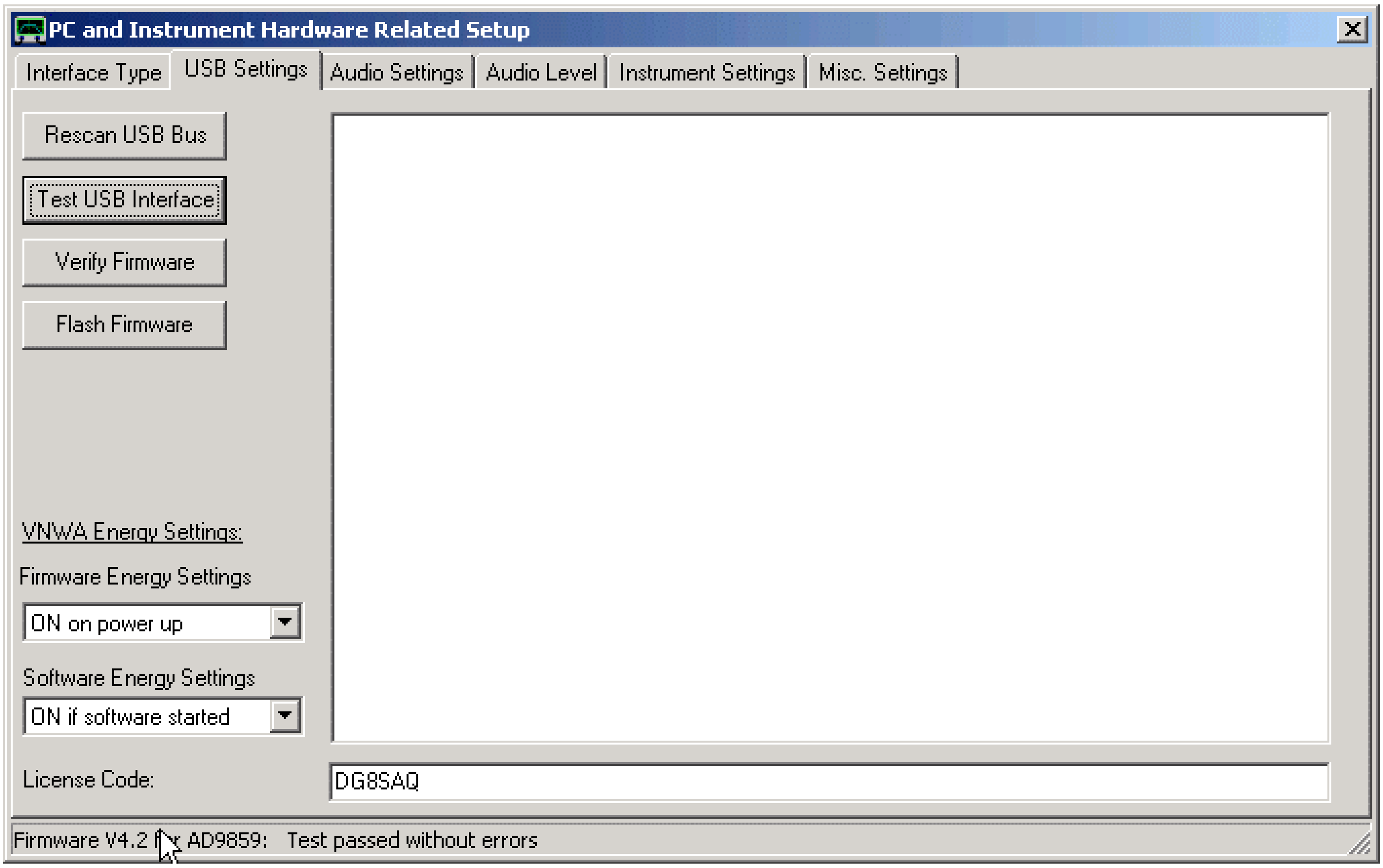
7. Flash firmware v4.8, power-cycle VNWA and press "Rescan USB Bus"

8. Flash firmware v4.9, power-cycle VNWA and press "Rescan USB Bus"

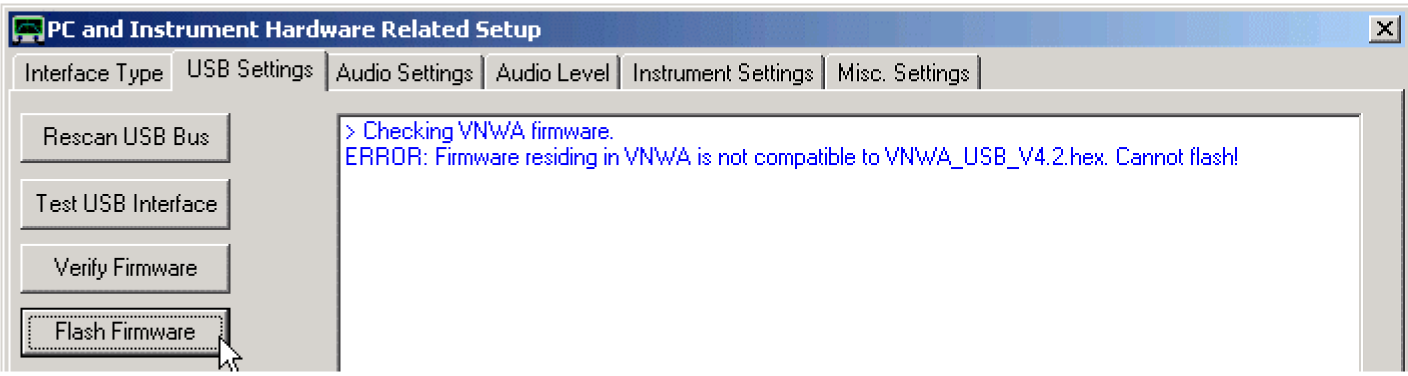
▸▸▸ **Note:** The flashing procedure is very safe. Nevertheless **avoid power failures during flashing**!
▸▸▸ **Note:** All **calibration files will remain valid** also after the firmware update! There is no reason not to do a firmware update if a new firmware is available.

1. Check the firmware version you are running:

You do this by using the "**Test USB**" feature in Setup-USB Settings. The result will be displayed in the bottom status line:

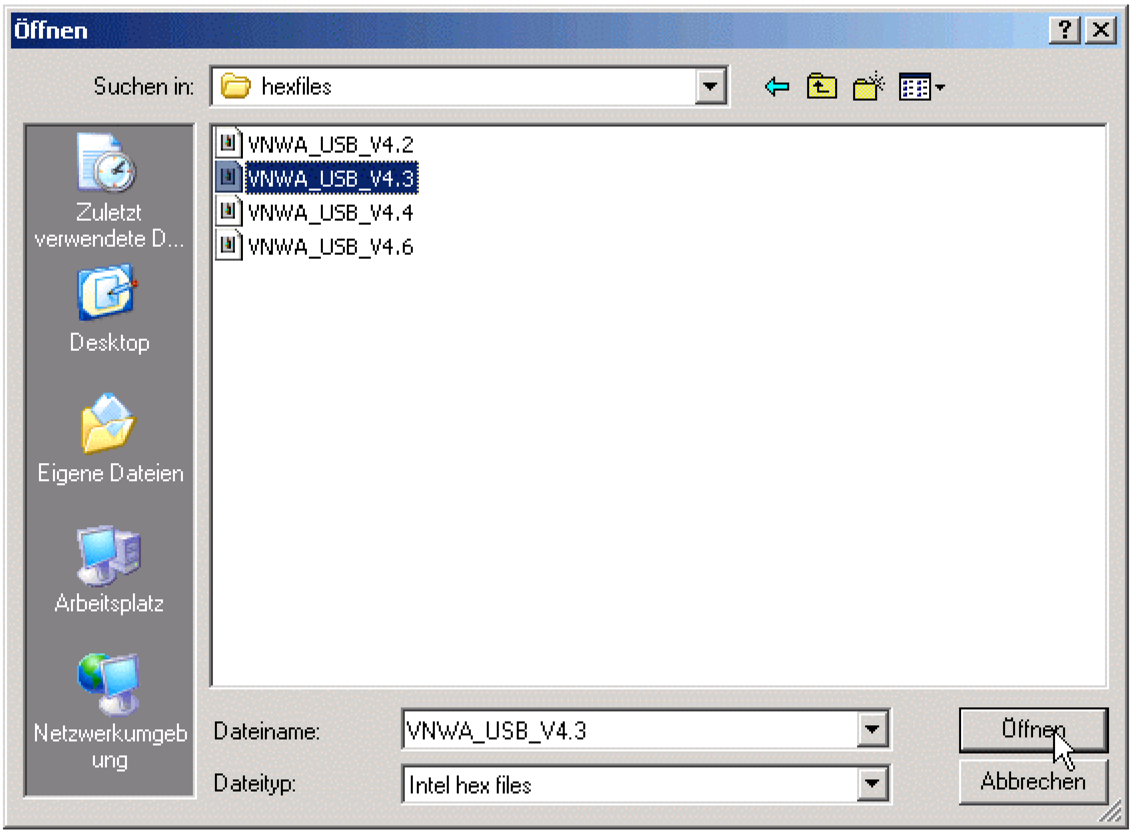


It can be seen in the bottom status line (see mouse pointer above), that firmware version v4.2 is running.
▸▸▸ **Note:** Not all firmware versions are compatible with each other, i.e. not every firmware version can be overwritten with any other firmware version. Basically, you have to do the upgrades in the order of the firmware version numbering. If you attempt to flash an incompatible firmware, you cannot damage anything, but you will see the following error message:

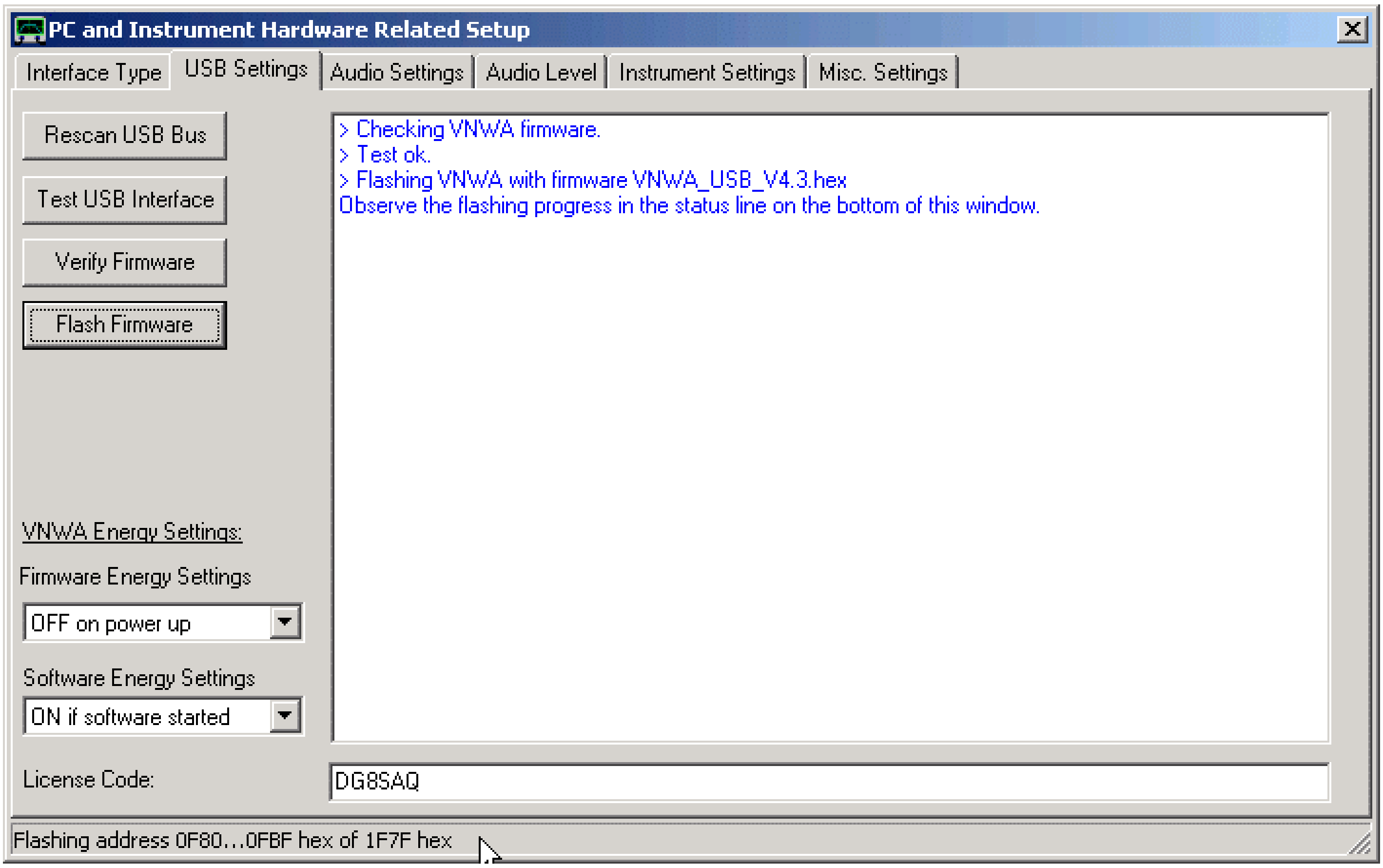


2. Flash new firmware:

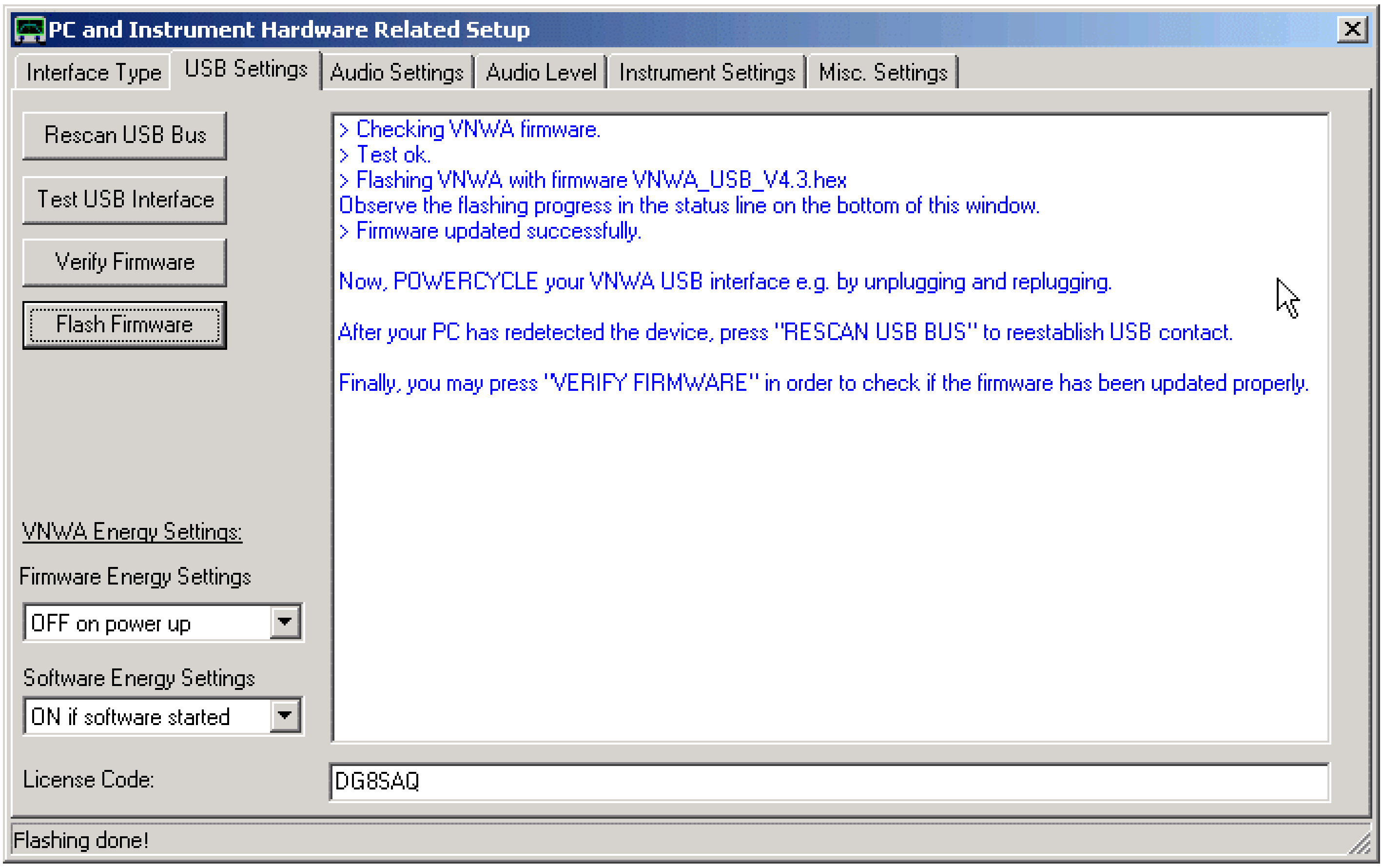
Press the "**Flash Firmware**" button and select the desired firmware version:



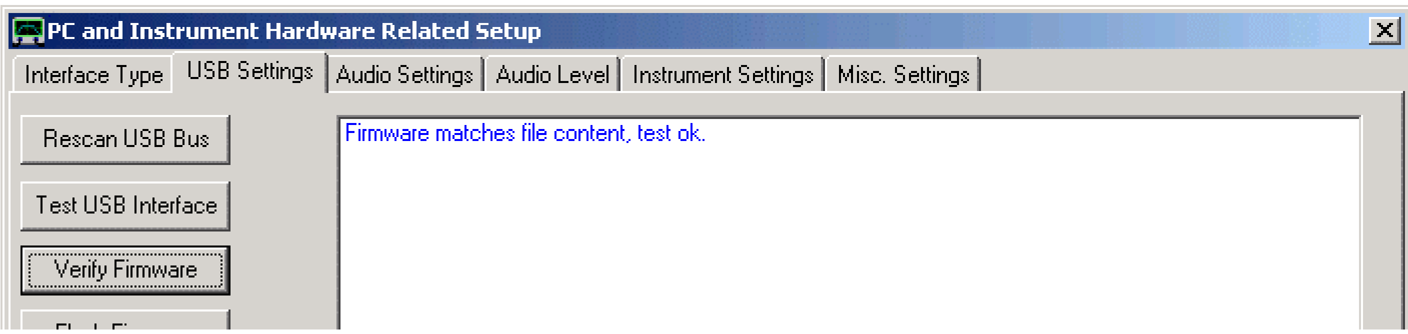
After pressing the "**Open**" button, you can observe flashing progress in the bottom status line of the setup window:



After the flashing process is over, you will see further instructions on the screen:



a) **Powercycle** the USB interface in order to generate a reset condition for the AVR controller. This is necessary to grant stability. Some firmware versions (v4.3, v4.5) also need this hardware reset to trigger some further internal reprogramming action.
b) **Rescan USB Bus**. This is necessary to reestablish contact between the VNWA application and the USB interface after the hardware reset in a). Closing and reopening the VNWA application will do the same job but can be avoided by rescanning the USB bus.
c) Optionally, you can verify the firmware update by pressing the "**Verify Firmware**" button and selecting the file that is to be compared to the firmware residing in the VNWA interface.
Successful verify will produce this output:

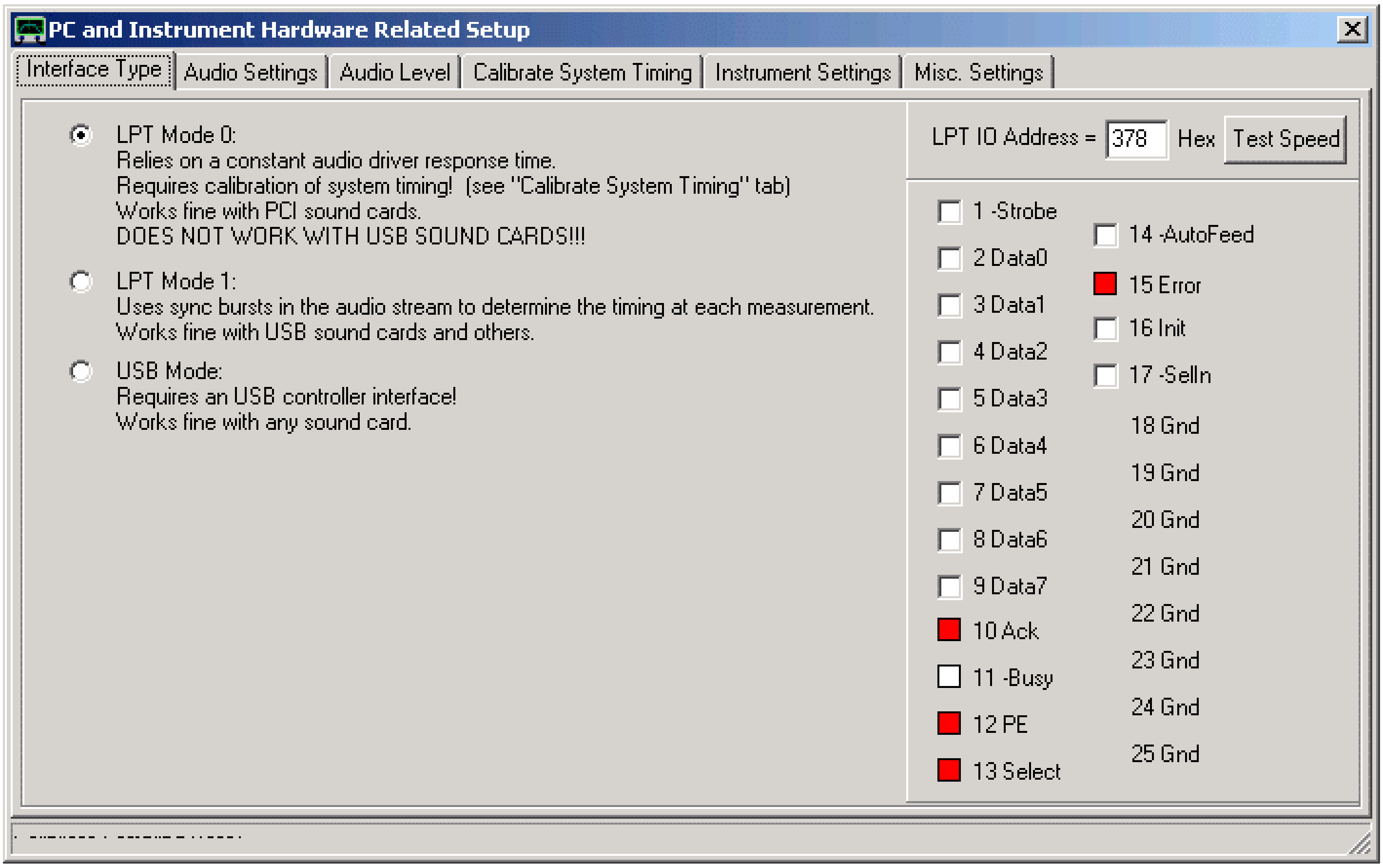


▸▸▸ **Note: Some firmware versions cannot be verified** after flashing (they will produce a compare error), because they will reprogramm themselves after power cycling. Todate these are v4.3 and v4.5.

▸▸▸ **Note: Firmeware version v4.5 requires installation of a new USB driver.**

Starting the Software, Choice of Interface

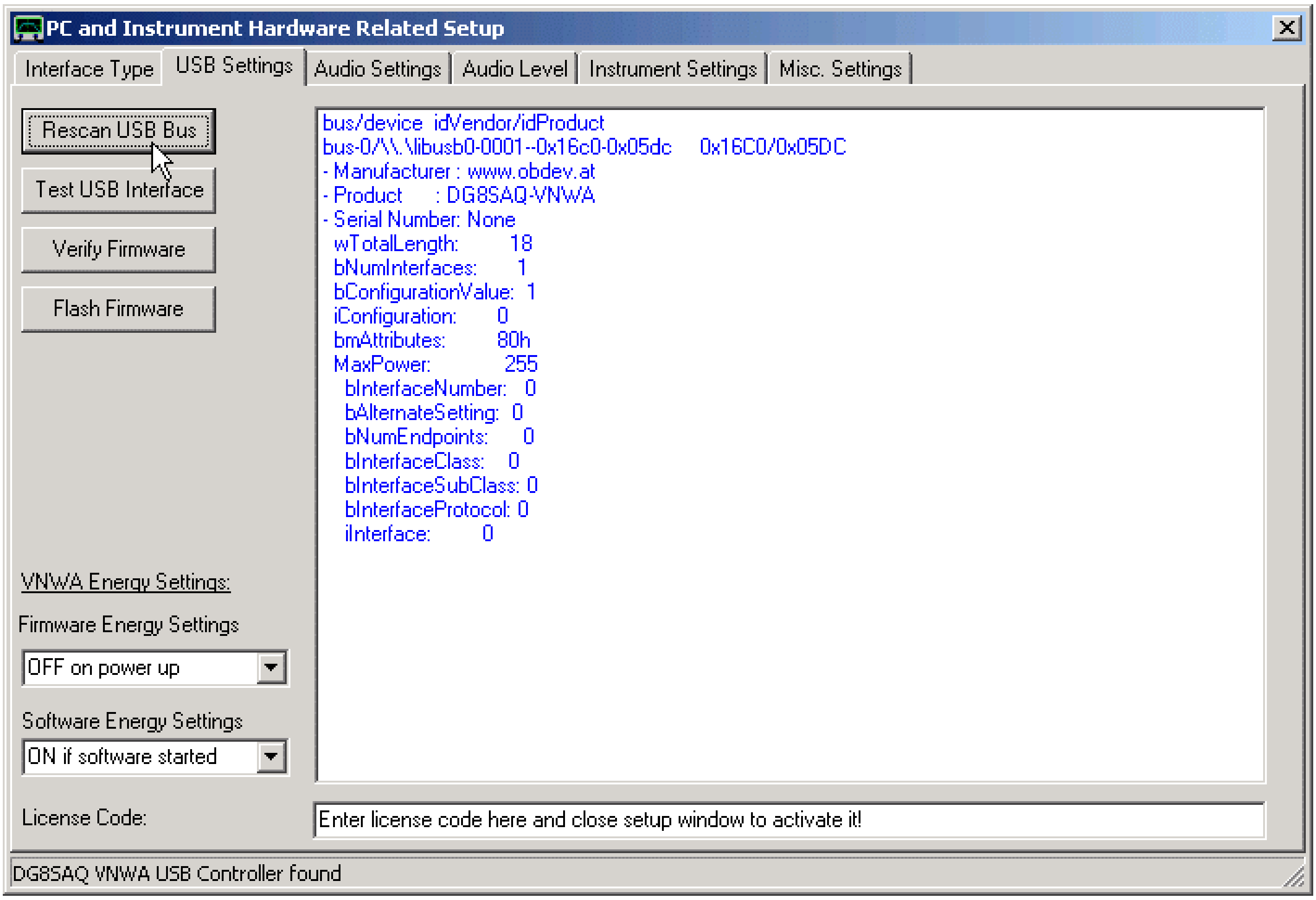
To start the VNWA software, double-click VNWA.exe. Look here for special issues on Windows Vista.
On program start two windows should open.One of looks like this:



Check the desired interface type. Make sure to enter the correct address of your LPT port (378hex is the most common choice, see your PC BIOS). The button "Test LPT" allows you to evaluate the speed of your LPT port. You should obtain values between about 1us and 2us per byte. This is no functionality test of the LPT port hardware, though! You can actually test the LPT hardware with the feature to the right. You can toggle the states of the LPT lines and observe the changes with a scope. You can also make a loopback connector and observe the changes on the input lines.
▸▸▸ **Note: USB to LPT converters won't work** for this application due to latency of the USB transfer protocol.

Before you select *USB Mode*, the USB_VNWA interface must be plugged in!
If you select **USB Mode**, the LPT test features disappear and the **USB Settings tab** appears.

USB Settings (only applicable to the USB_VNWA interface)



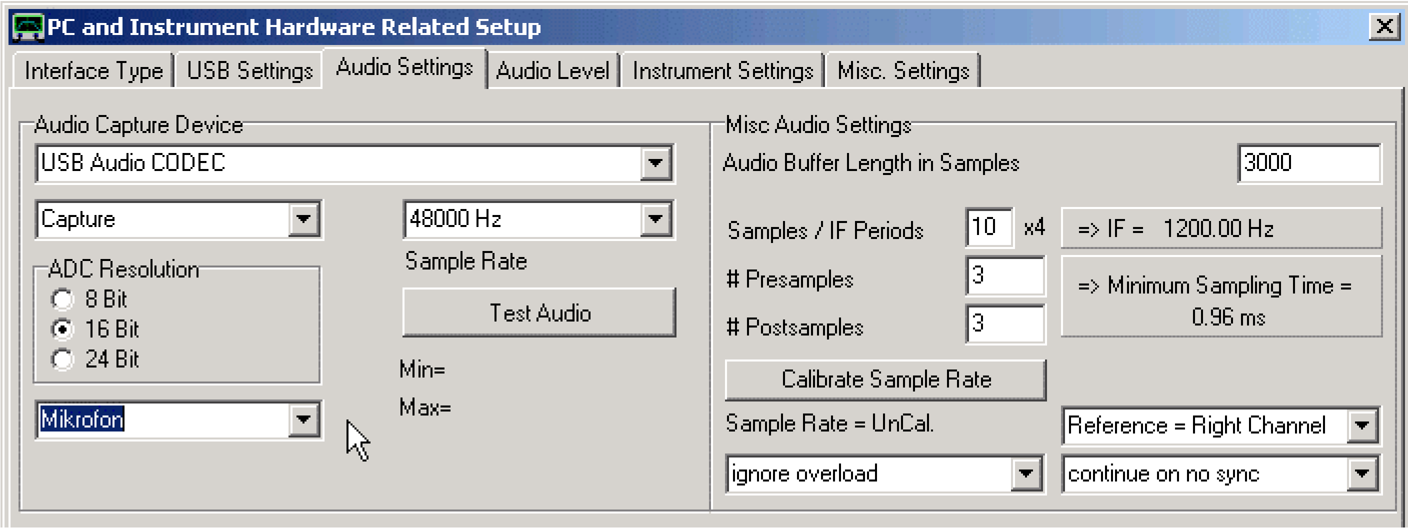
Press **Rescan USB Bus** to see if the USB driver and the USB_VNWA interface are being properly detected. You should see the above screen in this case.

▶▶▶ **Note:**Before continuing, you must enter the **license code** you have received together with your USB_VNWA interface into the bottom field. After this **you must close and reopen the hardware setup** window in order to activate the license key!

After activation you can press **Test USB Interface** to verify flawless communication.

Audio Settings

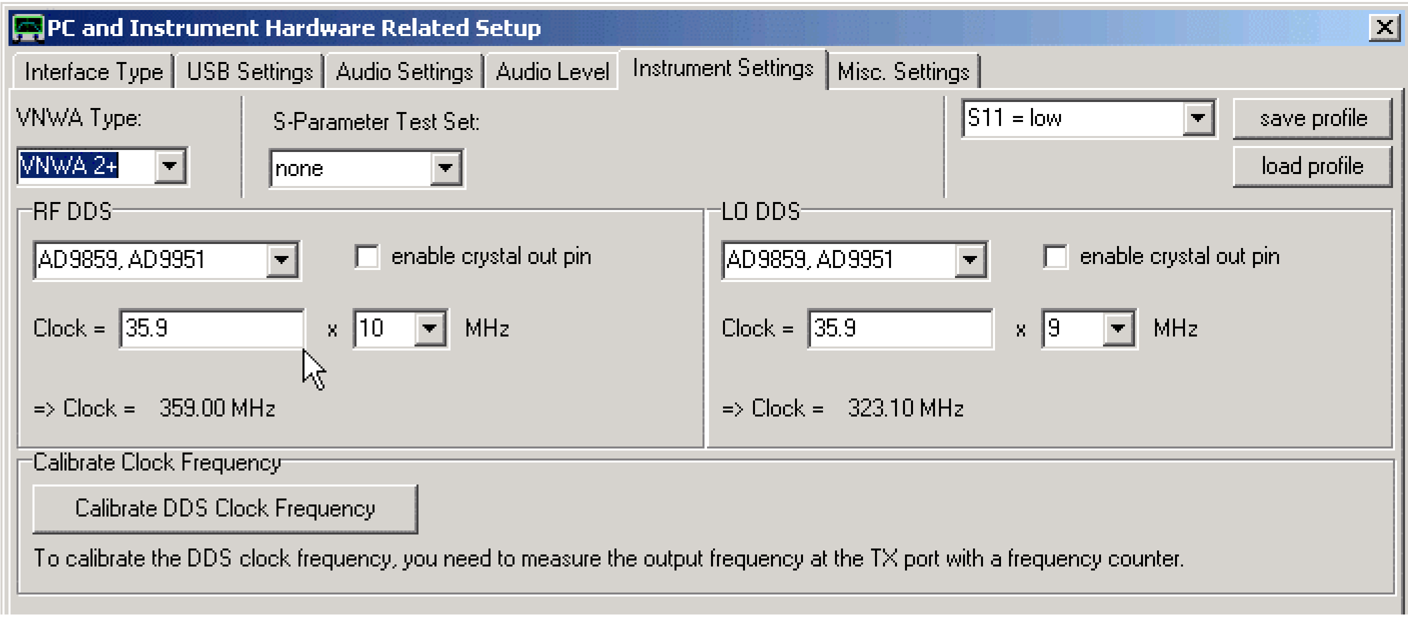
Next move on to the **"Audio Settings"** tab:



Select your prefered audio capture device on the left. 48kHz is a good choice of a sampling rate, which is supported by all modern sound cards. Lower sampling rates may be advantageous for very low frequency measurements, but limit the sweep speed. The indicated "Misc Audio Settings" are good choices. If you have a slow CPU you might want to increase the Audio Buffer Length. This will lead to fewer graphics updates.

Instrument Settings / External Hardware

Next move on a few tabs further to **"Instrument Settings"**:

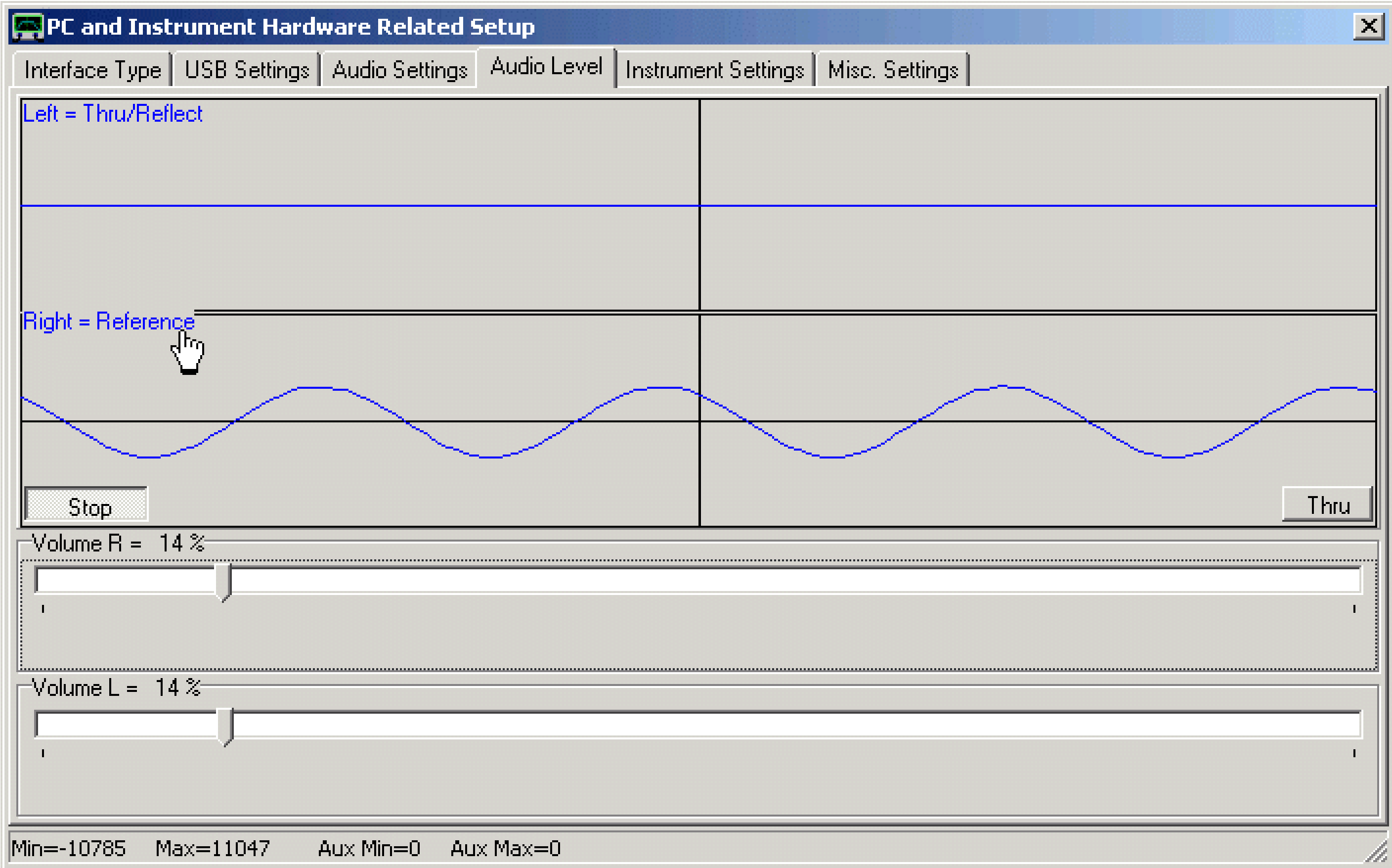


Here you can specify the hardware inside your VNWA. If you have a different crystal frequency, modify the data according to your hardware. The displayed frequency of 35.9 MHz is approximately obtained if a standard 12 MHz crystal is used in 3rd overtone mode. If you want to calibrate your DDS clock oscillator frequency, press the according button and follow the instructions. You need a frequency counter to do so. Note, that the two frequency multipliers need to be as high as possible and different from each other. If you want to use the frequency range 600 MHz...1.3 GHz, you need to select multipliers **auto**. When you do so, the prefilled auto clock multiplier table pops up. Simply close it as the default values should be ok. Now your PC is ready to send control data to your VNWA board and it is time to connect it to the PC, if you haven't done so already.

▸▸▸ **Note:** With the displayed clock multiplier selections you stay within the Analog Devices DDS specification, but you can only measure up to about 300 MHz.

Adjusting the Audio Level (no adjustment available for the DG8SAQ USB_VNWA interface or for Windows Vista)

Move on to the **"Audio Level"** tab and press the button "Test Audio". You might want to increase the window size for better visibility. You should see the following screen:

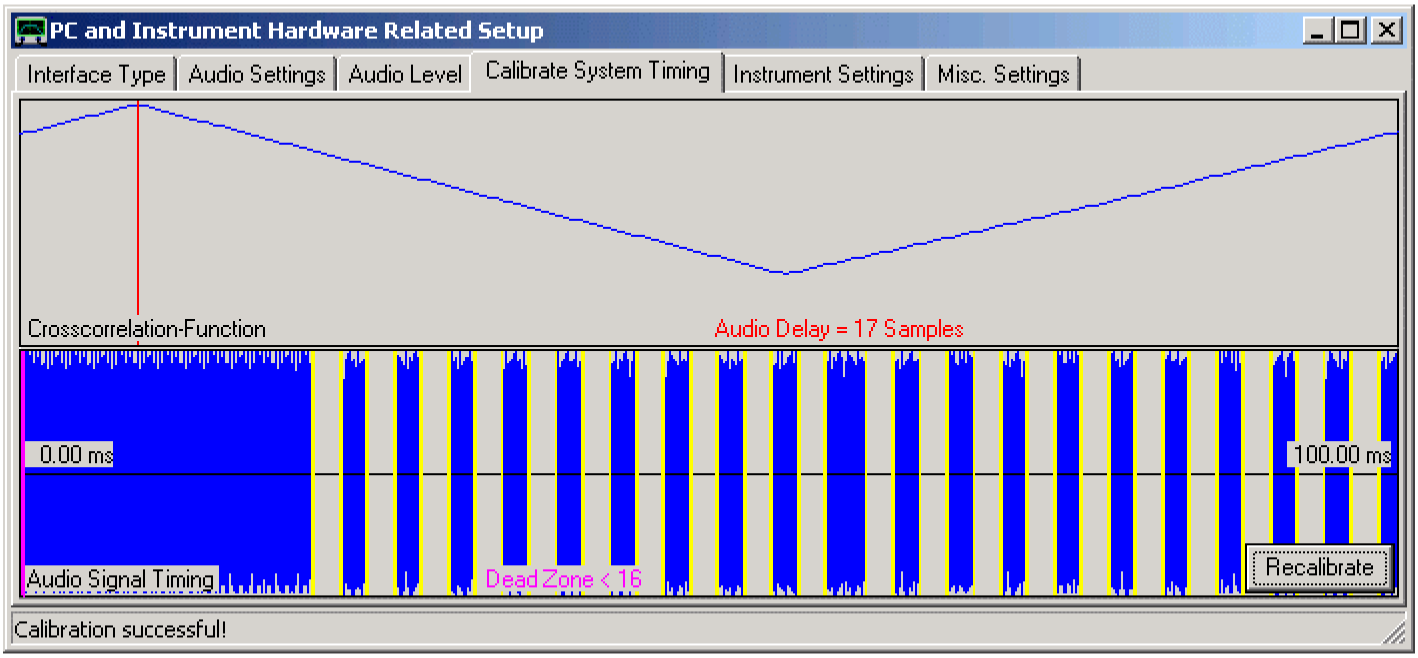


Left and right audio channels are monitored while the VNWA is set to a fixed frequency. On the above screen, you can see that the reference signal arrives on the right channel (bottom curve, if it's the left channel in your case, make sure to change the setting in the Audio Settings/right block accordingly). Set the volume slider for the reference channel such, that by no means clipping occurs. A good rule of thumb is to set the volumes such, that the amplitude spans 50% of the maximum amplitude. The left channel (top) shows a flat line, as the Thru signal is zero while the VNWA Tx and Rx ports are disconnected from each other. Press the right lower button showing "Thru". It will change to display reflect. At the same time, you'll see the Reflect signal on the left channel (top). Adjust the level with the according slider as for the reference signal. If you connect a short or a load resistor to the TX port, you can observe how the amplitude and the phase of the reflect signal changes. If you now connect the TX port with the RX port via a coaxial cable, the reflect signal should disappear, as the 50 Ohms input resistance of the RX port does not reflect RF power. If you press the bottom right button again to show "Thru", you should see a strong thru signal instead. If its amplitude appears too large, reduce it with the according slider. Don't increase the amplitude at this time, as you woud also increase the reflect signal amplitude, that you have already adjusted.
▸▸▸ **Note:** No sliders will be available for the DG8SAQ USB_VNWA interface or for Windows Vista!

If you are done, press the **"Test Audio"** button again to stop sampling. Note, that the other tabs are deactivated while the sound device is capturing! Also, you cannot close the setup window while sampling.

Audio Driver Timing Calibration (LPT Mode0 only) and Sample Rate Calibration

If you have selected interface type "LPT Mode 0" in the beginning, you need to calibrate the sound driver latency next. In this case move on to tab **"Calibrate System Timing"** (for "LPT Mode 1" you can skip this step). Press the button "Recalibrate". You should see the following picture:



Here, the DDSes are switched on and off and the data stream is analyzed to recover the switching moments. The vertical yellow lines should exactly match the changes in the blue audio stream. The cross correlation function (top) should have a sharp maximum. In the above example, an audio delay of 17 samples is determined. Press the recalibrate button a few times and observe the calculated audio delays. If they are constant to within about +-1 sample, then this operation mode is feasible for your PC. If not, then you need to run "LPT Mode 1". The latter is the case for all USB audio devices.

Next, you need to calibrate the sound card sampling rate. Move back to the tab **"Audio Settings"** and press the button "Calibrate Sample Rate" in the right lower half of the window. A blue progress bar should appear for about 30 seconds. If the calibration is successful, you should see the determined sampling rate just below the button. It should be very close to the one specified in the according selection box. The VNWA must be connected and running in order to perform this calibration.

Master Calibration Filename and Debug Settings

You may want to inspect the last tab **"Misc. Settings".**

Here, you may enter a filename, which is used to store an instrument master calibration, which will be loaded and interpolated automatically, in case no calibration for the actual sweep settings has been performed.
DO NOT TOUCH THE DEBUG SETTINGS UNLESS YOU EXACTLY KNOW WHAT YOU ARE DOING! All debug settings need to be unchecked for proper VNWA operation.

Now close the setup window and close the main window by pressing the buttons. Closing the main window will cause the entered data to be stored in ini-files, which you will now find in your VNWA programm directory.

This setup is required only once at first program start. On the subsequent program starts, the necessary data will be read from these ini-files and the setup menu will not be entered automatically any more. But you can still enter and modify the setup manually via the menu "Options-Setup".

The following procedures apply to Vista and Windows7 32 bit and 64 bit operating systems. For 64 bit operating systems a special set of USB-driver is required, though. Please consult the **"Driver Compatibility Tables and Driver Installation"** page for details.

▸▸▸ **Help system note:** When you see this text on a Vista or Windows7 machine, you have already figured out how to **configure your OS to support old helpfile formats**. By defaullt Vista and Windows 7 do NOT support *.hlp help files.

▸▸▸ **LPT note:** In order to use LPT mode on 32 bit Vista or Windows 7, you must run the VNWA application with administrator rights. See here for details. Note, that you **cannot use LPT mode on 64 bit OSes**, as the LPT drivers used by VNWA are not digitally signed by Microsoft.

▸▸▸ **USB audio codec note:** If you use the built in audio codec of the **DG8SAQ USB_VNWA interface**, you must **manually configure the audio codec for stereo operation**.

Vista Help File Fix

In order to view this helpfile on a Vista machine, you need to download and install the appropriate **WinHlp32.exe** for your operating system.

See http://support.microsoft.com/kb/917607 for details!

Windows7 Help File Fix

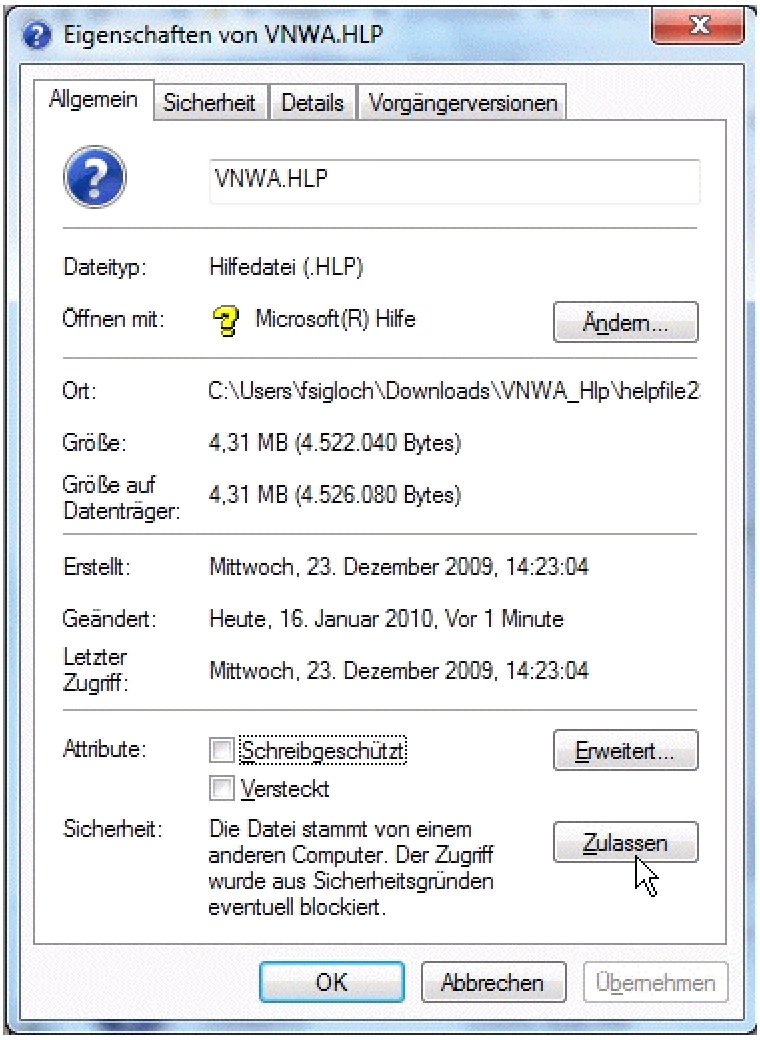
Finally, Microsoft has provided a fix for Windows7 also to support this help file format. In order to view this helpfile on a Windows7 machine, you need to download and install the appropriate **WinHlp32.exe** for your operating system.

See Windows Help program (WinHlp32.exe) for Windows 7 for details!

An alternate workaround is possible: You need to get the file **WinHlp32.exe** from a WindowsXP machine and copy it to your VNWA software directory. Next, modify VNWA.ini with a text editor by changing the line "HelpByExe=0" to "HelpByExe=1".
This done, the helpfile will open normally from within VNWA.
This fix will also work for Vista in combination with VNWA, but it will not fix help file problems for third party software.

Vista and Windows7 Security Settings to allow to open VNWA.hlp

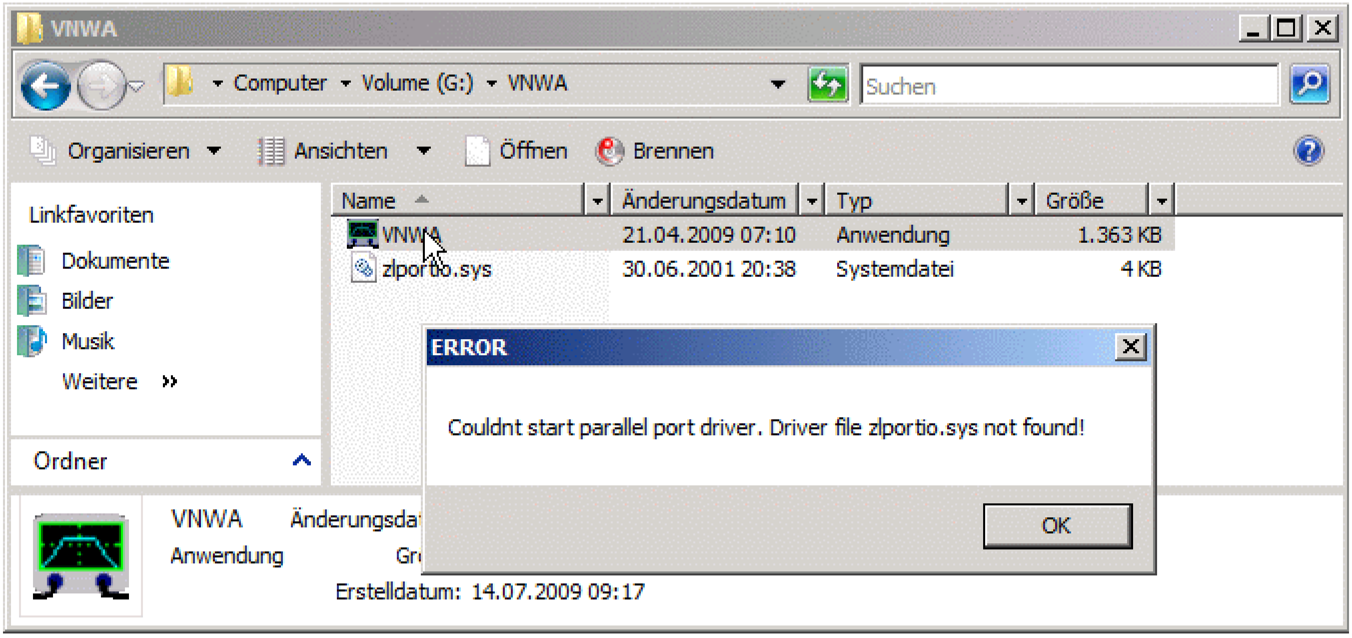
Even after having installed the Microsoft help fix you might not be able to open the helpfile with WinHlp32. In this case, the help file is blocked by Windows for security reasons. In order to qualify VNWA.hlp as safe, **right-click VNWA.hlp** and select **properties**. You will see the following window.



At the very bottom you see the item **Security** and a statement telling you file access is being blocked. **Press the release-button** right next to it (see mouse pointer).
You might have to repeat this step with the accompanying content file **VNWA.cnt**.

LPT mode for Vista or Windows 7 (32 bit only!)

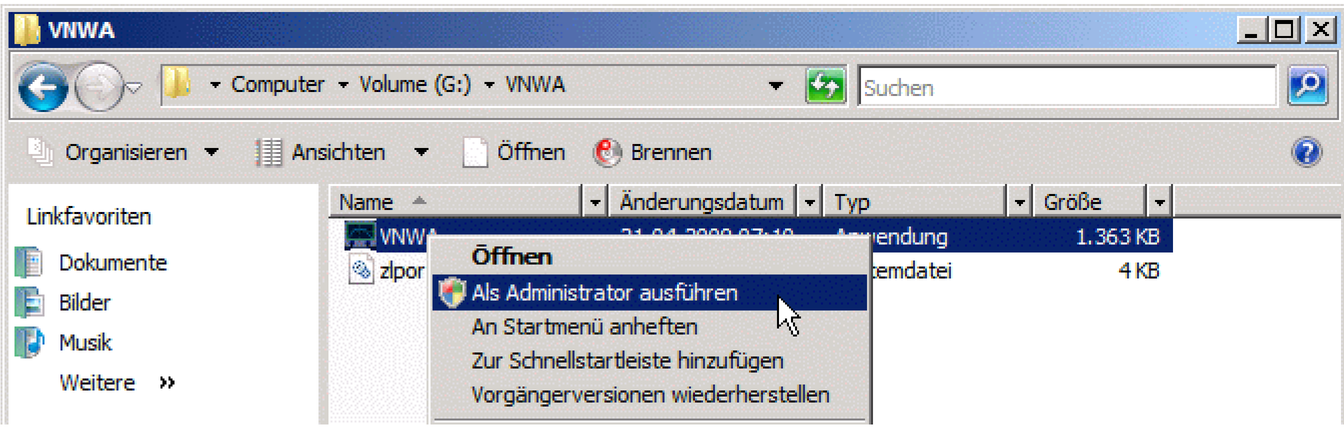
When the VNWA software is used in combination with the LPT driver **zlportio.sys** on a Windows Vista or Windows7 machine, then **by default Windows blocks the port driver** and upon double-clicking VNWA.exe, this is what you will see:



There are two solutions to this problem:

Solution 1:

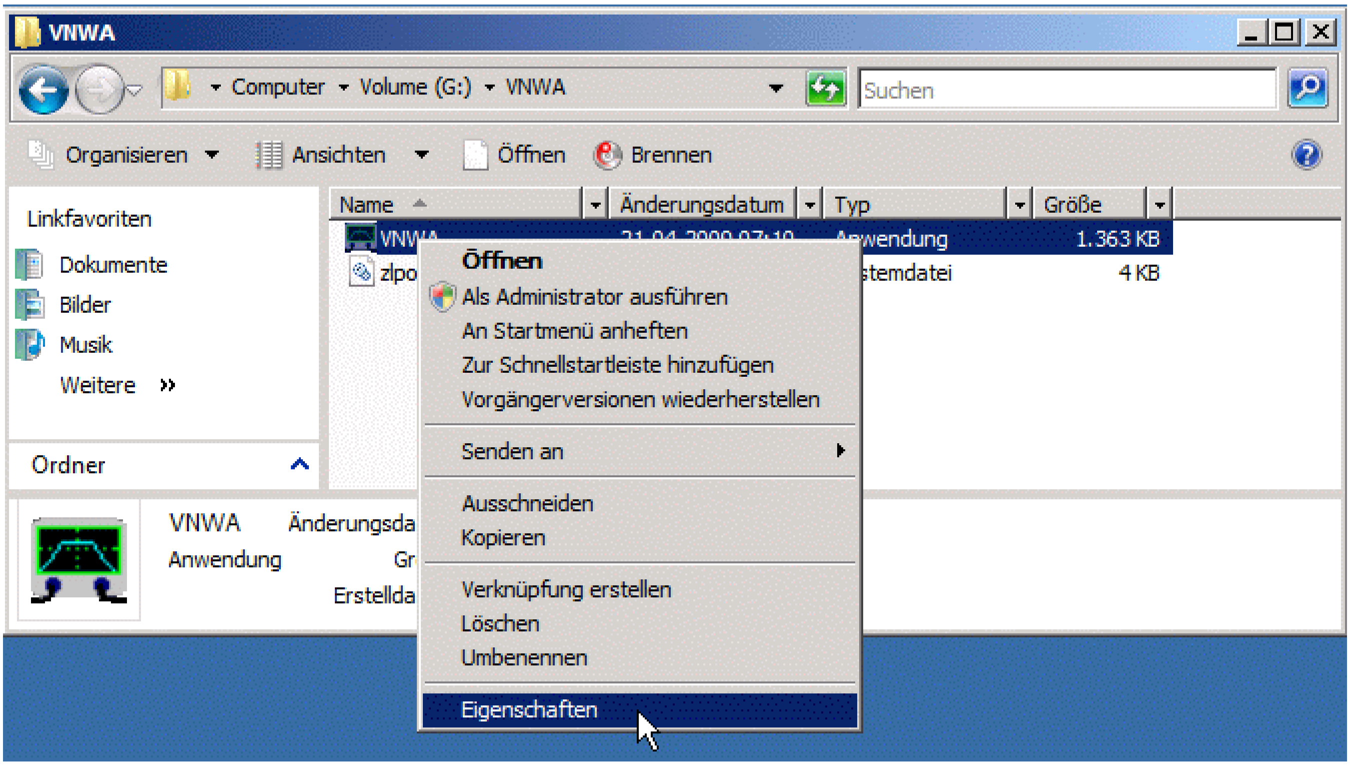
Right-click VNWA.exe and select **"run as administrator"**:



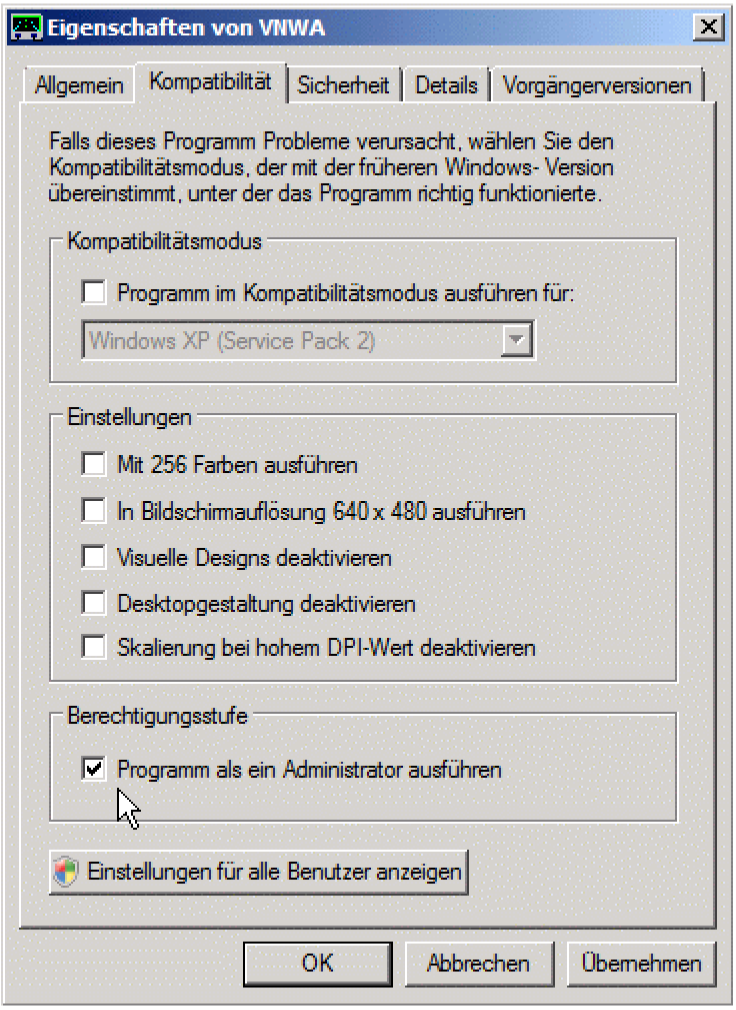
Confirm the warning popping up by agreeing to start the program. Then VNWA.exe should be started flawlessly with LPT support. Note, that you have to do this upon every program start. More convenient is solution 2:

Solution 2:

Right-click VNWA.exe and select the menu item **"properties"**:



Select the tab-sheet **"compatibility"** and activate **"run program as administrator"**. Do **not** select any compatibility mode! Confirm with **"ok"**.



From now on you can double-click VNWA.exe to run it with administrator rights. A warning window will still pop up upon every program start unless you deactivate it inside Windows.

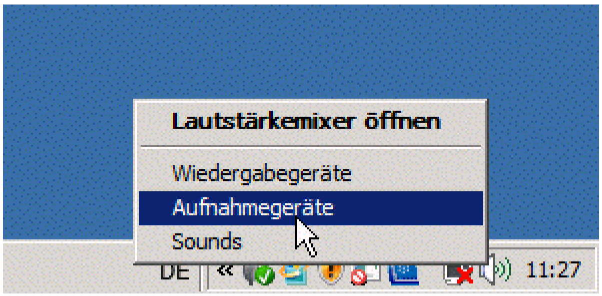
▸▸▸ **Warning:** In Windows 7 / Vista the **name of the USB Audio Codec depends on which USB port** you plug the VNWA into. This means, if you do below configuration with the VNWA connected to one specific USB port, you have to redo it again if you connect the VNWA to a different USB port. Moreover, the VNWA software will not be able to autodetect the USB audio codec, when you change connection from one USB port to another. You have to manually select the correct sound codec in "setup - audio settings" in this case. Summarizing, it is a good idea to **connect the VNWA always to the same USB port**.

Configuring the USB Audio Codec in Vista and Windows 7

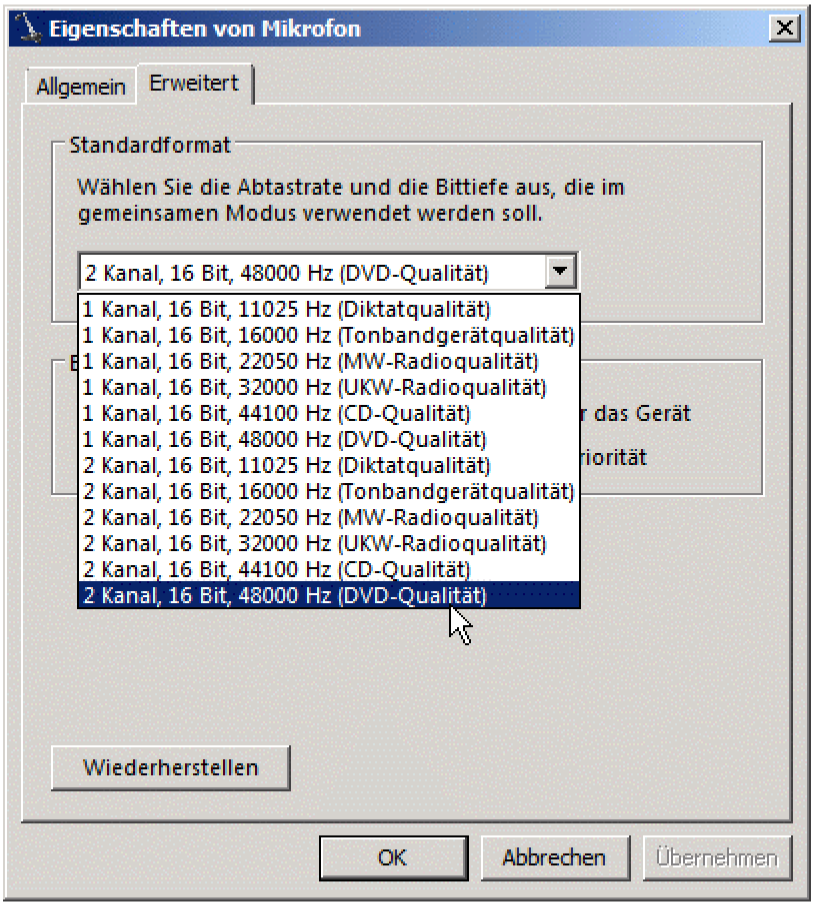
Note, that some people have reported they had to run VNWA/USB in administrator mode as described before for the LPT mode in order to make it work. This is not generally the case but might be worth a try in case of problems.

Configuring the audio codec:

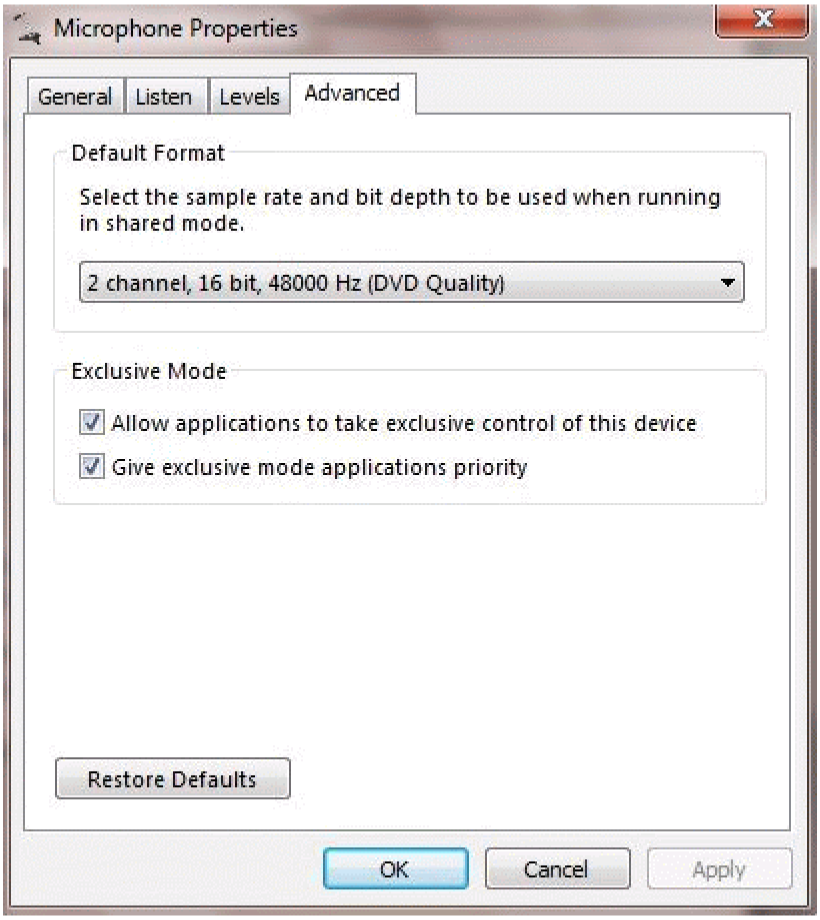
1. Right-click the the little loudspeaker icon on the lower right Windows bar of your desktop and select **"capture devices"** from the popup menu:



2. Highlight the USB audio codec by clicking on it as seen above and press the **"properties"** button:



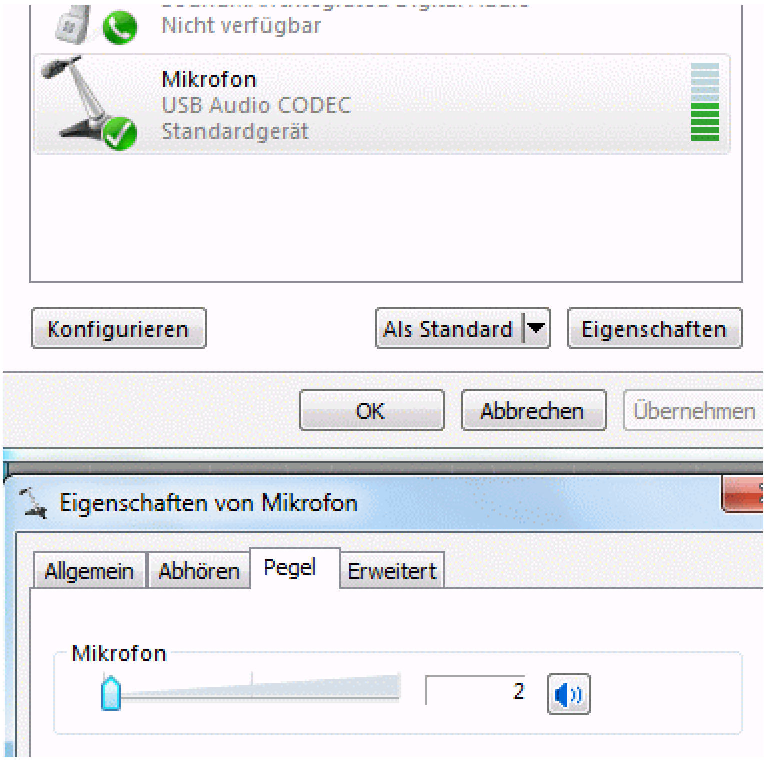
3. Select **2 channel 16 bit 48000Hz** as operation mode for the codec as seen above.



4. Confirm by pressing the **ok-button**.

Windows 7 Volume Control

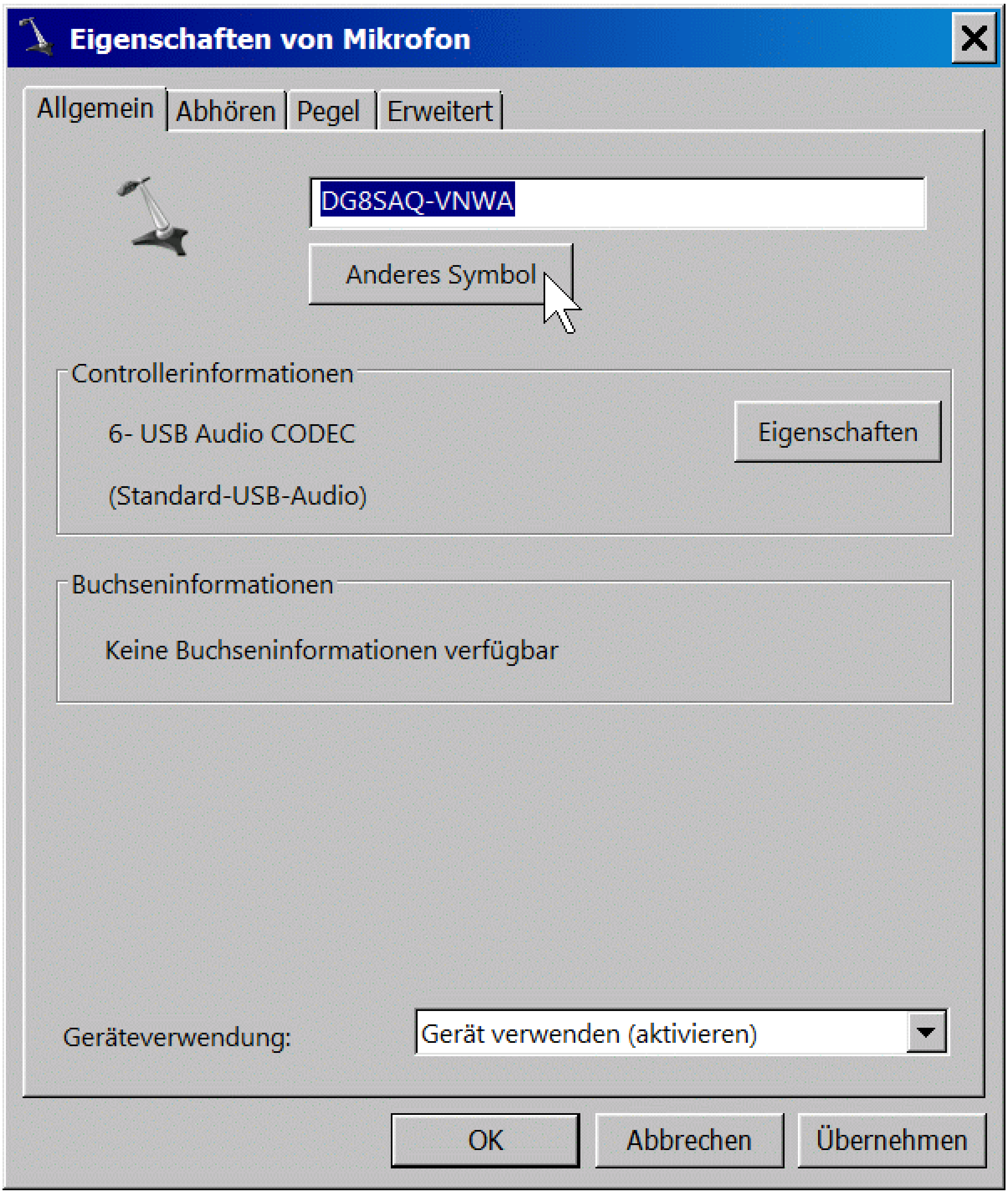
In contrast to all other supported operating systems, **Windows 7 allows to control the recording volume of the USB codec**. Note, that this is only a software control, which allows to multiply the sample data that the codec is sending by an amplification value. But apparenty, numerical clipping will occur. By default, the volume setting is reportedly too sensitive, so reduce the volume appropriately by highlighting the microphone device of the USB codec, pressing the "properties" button and selecting the "level" tab.



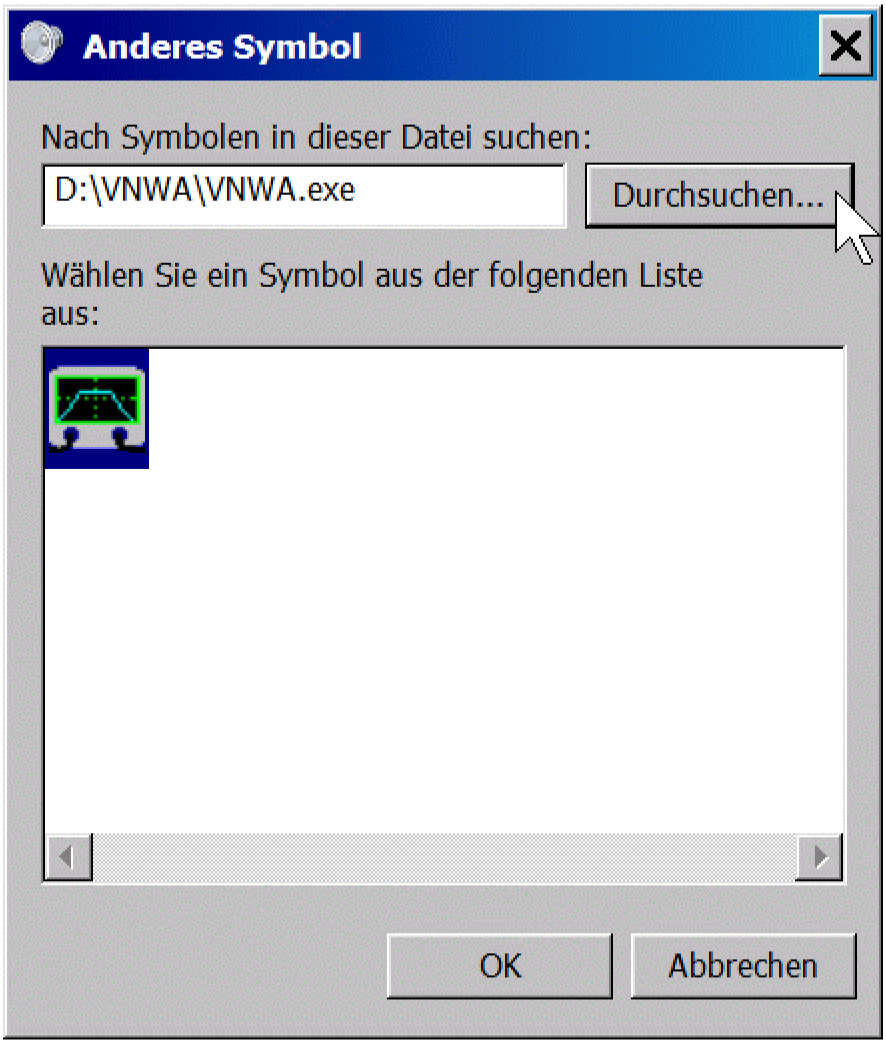
Levels between 2 and 4 are reported to work ok.

Not necessary but nice: Customizing the VNWA Audio Device in Windows 7

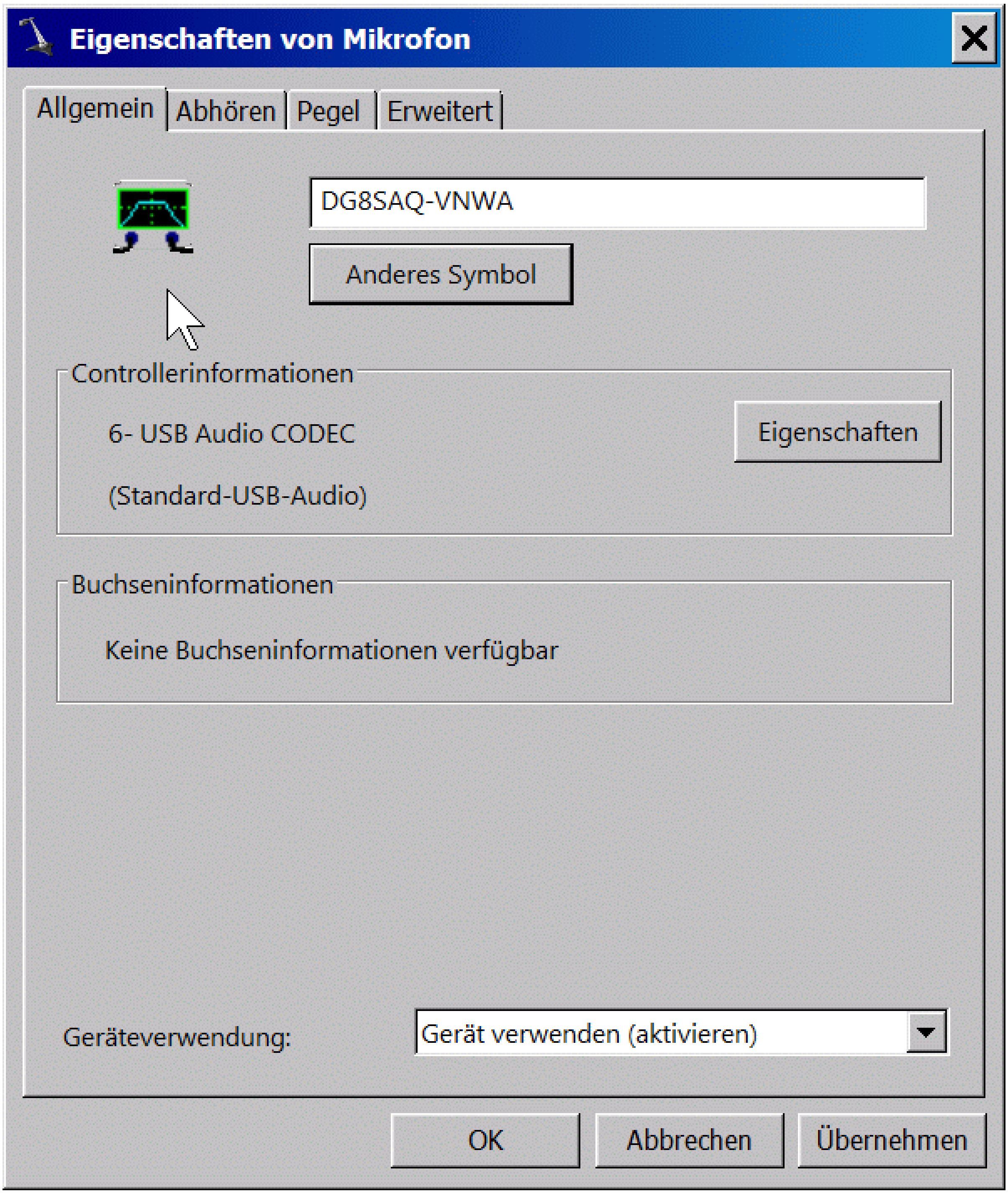
Windows 7 offers the possibility to change the name and symbol/icon of the USB audio device:



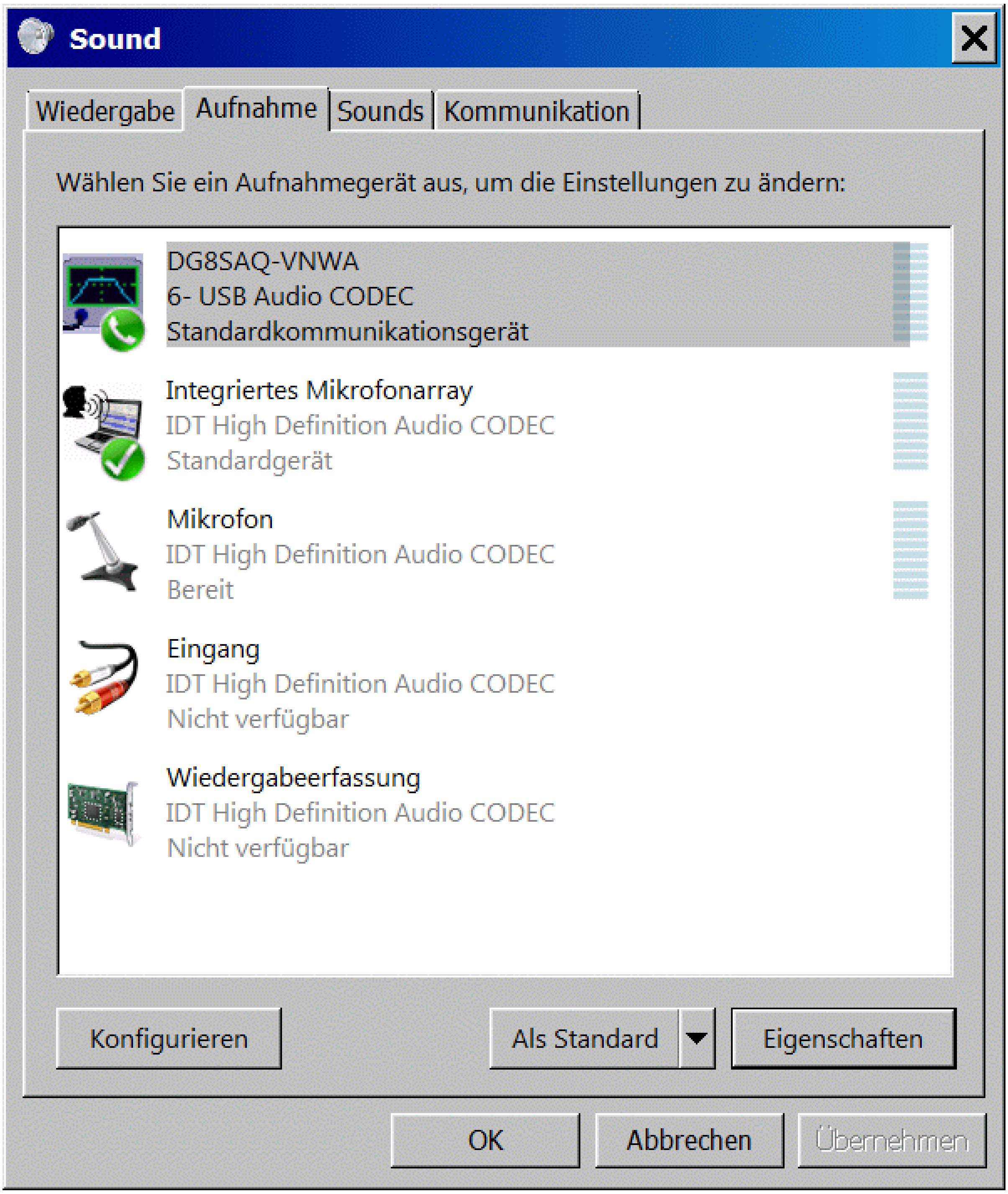
To change the name, simply replace the original name by something more descriptive like "DG8SAQ-VNWA" as shown in above example. The icon can be exchanged by pressing the button "other icon" (see mouse pointer in above screen shot). After pressing the button, a file selection menu opens. Use it to select the VNWA application file (VNWA.exe):



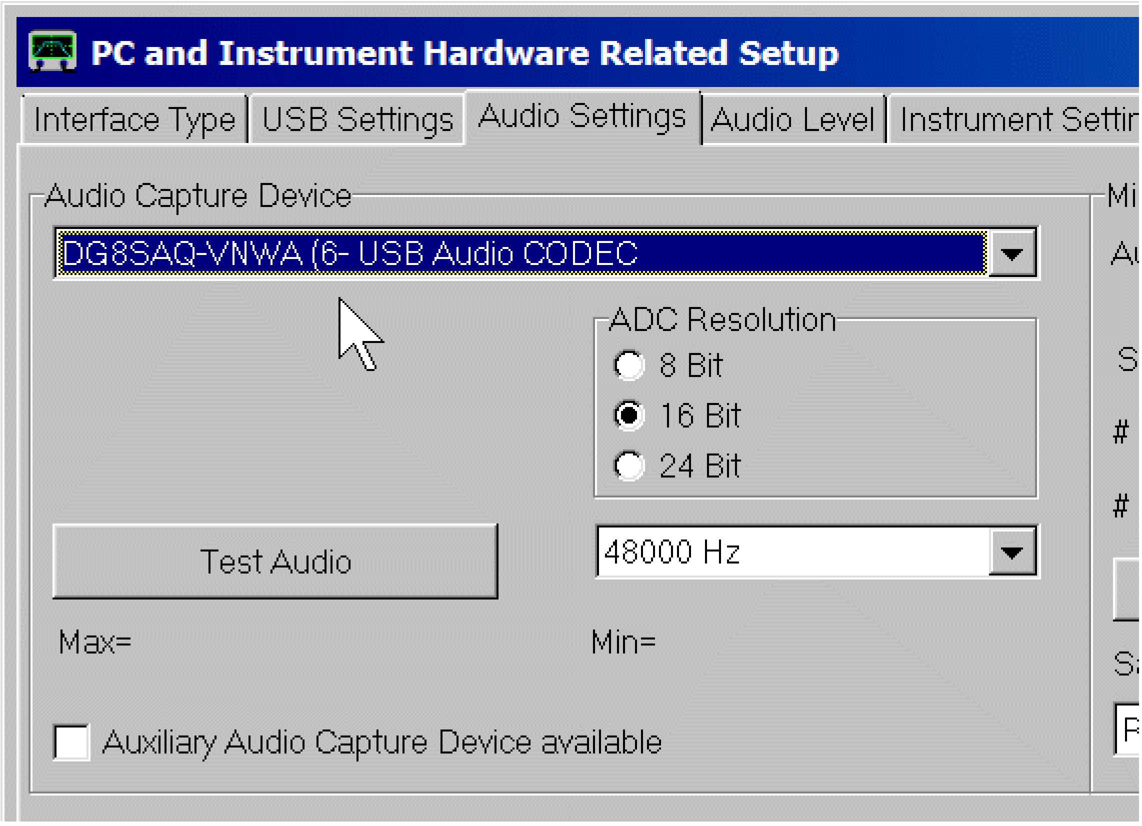
Confirm with "OK" and the USB audio input device will show with the VNWA icon:



Press the "Accept" button and then "OK" and the new name and icon are saved:



From now on you will see the VNWA USB codec with its new name in the VNWA audio setup:



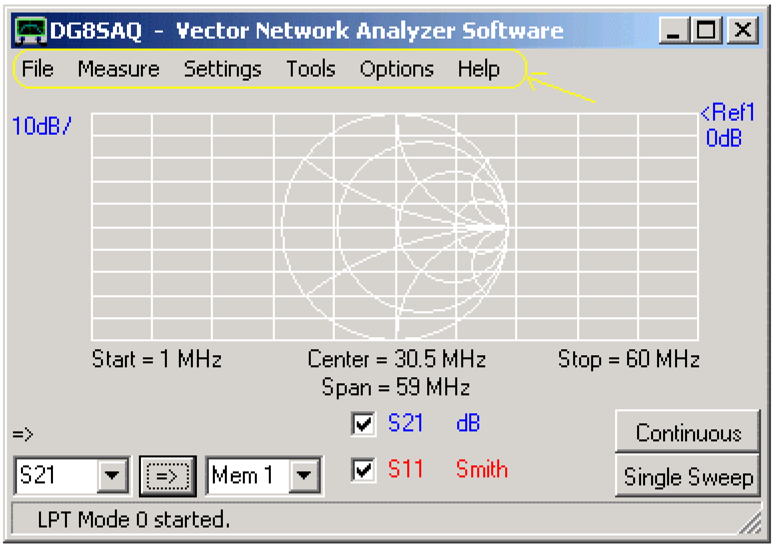
▸▸▸ **Note:** The new name and icon is linked to the VNW USB codec AND the specific USB port it is connected to. If you reconnect your VNWA to a different USB port, it will show with a different name and icon. Connecting it back to the original port, it will show the freshly installed name and icon, though.

=> Always connect your VNWA to the same USB port. If you change the port, you will have to redo the VNWA audio settings AND the Windows7 audio settings!

Activating N2PK Support

In order to activate N2PK Support, you need to create file named **N2PK.ini** with arbitrary contents in the program folder prior to program start.
You can do so by e.g. typing *echo > n2pk.ini* on the console prompt.

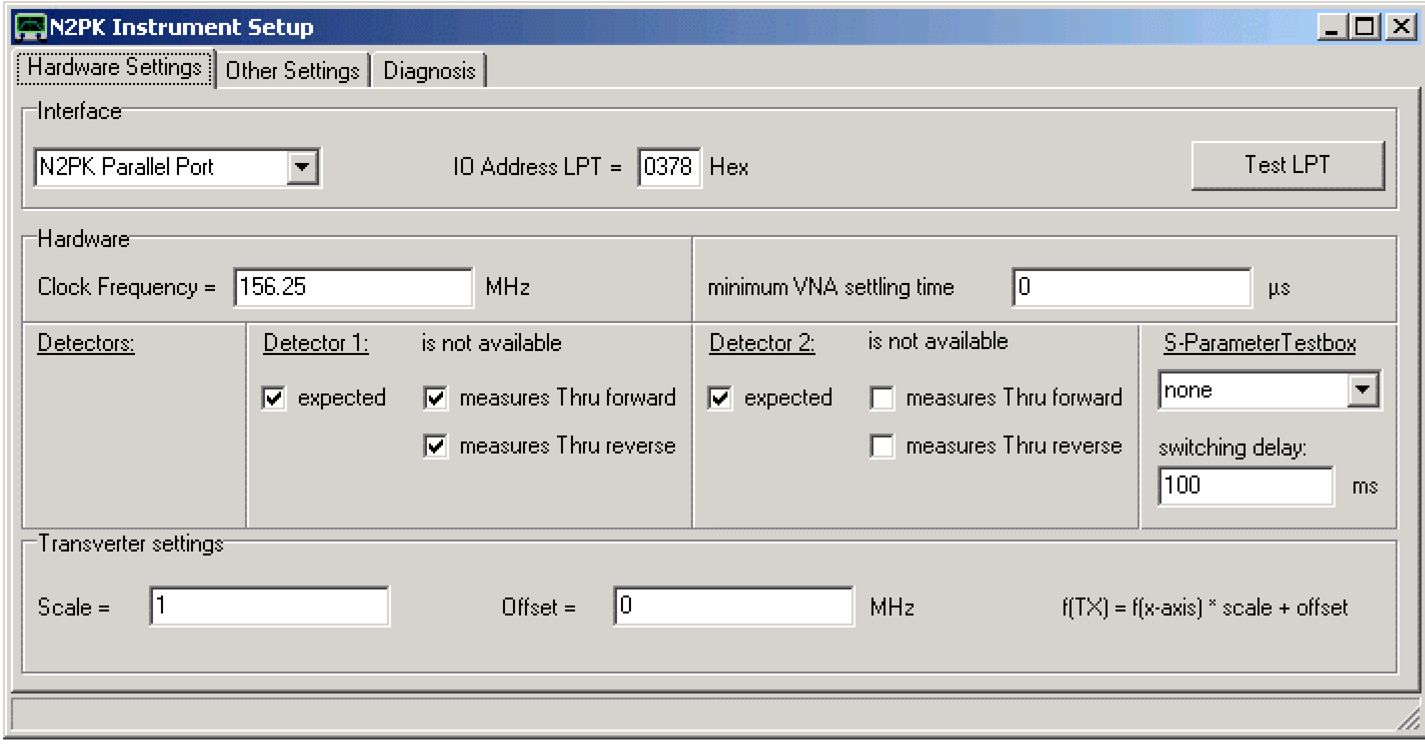
Then start the VNWA software by double-clicking VNWA.exe. Two windows should open. One of them is the VNWA setup window. Close it by pressing the [X] button.
The remaining Window is the VNWA main window with its main menu:



Activating N2PK Mode

Connect your N2PK VNA to your PC parallel port interface and power it up. To activate N2PK mode, select from the main menu "Options" , "Select Instrument" and "N2PK VNA". The software will search for the N2PK hardware, but likely it will not find it as we haven't provided the setup data yet. If the software claims that it hasn't found detectors, quit the notification. After having done so, the N2PK setup window will automatically open:

N2PK Setup



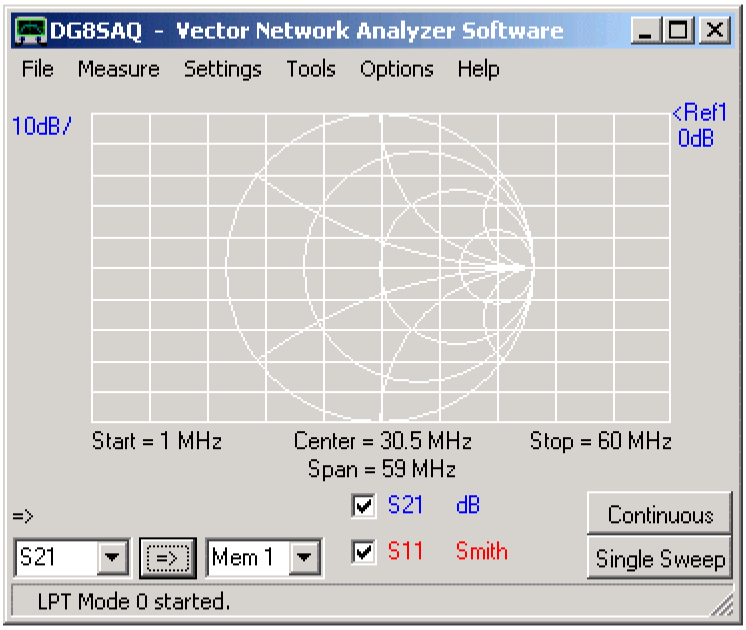
Enter the required data. If you use the LPT interface mode, make sure to enter the correct LPT address.
If you want to use the G8KBB USB interface mode, make sure to copy the files **vnadll.dll** and **delphivna.dll** into the VNWA software directory prior to first program start.

In the "Other Settings" tab, you can specify a master calibration filename, which will be automatically loaded upon program start and which will be used to interpolate corrections in case no calibration has ben performed.

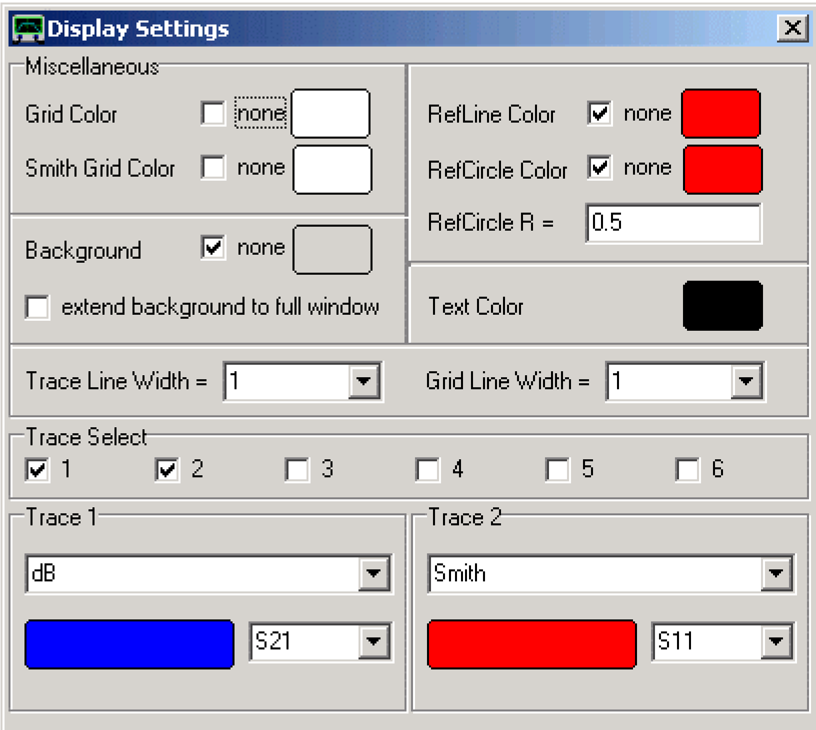
You can use the functions in the "Diagnosis" tab to ckeck proper operation and communication between PC and N2PK hardware.

If done, close the setup window and close the main window by pressing the  buttons. Closing the main window will cause the entered data to be stored in ini-files, which you will now find in your VNWA programm directory.
This setup is required only once at first program start. On the subsequent program starts, the data will be read from these ini-files and the setup menu will not be entered automatically any more. But you can still enter and modify the setup manually via the menu "Options-Setup".

After having gone through the hardware setup procedure after first program start, and after having **restarted the VNWA software**, you should see the following screen, which I subsequently call the main graphics window:



Select the menue item **"Settings"-"Diagrams"-"Display".** You'll see the following window:

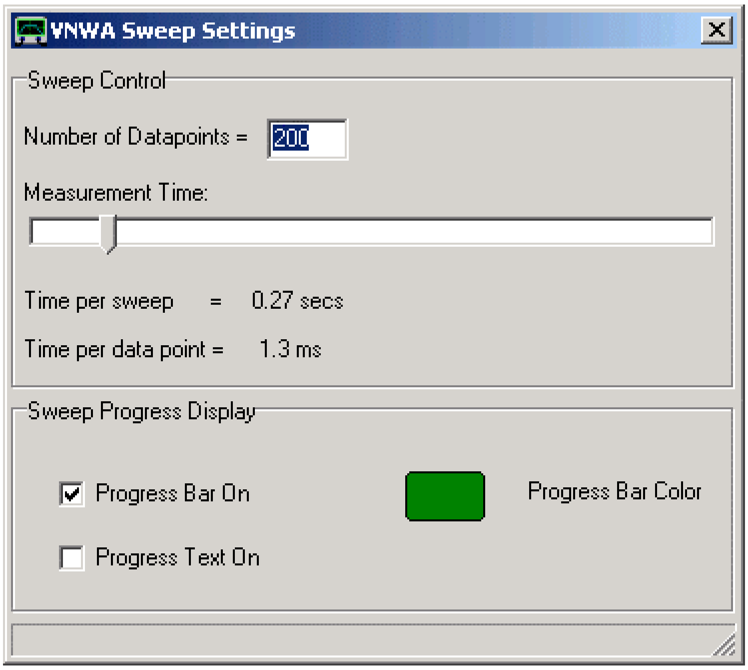


Here, you can customize colors, line widths and select, which traces (=curves, maximum 6) and types of traces to to be displayed.

To save CPU time it is recommended to select Background "none", which leads to a grey windows type background color. I prefer grid and smith diagram grid colors to be white on a grey background. To change colors, left-click the color boxes and select a color of your choice.
You can display **two horizontal freely placable reference lines** if you like. If you don't need them, check "RefLine Color " "none".

You can select the traces you want to display with the **"Trace Select "** checkboxes. When checking one of the traces, a trace information field pops up, where the memory space (S21, S11, S12, S22, Memory1...4) and the display type (dB, Smith...) can be selected.

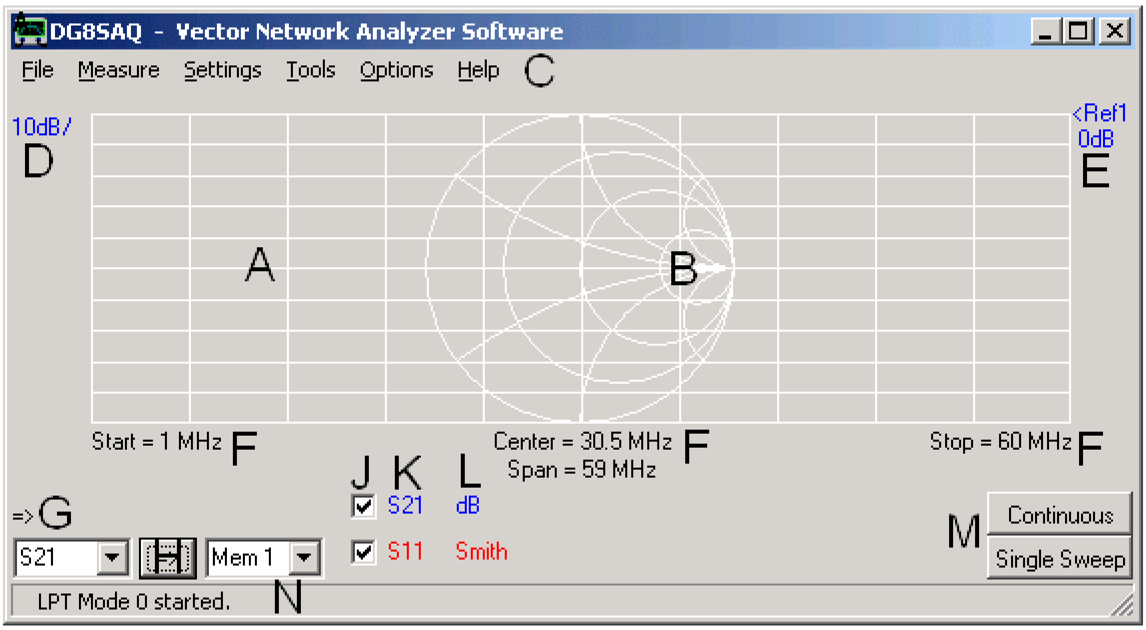
When you are done with these settings, close the Display Settings window and open the menu item **"Settings" - "Sweep Settings"**:



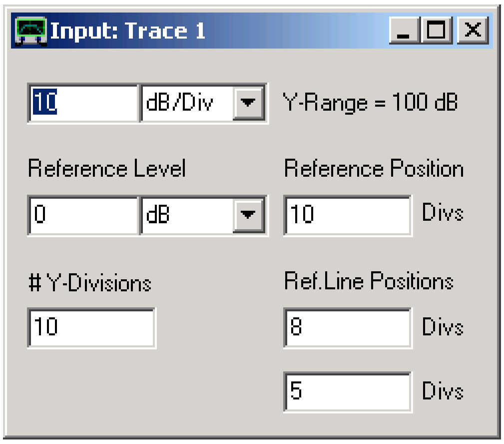
Select the **number of data points** (min 1, max 8192) and the **sampling time per data point** (min 0.2 ms, max 100 ms) for your measurement.
▶▶▶ **Note:** Sweep rates below 1ms / frequency point are only available in USB mode.
▶▶▶ **Note:** The sample rate slider will block on the left at the fastest possible sweep rate. If you want to sweep faster you need to increase the IF by decreasing the #samples per IF period and/or increasing the sample rate.

Select a **sweep progress display** option if you like. This is particularly usefull, when doing slow sweeps.

Close the Display Settings window when done. You should see the main graphics window again with your new colors.



A **dB vs. frequency grid (A)** and a **Smith chart (B)** are visible. Above, you find the **main menu (C)**. The top left blue **scale label** (D) indicates the scales per division for the to be blue trace. The right blue **"reference label" (E)** indicates the reference level and reference postion of the to be blue trace. To change the scale, the reference level or the reference position, double-click either of the labels D or E, and modify the data in the window which is popping up:

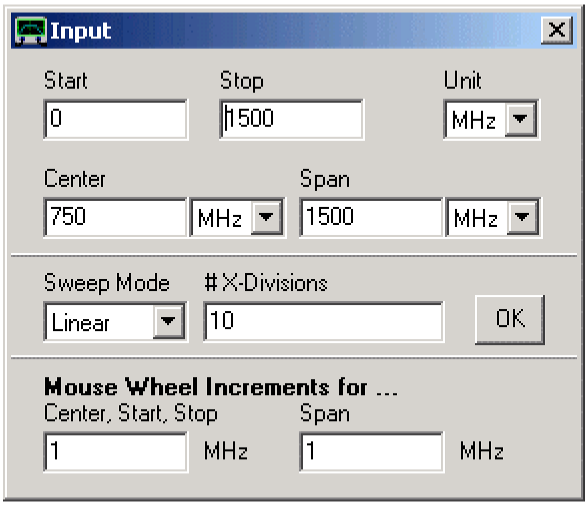


Also, the number of horizontal grid lines and the position of the optional reference line can be changed here. Close the window when done.
An alternative way to change the scale/division is to hold the mouse pointer over the corresponding reference label (E, the shape of the pointer will change) and turn the mouse wheel.
Yet another possiility is to right-click the label.

An alternative way to change the reference level of a trace is to move the mouse pointer to the according reference label of the trace on the right hand side of the grid (E, mouse pointer shape will change), then press the left mouse key and move the mouse up or down while keeping the left mouse key pressed. This way, the traces can be moved up or down the screen with the mouse in realtime.
Yet an alternative is to right-click the reference label.

A very useful feature is the **"get scales from"** feature accessible on right-click on any trace label (D,E,L). It allows to copy all scale information from one trace to another, e.g. when you want to save S21 to Memory 1 and then display Memory 1 with the same scales as S21.

In order to set the measurement start and stop frequencies, double-click on one of the **frequency labels** below the grid (F). The following **frequency "Input"** window will pop up:



Enter start and stop frequency or alternatively center frequecy and frequency span.
Also, the number of vertical grid lines and the sweep mode (linear, logarithmic, listed sweep) can be changed here. Note, that the start, stop, center frequency and the frequency span can also be changed with the mouse wheel by holding the mouse pointer over the corresponding main window frequency label below the grid (F).and turning the mouse wheel. The increment values can also be set in the frequency "Input" window.

Close the window when done.

If you have calibrated your instrument, it is now ready for sweeping (which means measuring). To do so, press the button **"Single Sweep"** (M) if you only want to aquire a single frequency sweep or the button **"Continuous"** (F) for continuous sweeping. Both sweeps can be interrupted by pressing the same button again.
▸▸▸ **Warning:** Your measurement results will be quite meaningless unless you have calibrated the instrument!

Each trace can be switched on or off by checking/unchecking the checkboxes below the grid (J).
You can also change the displayed **memory space (S21, S11, S12, S22, Mem1...Mem4)** and the **display type (dB, Smith, mag, phase...)** by right-clicking the labels next to the checkboxes (K, L). Alternatively, double-click the label K or L.

Trace **data can be copied** from one memory space to another (e.g. S21 to Memory 1) with the button H. The combo box to the left of the button selects the source, the one on the right selects the target memory space for the copy process.

Markers can be added by right-clicking into the grid field or into the smith chart. Marker positions can be changed by dragging the markers with the mouse (pointing the mouse onto the marker triangle, pressing the left mouse key and keeping it pressed while moving the mouse).

The **"File"** menu allows to **save or print the screen**, **to save or load S-parameter files** in Touchstone format (s1p, s2p) or **save/load calibrations**.
S-parameter files can also be loaded most conveniently by dragging them from any file browser and dropping them onto the VNWA main window.

General

Data inside the VNWA software is organized in distinct **memory spaces**, which generally contain only S-parameters, but **no frequency information**. All memory spaces (except the "Plot" spaces) share the same frequency frame data given by the x-axis of the main diagram in unzoomed mode.

Available memory spaces:

S21 = measured data S21
S11 = measured data S11
S12 = measured data S12
S22 = measured data S22
Mem1 = data space Mem1 data
Mem2 = data space Mem2 data
Mem3 = data space Mem3 data
Mem4 = data space Mem4 data
Plot1 = data space Plot1 data
Plot2 = data space Plot2 data
Plot3 = data space Plot3 data
Plot4 = data space Plot4 data
s_11...s_33 = measured 3-port data
Cust1...Cust6 = custom calculated data

▸▸▸ **Note:** If the frequency frame data is changed, the memory space data remains unchanged (except for the "Plot" data spaces), thus, you will see the same traces with changed frequency axes, which generally invalidates data.

▸▸▸ **Note:** If the number of frequency points is changed, the **memory space data will be erased** (except for the "Plot" data spaces).

▸▸▸ **Note:** Data can be copied from almost any memory space to almost any other memory space by means of the **"=> button"** H.

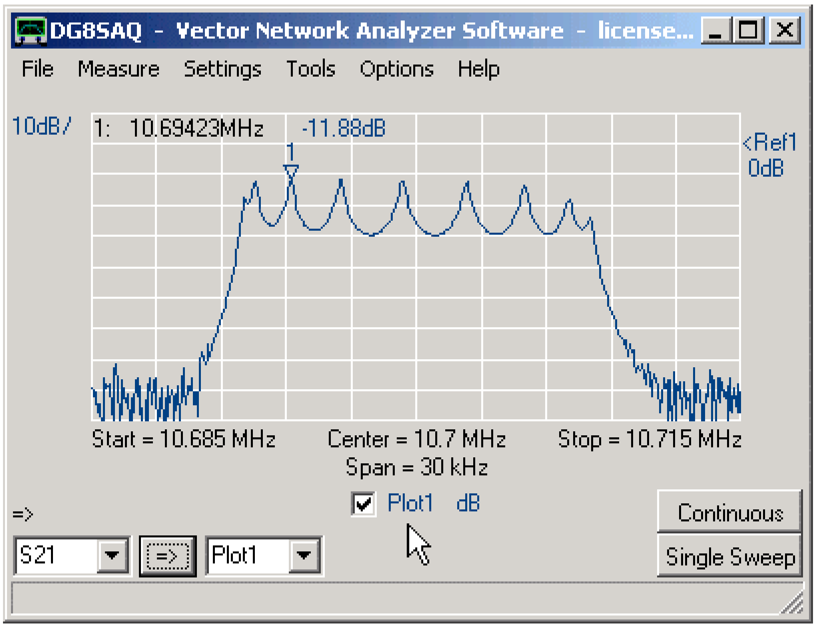
▸▸▸ **Note:** Memory space data can be exported to a file or imported from a file by means of the "File"-"Export Data" or "File"-"Import Data" menu.

Special features of the Plot memory spaces

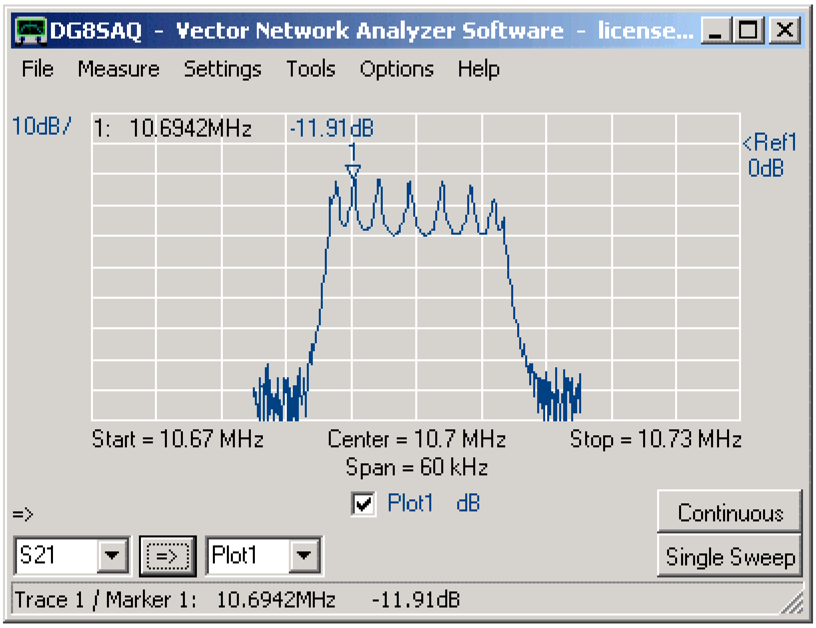
The memory spaces Plot1...Plot4 each contain an individual frequency grid memory. While importing data into e.g. Mem1 causes the frequency grid for all traces (except Plot* traces) to be modified, importing data into the Plot* memory spaces leaves the main frequency grid untouched. The Plot* data remains valid upon changing the main frequency grid (frequencies, lin/log scale) or the number of frequency data points. Note, that marker data for the Plot* memory spaces is interpolated to the main frequency grid.

Example:

1. Copy any data to Plot1 and display it:



2. Increase the frequency span and the number of data points:



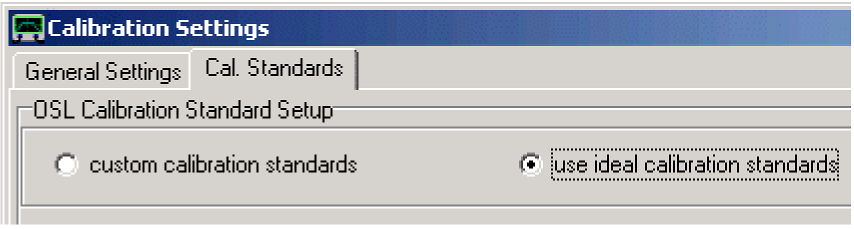
You will still see correct data plotted into the valid frequency span. Note, that marker levels have slightly changed due to the frequency grid interpolation.

[Calibration Standard Setup]

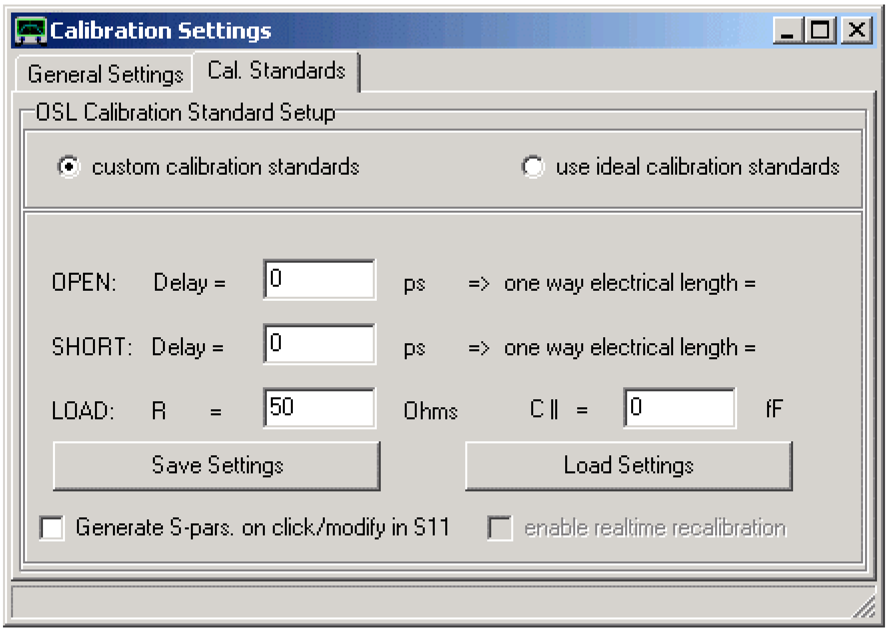
[Performing a Calibration for a Two Port S-Parameter Measurement]

[Master Calibration]

Todate, the VNWA software allows for SOLT (Short, Open, Load, Thru) calibration. Before actually performing an instrument calibration, you need to specify the RF properties of the calibration standards you are going to use. To do so, select from the VNWA main menu "Settings" the submenu "Calibration Kit Settings". The following window will open:



If you do not know anything about your cal standards, select "use ideal calibration standards". In this case, the load standard is assumed to be 50 Ohms, the open and short standards are assumed to be ideal open and short circuits of zero length.
For high precision measurements in the VHF and UHF range, this is generally not good enough. In that case you need to obtain a more precise model of your cal standards.
Select "custom calibration standards". In this case, you will see the following screen:



Enter the electrical delays of your open and short standards as well as the DC resistance and shunt capacitance of your load standard. You can accurately measure the DC resistance of your load standard with a 4 wire Kelvin probe. In order to obtain the other parameters, you need a reference VNWA to measure them. For comparison between the measured reflection coefficients from the reference VNWA and the model data, you can simulate the model calibration standard reflection coefficients by activating the checkbox "Generate S-parameters on click/modify in S11". S11 data will be automatically updated whenever you modify the calibration standard model parameters. E.g. if you display the measured reflection coefficient of your Open in memory space Mem1 and compare it to the simulated reflection coefficient in memory space S11, you can modify the parameter "Open delay" (can also be modified with the mouse wheel), until both curves match.
▸▸▸ **Hint:** You will only see a difference in delays, if you look at the phase.

▸▸▸ **Hint:**
If you do not have access to a reference VNWA, then you might use the realtime recalibration feature on the bottom right right (greyed in above figure). To do so, you must previously have performed an SOL reflection calibration with your unknown calibration standards and you need to have measured the reflection coefficient an about 30cm long open ended or shorted coaxial transmission line. It must be measured on your VNWA calibrated with your unknown cal standards. Make sure, the measured line reflection coefficient is displayed in S11. If a valid calibration is available, then you are allowed to activate the **"enable realtime recalibration"** checkbox. When activated, if you click

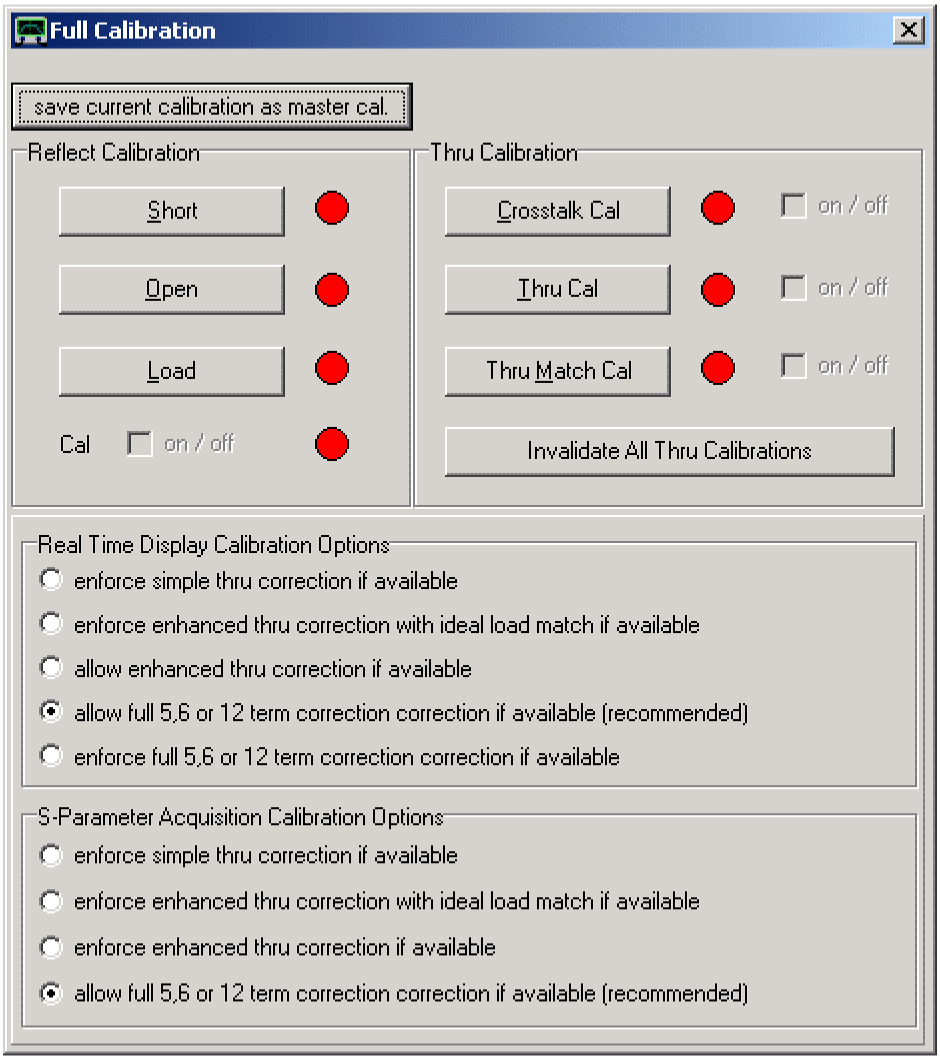
on one of the cal standard parameter fields and turn the mouse wheel, the parameter will change and the calibration will be recalculated in realtime. You will see this by a changing S11 trace. This way you can actually tune the cal standard parameters for lowest port mismatch ripple of your transmission line measurement in amplitude (dB) and phase progression (-continuous phase/f). Make sure you keep one delay (e.g. that of the Open standard) at a fixed value (e.g. =0). This will define the position of the calibration plane.

When measuring S-parameters of a two-port device (S11, S21, S12, S22), care has to be taken, that calibration planes for the reflect (S11, S22) measurement and for the through (S21, S12) measurement are identical, i.e. the phase values for all 4 S-parameters are consistent.

In the following, I describe the procedure, I use to calibrate my VNWA. This is not the only possible way to perform a proper calibration, but if you stick to this method, you can be sure to obtain a correct calibration.

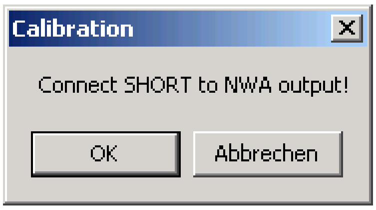
▶▶▶ **Important Note:** Before performing a calibration measurement, make sure, that the **instrument sweep rate is set to the lowest value** you intent to use the calibration with, otherwise your later measurement results will be dominated by the noise of your calibration measurements.

In order to initiate a two-port calibration, click the menu **"Measure"** and **"Calibrate"**. You'll see the following window popping up:

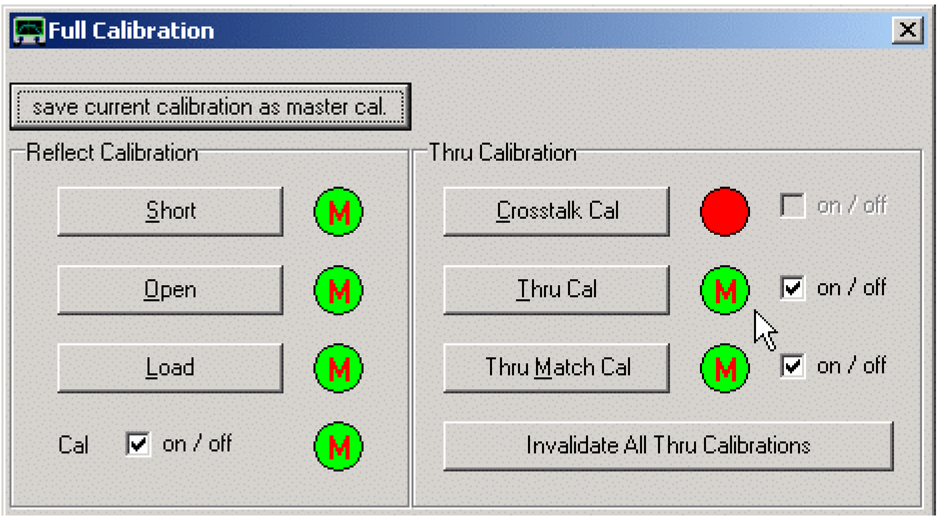


The radio menus on the bottom allow to experiment with various degrees of mathematical corrections. The recommended settings are the ones shown above. If you don't intend to play with those settings, reduce the window size, such that you don't see them any longer.

When you press the buttons next to the red lights, you'll see a window popping up which tells you, which cal standard to connect, e.g. when pressing on "Short", you'll see this window:



Now, connect the calibration standards, which you are asked for, and press ok when done. Output is synonymous for TX port and input for RX port. If you press "abort", no measurement will be performed. You can repeat any calibration measurement at any time. You can calibrate in any order you like. After a successful calibration sweep, the corresponding red lamp turns green indicating, that valid calibration data is available:

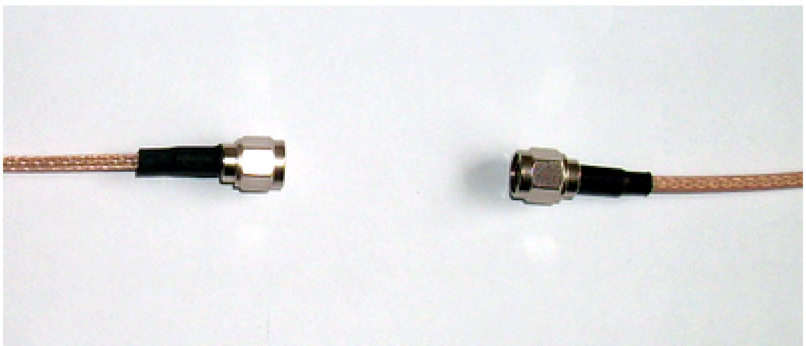


Note the red "M"s inside the green lamps. These indicate that no master calibration is available yet. By clicking on the green lamp, you can invalidate a calibration measurement and the green lamp turns red again. You can also switch corrections off and on again without invalidating the calibration measurements using the checkboxes next to the lamps.

In the following I will exactly describe my cal standards and how the calibration is to be performed.

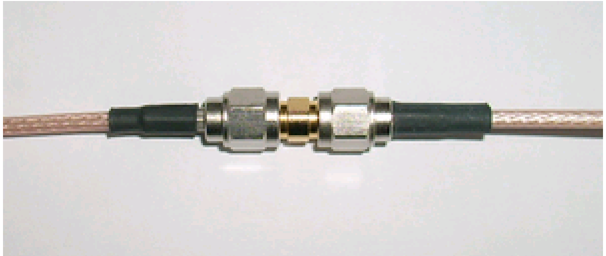
Thru Calibration:

Starting point are the two coaxial cables comming from the VNWA TX port (output, left) and going to the VNWA RX port (input, right) with two SMA male connectors.:



In case you run the **"Crosstalk Cal"**, just leave the two cables separated from each other as shown above.
▸▸▸ **Note:** Normally a crosstalk calibration can (and should) be omitted, as the VNWA's crosstalk level is below the noise floor. In this case, a crosstalk calibration only increases the noise level by 3dB. Use the crosstalk calibration to eliminate crosstalk of test adapters if necessary.

For the **"Thru Cal"** you need to connect the two cable ends. Obviously, you need an SMA female to female adapter as can be seen in the following picture:

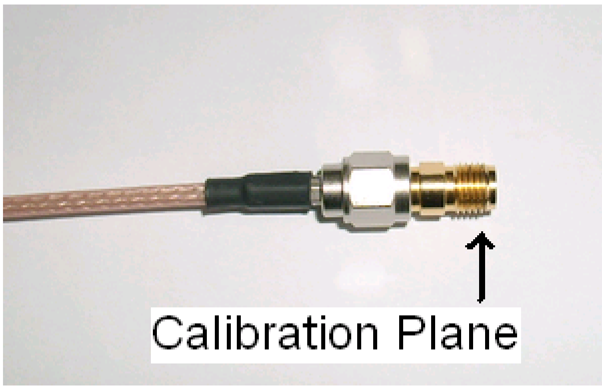


Note that the adapter introduces some phase shift because of its finite electrical length that has to be accounted for in the further reflection calibration.

The **"Thru-Match"** calibration step requires the same thru standard as the thru calibration. While the thru calibration step measures transmission, the thru match calibration step measures reflection of the RX port in order to numerically compensate deviations from a perfect 50 Ohms RX port match.

Reflection calibration:

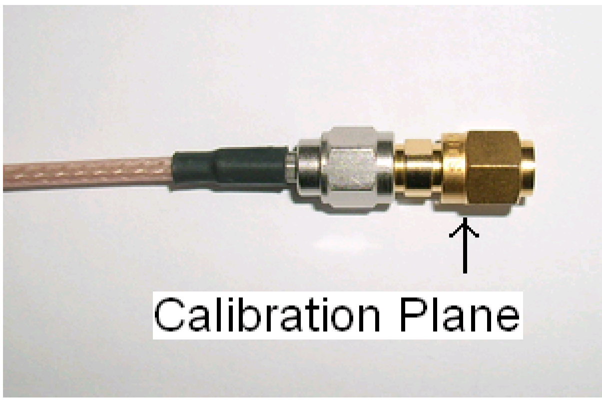
My way to deal with the additional phase shift of the thru adapter is to leave it in the system when performing the reflection calibration. The following picture shows my open "standard", which is the bare female SMA side of the thru adapter:



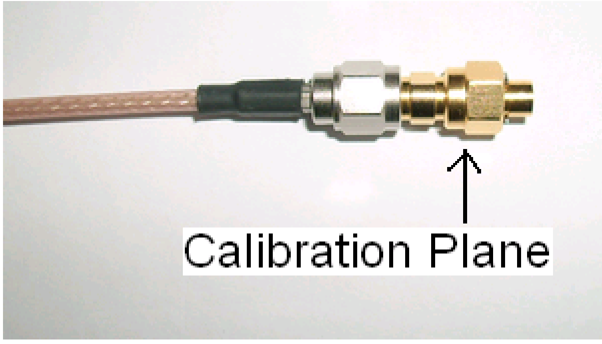
"Open" calib ration standard

This nails the reference plane to the place where the RX cable connector was sitting during the thru calibration and ensures zero phase difference between S11 and S21.

The thru adapter is also in place when measuring the "Short" and "Load" standards, which are both commercial products with male SMA connectors.



"Short" calib ration standard

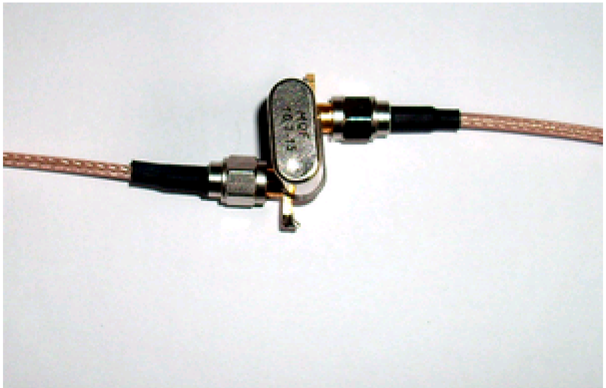


"Load" calib ration standard

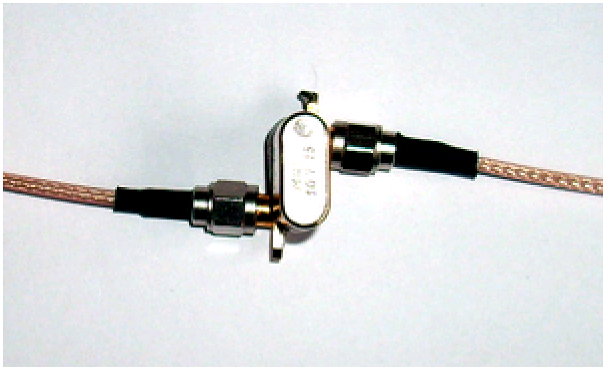
The "Short" standard is an SMA male connector with a built-in short circuit. The "Load" standard is an SMA male connector with a built-in 50 Ohms resistor.

Actually, for open, short and load standards, the calibration planes will still not be exactly at the same position as internal builds differ slightly. These slight errors can be corrected for by entering proper calibration standard parameters if known.

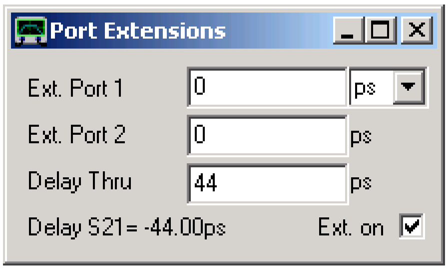
Finally, you want to remove the Thru adapter so you can connect the VNWA cables to your test object (a monolythic crystal filter in below picture)
in forward direction to measure S11 and S21



and reverse direction to measure S12 and S22.



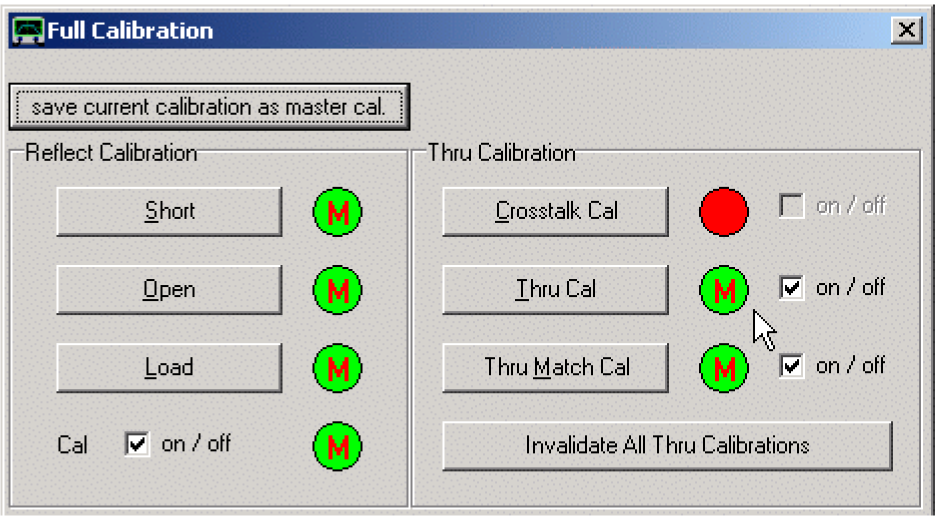
The only remaining problems are, that your reference plane still sits at the position where the end of the Thru adapter was, which would be well inside your test object now., and, in case of a two port calibration, your thru adapter for the thru calibration step should have been twice as long as it actually is.Both effects can be compensated in the port extension menu invoked by "Measure"-"Port Extensions":



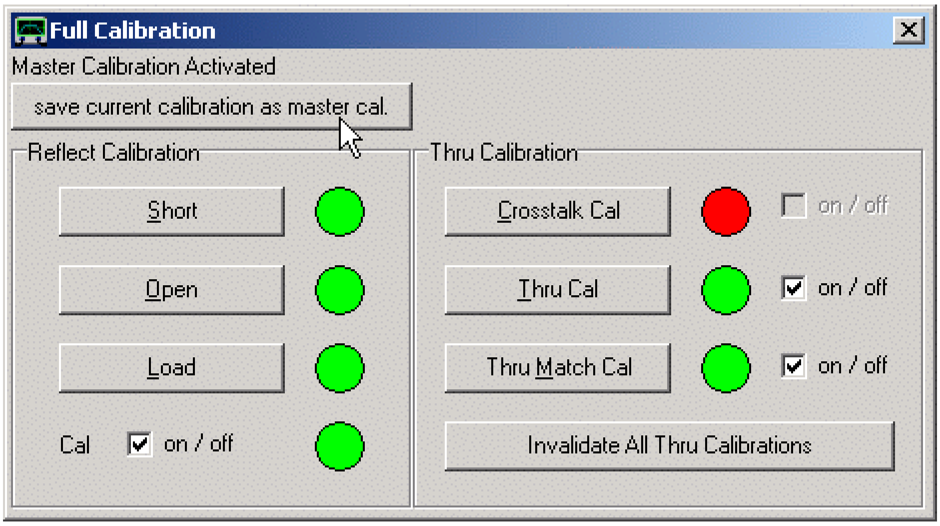
Here, you must enter the electrical delay of the Thru adapter into the "Delay Thru" field, which is 44 picoseconds for mine. All, S11,S21,S12 and S22 measurements will be corrected by this delay, if the "Ext. on" checkbox is checked. You can monitor the application of the correction by a slight rotation of your previously measured data in the Smith Chart .

Additionally, you can move calibration planes for forward and reverse measurements independently, by entering delays into the "Ext. Port 1" and "Ext. Port 2" fields. This might be useful, when you want to correct for different input and output connector lengths of your test object or if you want to move the calibration planes beyond the connectors to the solder pads of your component under test. A positive port extension will move the calibration plane away from the VNWA.

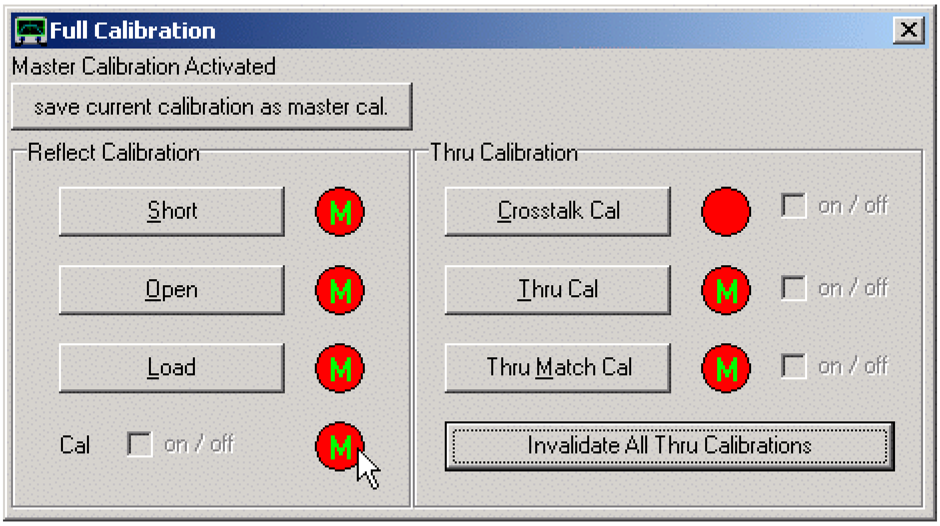
As long as you haven't performed a calibration, the VNWA will yield quite arbitrary and useless results. Once you have decided, what frequency range to sweep and how many data points to collect, it's time to do the calibration. **When you modify the number of data points or the frequency span, the calibration will be invalidated and is thus lost.** To avoid the necessity for repeated calibrations for standard moderate precision measurements, the master calibration feature has been implemented. The idea is as follows: Calibrate the instrument once with the maximum number of points (8192) and the maximum span (1 kHz...1.3 GHz):



Then make this calibration the master calibration by pressing the according button in the calibration window. You are asked for a filename to store the calibration for later reuse after program restart. Successful activation of a master calibration is indicated by the red "M"s inside the green lamps turning green and thus vanish:

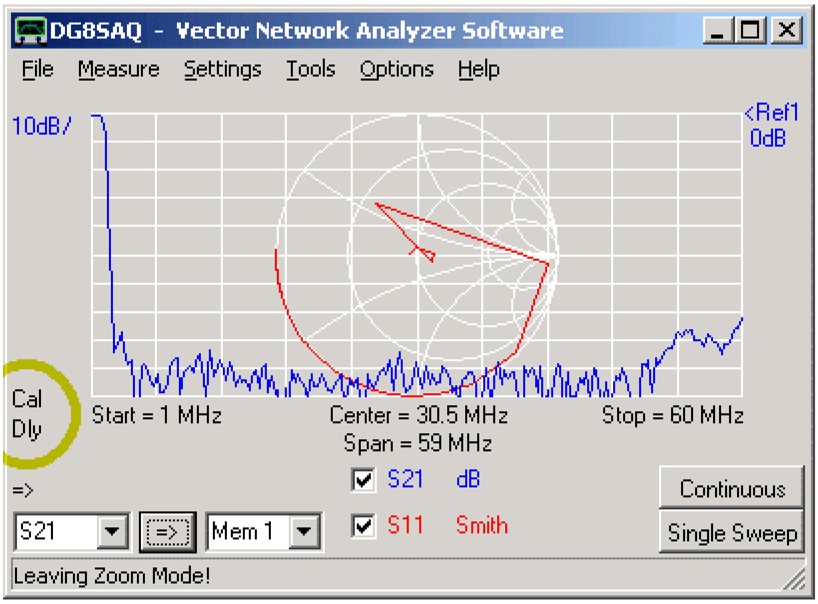


When you change the number of points or the frequency span, and the calibration is thus invalidated, the master calibration data is used to interpolate new calibration data. In this case, the instrument will still deliver reasonable results, even though it is not calibrated. This can also be seen from the calibration window which shows green "M"s inside red lamps indicating no calibration but an activated master calibration:



▸▸▸ **Hint:** You can have more than one master calibration by selecting different file names. You can reload an existing master calibration file in the main menu "Setup" and "Misc-Settings" tab.

Measurement:
Your instrument is now set up and calibrated and showing the main graphics window. It is ready for **sweeping**, which means measuring.
To do so, press the button **"Single Sweep"** if you only want to aquire a single frequency sweep or the button **"Continuous"** (F) for continuous sweeping. Both sweeps can be interrupted by pressing the same button again.
The picture below shows the measurement result of a low pass filter.



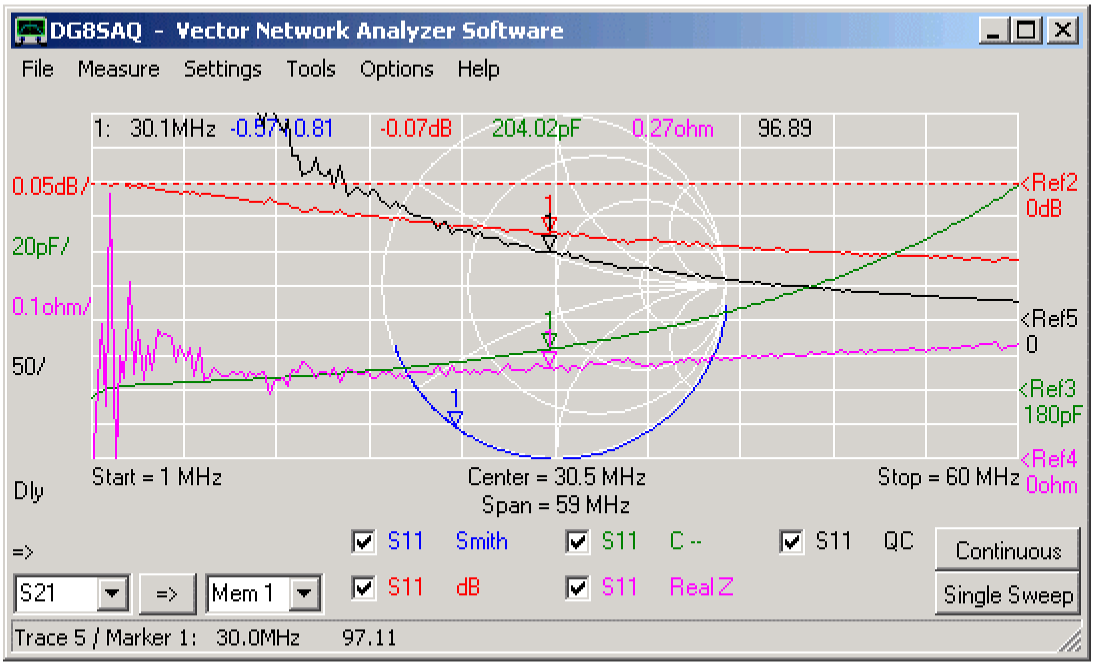
Note the two labels which are circled yellow:

The **Cal**-label indicates a valid calibration. If that label text shows **MC** instead, there is no valid user calibration, but a Master Calibration is loaded. You can double-click the label to activate the calibration menu. Right-click to switch off the calibration. If the user only performs a thru calibration, the software combines the user thru calibration with the master reflect calibration (if available).

The **"Dly"** label indicates, that a port extension (delay) is being applied. Double-click it to enter the port extension menu or right-click to switch off port extensions.

Note the **arrow** below the yellow circle. It indicates the measurement direction:
=> means forward measurement. Measurement data is stored in data spaces S11 and S21.
<= means reverse measurement. Measurement data is stored in data spaces S12 and S22.
As the VNWA2.* hardware cannot switch the measurement direction, you need to turn around the DUT manually for a reverse measurement.

Display modes:
Data can be displayed in various ways. The software can recalculate S11 e.g. to **VSWR, impedances, capacitances, inductances, Q-values**... . The following screenshot shows a reflection measurement (S11) of a wired 180pF capacitor:



▸▸▸ **Hint:** If you change the display type (e.g. from dB to VSWR) and you won't see your trace because of unfit scales, the fastest way to see something is to perform an **autoscale** operation. To do so, right-click the according scale-label and select autoscale.

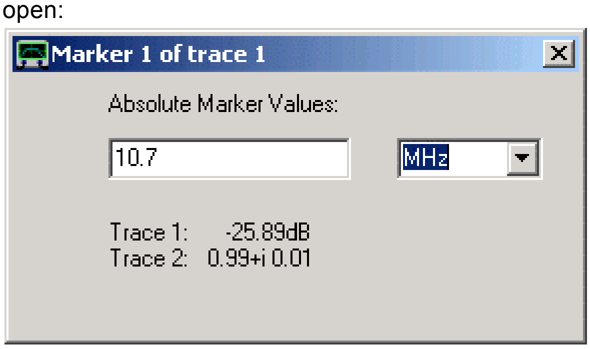
Markers

Up to 9 markers can be used. To add a marker, right-click onto the main grid at the frequency, where you want the marker to show up and select **"add normal marker".**

Available marker types:

- *normal marker*
- *maximum marker* (automatically jumps to the maximum of the first displayed trace)
- *minimum marker* (automatically jumps to the minimum of the first displayed trace)
- *bandwidth marker* (places a maximum marker, two markers down one vertical division below the maximum marker and a center marker between the bandwidth markers)
- *delta marker* (displays the frequency distance and the vertical distance to the last normal marker)

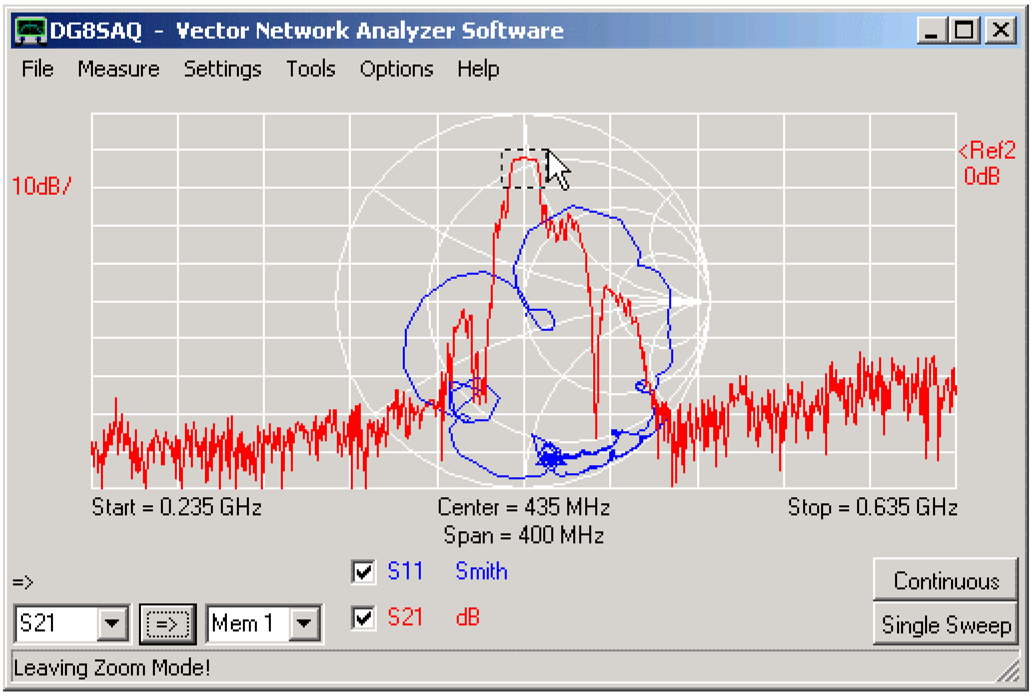
▸▸▸ **Hint:** To see a summary of marker positions or change the marker frequency, double-click onto a marker.
▸▸▸ **Hint:** You can also drag the markers around with the mouse.
▸▸▸ **Hint:** To **delete** one or all markers, right-click into the display grid and select "delete last marker" or "delete all markers".
▸▸▸ **Hint:** You can enter an exact marker frequency after double-clicking onto the marker. The following window will open:



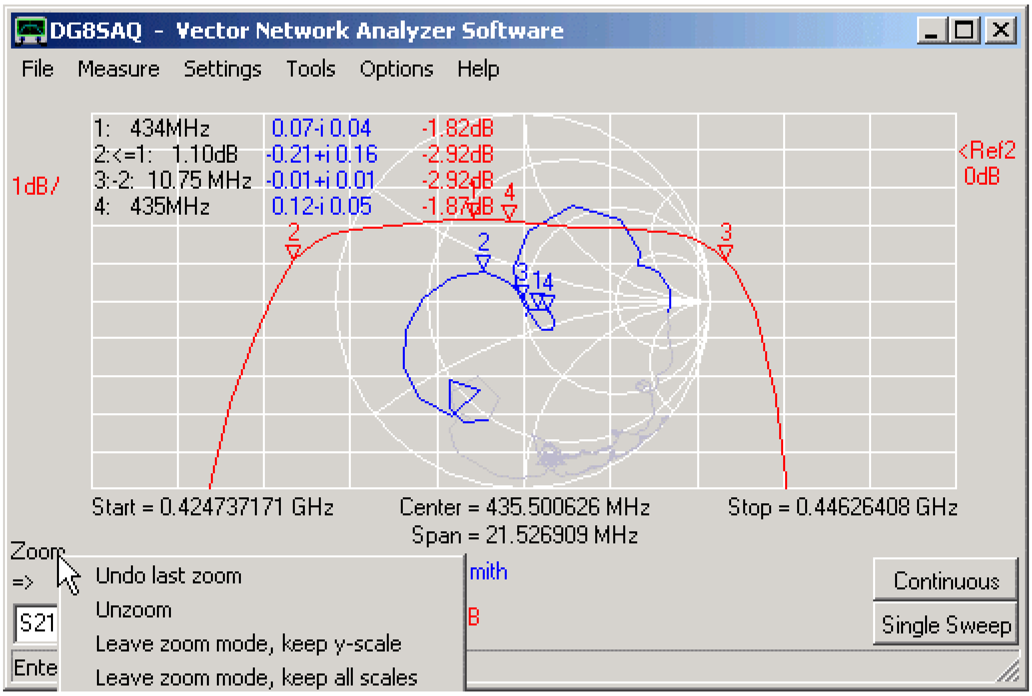
▸▸▸ **Hint:** In the above picture, you also see one of the two available horizontal dashed **reference lines**. You can move them up and down with the mouse. You can change their color by right-clicking on them.

Zoom

You can also zoom into a trace. To do so, left-click to one corner of an imagined box in the main grid, which you want to enlarge and draw the so called zoom-box with the mouse. While you keep the left mouse button pressed and you move the mouse, you will see the zoom-box. In the following example, we attempt to zoom into the measured filter's passband:



As soon as you release the mouse button, the zoom-box will be enlarged to completely fill the main grid.



You see the zoomed passband. Note, that I have added **bandwidth markers** in the above image as an example.

▸▸▸ **Note:** You can do multiple **consecutive zooms**.

▸▸▸ **Note:** When zooming, the measurement span and measured number of data points remains untouched. Only part of the data is displayed. When sweeping in zoomed state, you still sweep the whole unzoomed frequency range, but only part of it is displayed.

▸▸▸ **Note:** You cannot zoom inside the Smith chart. But as you can see in above screenshot, the Smith chart data outside the zoomed frequency range is greyed.

▸▸▸ **Note:** You can **unzoom** by right-clicking the **zoom label** near mouse pointer. If you right-click it, the above shown **unzoom menu** pops up.

You can
- ***Undo last zoom*** = return to the state before the last of consecutive zooms
- ***Unzoom*** = restore the original x- and y-scales before all consecutive zooms
- ***leave zoom-mode keep y-scale*** = restore the full frequency span but keep the zoomed vertical scale
- ***leave zoom-mode keep all scales*** = keep the zoomed frequency span and the zoomed vertical scale. The outside data is lost, the visible data is interpolated to the full number of data points grid. A sweep after this will only sweep the visible frequency span.
- ***Overlay Unzoomed*** = if selected, overlay an unzoomed and greyed version of the zoomed data with the zoomed range highlighted.

▸▸▸ **Hint:** You can also unzoom by right-clicking the main graphics grid.

The VNWA software allows to perform a realtime **FFT** on measured or imported data from frequency to time domain, display data in time domain, manipulate data in time domain by **gating** and perform an **inverse FFT** back to frequency domain.

A special topics section on time domain measurements demonstrates examples of both usages:

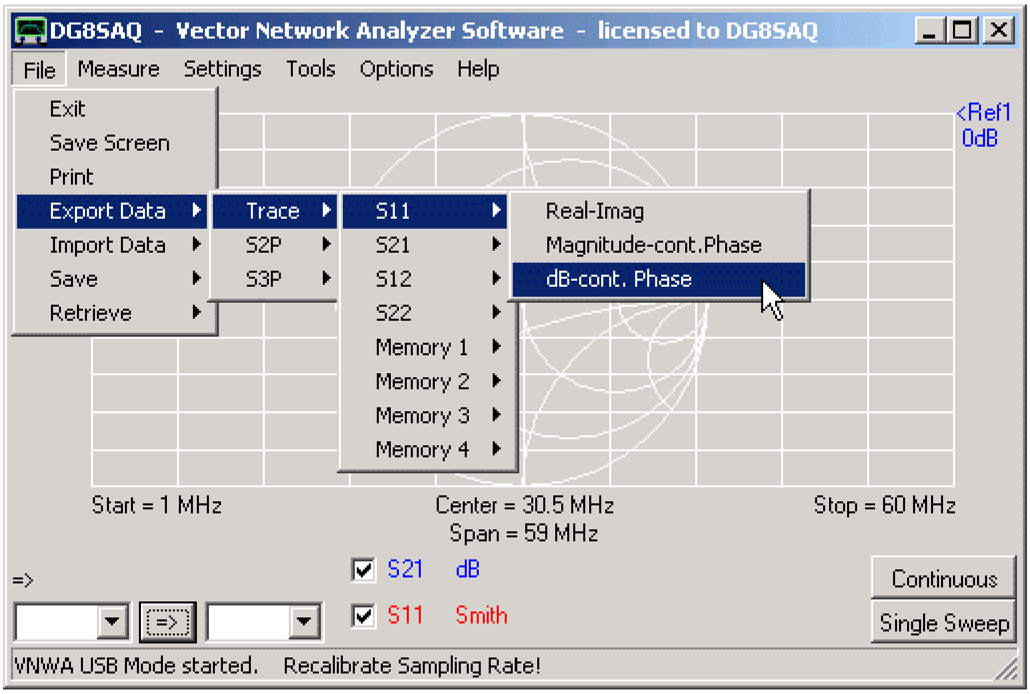
Time domain reflectometry is a useful technique to search defects in transmission lines.

The step response is useful to determine impedance variations along a coaxial cable.

Gating is used to separate responses depending on their arrival times, e.g. to separate the slow mechanical response of a crystal filter from the fast electromagnetic feedthrough of the test board.

The VNWA main menu **"File"** offers the following functions:

Exit
Save Screen
Print
Export Data
Import Data
Save
Retrieve



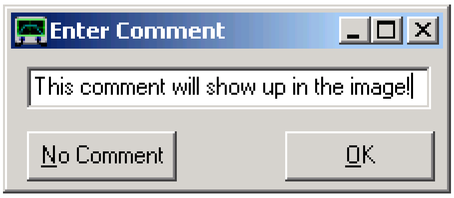
Exit

Exit the VNWA software. All settings will be saved for the next session.

Unsaved data will be lost!!!.

Save Screen

Save the current main graphics window into a bmp, jpg or png file. When saving, you will first be asked for a file name, then an input mask for an optional image comment pops up:

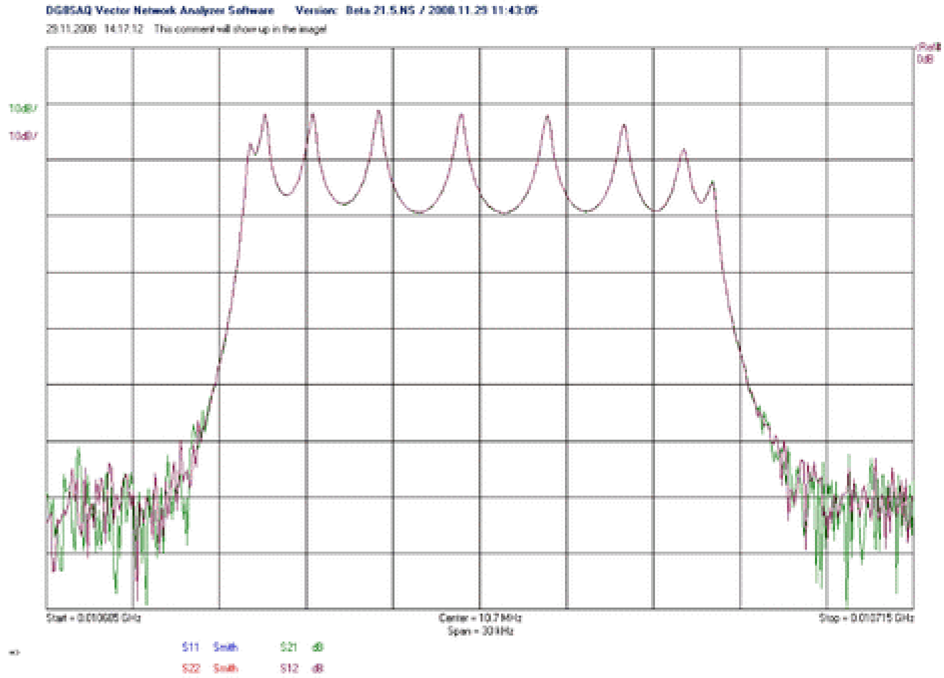


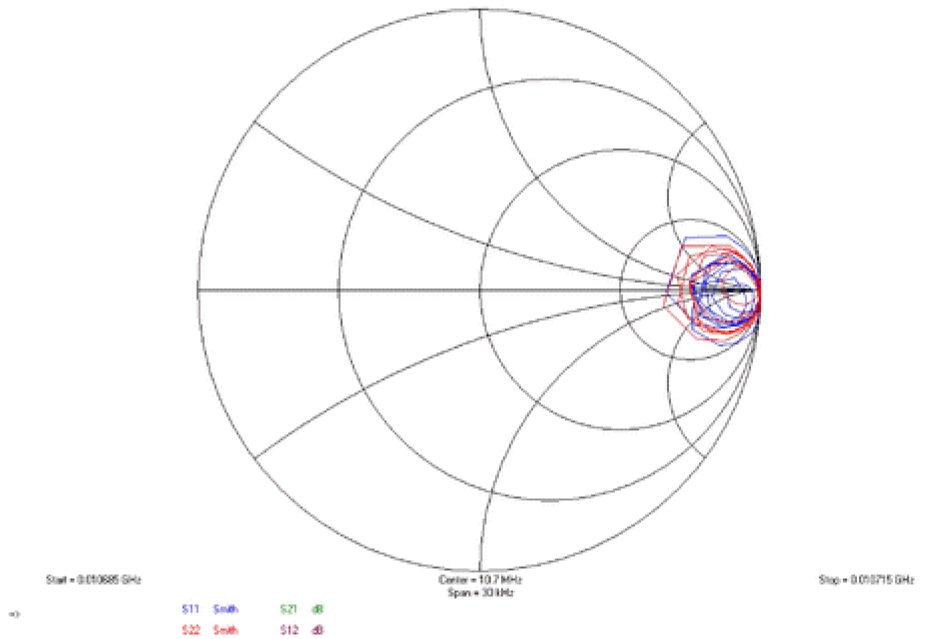
Enter a **comment** or press no comment. The exported image with the above comment will look like this:

Note, that the buttons and menus are suppressed. The image size in pixels is identical with the VNWA window client size. If you want the saved image to be bigger, increase the VNWA window size.

Print

Print the current main graphics window. Like in "Save screen", you can enter a comment showing up in the printout. The printout will look like this.

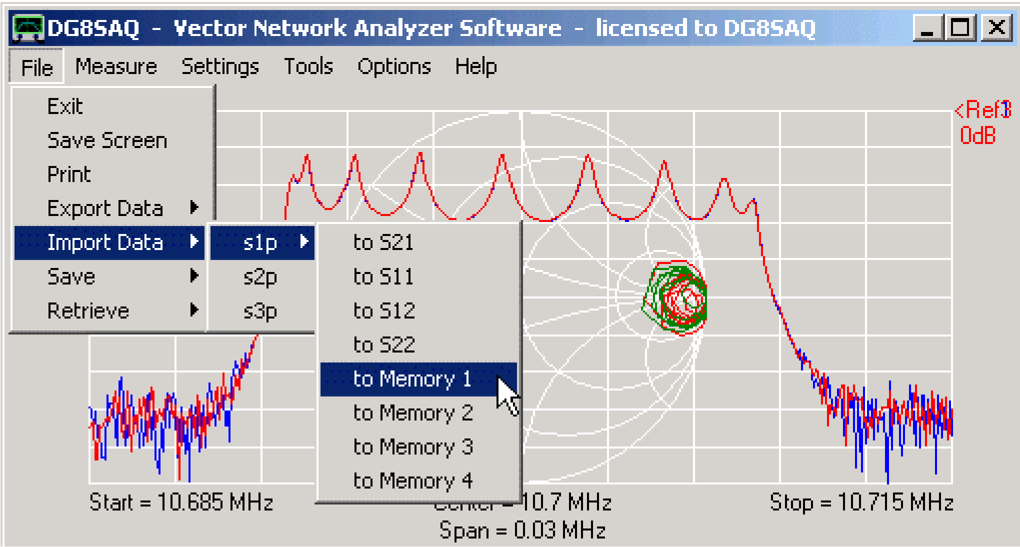

DG8SAQ Vector Network Analyzer Software Version: Beta 21.5.NS / 2008.11.29 11:43:05
29.11.2008 14:17:12 This comment will show up in the image!
10dB/
10dB/
Start = 0.010685 GHz
Center = 10.7 MHz
Span = 30 kHz
Stop = 0.010715 GHz
S11 Smith
S21 dB
S22 Smith
S12 dB


Start = 0.010685 GHz
Center = 10.7 MHz
Span = 30 kHz
Stop = 0.010715 GHz
S11 Smith
S21 dB
S22 Smith
S12 dB

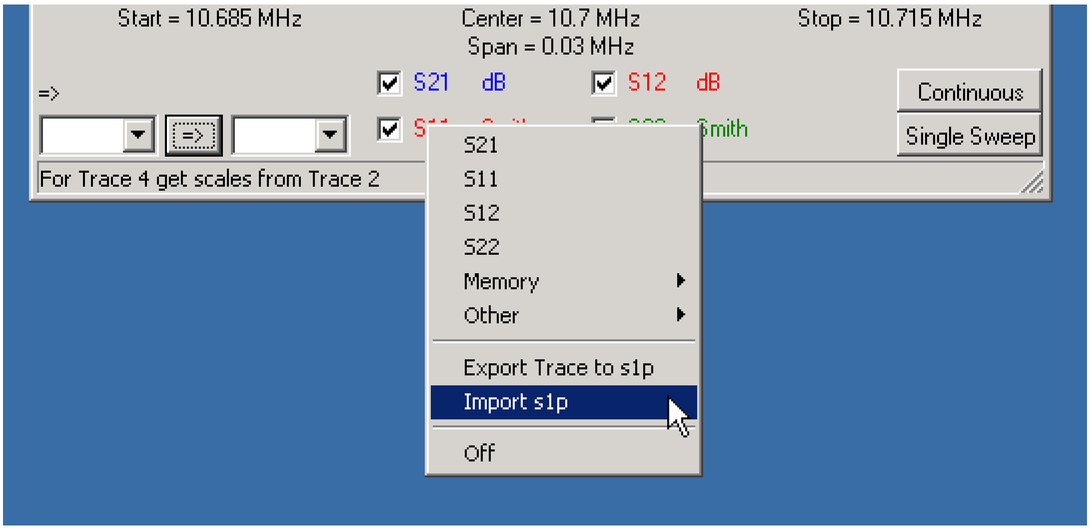
Import Data

VNWA can import certain S-parameter files in **Touchstone** format (s1p, s2p, s3p) and csv-format. To do so, select the main menu **"File"-"Import Data"**-... .

▸▸▸ **Note:** Importable files must contain S-parameters normalized to 50 Ohms.



▸▸▸ **Hint:** You can directly import trace data into a specific memory space by right-clicking the trace label (e.g. S11 in below example) and selecting "Import s1p". Thus, the s1p file wiull be loaded into the S11 memory space.

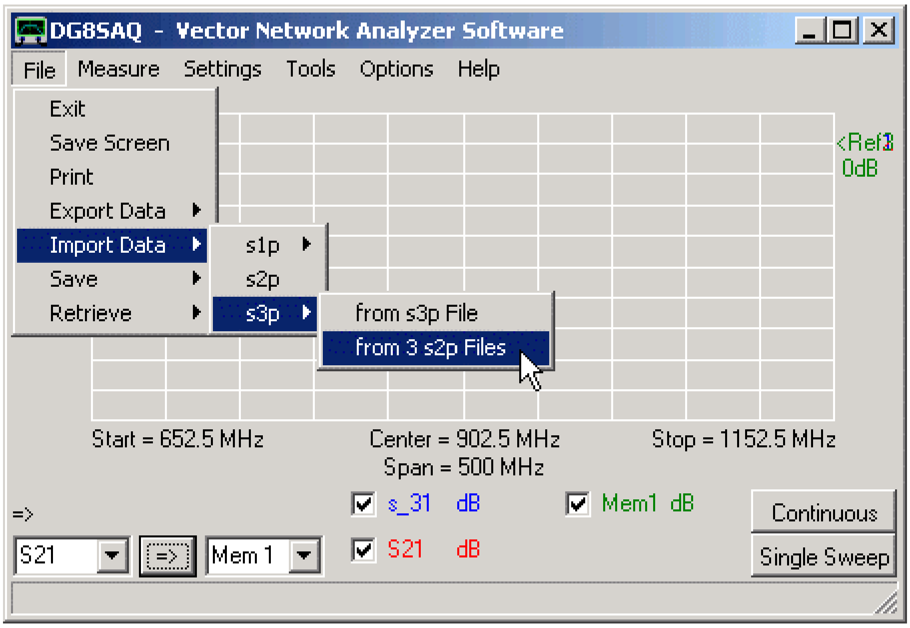


▸▸▸ **Hint:** You can also import S-parameters by dragging s1p, s2p or s3p-files from any file browser and dropping it onto the VNWA application.

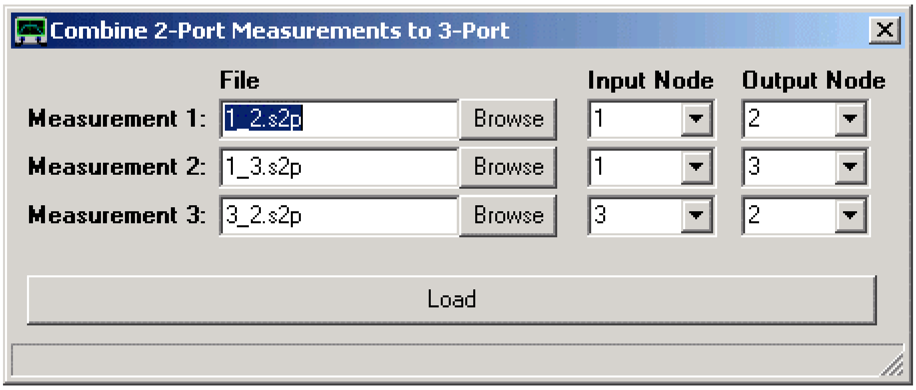
▸▸▸ **Hint:** VNWA will accept s1p, s2p or s3p file names as runtime parameter. A second numeric parameter specifies the data space where an s1p record is stored (1:S21, 2: S11...). Thus, if you associate e.g. the file ending s2p with VNWA.exe, then you can start VNWA and load an s2p file by double-clicking onto the file.

▸▸▸ **Hint:** You can also import 3-port data from 3 2-port measurements with the third port terminated by 50 Ohms.

It is possible to automatically combine 3 2-port measurements or data files to 3-port data. Note, that the unused port must be terminated with 50 Ohms in order to obtain valid 3-port S-parameters.
To do so, select the menu highlighted below:



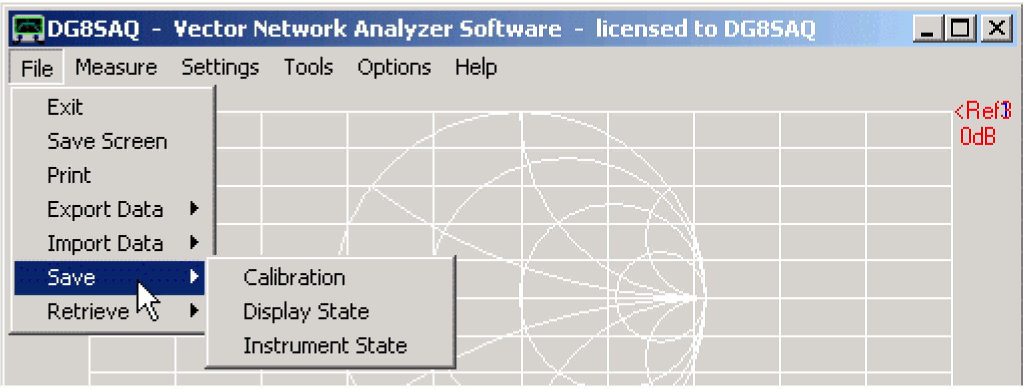
In the popping up window, you may specify 3 filenames and the corresponding DUT port numbers.



Pressing the **load button** will load the 3-port buffer.
Note, that in above example S33 from the second file import will be overwritten by S33 from the third file import.

Save:

VNWA allows to save various software states:



Save Calibration

You can save a calibration to a *.cal file. You are asked for a file name.
If the instrument is not calibrated, the command will be ignored.

Save Display State

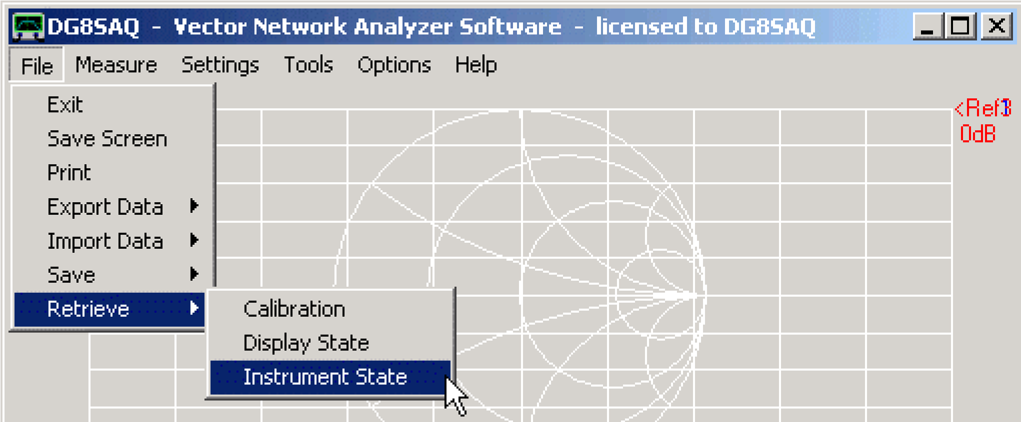
You can save the display state, i.e. the color scheme, displayed kinds of traces, grids and units into a file. You are asked for a file name. If you enter a file extension, it will be ignored, as the extension of the backup file is predefined.

Save Instrument State

You can save the complete instrument state including display state, hardware setup, calibration, multiplier table... into a set of files. You are asked for a file name. All backup files will start with the name specified by you. Predefined extensions will be added automatically.

▸▸▸ **Note:** The master calibration will NOT be saved!

VNWA allows to save and retrieve various software states:



Retrieve Calibration

You can retrieve a previously saved calibration from a *.cal file. When invoking this command, a file browser opens to select the *.cal file.

▸▸▸ **Warning:** If the number of data points or the frequency span of the calibration differs from that of your displayed data, **all data in the display buffers will be lost**.

Retrieve Display State

You can retrieve a previously saved display state, i.e. the color scheme, displayed kinds of traces, grids and units from a backup file. A file browser will open to assist you selecting the backup.

Retrieve Instrument State

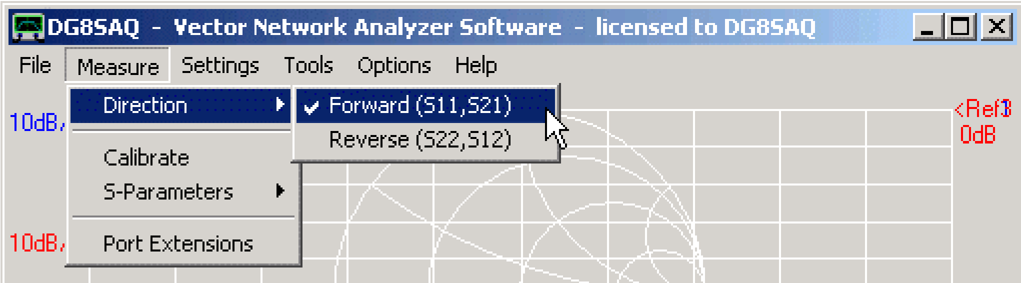
You can retrieve a previously saved complete instrument state including display state, hardware setup, calibration, multiplier table... from a set of backup files. You are asked for a file name. A file browser will open to assist you selecting the backup. You only need to select one of the backup files. The remaining files will automatically be found. Missing backup files will be irnored.

▸▸▸ **Warning:** If the number of data points or the frequency span of the calibration differs from that of your displayed data, **all data in the display buffers will be lost**.

▸▸▸ **Note:** The master calibration will NOT be overwritten!

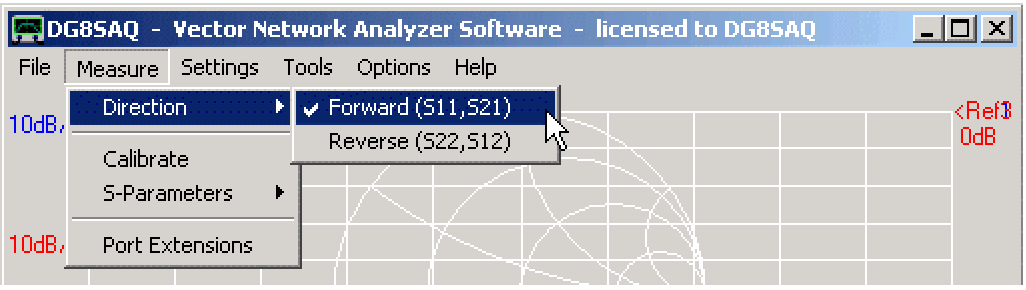
The VNWA main menu **"Measure"** offers the following functions:

Direction
Calibrate
S-Parameters
Port Extensions



Direction:

Use this menu item to select the measurement **direction forward** or **reverse**.



▸▸▸ **Hint:** The measurement direction can also be altered by clicking on the arrow label **=>** on the lower left side of the VNWA main window:

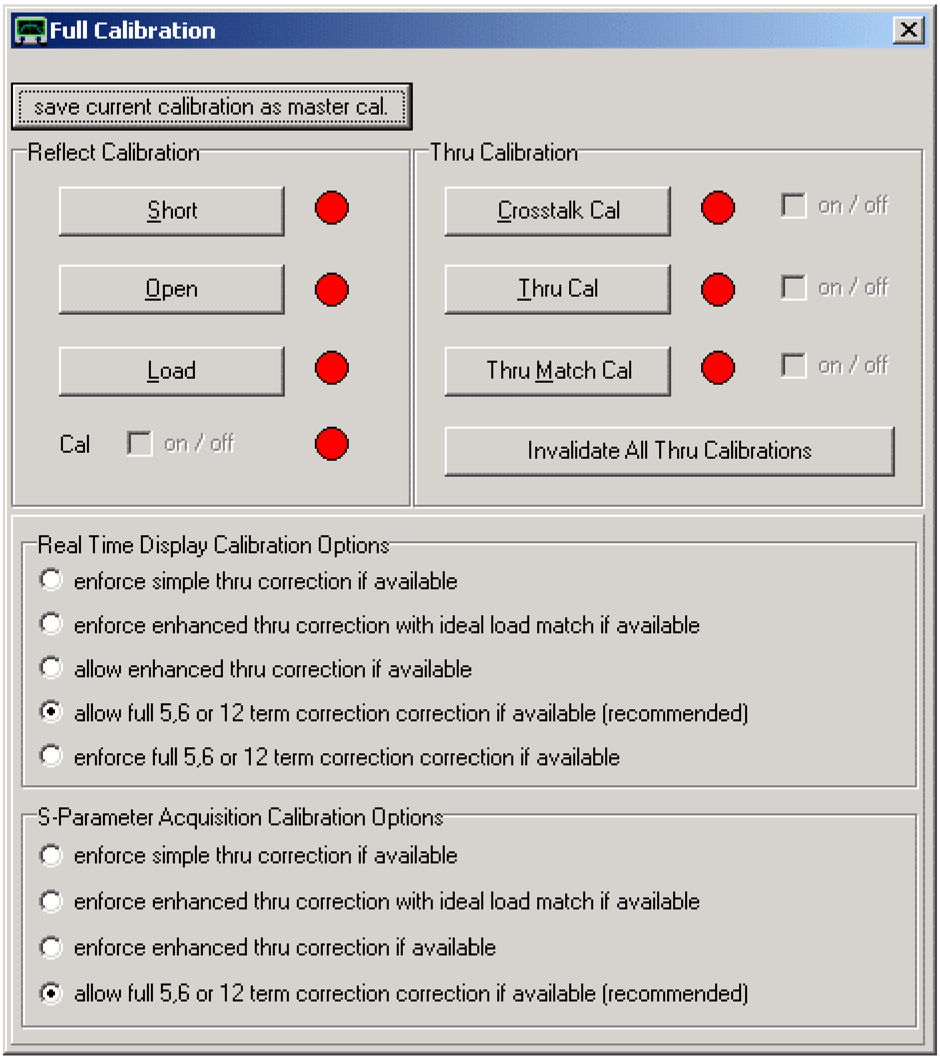
▸▸▸ **Note:** Without usage of an **S-parameter test set** the direction menu only determines, in which data spaces the measured data is stored. The function of the Tx port and Rx port remains unchanged. This means, **the VNWA2.x cannot automatically alter the signal flow direction.**
If you want to measure S12 and S22 in this case, you need to **manually** turn the DUT around, e.g. **exchange input and output terminals of the DUT**.

▸▸▸ **Note:** The VNWA software supports usage of an S-parameter test set, which actually allows for automatic change of signal flow direction. If activated, the direction menu will actually alter the signal flow direction.

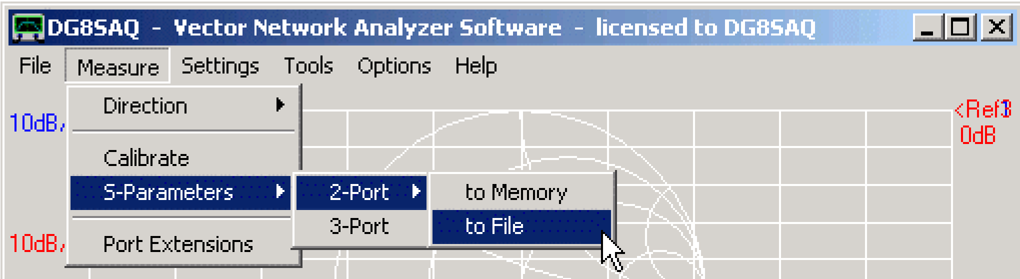
▸▸▸ **Note:** The direction menu will be disabled in the presence of an S-parameter test set **with automatic direction switching** being activated.

Calibrate:

The "Calibrate" menu allows to calibrate the VNWA using calibration standards. A detailed calibration guide can be found here.



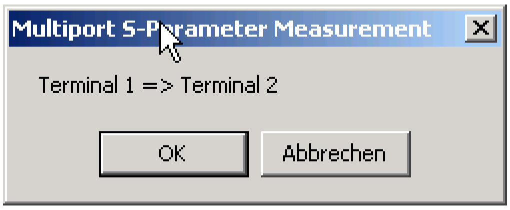
This menu allows you to **measure S-parameters of a 2-port or a 3-port device**, e.g. of a crystal filter (2-port) or a balun (3-port).



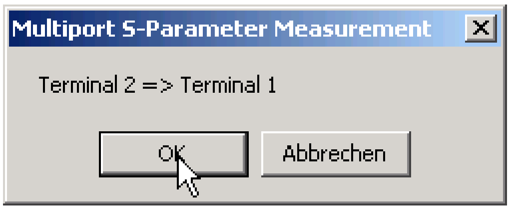
2-Port

You can select, if you want to store the acquired data in the internal s2p-buffer (=memory) or additionally store it in a file on your hard disk.
▶▶▶ **Note:** Acquired or loaded 2-port S-parameters will be stored in the VNWA data spaces (S21,S11,S12,S22) and **additionally in an internal s2p buffer**. This is necessary for recalculating S-parameters to new varying normalization impedances, e.g. like in the matching tool.

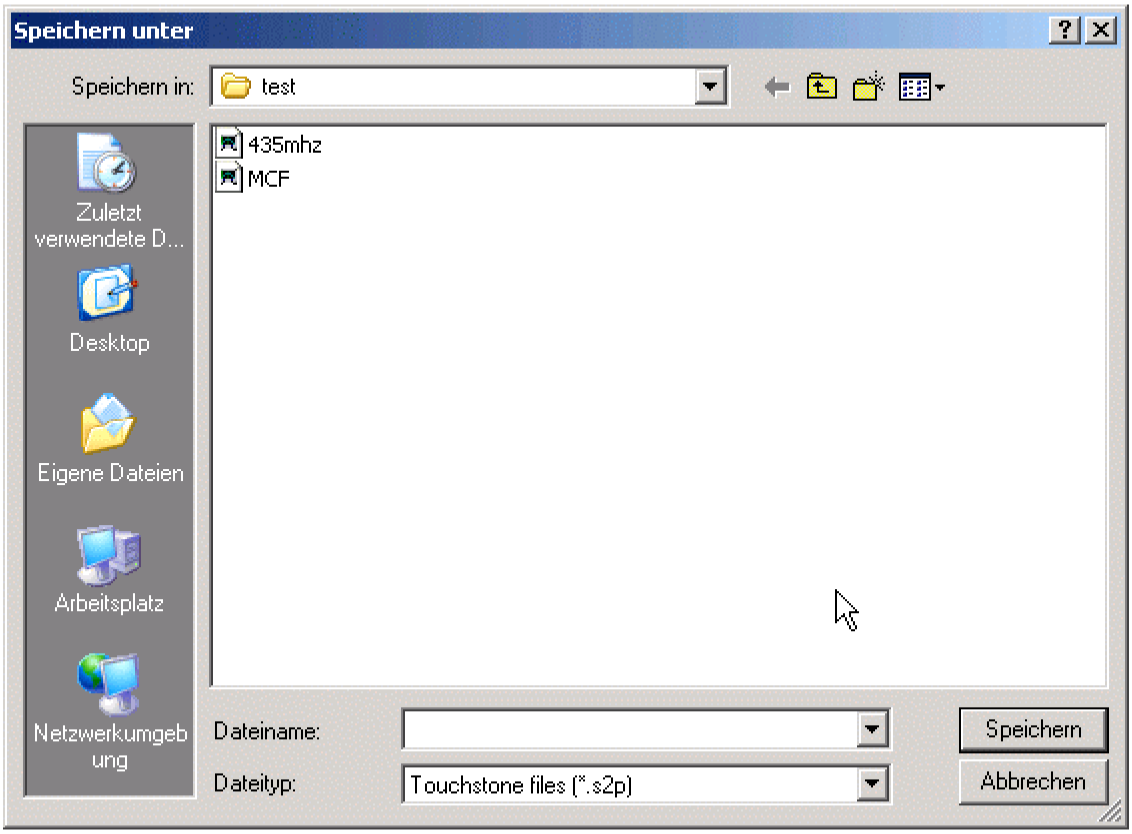
If you invoke the measurement, you need to confirm by pressing the **ok**-button, that you are ready to measure.



After the forward measurement (S11,S21) has finished, you need to manually turn the DUT and confirm with the **ok** -button again, that you are ready for the reverse measurement.



If you have selected to measure **"to File"** then you are asked to select a file name:

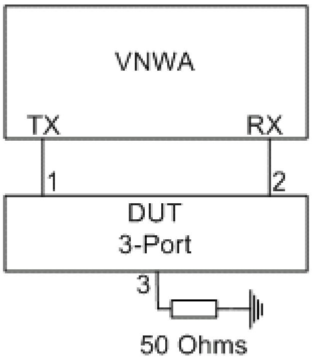


The data is saved in Touchstone s2p format.

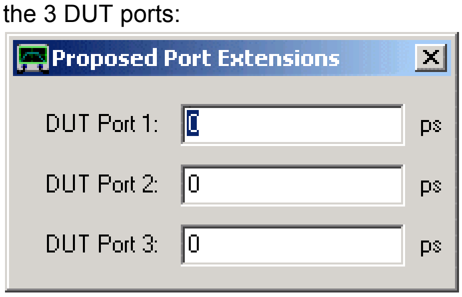
▸▸▸ **Note:** If you have decided to measure **"to Memory"**, you can still save your measured data manually.

3-Port

3-port measurements basically work like 2-port measurements, except that **you must terminate the unused port with a 50 Ohms resistor**:



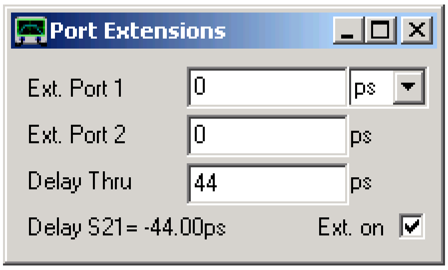
▸▸▸ **Note:** If port extensions are activated upon measurement start, you may specify 3 different port extensions for the 3 DUT ports:



▸▸▸ **Note:**Measurement data will be collected into an internal s3p buffer (=memory containing s11,s12,s13,...,s33). There is a distinction between e.g. **S11** (2-port data) and s11 (3-port data). These are stored in different memory spaces. This is necessary when analyzing 3-ports by reducing them to a 2-port with a balanced port by means of the 3-port analyzer tool.

▸▸▸ **Note:** You can save your measured data manually to a 3-port Touchstone *.s3p-file .

Here, you can add **delays** to your measured data as well as account for the **finite length of your Thru calibration standard**.



▶▶▶ **Example:** You perform a calibration at the end of your test cable. But between your test cable and you DUT, there might be another low loss 50 Ohms transmission line (e.g. a stripline on the test PCB). If you want to know the reflection coefficient of your DUT without the additional transmission line, you can simply enter the signal delay caused by the additional line into the **"Ext. Port 1"** field and check the box **"Ext. on"**. The delay of the transmission line can easily be found by creating a temporary short circuit at the DUT and tune "Ext. Port 1" such that the measured reflection coefficient shows up at the short circuit point in the Smith chart.

▶▶▶ **Note:** Entered delays are **one way delays**. Reflection data will be corrected by 2x the delay, as reflected signals travel through the delaying transmission line twice (forth and back!). Transmission data will only be corrected by 1x the delay as a transmitted signal will pass the delaying transmission line only once.

▶▶▶ **Note:** Positive delays move the calibration planes further away from the VNWA, negative delays move them closer to the VNWA.

▶▶▶ **Hint:** You can also tune the delays with the **mouse-wheel**.

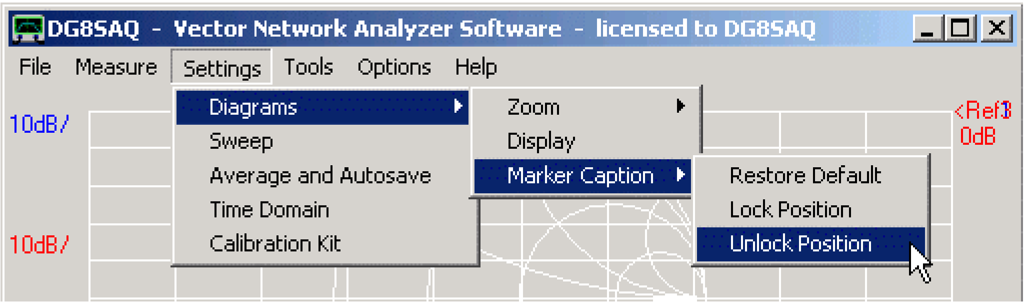
▶▶▶ **Note:** Delays for forward and reverse measurements can be chosen independently.

▶▶▶ **Note:** The "Delay Thru" is the delay of your Thru standard and should not be changed once it is determined.

▶▶▶ **Note:** All delays also influence the phases of the transmission measurements.

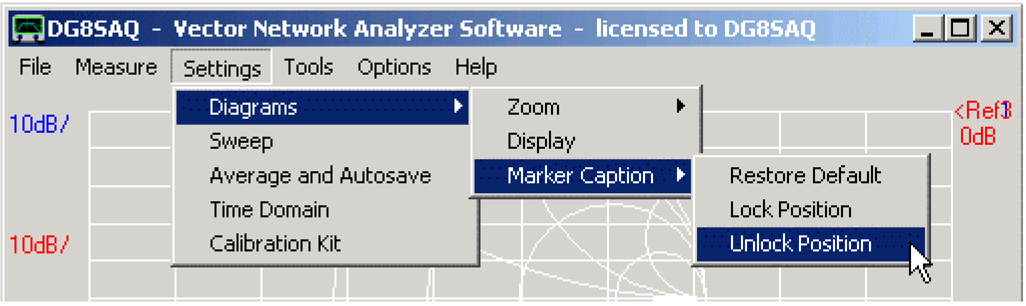
The VNWA main menu **"Settings"** offers the following functions:

Diagrams
Sweep (VNWA)
Sweep (N2PK-VNA)
Average and Autosave
Time Domain
Calibration Kit



The VNWA main menu **"Settings - Diagrams"** offers the following functions:

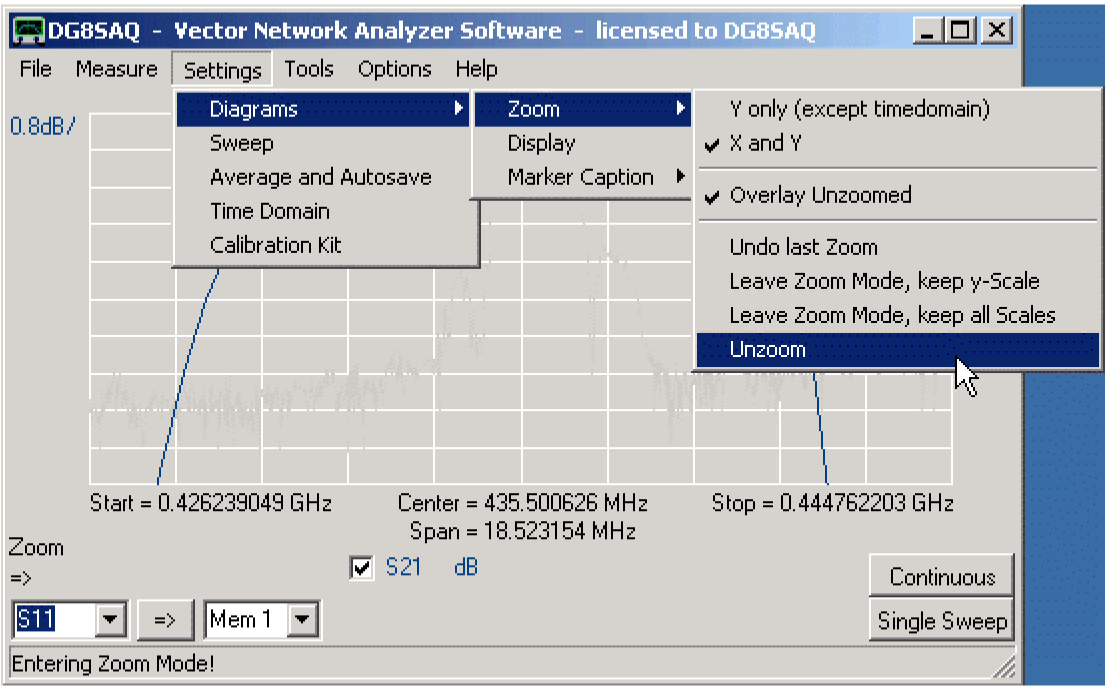
Zoom
Display
Marker Caption



The **zoom** settings allow to control **box-zooms**, which can be accomplished by drawing a box with the mouse keeping the left mouse button pressed around the trace feature of interest.

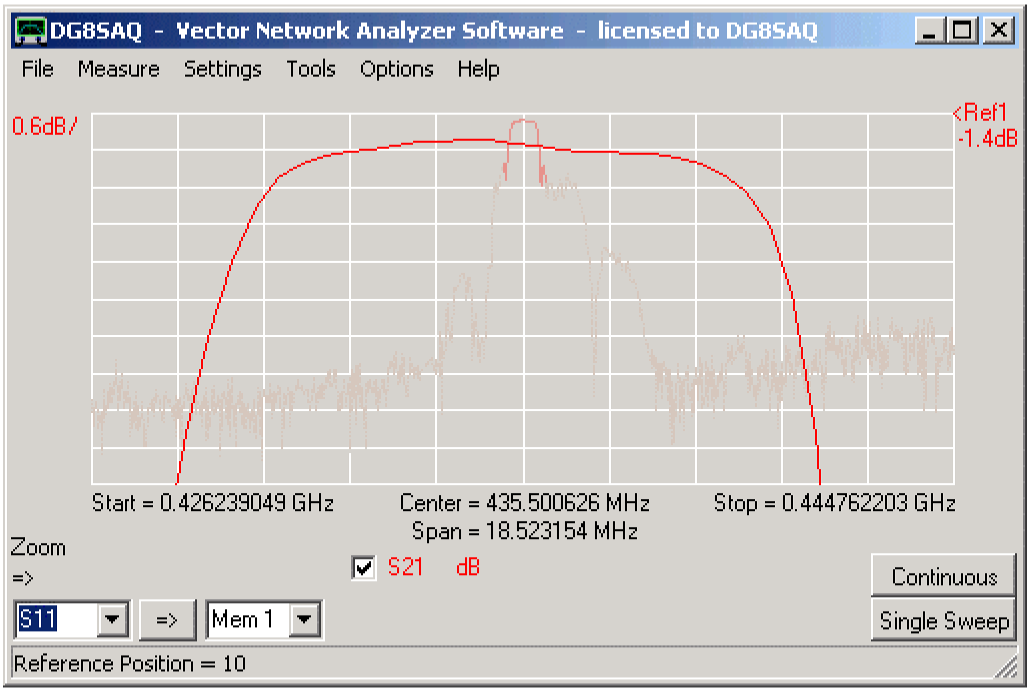
▸▸▸ **Note:** Zooming into Smith charts is not implemented.

▸▸▸ **Note:** The zoom menu items below the separator line are only visible in zoom mode, i.e. when a box zoom has been performed. **Zoom mode is indicated by the "Zoom-label"** on the lower left of the main window (see below screenshot).



Choices:
- ***Y only ...*** = only zoom vertically, leave the frequency axis unchanged (not applicable in time domain mode).
- ***X and Y*** = zoom frequency axis and y-axis

- ***Overlay Unzoomed*** = If selected, a greyed version of the unzoomed data is displayed with the zoom range highlighted together with the zoomed trace:

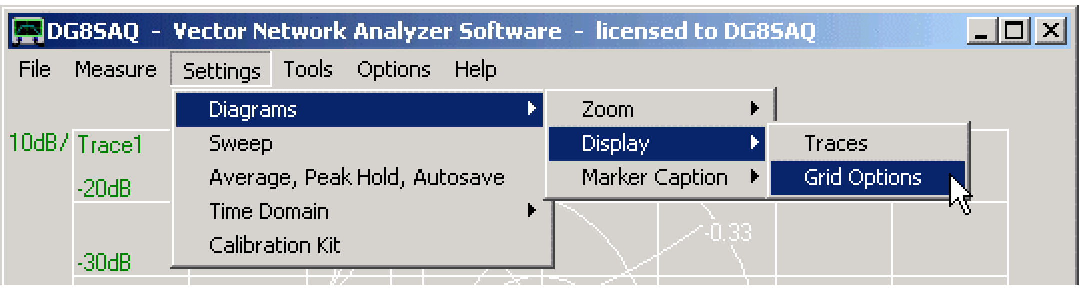


The following choices are only visible in zoom mode:
- ***undo last zoom*** = restore the x- and y-scales of before the last zoom
- ***leave zoom-mode keep y-scale*** = restore the full frequency span but keep the zoomed vertical scale
- ***leave zoom-mode keep all scales*** = keep the zoomed frequency span and the zoomed vertical scale. The data outside the last zoom box is lost, the visible data is interpolated to the full number of data points grid. A sweep after this will only sweep the visible frequency span.
- ***unzoom*** = restore the original x- and y-scales of before all consecutive zooms

▸▸▸ **Hint:** You can also unzoom by right-clicking the main graphics grid.
▸▸▸ **Hint:** You can also unzoom by right-clicking the zoom label on the main window.

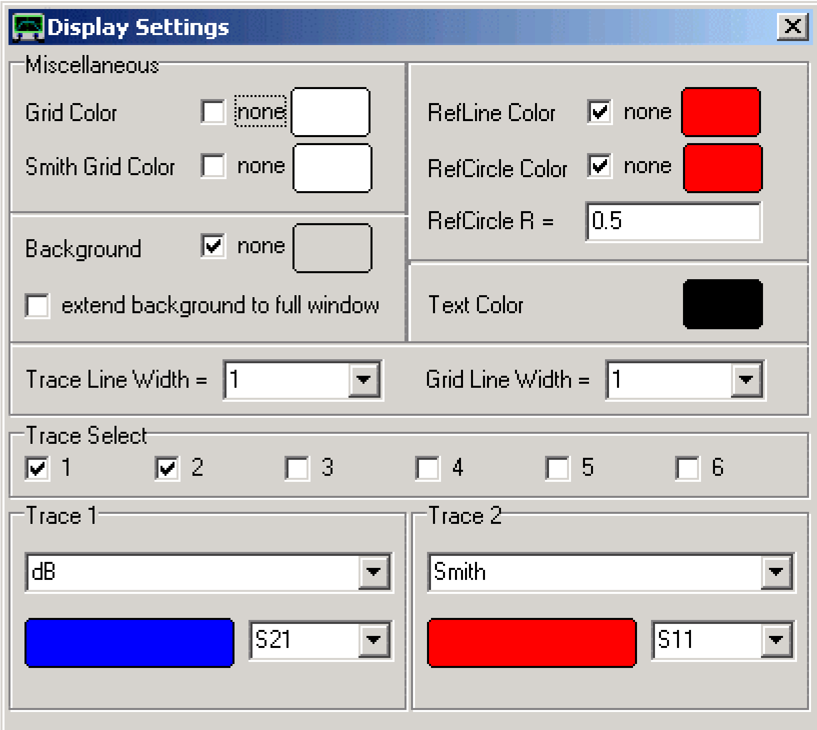
The VNWA main menu **"Settings - Diagrams - Display"** offers the following functions:

Traces
Grid Options



The **"traces"** settings allow to **control most displayed features on the main window**.

On clicking the menu item **"Settings - Diagrams - Display - Traces"**, the display settings window will open:



In the upper part of the miscellaneous panel the displayed**colors** can be configured. If "none" is selected, the feature is either not displayed or the standard windows color (grey) is used. To change a color, click onto the appropriate color field and select the desired color from the popping up color select menu.

▸▸▸ **Hint: Two horizontal freely movable reference lines** can be displayed. If these are not needed, check "RefLine Color " "none". These lines can either be positioned in the y-scale menu (accessed by double-clicking the y-scale labels (D,E) or by dragging them up or down with the mouse.

▸▸▸ **Hint: A user definable reference circle** can be displayed in the Smith chart. If it is not needed, check "RefCircle Color " "none". The circle diameter can be specified by the "RefCircle R" parameter. R = 1 puts the circle at the edge of the Smith chart.

▸▸▸ **Hint:** The traces' and grids' **line widths** can be specified to improve visibility outdoors in the sunlight. The default linewidth is 1 (= 1 pixel, thinnest lines)

The **"trace select checkboxes"** select, which traces (=curves, maximum 6) are displayed.
When checking one of the traces, an according trace information field pops up, where the **trace type** (S21, S11, ...) and the **display type** (dB, Smith...) can be selected.

Available trace types:

S21, S11, S12 , S22	=> measured data
Memory 1...4	=> stored data
Unitarity	=> $\lvert S11\rvert^2+\lvert S21\rvert^2$ = fraction of power not dissipated
M.3/M.4	=> Memory 3 / Memory 4
S21/M.3	=> S21 / Memory 3
S11/M.4	=> S11 / Memory 4
TimeDomain	=> inverse FFT of selected data space, see example.
Gated F-Domain	=> FFT of gated time response, see example.
Custom1...6	=> arbitrary mathematical expression containing data space variables, see example.
s11,s12,...s33	=> 3-port S-parameters

Available display types:

dB	
Re dB	=> real part shown in dB
Im dB	=> imaginary part shown in dB
smith	
VSWR	
sVSWR	=> signed VSWR, negative for reflection coefficients > 1
real part	
imaginary part	
lin. magnitude	
phase	=> range -pi ... pi
continuous phase	=> extended range such that phase function becomes continuous
-cont. phase/f	=> minus continuous phase / frequency
group delay time	

The following display types are only visible for trace types S11 and S22. Z denotes the **impedance** and Y the admittance calculated from the reflection coefficient.

real Z	
imag Z	
| Z |	
C --	=> series equivalent circuit capacitance, series resistance is real Z.
L --	=> series equivalent circuit inductance, series resistance is real Z
real Y	
imag Y	
| Y |	
R ||	=> parallel equivalent circuit resistance
C ||	=> parallel equivalent circuit capacitance
L ||	=> parallel equivalent circuit inductance
QC	=> quality factor of capacitor
QL	=> quality factor of inductor
smith renormalized	=> smith chart renormalized to user selectable complex impedances (power wave representation), see example

▶▶▶ **Hint:** Trace types and display types can also be changed in the main VNWA window by right-clicking the trace-type labels (K) and the display type labels (L) respectively.

▶▶▶ **Hint:** The **display settings menu** can also be accessed by double-clicking the trace select checkboxes (J), the trace-type labels (K) (except for trace type "custom", where the custom menu is invoked and for trace types "time" and "gated", where the time domain settings window is invoked.) and the display type labels (L) .

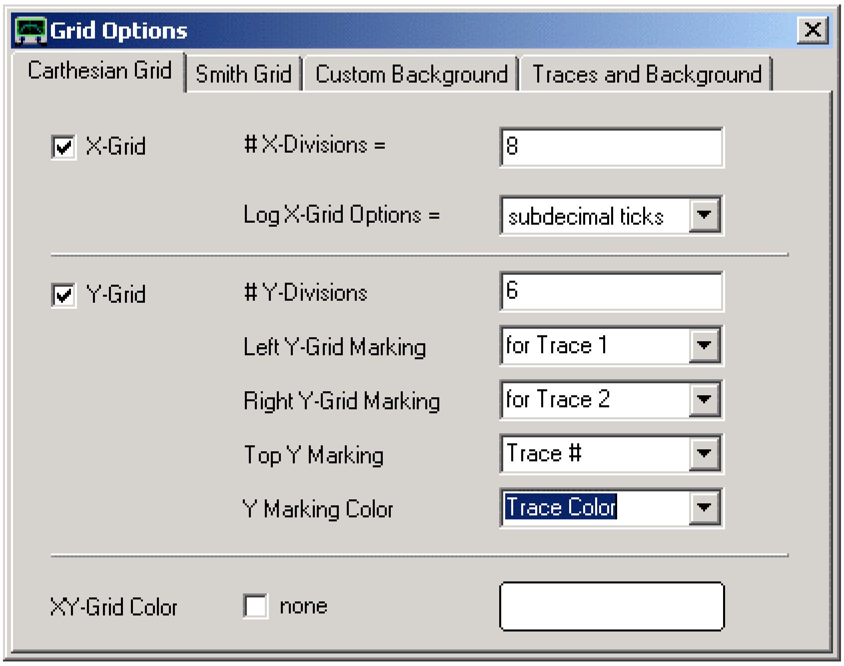
▶▶▶ **Note:** Traces can be switched on or off with the trace select checkboxes (J).

The **"Grid Options"** settings allow to **control the appearance of the x-y grid (=Carthesian grid) and the Smith grid**. Moreover, the **trace and the background appearance** can be controlled here.

On clicking the menu item **"Settings - Diagrams - Display - Grid Options"**, the grid options window will open.
▸▸▸ **Hint:** The same functionanity can be accessed by **right-clicking the grid** on the main VNWA window and selecting **Grid Options**.

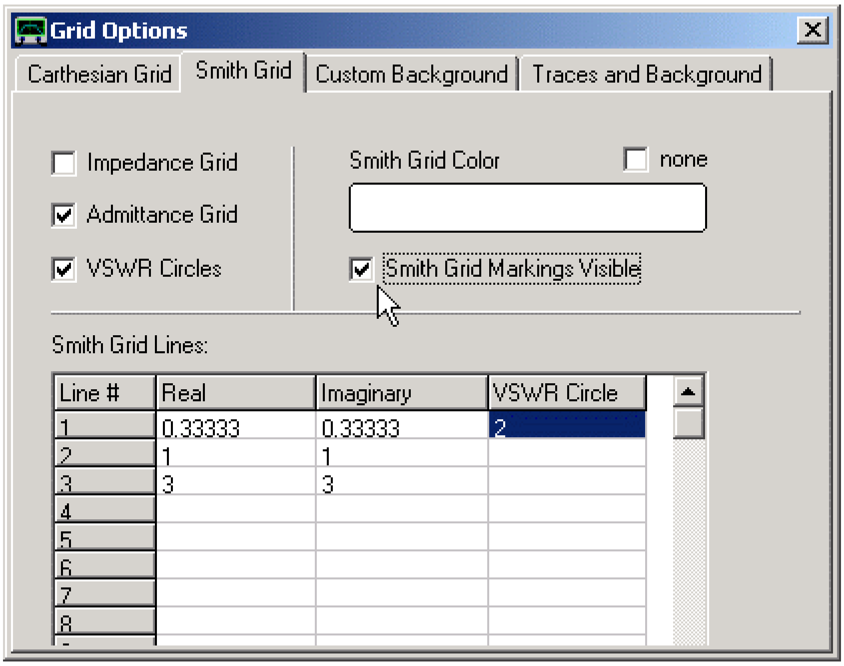
Many of the grid options are accessible in other windows, e.g. in the trace settings. All grid related settings are summarized here.

Carthesian Grid Options:



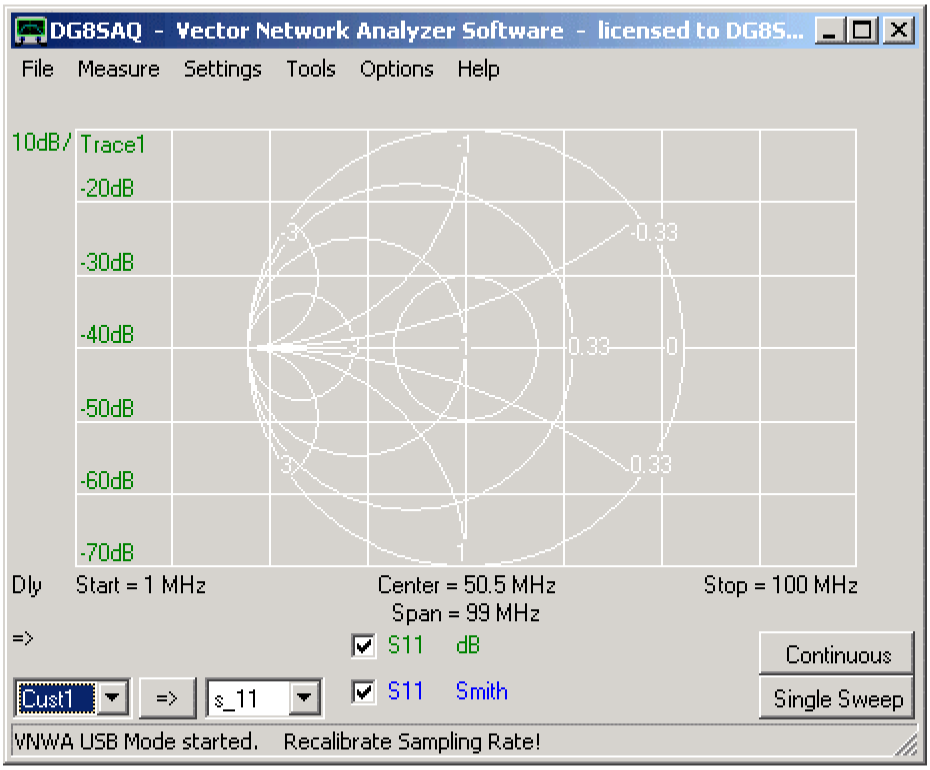
New grid options only accissible here are **left and right grid markings**, which can be chosen for two traces and which can be configured to show the trace number and to appear in the trace color.

Smith Grid Options:



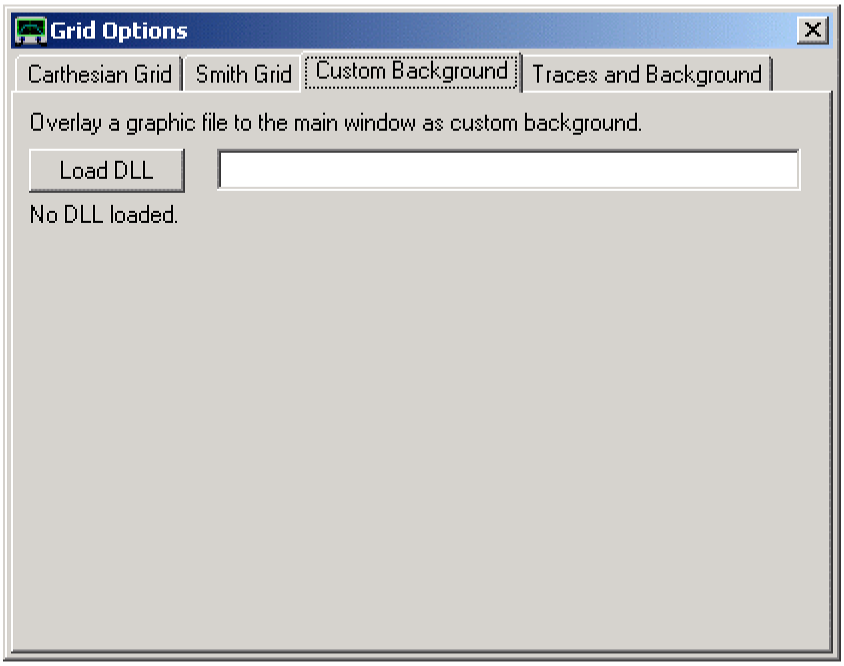
The Smith grid options allow to configure the Smith grid in various new ways:
- An **admittance Smith grid** (see below screen shot) can be selected in addition or alternatively to the default impedance Smith grid .
- **VSWR circles** can be added.
- All **grid lines are fully user configurable**. You can add or remove grid lines by adding or removing entries in the Excel-like table.
- **Smith grids** can be configured to show **markings** (see mouse pointer in above screenshot and below screen shot).

With above settings, the main window appears like this:



Note that the carthesian grid does not show right markings here, since trace 2 does not show in the carthesian grid but in the Smith grid.

Custom Background Options:

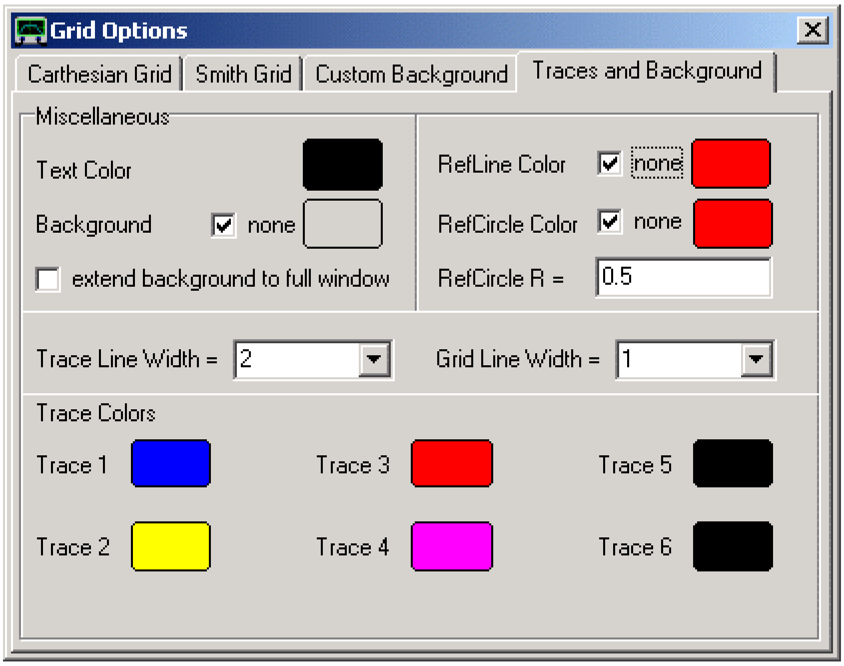


The custom background options allow to **load an image to appear as background on the VNWA main window**.
Image file import is implemented in an optional external dynamic link library (dll) to avoid incompatibilities with older Windows versions (Windows98 and Windows2000 do not support GDI, which is used to read and display *.svg-files).
Two different dlls are provided for reading image files:

svg_dll.dll supports import of scalable vector graphics files (*.svg)
gfx_dll.dll supports import of most pixel graphics files like *.bmp, *.jpg, *.png... .

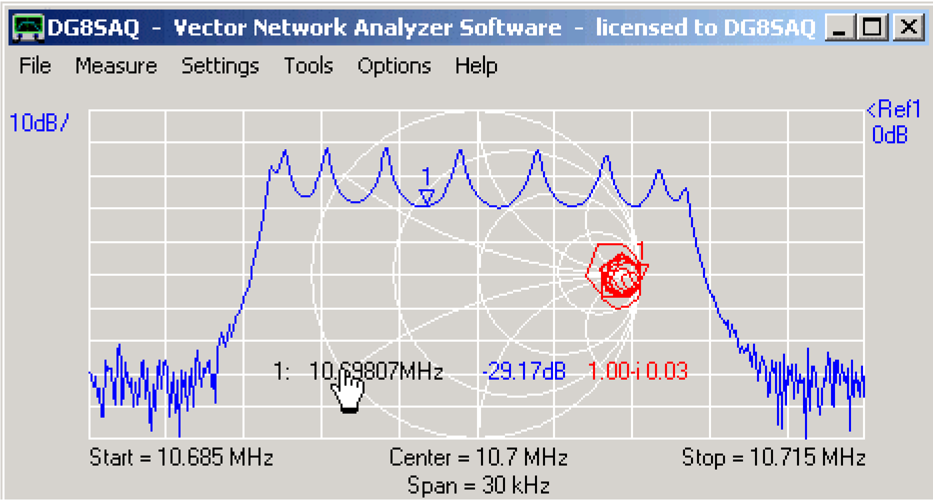
Before importing an image, an appropriate dll-file must be loaded by pressing the "Load DLL" button and selecting the dll-file.
Once the dll is loaded, the controls for loading and manipulating an image appear.

Traces and Background Options:

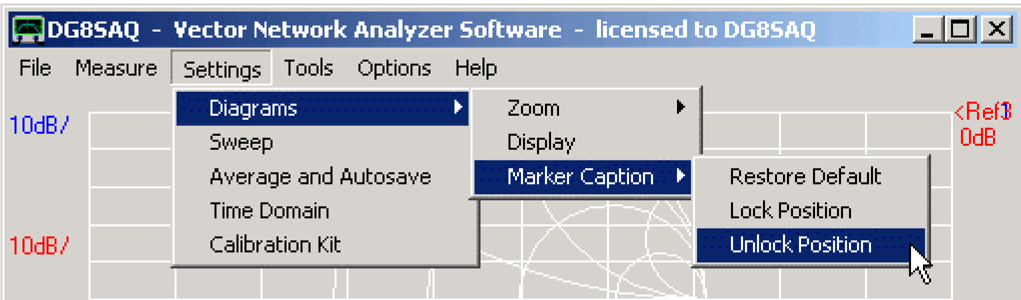


The traces and background menu allows to set the colors of traces, grid, background and reference lines and circles and do some related settings. This menu is redundant to the "Trace Options" menu.

▶▶▶ **Note:** The marker captions can be moved on the main window by pointing onto the frequency part and dragging the captions around with the mouse, while keeping the left mouse button pressed.

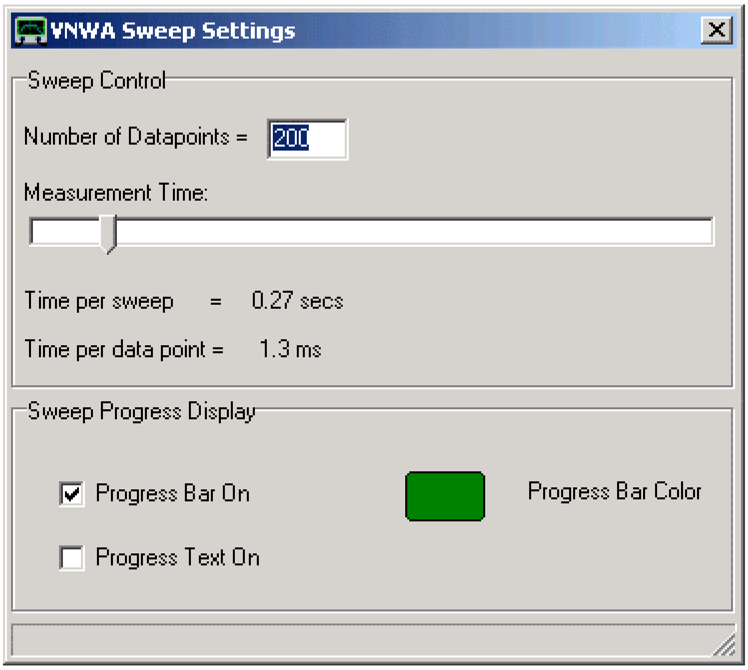


The **"marker caption"** settings allow to control the behavior of the marker captions on the main window.



restore default = places the marker captions to the default position in the upper left corner and makes it movable.
lock position = locks the marker caption position such that it cannot be moved with the mouse
unlock position = makes marker caption movable by the mouse after "lock position" again

The **"sweep settings"** allow to specify the **number of measured data points** as well as the **measuring time per data point**. From these values, a best guess **sweep time** is calculated, but as Windows is often doing unpredictable things, usually the sweep time is a bit longer in LPT mode. The prediction for USB mode is accurate.

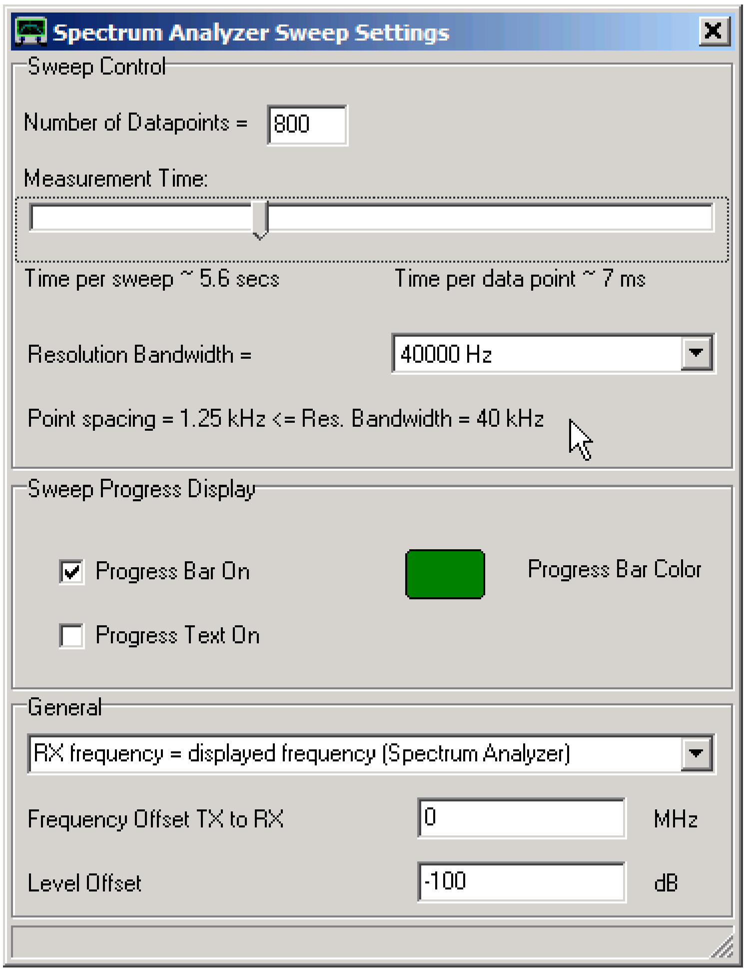


▸▸▸ **Note:** Except for the fastest (<=1ms) time settings, **50% of the specified time are reserved for the DUT to settle** to the new frequency. The remaining 50% are used for measurement. In the <=1ms cases, the DUT settling time is specified in the setup window by the # of presamples.

▸▸▸ **Note:** Sweep rates below 1ms / frequency point are only available in USB mode.
▸▸▸ **Note:** The sample rate slider will block on the left at the fastest possible sweep rate. If you want to sweep faster you need to increase the IF by decreasing the #samples per IF period and/or increase the sample rate.

Two kinds of **sweep progress indicators** can be switched on or off, namely a graphical progress bar appearing at the lower edge of the main VNWA window grid and a progress text appearing in the status line at the very bottom of the VNWA main window.

In **SA-mode (=spectrum analyzer mode)** the **"sweep settings"** allow to specify various settings.



1. The number of measured data points.
▸▸▸ **Warning:** Note, that in SA mode (unless used with tracking generator) the **frequency point spacing must be smaller than the resolution bandwidth**! If this is not obeyed, spectrum gets lost, e.g. there might be spectral lines but you won't see them. If settings are inappropriate, the **status text** (left of the mouse arrow in above screen shot) will turn red.

2. The measurement time per data point: From these values, a best guess **sweep time** is calculated, but as Windows is often doing unpredictable things, usually the sweep time is a bit longer in LPT mode. The prediction for USB mode is accurate.
Note, that the minimum sweep time depends on the resolution bandwidth. The slider will lock at the minimum allowed measurement time.
Note, that measurement times below 1ms / frequency point are only available in USB mode.

3. The resolution bandwidth.
▸▸▸ **Warning:** If the resolution bandwith is smaller than the frequency point spacing, then the status text below the entry field turns red (see 1.).
Note, that changing the resolution bandwidth to lower values might automatically increase the measurement time setting if necessary.

4. Two kinds of **sweep progress indicators** can be switched on or off, namely a graphical progress bar appearing at the lower edge of the main VNWA window grid and a progress text appearing in the status line at the very bottom of the VNWA main window.

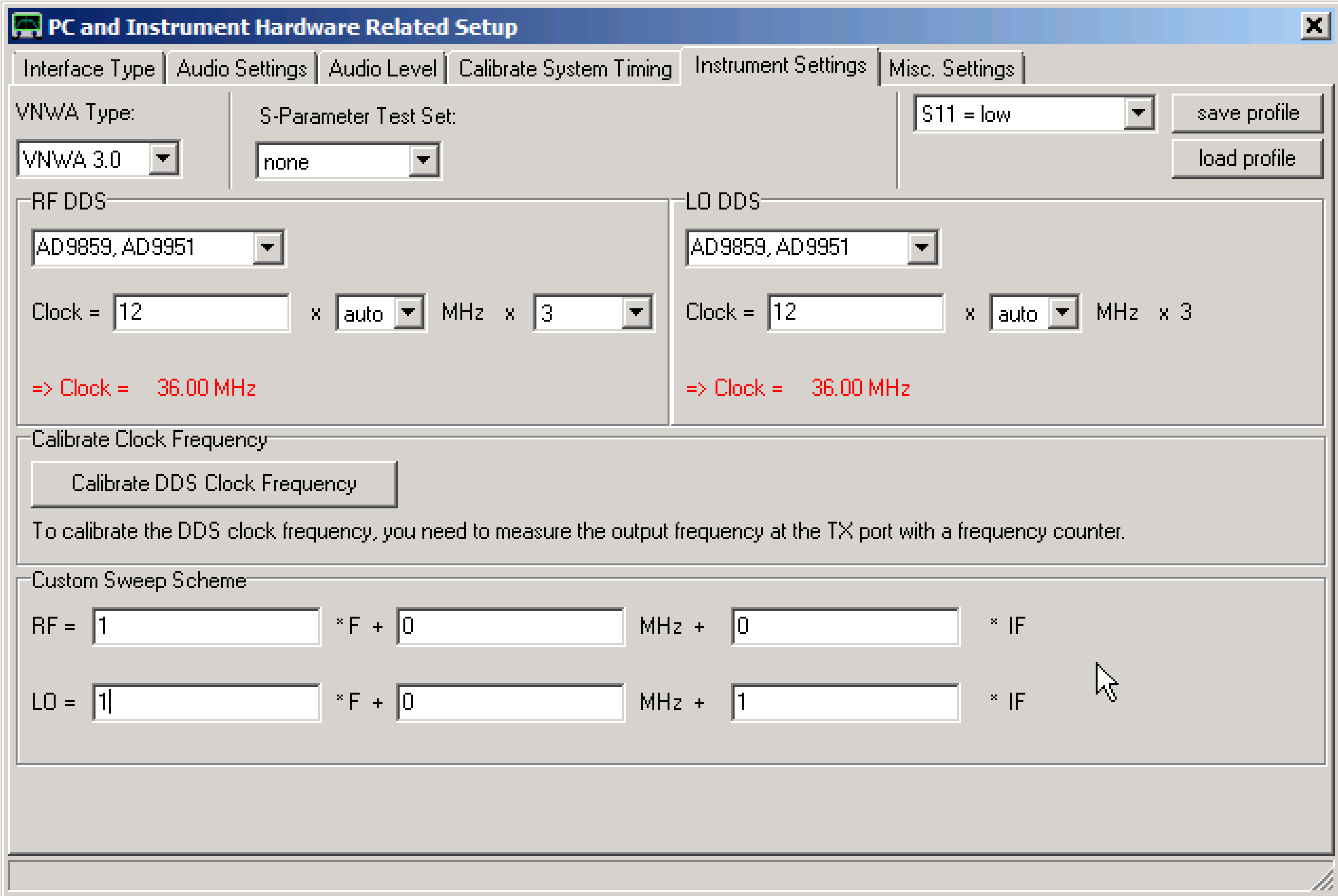
5. A frequency axis option: Three options are available:

a) **RX frequency = displayed frequency (Spectrum Analyzer)**

This option will set the main window frequency axis to the receive frequency of the RX such, that spectral lines are shown properly centered to their center frequency. Note, that the TX frequency is offset by the VNWA IF in this case.

b) **TX frequency = displayed frequency (Tracking Generator)**
This option will set the main window frequency axis to the transmit frequency of the TX such, that in tracking generator mode the measured frequency response is shown properly. Note, that the RX frequency is offset by the VNWA IF in this case.

c) **custom frequency scheme (see Setup - Instrument Settings)** This option allows the user to customize the frequency axis to e.g. account for external frequency converters or frequency multipliers. The customization is specified in the **"custom sweep scheme"** input field in "Setup"-"Instrument Settings":



.

RF denotes the frequency of the RF DDS.
LO denotes the frequency of the LO DDS.
F denotes the frequency axis.
IF denotes the RX intermediate frequency.

The above example shows a situation, where the TX frequency (=RF) is displayed and the LO of the RX is offset by the IF above the TX.

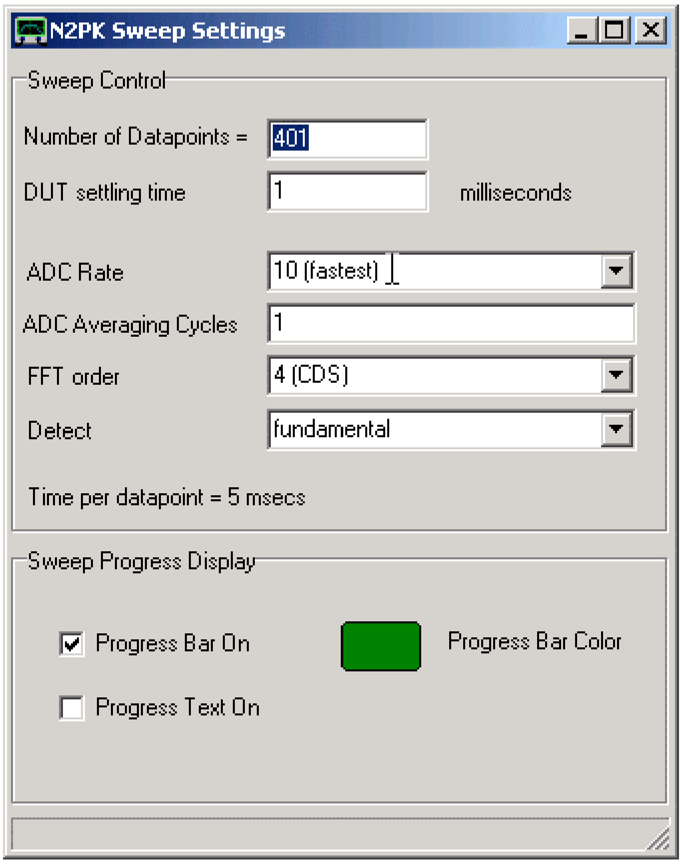
Note, that the above shown "custom sweep scheme" input field (next to the mouse pointer) is only visible if option c) (custom frequency scheme) is activated.

6. A frequency offset to detune the TX signal by the specified offset from the RX frequency (available in LPT mode only). The displayed frequency axis shows the RX frequency. Say, you display a frequency range of 1...10 MHz and the specified offset is 10 MHz. Then the TX sweeps like a **tracking generator** the range of 11...20 MHz. This feature is useful to measure transfer functions of **converters and mixers** using an additional external local oscillator. Note, that the frequency of the **external LO** must be identical with the selected frequency offset to within the resolution bandwidth. Note, that todate no calibration can be performed if the offset is not zero. For zero offset, a regular through calibration is possible, but then you might rather use the VNWA mode to measure transmissions.

7. A level offset to adapt the displayed amplitude levels e.g. to show dBm. The offset level must todate be determined with an external oscillator with known output level, as it depends on the sound card sensitivity.

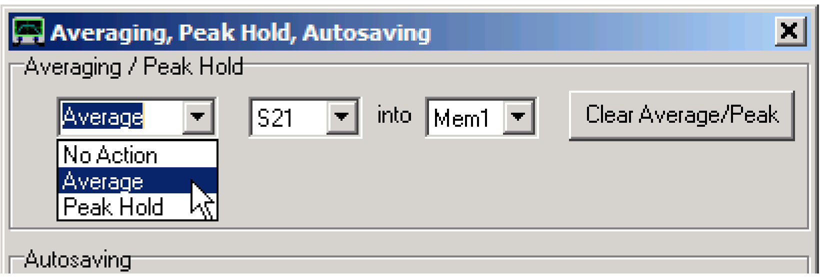
The **"sweep settings"** allow specification of the **number of measured data points** as well as the **measurement timing**.
Moreover, the **analysis mode** (normal, CDS, higher order FFT and harmonic detection) is specified here.
From these values, the time per data point and the sweep time are calculated and displayed.

Finally, two kinds of **progress indicators** can be switched on or off, namely a graphical progress bar appearing at the lower edge of the main VNWA window grid and a progress text appearing in the status line at the very bottom of the VNWA main window.

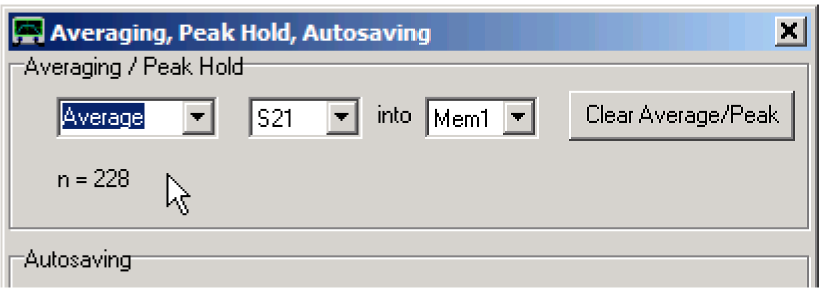


The " Average, Peak Hold, Autosave" settings allow to switch on **trace averaging**, the **peak hold function** and **autosaving** after each sweep.

Averaging and Peak Hold can be activated with the top left dropdown menu box:

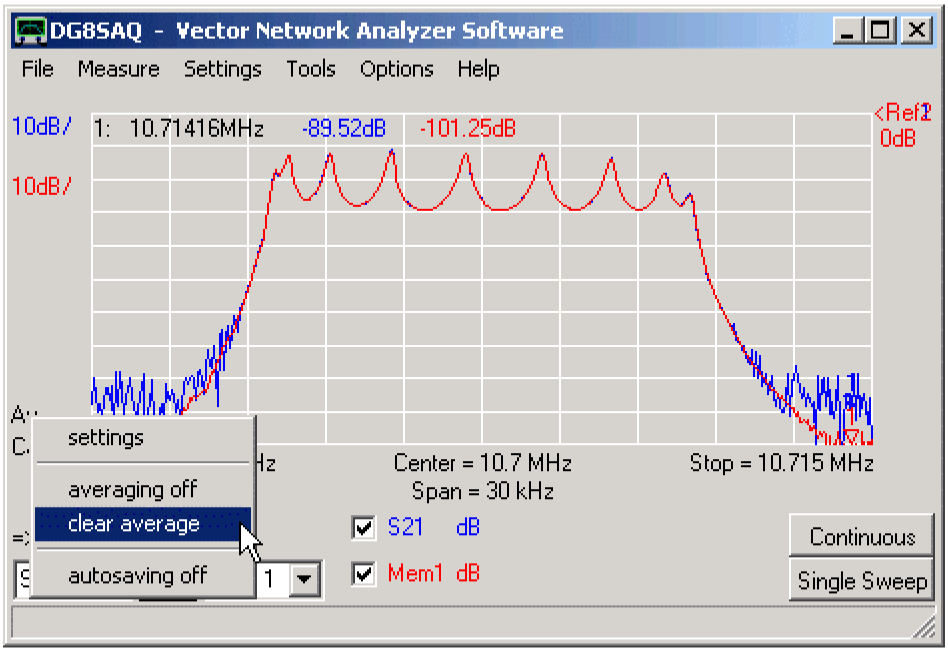


Averaging



In the above example, Mem1 accumulates the average of consecutive S21 sweeps. n = 228 means, that already 228 sweeps have been averaged up to now. This number is reset to zero when pressing the "Clear Average" button. At the same time, the the averaging memory space (Mem1 in above example) is cleared.

The effect of averaging can be seen in the following image:



Averaging effectively reduces the noise. The averaged red trace is a lot less noisy than the single sweep blue trace.

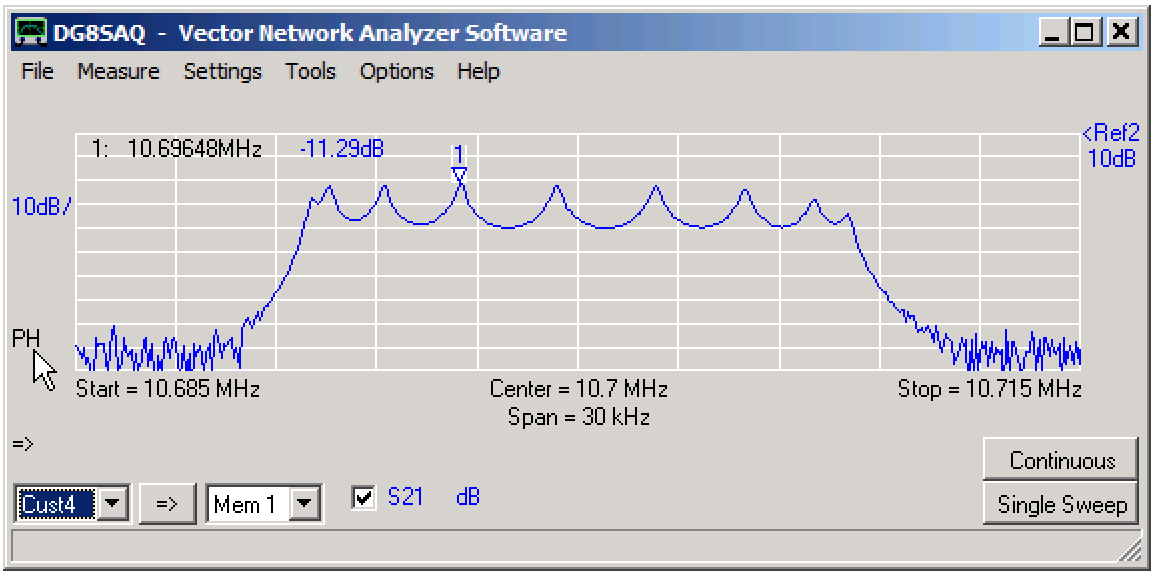
▸▸▸ **Note:** You can also access some averaging functions by right-clicking the "averaging label" on the main window (below the upper left corner of the above pupup menu).

▸▸▸ **Note** The averaging label is only visible, if the averaging function is activated.

▸▸▸ **Note:** Holding the mouse pointer over the averaging label will invoke a pop-up text with the current average number n.

Peak Hold

The peak hold function will determine after each sweep for every frequency point the bigger amplitude of the recent sweep and the stored sweep and stores this maximum value of the two. Thus over the time and consecutive sweeps, at each frequency point, the biggest ever encountered amplitude will be accumulated. This function is well known in spectrum analyzers. It may also be useful for tuning filters. Trace selection and reset functionality is the same as for averaging. If the peak hold function is activated, this is indicated by the PH label in the main window.



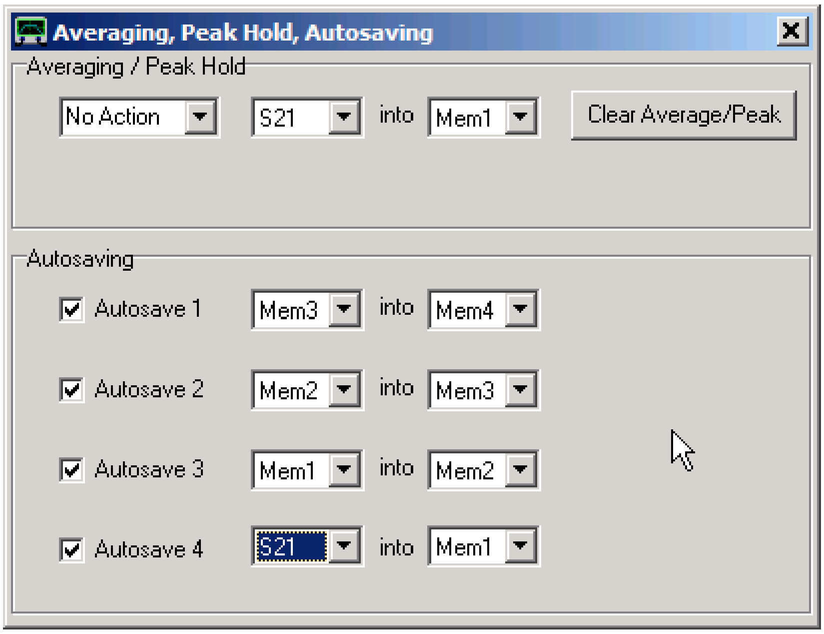
▸▸▸ **Note:** You can also access the peak hold functions by right-clicking the "PH label" on the main window (see above).

▸▸▸ **Note** The PH label is only visible, if the peak hold function is activated.

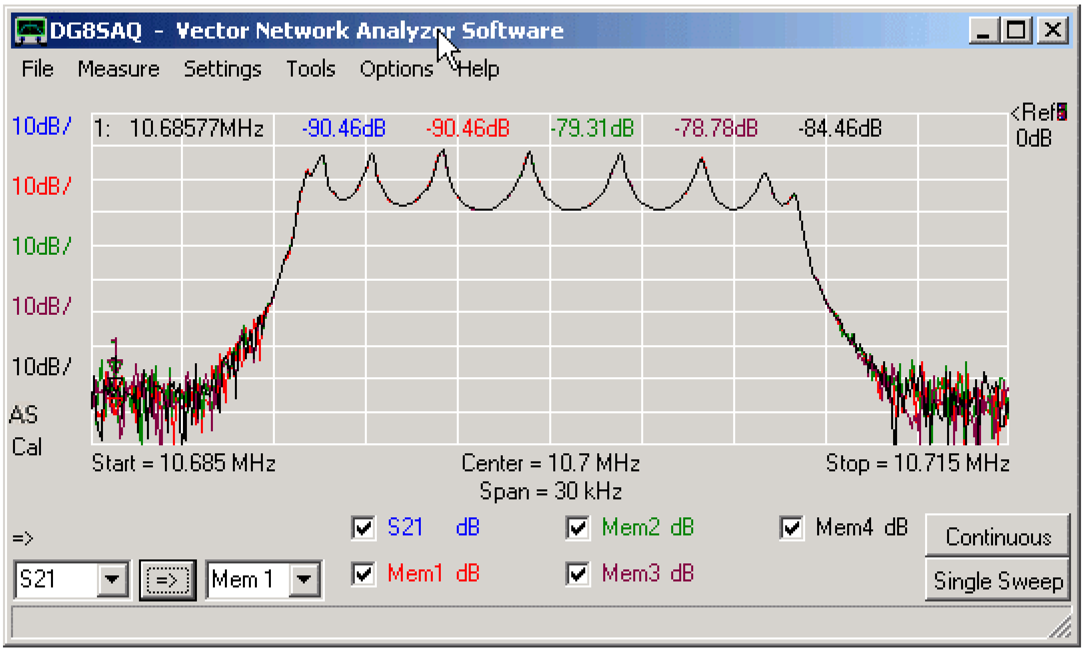
▸▸▸ **Note:** Holding the mouse pointer over the PH label will invoke a pop-up text with the current sweep number n.

Autosaving

Autosaving means automatically saving data at the end of each sweep to another memory space.



The effect of the above settings is, that the last four sweeps are automatically displayed and updated after each sweep:



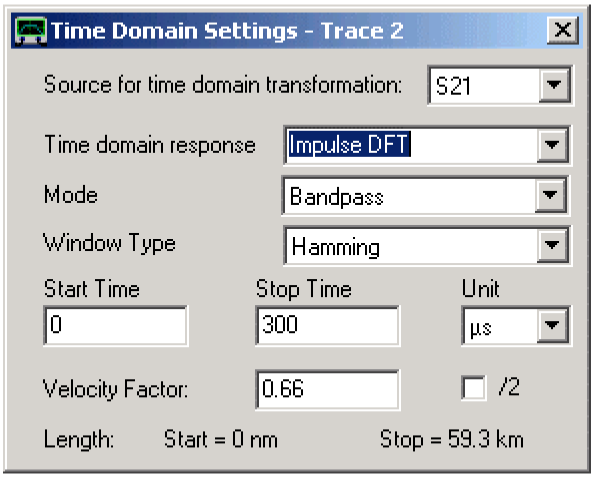
This feature is useful when tuning devices, where it is advantageous to see the changes from sweep to sweep.

The activated "auto saving" function is indicated by the AS label on the left side of the main window (see above).

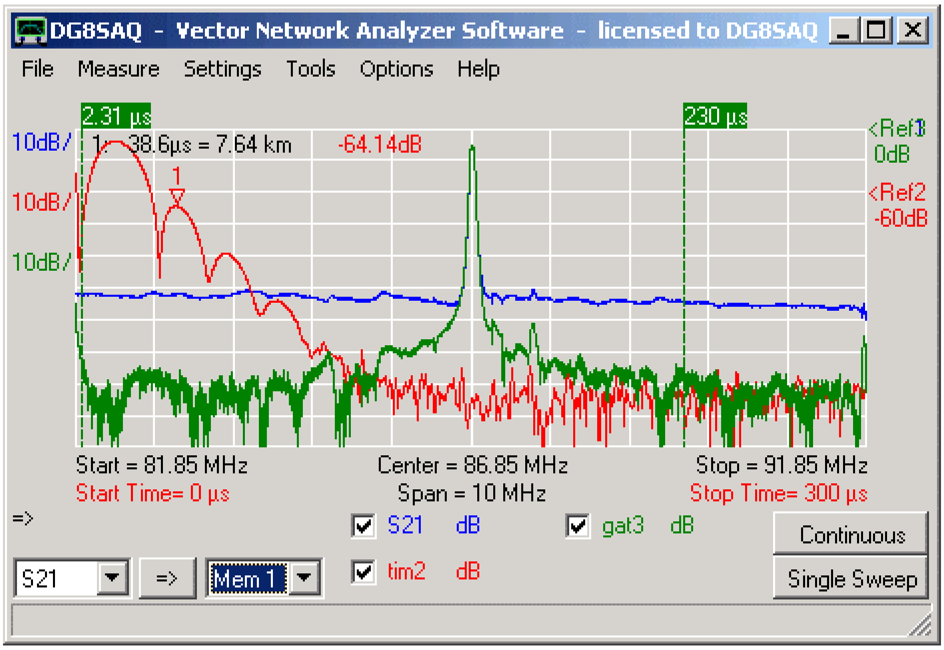
▸▸▸ **Note:** You can also access some autosave functions by right-clicking the "AS label" on the main window (see above).

▸▸▸ **Note** The AS label is only visible, if the autosaving function is activated.

The "time domain settings" allow display and analysis control of time domain data.



With the above settings, after each sweep an inverse discrete Fourier transform (DFT) of the S21 data will be calculated and displayed in a time range of 0...300 us if trace type "time" is activated (see below, red trace 2). The frequency data is weighted with a Hamming window function prior to calculating the Fourier transform in above example.



Time markers display time, distance and level. The distance is calculated from the time with the vacuum speed of light and a velocity factor. This is useful to determine lengths of coaxial cables.
The /2 checkbox allows to get proper lengths on reflection measurements, where the signal travels through the cable twice (forward and backward).

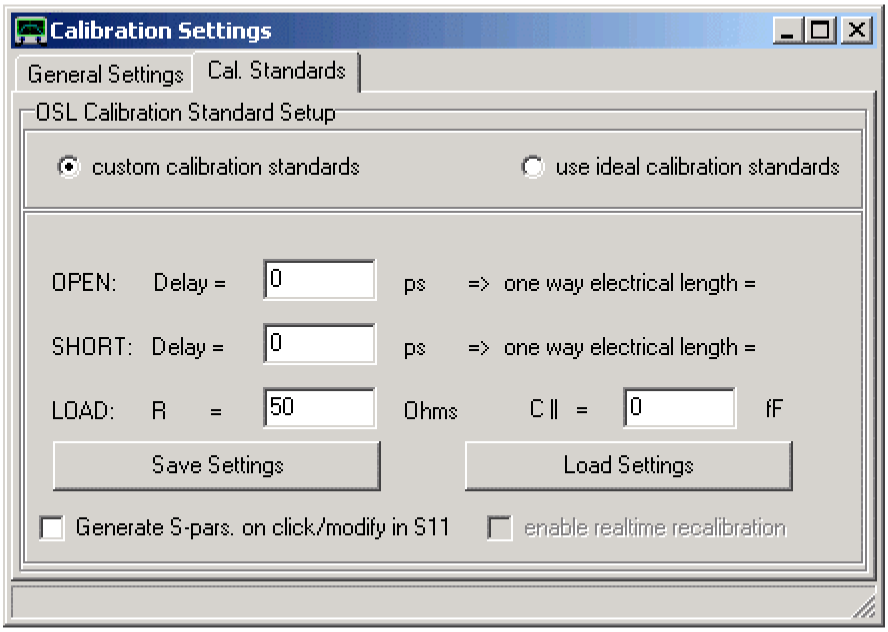
▸▸▸ **Hint:** You can access the time domain settings also by double-clicking the red start and stop time labels or the Time or Gate label in the main window.

▸▸▸ **Hint:** You can separately add frequency and time markers.

▸▸▸ **Hint:** Besides the impulse response, also the **step response** of a DUT can be displayed selectable with the **"Time domain response"** dropdown menu. This is useful to determine transmission line impedances and impedance variations.

For an introducion to time domain measurements and practical examples see here.

The calibration kit settings allow to specify the parameters of the user calibration kit.



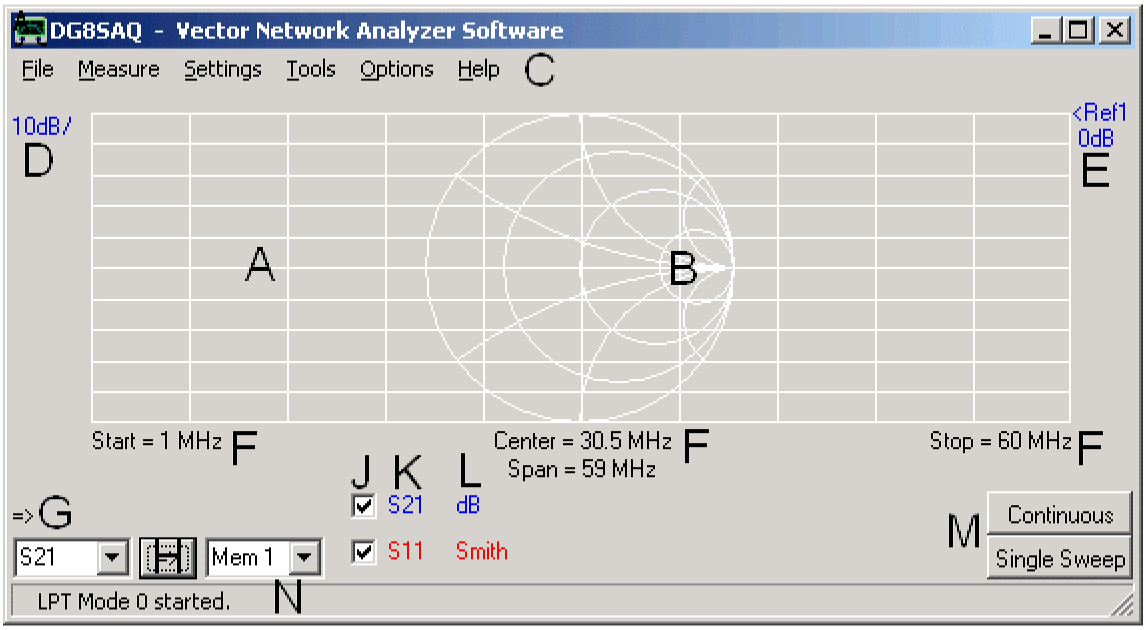
Above, the parameters of an ideal calibration kit are shown.

▸▸▸ **Note:** **Delays are two way delays.** Lengths are calculated from half the delays. If the calibration standard is longer than the position of the desired calibration plane indicates, a **negative delay** is to be entered.

▸▸▸ **Hint:** Calibration kit data can be saved to and loaded from a *.ckf file.

See here for details on calibration.

▸▸▸ **Note:** This menu can also be used to extract calibration kit parameters from measurements. See here how to extract calibration kit parameters.


DG8SAQ - Vector Network Analyzer Software
File
Measure
Settings
Tools
Options
Help
C
10dB/
D
A
B
<Ref1
0dB
E
Start = 1 MHz
F
Center = 30.5 MHz
Span = 59 MHz
F
Stop = 60 MHz
F
J
K
L
S21
dB
S11
Smith
=>
G
S21
H
Mem 1
M
Continuous
Single Sweep
LPT Mode 0 started.
N

The VNWA's built-in ***matching tool*** allows to **simulate the 2 port DUT's transfer characteristics under arbitrary impedance termination conditions** and to **calculate matching networks**.

This is useful for the application of non 50 Ohms filters like crystal filters, where matching networks need to be optimized.

▶▶▶ **Note:** The original S-parameters normalized to 50 Ohms are stored in a separate internal S2P-buffer and remain untouched during the simulation. Simulation results will be stored and displayed via the display memory spaces (S11, S21, S12, S22), which will change in turn. The original 50 Ohms S-parameters can be restored from the S2P-buffer into the display buffers by the main menu command "Tools"-"Restore-Unmatched".
▶▶▶ **Note:** The matching tool can only be invoked, if the internal S2P-buffer contains valid S-parameters. You can copy the display memory spaces (S11, S21, S12, S22) into the internal 2-port buffer and thus make them valid S-parameters with the main menu command "Tools"-"Copy Display Data to S2P Buffer".

A detailed example is found here.

"Restore Unmatched" copies S-parameters from the S2P-buffer into the display buffers (S11, S21, S12, S22) .

▸▸▸ **Note:** The "restore unmatched" can only be invoked, if the internal S2P-buffer contains valid S-parameters.

"Copy Display Buffer Data to S2P Buffer" copies the display memory spaces (S11, S21, S12, S22) into the internal S2P-buffer and thus makes them valid S-parameters, which can be used in conjunction with the built-in matching tool.

The **"crystal analyzer"** tool allows to extract equivalent circuit model parameters from the measured reflection coefficient of a crystal resonator or similar resonator (SAW, ceramic,...)

See here for a detailled example.

Three port RF devices like baluns or SAW-filters with e.g. balanced outputs confront the user with a complex characterization task:
On the one hand, one would like to know e.g. the insertion attenuation from single ended input to differential outputs and the complex differential output impedance (**differential mode**). On the other hand, the **common mode attenuation** is of interest.

The 3-Port analyzer tool allows to perform both characterizations on a set of imported or measured 3-port S-parameters.

A step by step example of performing a 3-port analysis is found here.

This is a tool for **evaluating complex expressions** written by Simon Bucheli from http://www.tyberis.com/
Thanks for allowing me to integrate this tool into VNWA!

The tool basically works like a pocket calculator, but it can calculate with **complex numbers**.

▸▸▸ **Hint:** The tool can **access data in the VNWA data spaces** (S11,S21,...,Mem1...4) and **marker frequencies**.

Usage

Evaluation of simple expressions
sqrt(2)
e^(j*pi)+1

Definition of simple variables (immediate evaluation) - use "="
x = 1
a = 1; b = 2; c = 3

Definition of expression variables - use ":="
x1 := a+b
x2 := a-b

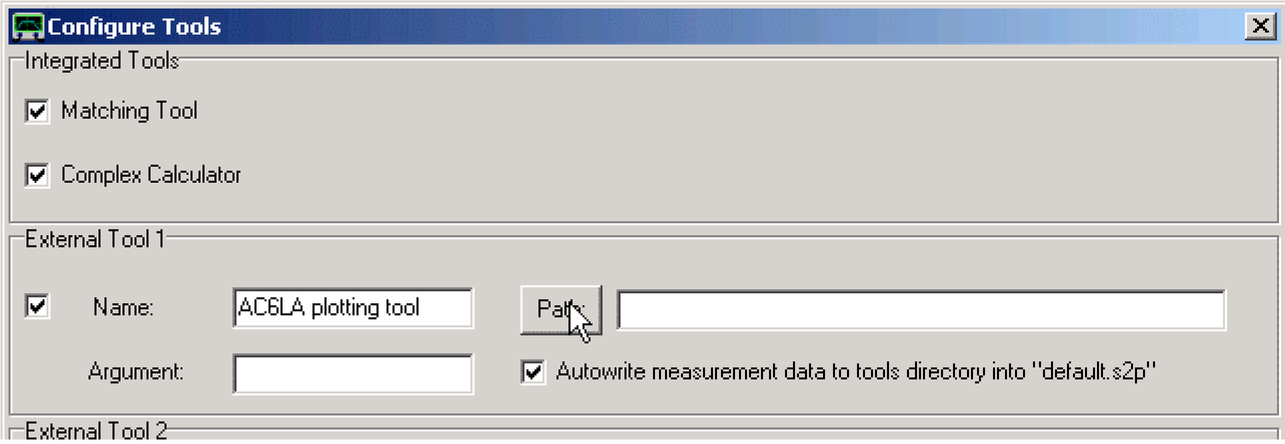
Implemented Functions

abs
arcsin
arccos
arctan
arccot
arg or **Arg** *example: arg(exp(j*5))=5*
cos
cot
conj
ceil
deg conversion radiants to degrees
db or **dB** *dB(x)=20*log(abs(x))*
exp
floor
heaviside
im *example: im(2+3*j)=3*j* (unusual definition of the imaginary part)
imag *example: imag(2+3*j)=3* (usual definition of the imaginary part)
ln
log
lb
mag *mag(x)=abs(x)*
marker or **Marker** or **m** or **M** *example: marker(3) = frequency [Hz] of marker 3*
mem1 or **Mem1** *example: mem1(200e6) = value of data space Mem1 @ 200 MHz*
mem2 or **Mem2**
mem3 or **Mem3**
mem4 or **Mem4**
round
rad conversion degrees to radiants
re *example: re(2+3*j)=2 = real part*
sin
sqrt
sqr
sign
s21 or **S21** *example: S21(10e6) = value of data space S21 @ 10 MHz*
s11 or **S11** *example: S11(m(2)) = value of data space S21 @ marker 2*
s12 or **S12**
s22 or **S22**
s2z converts a reflection coefficient normalized to 50 Ohms to a complex impedance. *example: s2z(0) = 50*

tan

z2s converts a complex impedance to a reflection coefficient normalized to 50 Ohms. *example:* *z2s(0) = -1*

The **"configure tools"** menu allows to **customize the tools menu** by showing or hiding the built-in tools and/or by **registering up to four external tools** to the **"Tools"** menu. An example is presented in section "Interfacing ZPlots".



External tools are characterized by four parameters which can be selected by the user:

Name: The name of the external tool that will be shown in the "Tools" menu.

Path: Path including the application file name of the external application. Valid applications are files and links that can be started in Windows by double-clicking.

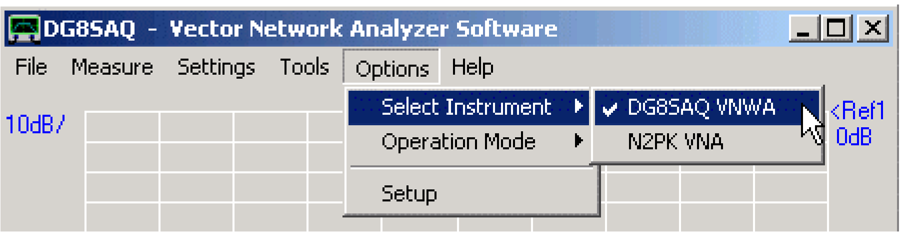
Argument: An optional runtime argument that can be passed to an external tool, e.g. a file name for a text editor.

Autowrite: When the "autowrite" checkbox is activated, current measurement data will be written to the directory specified by "Path" as default.s2p prior to launching the external tool and also every time a sweep cycle is completed. This allows automatic data transfer to external tools.

▸▸▸ **Note:** You have to **activate an external tool by the checkbox** on the very left to make it visible in the tools menu.

The VNWA main menu **"Options"** offers the following functions:

Select Instrument only visible, if N2PK support is activated
Operation Mode
Setup (VNWA, default)
Setup (N2PK-VNA, only activated if N2PK instrument is selected)

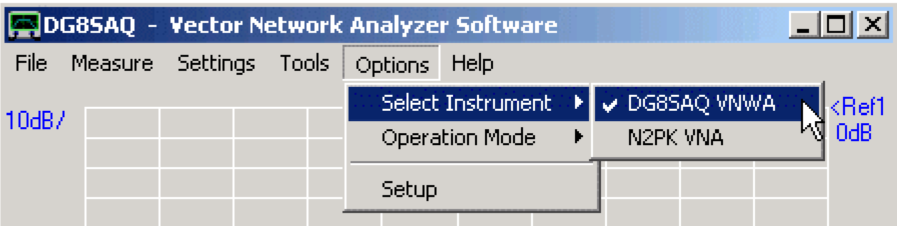


The menu item "Select Instrument" allows to select, which kind of hardware architecture the software VNWA.exe is to control.

▸▸▸ **Note:** This menu item is only visible, if N2PK support is activated

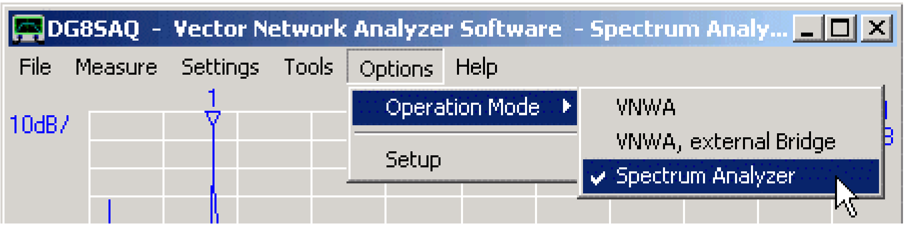
▸▸▸ **Note:** Default hardware is the DG8SAQ VNWA hardware.

▸▸▸ **Note:** Depending on the instrument selection, clicking the "Options"-"Setup" menu item will either open the VNWA setup window or the N2PK-VNA setup window.



▶▶▶ **Note:** This menu is only visible in VNWA instrument mode.

The menu item **"Operation Mode"** allows to select three different operation modes for the VNWA hardware:



1. VNWA Mode

This is the standard **vector network analyzer mode**, which allows to perform vector transmission and reflection measurements.

2. VNWA External Bridge

This is a mode for **reflection measurements only** as the RX input is used as bridge detector for an external reflection bridge. This mode might be useful, when measuring antennas in a strong BCI environment. For details see here.

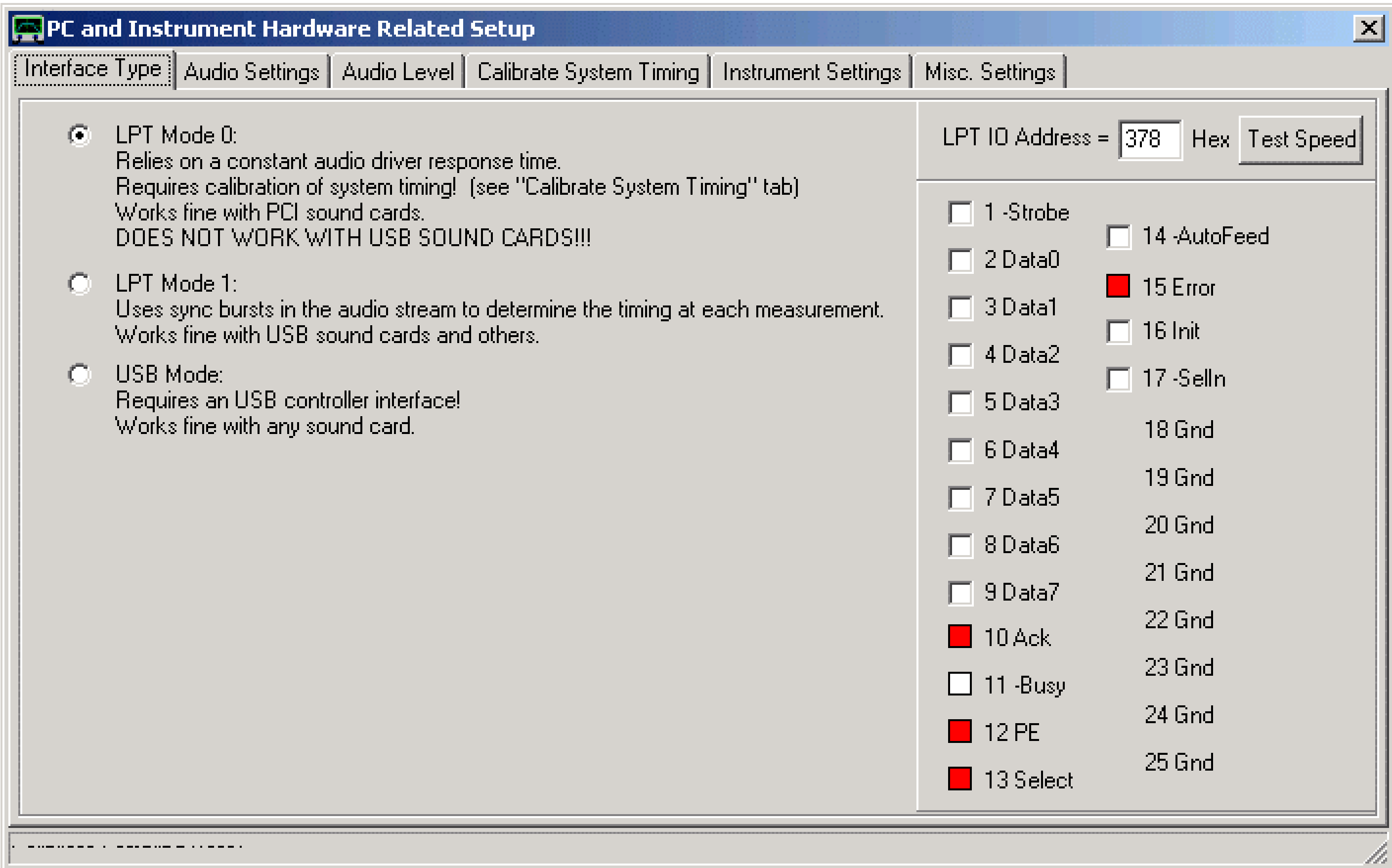
3. Spectrum Analyzer

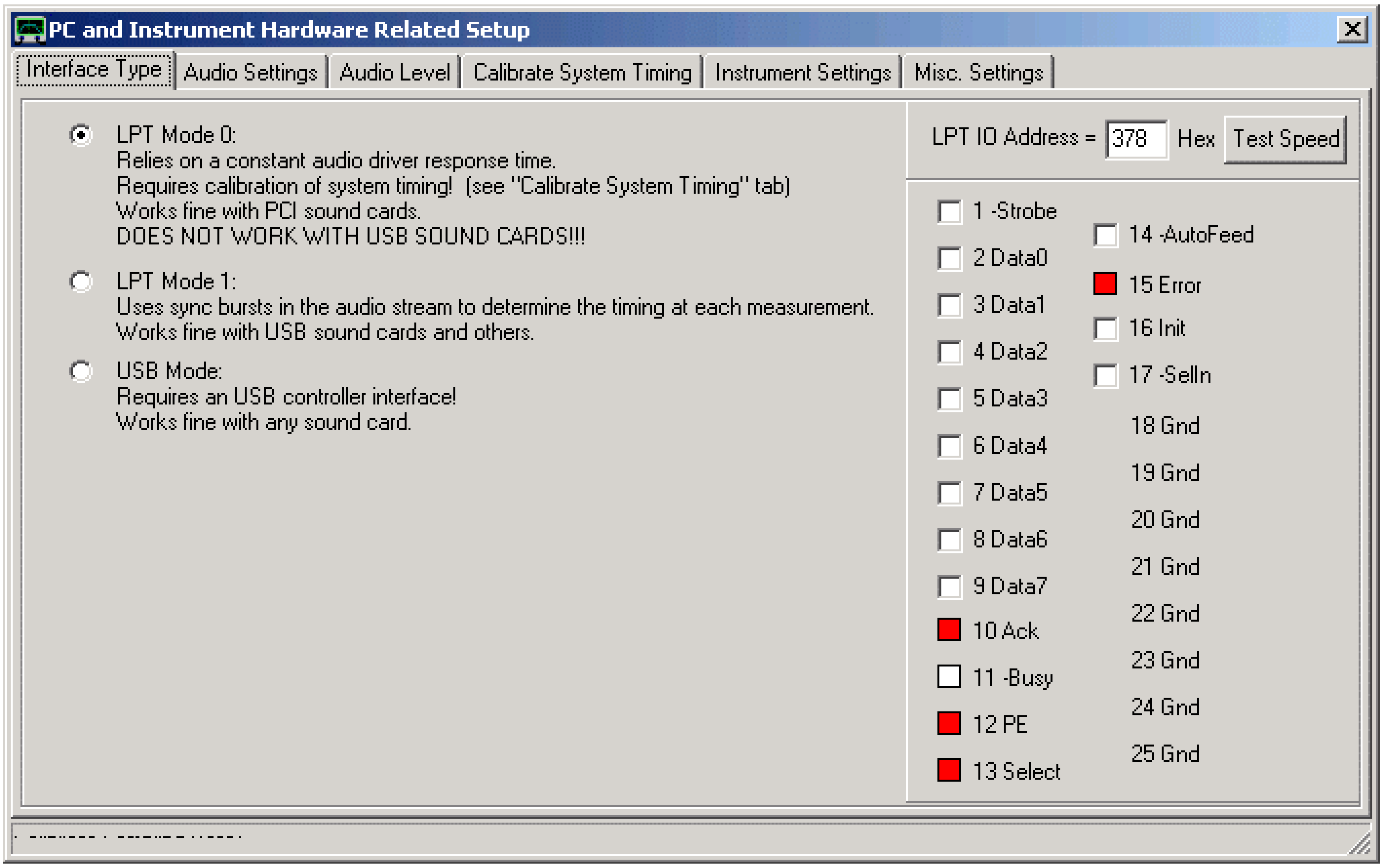
The VNWA hardware can also be used as a rudimentary spectrum analyzer. Details can be found here.

The VNWA setup window allows to specify all **VNWA hardware related settings**.
A quick start guide to do the setup can be found here.

Input is organized in several tabs, which open when they are clicked. Some of these tabs are context dependent, i.e. they are only visible if applicable to the selected interface type.

Interface Type
USB Settings
Audio Settings
Audio Level
Calibrate System Timing
Instrument Settings
Misc. Settings





The VNWA can be controlled via the PC parallel printer port (LPT) or via the DG8SAQ USB_VNWA interface.

For LPT control, there are two timing modes are available:
LPT Mode 0 is very experimental and requires a special system timing calibration.
LPT Mode 1 is the safe mode. If activated, the Calibrate System Timing Tab is **invisible**.

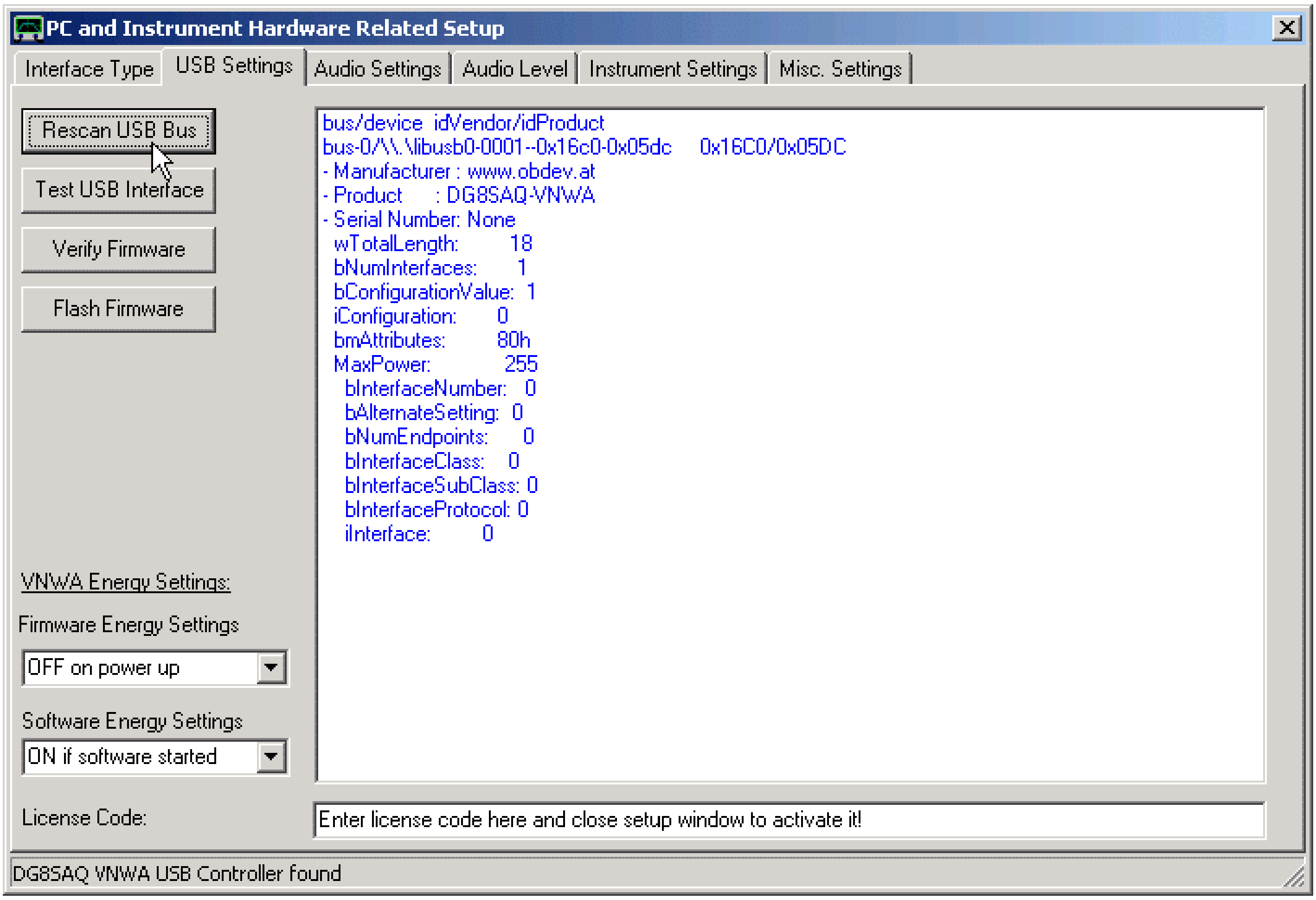
If **USB Mode** is activated, all LPT related controls are switched invisible.

-For proper operation via LPT, the correct **LPT IO address** must be specified here. Any address between 0 hex and FFFF hex can be entered.

-Also crucial for proper operation via LPT is the **minimum transfer speed**, the parallel port must achieve. This can be tested by pressing the **"Test LPT"** button. The test result is displayed in the status line on the very bottom of the setup window. **Note, that this test does not check if your LPT lines are actually switching!** Only the port driver software speed is determined.

-If you have trouble with your LPT interface, you may use the controls on the right for diagnosing it. The checkboxes allow to statically toggle any LPT port output line. The lamps allow to detect the levels of all LPT input lines. Note that **inputs will only be read when outputs are being changed**. If you don't have a scope for checking levels, you can make a loopback cable and control the switching action here. Note, that the controls resemble the LPT port pinout configuration.

▶▶▶ **Note:** This tab is only visible, if USB interface type is selected.



In this tab you can configure and test the **DG8SAQ USB_VNWA interface**.

Rescan USB Bus:
This button allows to scan the PC USB interfaces for the DG8SAQ USB_VNWA interface. If successful, the device properties are being listed on the right as shown above.
▶▶▶ **Note:** The proper USB device driver must already be installed for this test.

▶▶▶ **Note:** In order to use the following controls, the USB Interface must have been **activated** with a **valid license key**.

Test USB Interface:
With this control a communication test between PC and USB interface is initiated. The result will be displayed in the status line on the very bottom of the window.

Verify Firmware:
With this control you can compare the firmware inside your USB device with a hex-file in order to find out if the two are identical or not.

Flash Firmware:
With this control, a firmware update can be uploaded to the DG8SAQ USB interface.
▶▶▶ **Warning:** **You may damage your interface** if you upload unapproved firmware or if you unplug or power down your USB interface during firmware upload. See here for details of the upgrading procedure.

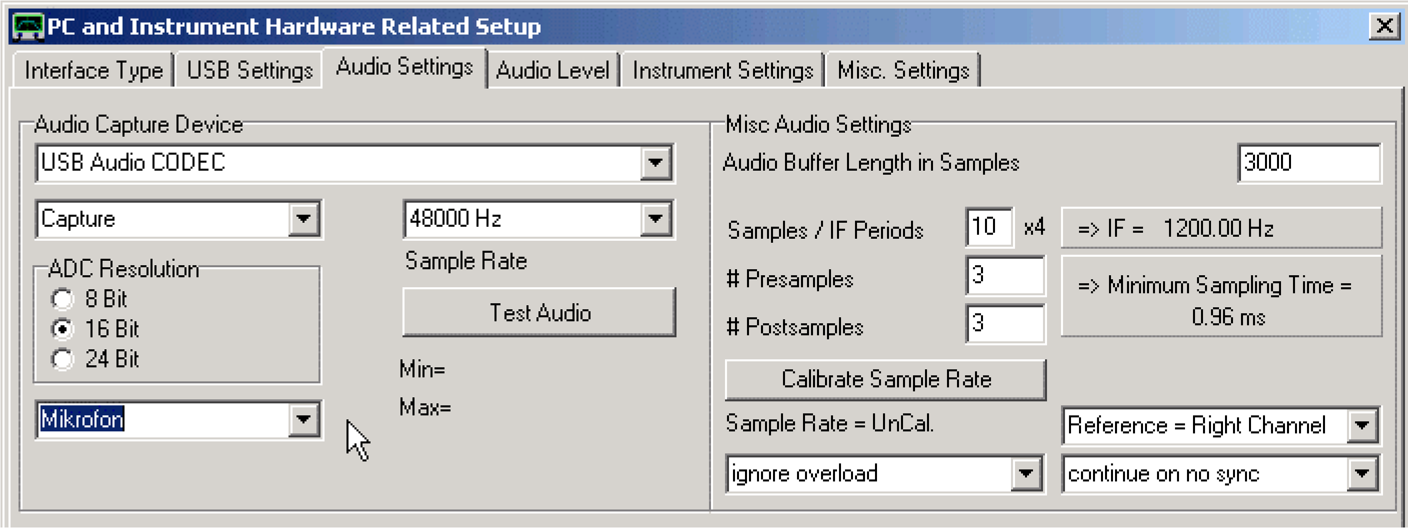
Firmware Energy Settings: These settings tell the USB firmware when to power up the VNWA independently of the VNWA software.
- *Off on power up:* The power supply for the VNWA will **not** be switched on, when plugging it into a powered

USB outlet or when powering up the PC.
- *On on power up:* The power supply for the VNWA will be **switched on automatically**, when plugging it into a powered USB outlet or when powering up the PC.
▶▶▶ **Note:** The former setting will help conserve energy, e.g. when using battery power. The latter guarantees for maximum warmup time of the instrument.

Software Energy Settings: These settings tell the VNWA software when to power up the VNWA.
- *VNWA always OFF:* The power supply for the VNWA is switched off at all times. This setting may be useful to reset the DDSes or to conserve energy.
- *VNWA always ON:* The power supply for the VNWA is switched off at all times. This setting is useful to guarantee maximum warmup time for the instrument.
- *On if software started:* Upon start of the VNWA software, the VNWA is powered up and upon program termination, the VNWA is powered down automatically. This setting may be useful to conserve energy in battery powered systems.

License Code:
Enter a valid license code here.
▶▶▶ **Important:** Immediately after entry of a valid license code, you will not yet be able to e.g. test the USB interface. You must **activate the license code** first! In order to activate the license code, you must **either close the setup window or rescan the USB bus.**

The audio settings tab allows to control all sound card related settings.



Audio Capture Device

In the audio capture device panel all sound card settings are being controlled. Select:

- the used *sound card*
- *recording* or capture, NOT playback!
- a *stereo line-in input*
- a standard *sampling rate*. 48 kHz is recommended. Lower sampling rated reduce the CPU speed but also degrade the signal to noise ratio.
- the *ADC resolution* of the selected sound card. If a lower resolution than available is selected, only the selected number of bits is used. If a higher resolution is selected, the unavailable bits are padded with zeros, but at the expense of CPU time. The 8 bit setting is only for educational purposes to demonstrate the effects of quantization noise.

Proper operation of the sound card can be tested with the **"Test Audio"** button. The labels below the button show the smallest an largest sample within a data block. The audio data can be visualized in the "**Audio Level**" tab. Also, the recording levels can be set there.

▸▸▸ **Note:** Some of the Audio Capture Device controls are being suppressed on a Vista machine as these settings must be done via Windows Vista in this case.

▸▸▸ **Note:** Most controls will become inactive when "Test Audio" is activated. Also, the setup window cannot be closed at once in this case.

Misc. Audio Settings

Available controls:

- **"Buffer length in Samples"** specifies the length of each ring buffer segment. Recommended is 3000. Shorter length means more frequent screen update at the expense of increasing CPU load. Experiments show, that windows cannot cope satisfactorily with buffers shorter than 800 samples.

- **"Samples / IF period x4"** specifies the IF frequency. Note, that the entered number is multiplied by 4. So, if 10 is entered, that means 40 samples per IF period. At a samle rate of 48 kHz, this leads to an IF of 48 kHz / 40 = 1.2 kHz, which is displayed next to the edit field. The highest selectable IF is 12 kHz at 48 kHz sampling rate. Note, that at high IFs, there is considerable channel to channel crosstalk in the sound card leading to performance degradation.

- **"# Presamles"** specifies the number of samples, that are skipped after a frequency change. Recommended is 3.

- **"# Postsamples"** specifies the number of samples, that are skipped before a frequency change. Recommended is 3.
Presamples and postsamples are safety margins to cope with variations between the sampling clock and the CPU clock. It is to be avoided, that data is analyzed, that was taken, while the VNWA received a frequency update due to the resulting digital noise.

▶▶▶ **Note:** A measurement is only possible, if at least one IF period + the presamples + the post samples fit into the selected measurement time per data point. The minimum necessary measurement time is displayed. If a shorter measurement time per data point is selected, there's a timing conflict and the software will refuse to sweep.

▶▶▶ **Hint:** For measurements in the 0.1...20 kHz range, it is advantageous to manually select the lowest possible IF and use a long measurement time per data point. This avoids interference of RF and IF in the sound card.

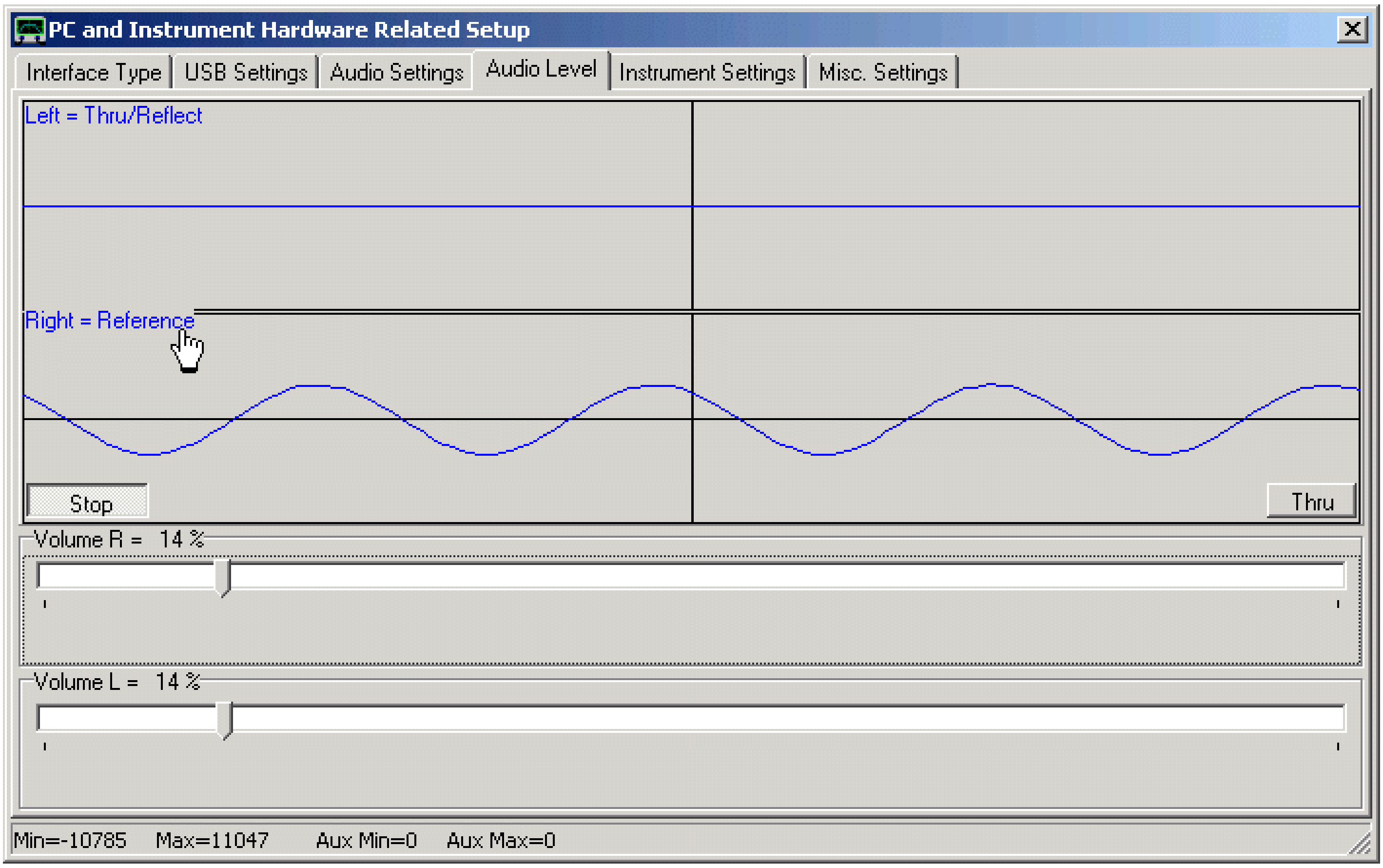
- **"Calibrate sampling rate"** button: It is required, that the sound card sampling rate is well known in order to retrieve the data from the correct time spots in the audio stream. Therefore, the sampling rate needs to be compared once to the CPU clock by pressing this control. Note, that a calibration is only possible if a VNWA is connected. The procedure requires 30 seconds.

- Pulldown control **"Reference = Right Channel"**: Use this button to control which audio channel the software is to interpret as reference channel.

- Pulldown control **"ignore overload"**: The VNWA software detects potential signal clipping in the audio stream. use this control to determine the action to be taken upon an overload condition.
Possible Choices:
-ignore overload
-stop after sweep on overload
-immediately stop on overload

- Pulldown control **"continue on no sync"**: This control only shows effect for LPT mode 1 and USB mode, which both use sync bursts. Occasionally, sync bursts can get lost due to other Windows traffic. Select if you want to continue sweeping, restart a sweep or stop sweeping if a sync is lost. Recommended is "restart sweeping".
Possible Choices:
-stop on no sync
-restart on no sync
-continue on no sync
-silent restart on nosync In silent restart mode, the "no sync" label on the main window is suppressed. This setting might be useful on slow PCs, where e.g. moving a marker might lead to a no-sync condition.

The audio level tab allows to visualize incoming audio data and controling the recording levels.



Controls:

- **"Start - Stop"** button: Use it to start and stop sampling. If the VNWA is connected, the reference signal should always be visible. In the above example, the reference signal comes in on the lower right channel.

- **"Thru - Reflect"** button: It toggles the instrument between thru and reflect measurement. In the above example, the instrument is in thru mode without a connection from TX to RX. Therefore no thru signal arrives.

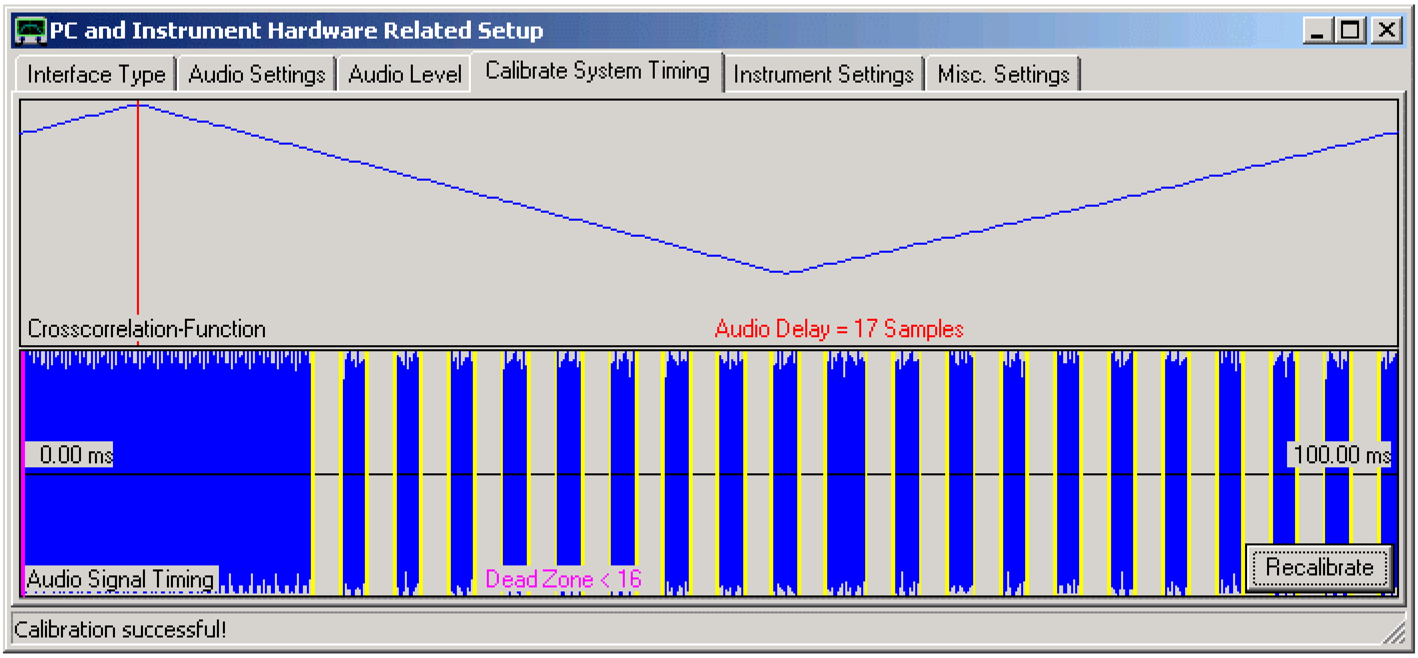
- **Volume sliders**: Use these to control the recording volumes. Make sure that the audio levels are at most about 50% of the displayed vertical span. At 100% span clipping occurs. Note that the difference between 50% span and 100% span is only 6dB. So it is not worthwile go go to the limit and risk clipping.

- **"Channel labels"**: Left-click onto the blue channel labels (see handpoint marker in above screendump) in order to change the Reference channel selection.

▸▸▸ **Note:** Volume sliders are not available on Vista and Windows7 machines. The volumes must be set via the Windows mixer in this case.

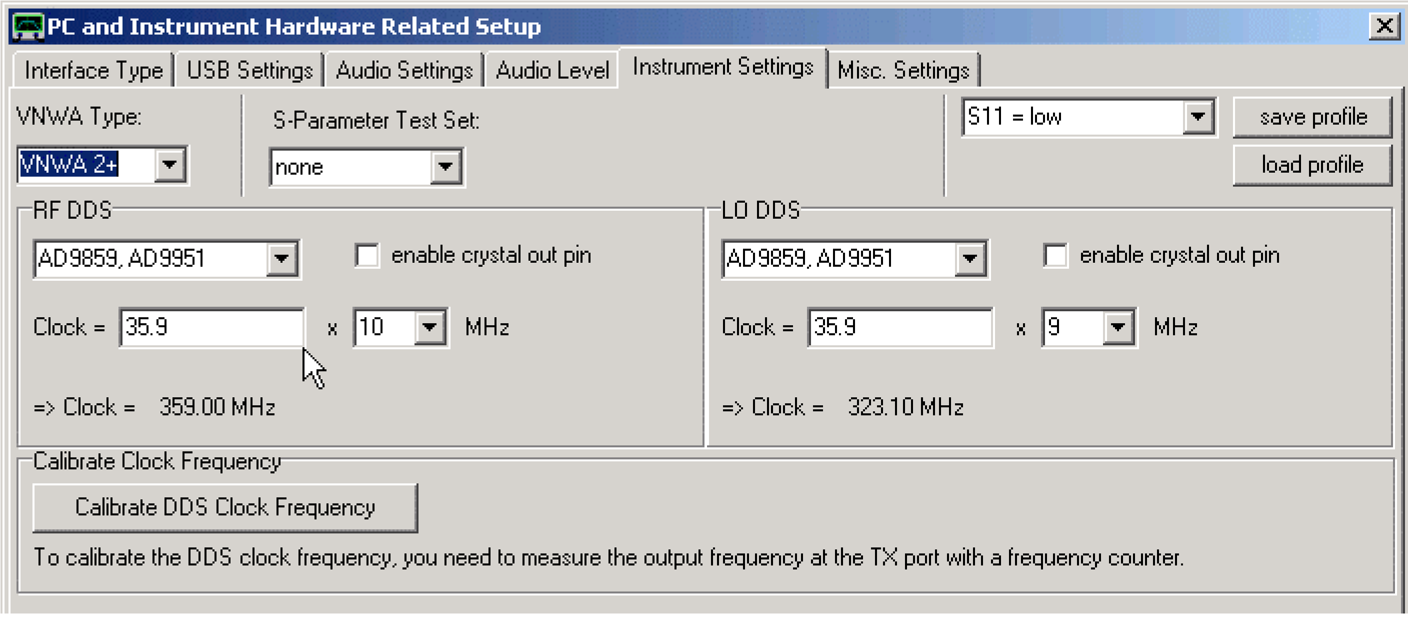
▸▸▸ **Note:** It is not possible to close the setup window during active sampling. An attempt to do so will stop sampling, but won't close the window. Try to close the window a second time in this case.

The "calibrate system timing" tab is only visible if LPT mode 0 is selected. It is used to determine the sound driver latency (=audio delay).



Press the "Recalibrate" button and observe the displayed audio delay (17 samples in the above example). Repeat this several times. If the audio delay varies more than +-1 sample over several measurements, LPT mode 0 is not suitable and you must select LPT mode 1.

The "instrument settings" tab allows to specify the details of the connected **VNWA hardware**.



General controls

- **"VNWA Type"**: For usage with VNWA2.1 upwards select *"VNWA2+"*.
-**S-Parameter Test Set**: Select none if you don't have one. If available, manual or automatic measurement direction control can be selected. Also, the switch response time (=switch delay) and the polarity of the control line can be chosen.
- **"Control line: S11 = ..."** pulldown control: For usage with VNWA2.1 upwards select *"S11 = Low"* as seen above.
- **"safe profile"** button allows to save the VNWA hardware settings in a *.prf profile file. This is only useful, when the same software is to control several VNWAs with different settings, as the settings are stored in ini-files anyway.
- **"load profile"** button allows to reload a VNWA hardware profile from a *.prf file.

DDS controls

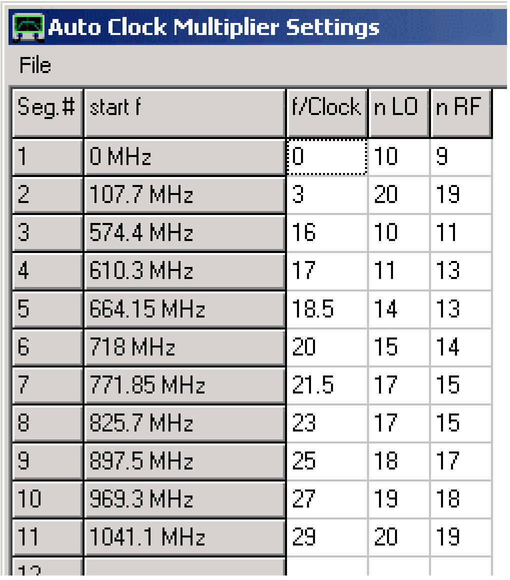
- **DDS Types:**
Select *AD9859, AD9951* on both DDSes for VNWA2.1 upwards.

- **"enable crystal out pin"**:
Uncheck for VNWA2.1 upwards.

- **"Clock"**:
Enter the DDS input clock, which is **identical on both DDSes** for VNWA2.1 upwards and amounts to about 3x the used crystal frequency.

- **clock multipliers**
must be as high as possible, but different such that the smallest common multiple is as high as possible. Obviously, for VNWA2.1 and upwards 20 and 19 is the best choice in this respect. On the other hand, VNWA electrical power consumption is proportional to the clock multipliers. So, if the VNWA is used in the short wave bands only and power consumption is an issue, use of smaller multipliers can be advantageous. **In order to cover the frequency range 600 MHz...900 MHz, dynamical switching of clock multipliers is mandatory**. Select clock multipliers **"auto"** select this feature. On doing so, the **"Auto Clock Multiplier Settings" window** will open.

- **"high/low side injection"** toggle switch:
This switch allows to select, if the LO = RF + IF (high side injection) or LO = RF - IF (low side injection) . For measurements in the kHz range high side injection is recommended. Otherwise, there should be no effect.



Auto Clock Multiplier Settings

File

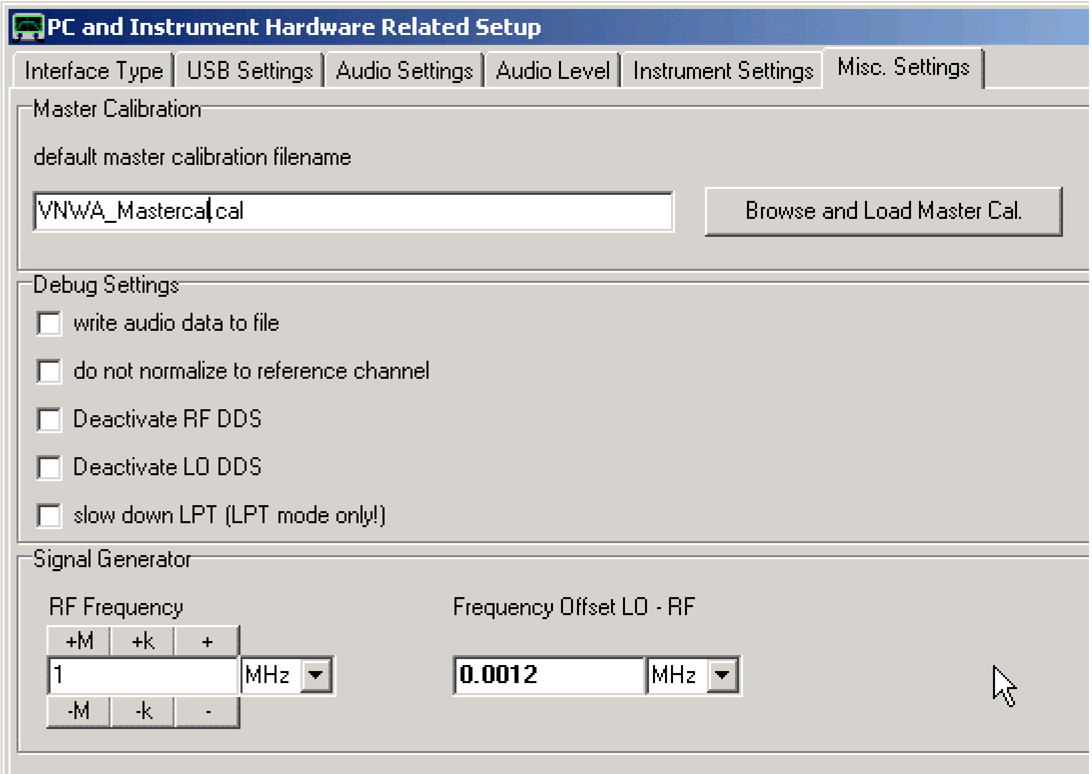
Seg.#	start f	f/Clock	n LO	n RF
1	0 MHz	0	10	9
2	107.7 MHz	3	20	19
3	574.4 MHz	16	10	11
4	610.3 MHz	17	11	13
5	664.15 MHz	18.5	14	13
6	718 MHz	20	15	14
7	771.85 MHz	21.5	17	15
8	825.7 MHz	23	17	15
9	897.5 MHz	25	18	17
10	969.3 MHz	27	19	18
11	1041.1 MHz	29	20	19

The **"Auto Clock Multiplier Settings" window** is already prefilled with proven values. Here, the clock multipliers nLO and nRF are specified for a list of frequency segments, where each spans from the specified start frequency to the start frequency of the next segment. Here, you could also specify a low frequency segment from e.g. 0...30 MHz with lower clock multipliers to conserve energy. Note, that you can insert or delete lines by right clicking on a line. Note, that there must not be empty lines. Also note, that you can save and reload this data to or from a *.mul file by using the "File" dialog.

Calibrate Clock Frequency

It is crucial for the precise function of the VNWA, that the DDS clock frequency is accurately specified. If not, the frequency axis unprecise, but worse, the IF is unprecise and even dependent on the measurement frequency. This might cause signal degradation together with the narrowband digital filters, which might show up in oscillatory noise or interference.

The **"Calibrate DDS Clock Frequency"** button allows to calibrate the clock with the aid of an external frequency counter. Pressing it, a 10 MHz signal will be generated at the VNWA's TX port. Measure it with a frequency counter as accurately as possible and enter it into the appearing frequency field. Having done so, press the appearing "Done" button and the software will calculate and update the DDS clocks.



"Default Master Calibration Filename":
Here, you can load an existing master calibration file or specify a master calibration filename to be loaded at the next program restart.

"Debug Settings":
These are used for VNWA software development and for educational purposes only!

▸▸▸ **WARNING:** Don't touch these debug settings unless you exactly know what you are doing! For proper VNWA operation, all checkboxes must be unchecked!

Options:
- *"write audio data to file"*: If selected, the raw stereo audio stream data is stored after each sweep to the file out.dat. => requires considerable processing time. A good free software tool to view the raw audio data is the plotting software GENPLOT.
- *"do not normalize to reference channel"*: If selected, the reference channel data is ignored. This is useful to evaluate absolute signal amplitudes of the DDSes and noise levels.
- *"Deactivate RF DDS"*: If selected, the RF DDS is shut down. This is useful to check the noise floor level of the instrument undisturbed by RF crosstalk. Requires USB firmware V4.2 or higher.
- *"Deactivate LO DDS"*: If selected, the LO DDS is shut down. This is useful to check the noise floor level of the IF signal path. Requires USB firmware V4.2 or higher.
- *"slow down LPT"*: If checked, communication via LPT is slowed down by a factor of 2.

▸▸▸ **Note:** The last two options show no effect on USB controlled VNWAs.

"Signal Generator"
With the *RF Frequency* control, you can directly control the frequency of the RF DDS. The *Frequency Offset LO - RF* control sets the frequency of the LO DDS accordingly.

Example 1:
If RF is set to 10 MHz and an offset of 1 MHz is selected, then the RF DDS is set to 10 MHz and the LO DDS is set to 11 MHz.

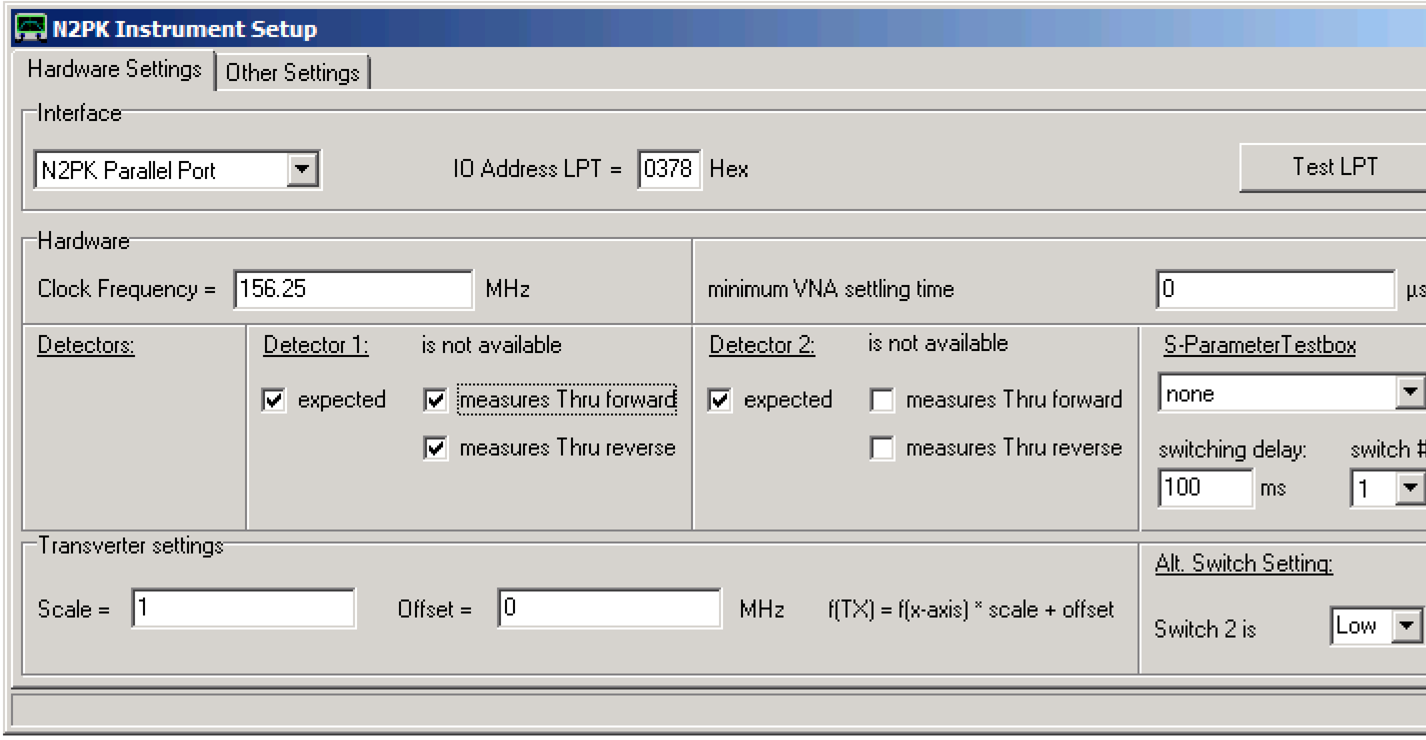
Example 2:

If RF is set to 5 MHz and an offset of -5 MHz is selected, then the RF DDS is set to 5 MHz and the LO DDS is set to 0 MHz, which means it is switched off.

Example 3:
If RF is set to 0 MHz and an offset of 5 MHz is selected, then the RF DDS is set to 0 MHz (=off) and the LO DDS is set to 5 MHz.

The "N2PK Instrument Setup" window allows to configure VNWA.exe to control an N2PK-VNA.

Hardware Settings Tab



In the **interface** section, select the interface of your choice (either **N2PK Parallel Port** or **G8KBB USB Interface**) and in the former case the port address. The "Test LPT" button allows to test the LPT speed. The result will be displayed in the very bottom status line and should be <2 us.

The **hardware** section specifies the **VNA clock frequency** and the available and used detectors. You can also specify if an **S-parameter test set** (automatic switch to invert the DUT) is available and if it should be used manually (user selecty measurement direction) or automatically (direction is switched automatically). Also, the **polarity** of the test set control can be selected. As most switches are built with slow mechanical relays, the software needs to wait until the switch is thrown. Specify the **switching delay** accordingly.

The **minimum VNA settling** time is not exactly a hardware setting. It specifies, how long the software waits after a DDS phase change for the VNWA to settle. 200us are sufficient. Entering 0 will invoke the minimum settling time influenced by the LPT transfer speed and can range from 100us to 200us.

The **"Transverter settings"** section specifies, how the main VNWA window frequency axis is to be scaled, if a frequency transverter is used together with the VNA. E.g. if scale=0.5 and offset=-400MHz, a specified measurement frequency range of 430...440 MHz will invoke a DDS output frequency of 15...20 MHz.

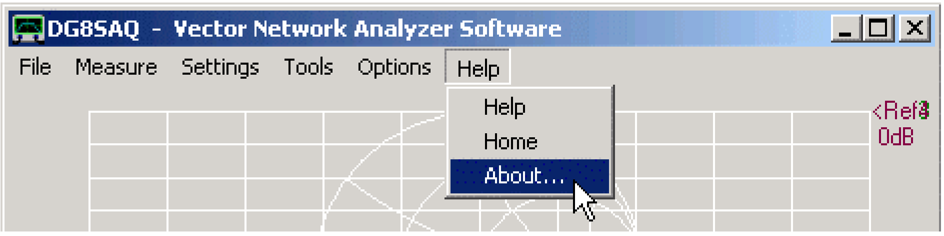
The G8KBB USB Interface allows to control two switching lines, one of which is used for controlling the S-parameter testset (usually switch #1, as is selected above). The other switch can be controled with the **"alt switch settings"** to e.g. control additional test heads. Note, that this feature is only functional in USB mode.

Other Settings Tab

Here, you can load an existent master calibration file or specify a master calibration filename to be automatically loaded on the next program start.

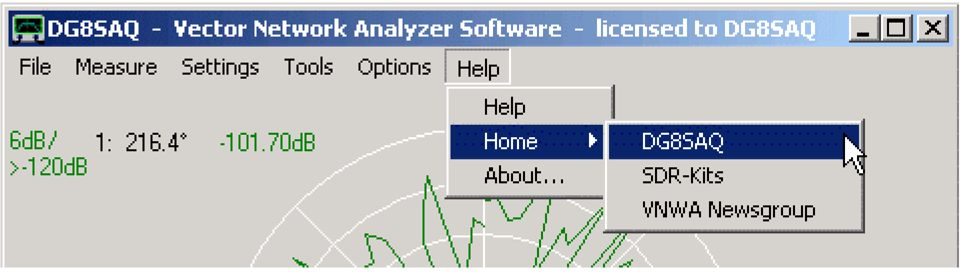
The VNWA main menu **"Help"** offers the following functions:

Help
Home
About



Select **"help"** to invoke **this help file** from within VNWA.
You must have copied VNWA.HLP and VNWA.cnt to your VNWA software directory in order to use this feature.

Select **"Home"** to access the **VNWA knowledge base**:

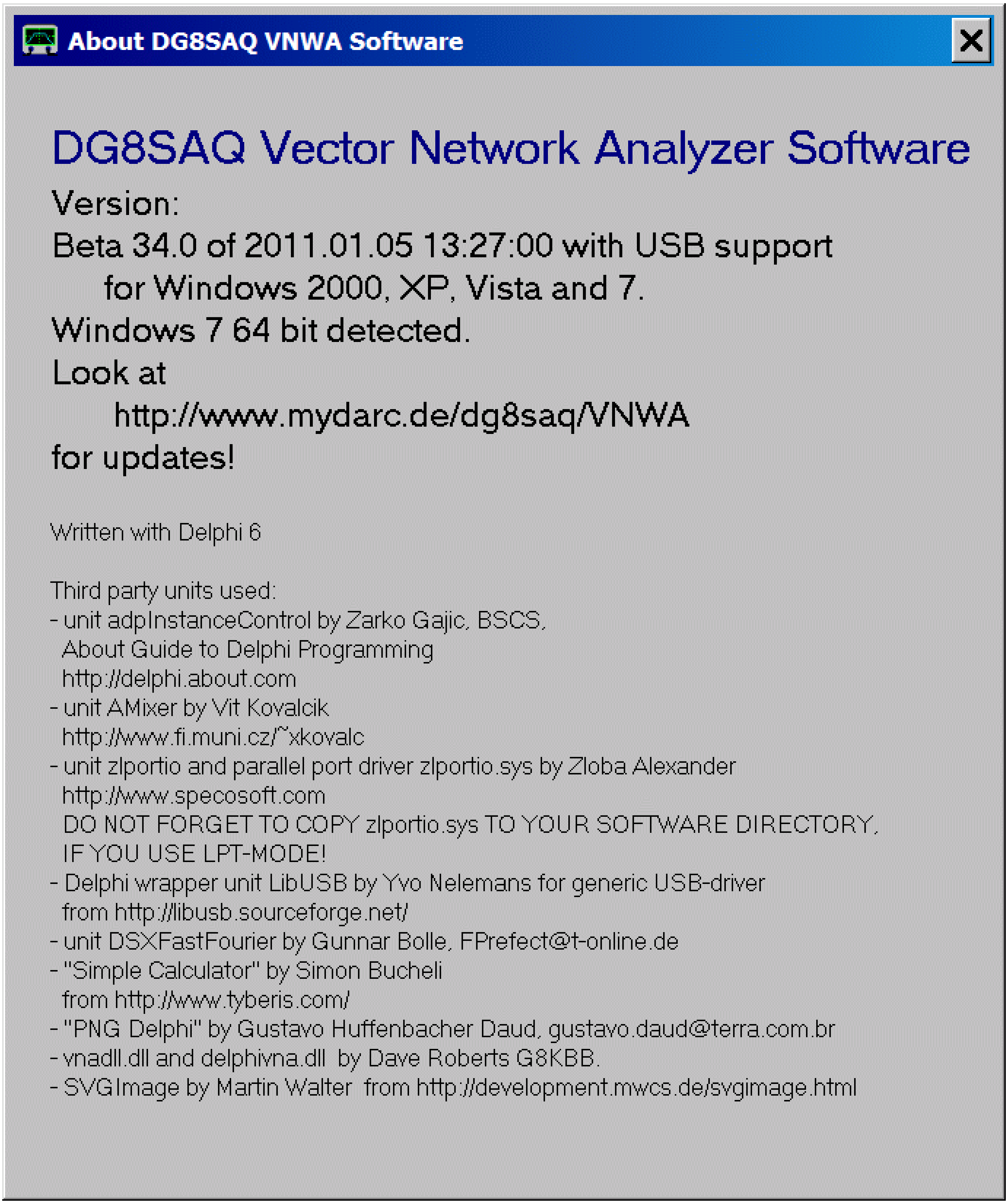


DG8SAQ will let you access DG8SAQ's webpage to e.g. to look for literature and links.

SDR-Kits will let you access Jan G0BBL's ordering and support webpage.

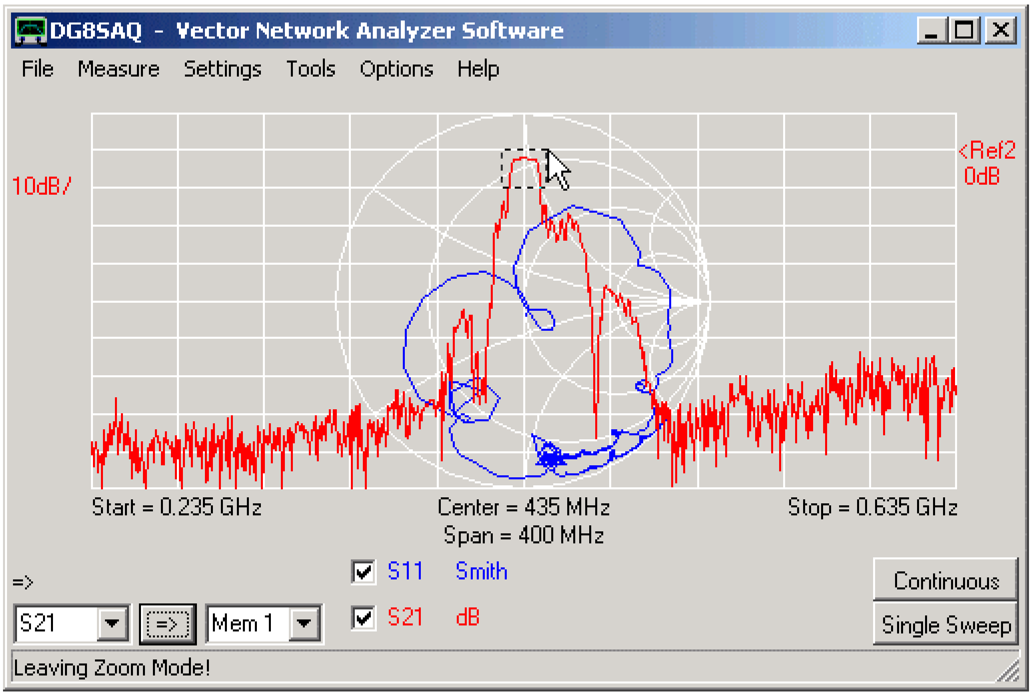
VNWA Newsgroup will guide you to the VNWA Yahoo forum, where you can post your questions and find software and driver updates.

Look here to determine your software version. **When reporting bugs, always report the software version including build date and time!**

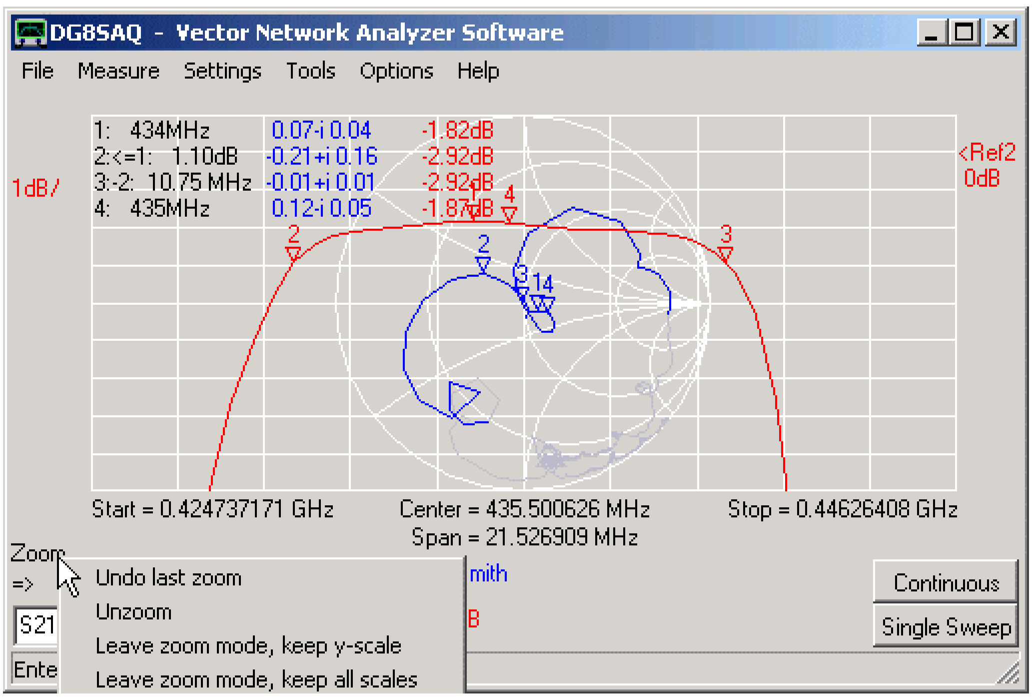


Thanks to all who contributed to this software!

You can zoom into a displayed trace in order to view an enlarged section of it. To do so, point the mouse cursor to one corner of an imagined box on the main window grid, which you want to enlarge, press the left mouse button and draw the so called zoom-box with the mouse keeping the left mouse button pressed continuously. While you keep the left mouse button pressed and you move the mouse, you will see the zoom-box taking shape. In the following example, we attempt to zoom into a measured filter's passband:



As soon as you release the mouse button, the zoom-box will be enlarged to completely fill the main grid.



▸▸▸ **Note:** You can do multiple **consecutive zooms**.

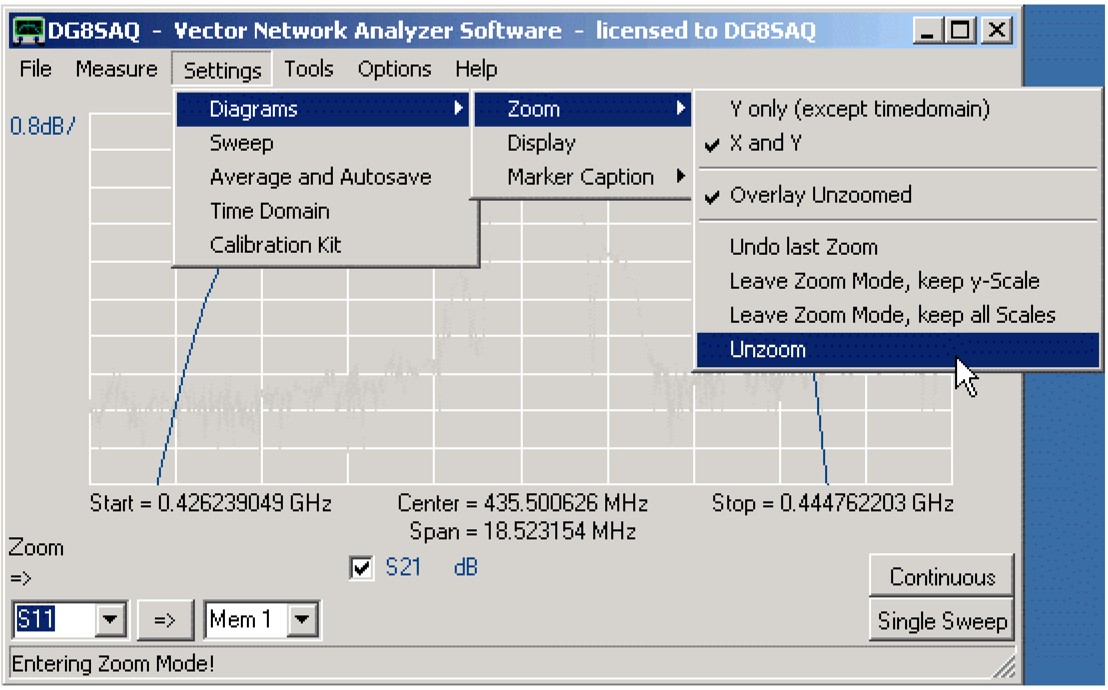
▸▸▸ **Note:** When zooming, the measurement span and measured number of data points remains untouched. Only part of the data is displayed. When sweeping in zoomed state, you still sweep the whole unzoomed frequency range, but only part of it is displayed.

▸▸▸ **Note:** You cannot zoom inside the Smith chart. But as you can see in above screenshot, the Smith chart data outside the zoomed frequency range is greyed.

▸▸▸ **Note:** You can **unzoom** by right-clicking the **zoom label** near mouse pointer. If you right-click it, the above shown **unzoom menu** pops up.

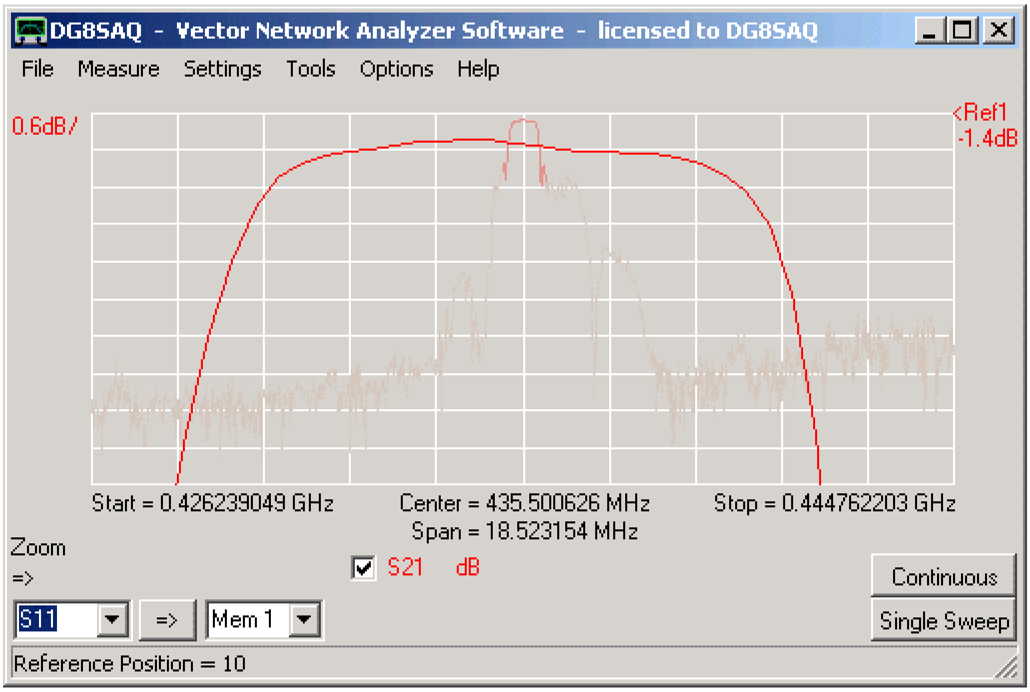
▸▸▸ **Hint:** You can also unzoom by right-clicking the main window grid.

▸▸▸ **Hint:** Zoom functions can also be accessed by the main menu **"Settings"-"Diagrams"-"Zoom"**:



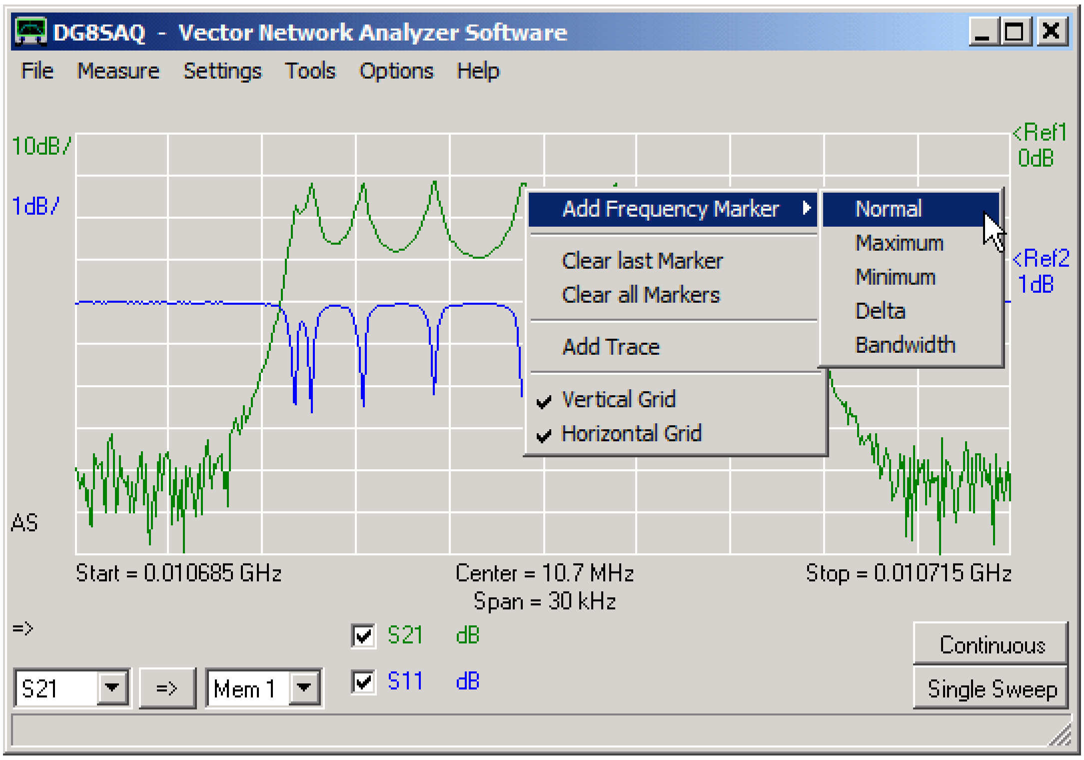
Choices:
- ***Y only ...*** = only zoom vertically, leave the frequency axis unchanged (not applicable in time domain mode).
- ***X and Y*** = zoom frequency axis and y-axis

- ***Overlay Unzoomed*** = If selected, a greyed version of the unzoomed data is displayed with the zoom range highlighted together with the zoomed trace:



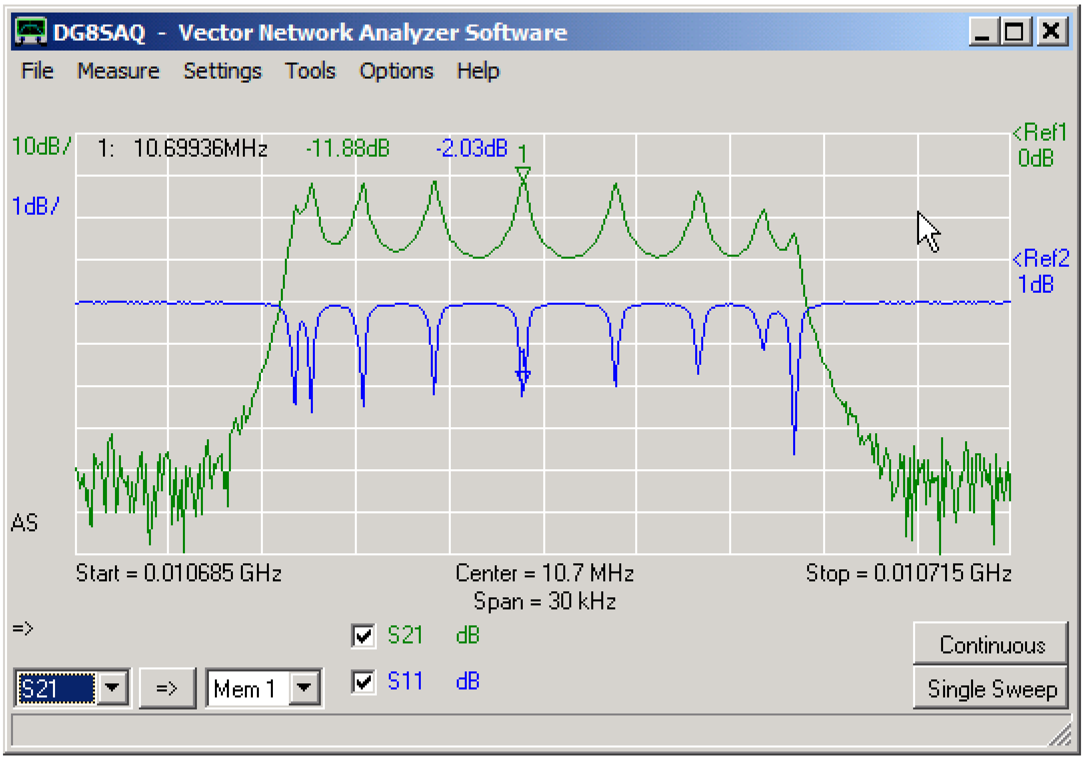
The following choices are only visible in zoom mode:
- ***undo last zoom*** = restore the x- and y-scales of before the last zoom
- ***leave zoom-mode keep y-scale*** = restore the full frequency span but keep the zoomed vertical scale
- ***leave zoom-mode keep all scales*** = keep the zoomed frequency span and the zoomed vertical scale. The data outside the last zoom box is lost, the visible data is interpolated to the full number of data points grid. A sweep after this will only sweep the visible frequency span.
- ***unzoom*** = restore the original x- and y-scales of before all consecutive zooms

In order to read numerical information from displayed traces, up to **9 markers can be placed** onto the traces. This is done by pointing the mouse cursor to the position (=frequency) in the plotting grid where you want the marker to be and right-click the mouse. A popup menu allows to select a marker type to be placed:



Note, that with the aid of above menu you can also delete the most recently placed marker (**"Clear last marker"**) or delete all markers (**"Clear all markers"**).

In the following a **"Normal" "Frequency Marker"** is placed:

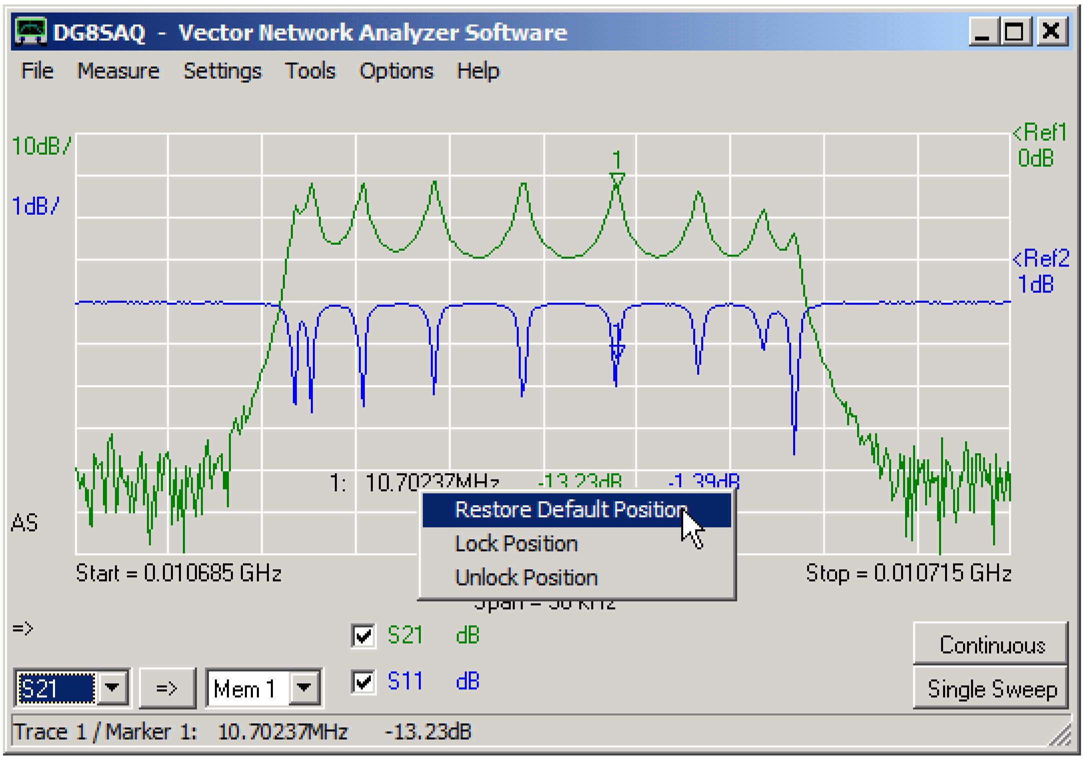


Note, that in addition to the actual markers a **marker caption** with marker number, marker frequency and marker values is displayed in the top left area of the plotting grid.
The user settable frequency unit of the center frequency is used as marker frequency unit (MHz in above example).

Available marker functionalities:

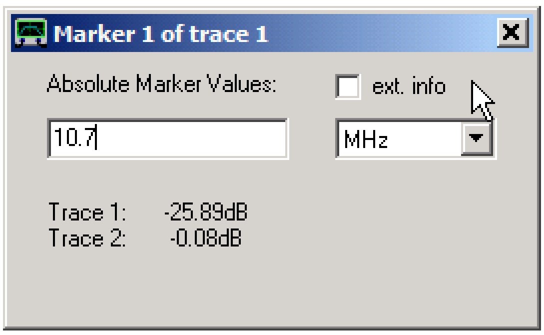
1. Markers can be moved with the mouse. To do so, point the mouse to the marker which you want to move. If the mouse pointer changes to a hand shape, press the left mouse button and drag the marker to where you want it to be.

2. The marker caption can be moved with the mouse in a similar fashion. To do so, point the mouse to the frequency part of the marker caption. When the marker pointer changes to a hand shape, press the left mouse button and drag the caption where you want it to be. The default position can be restored by right-clicking the freqency part of the marker caption. Also, the current position can be locked or unlocked:

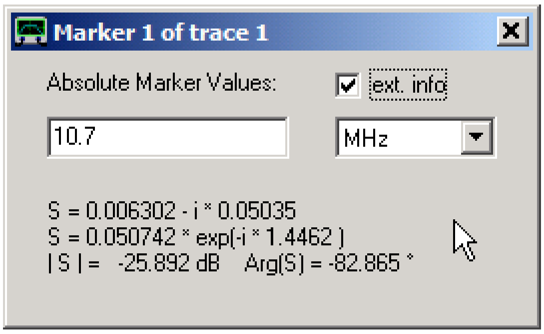


The same functionality is accessible via the main menu "Settings"-"Diagrams"-"Marker Caption".

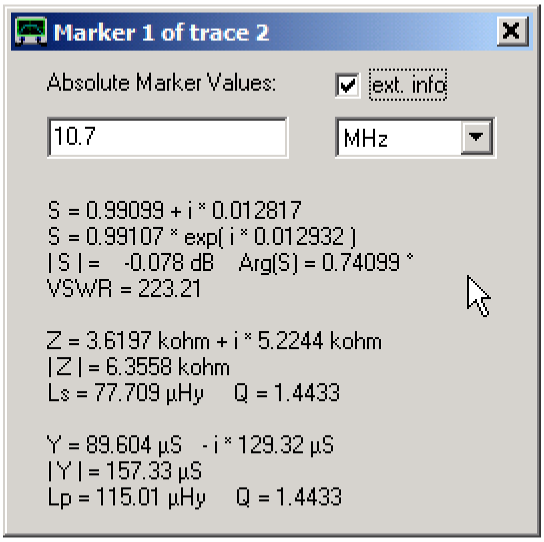
3. Extended marker information is available upon double-clicking a marker, e.g. marker 1 on the S21 trace in above diagrams. Also, the marker frequency can be set to an exact value (not for maximum, minimum or bandwidth markers), e.g. 10.7 MHz in below example.



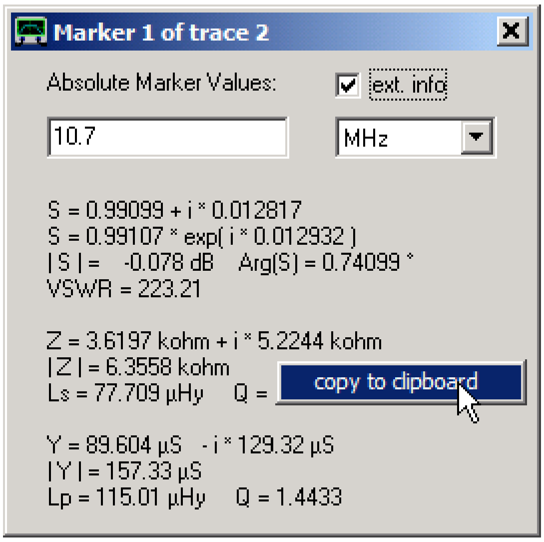
Checking the **ext. info** checkbox (next to mouse pointer above) will yield extended marker info:



Even more extended information is available for reflection trace markers (S11, S22), for which impedances and VSWR can be calculated:

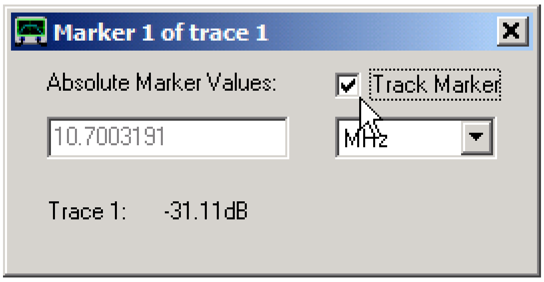


Note, that you can place the extended marker info window next to the main window and leave it open while sweeping. Whenever a trace is being updated or the corresponding marker is moved with the mouse, the extended marker info in the separate marker window is updated as well.
Note, that you can copy extended marker info to the Windows clipboard by right-clicking the extended marker window in order to paste it into any text document:



4. Tracking markers in spectrum analyzer mode:

Note, that extended marker information doesn't make sense in spectrum analyzer mode, as no phase information is available there. Some times it is desirable in SA-mode to lock the instrument center frequency to a spectral line that might slowly drift. This can be achieved by setting a maximum marker that detects the spectral line. Double-clicking it will open the following marker window:



Note, that you cannot modify the marker frequency as this is determined by the maximum of the marker trace. Checking the **"Track Marker"** checkbox will cause the software to set the measurement center frequency to the marker frequency just before starting a new sweep. This way, it is attempted to keep the maximum in the center of the measurement span. This tracking also works for minimum and all bandwidth markers. Sometimes it is more stable to track the 10dB center frequency than the maximum of a peak.

Available Markers

Depending on the displayed data (frequency data, time data, radar data) the following marker types are available in the markers menu:

Add Frequency Marker	(for frequency data and polar data, e.g. Smith charts)
Add Time Marker	(for time domain data)
Add RADAR Marker	(for angular data e.g. antenna diagrams)

▸▸▸ **Note:** Only frequency markers will yield an extended marker info window upon double-clicking.

Available Frequency marker types:

- ***normal marker***
- ***maximum marker*** (automatically jumps to the maximum of the first applicable displayed trace)
- ***minimum marker*** (automatically jumps to the minimum of the first applicable displayed trace)
- ***bandwidth marker*** (places a maximum marker, two markers down one vertical division below the maximum marker and a center marker between the bandwidth markers)
- ***delta marker*** (displays the frequency distance and the vertical distance to the last normal marker)

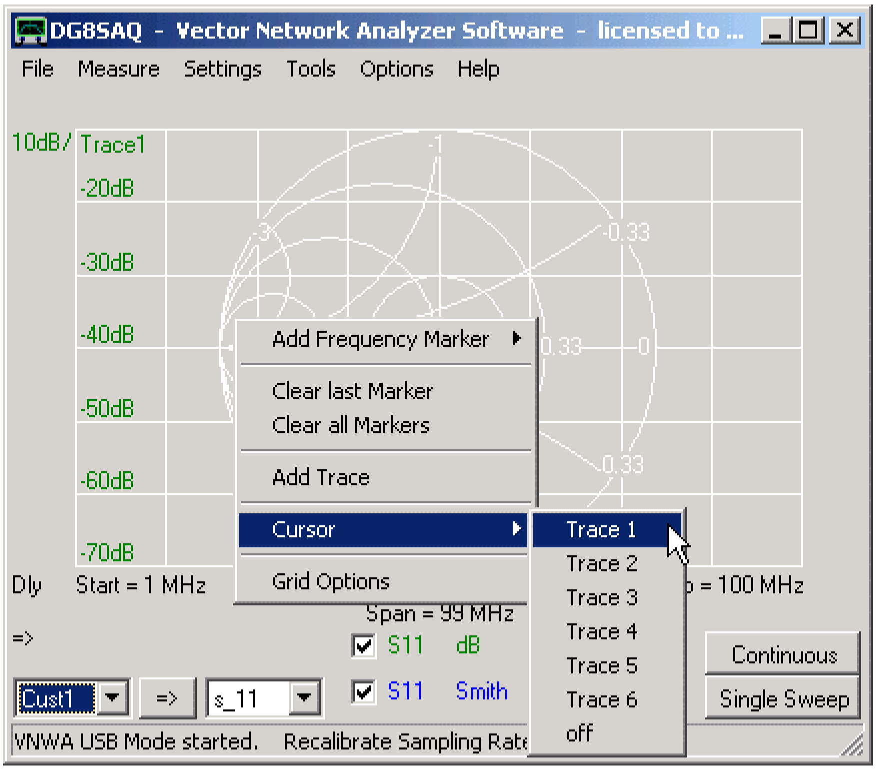
Available Time marker types:

- ***normal time marker***
- ***maximum time marker*** (automatically jumps to the maximum of the first applicable displayed time trace)
- ***delta time marker*** (displays the time distance and the vertical distance to the last normal time marker)

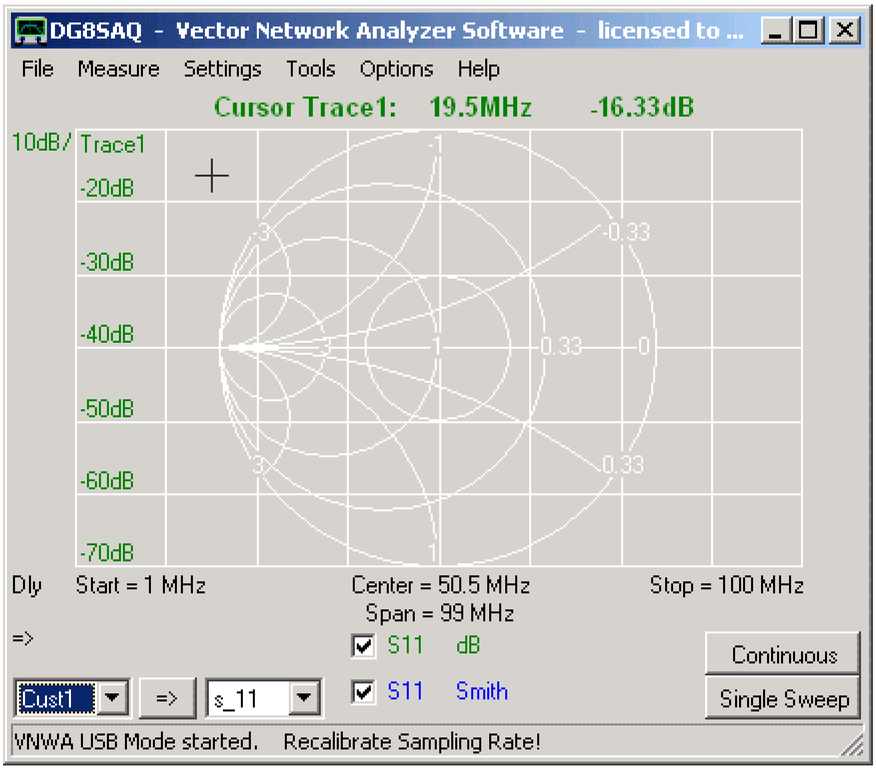
Available RADAR marker types:

- ***normal***
- ***maximum***(automatically jumps to the maximum of the first displayed RADAR trace)
- ***minimum*** (automatically jumps to the minimum of the first displayed RADAR trace)
- ***opening angle*** (places a maximum RADAR marker, two RADAR markers down one vertical division below the maximum marker and a center marker between the opening angle markers)
- ***delta marker*** (displays the angular distance and the vertical distance to the last normal RADAR marker)

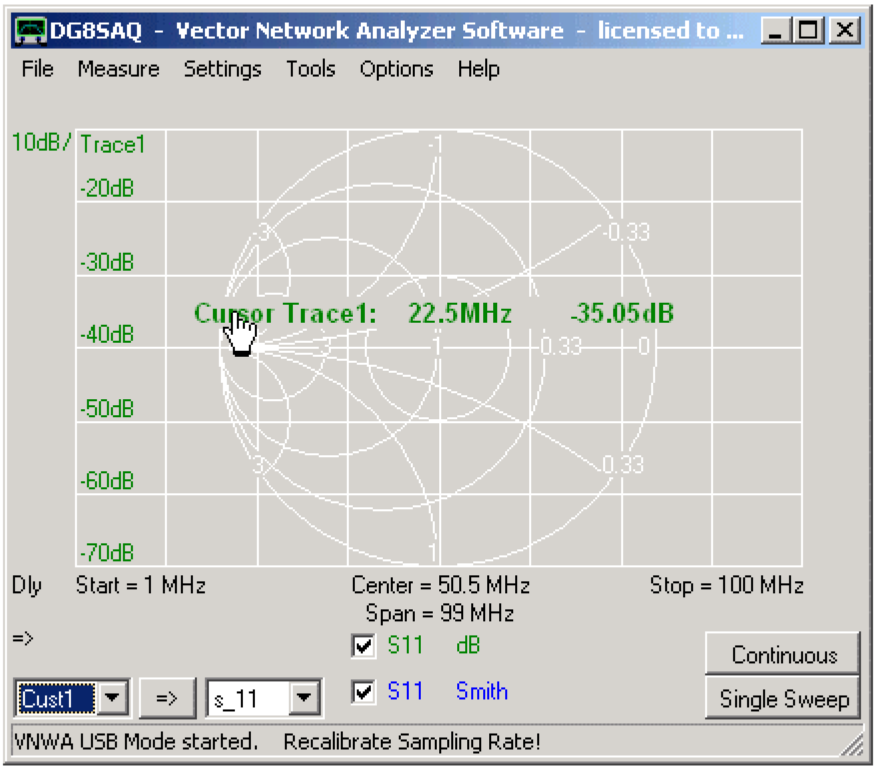
A **cursor** can be used **to determine grid coordinates** in a carthesian or Smith grid.
The cursor can be switched on by **right-clicking onto the grid**, selecting **"Cursor"** and selecting the trace for which the cursor is to be used for:



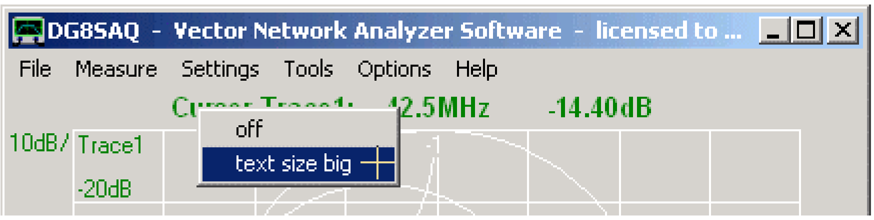
The mouse pointer will change to a cross and a **cursor label** colored with the trace color showing the cursor coordinates will appear right below the main menu:



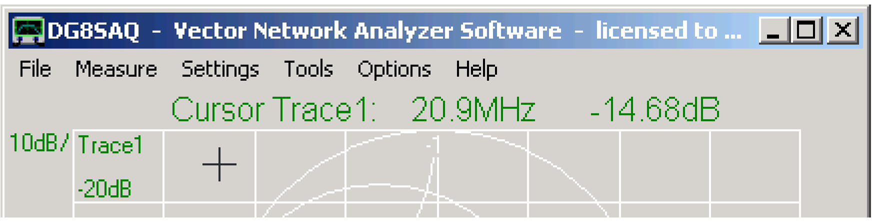
The **cursor label can be moved with the mouse** to any convenient position by holding the left mouse button pressed while pointing onto the label:



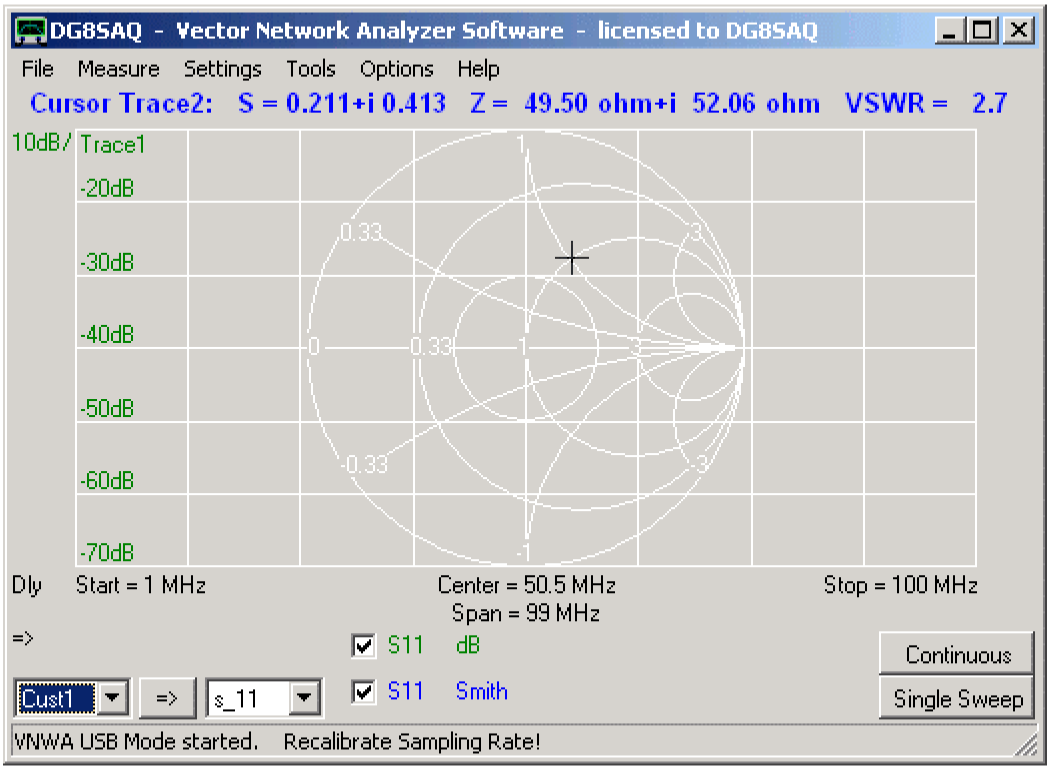
Right-clicking the label will open the label menu:



The label text size can be increased:

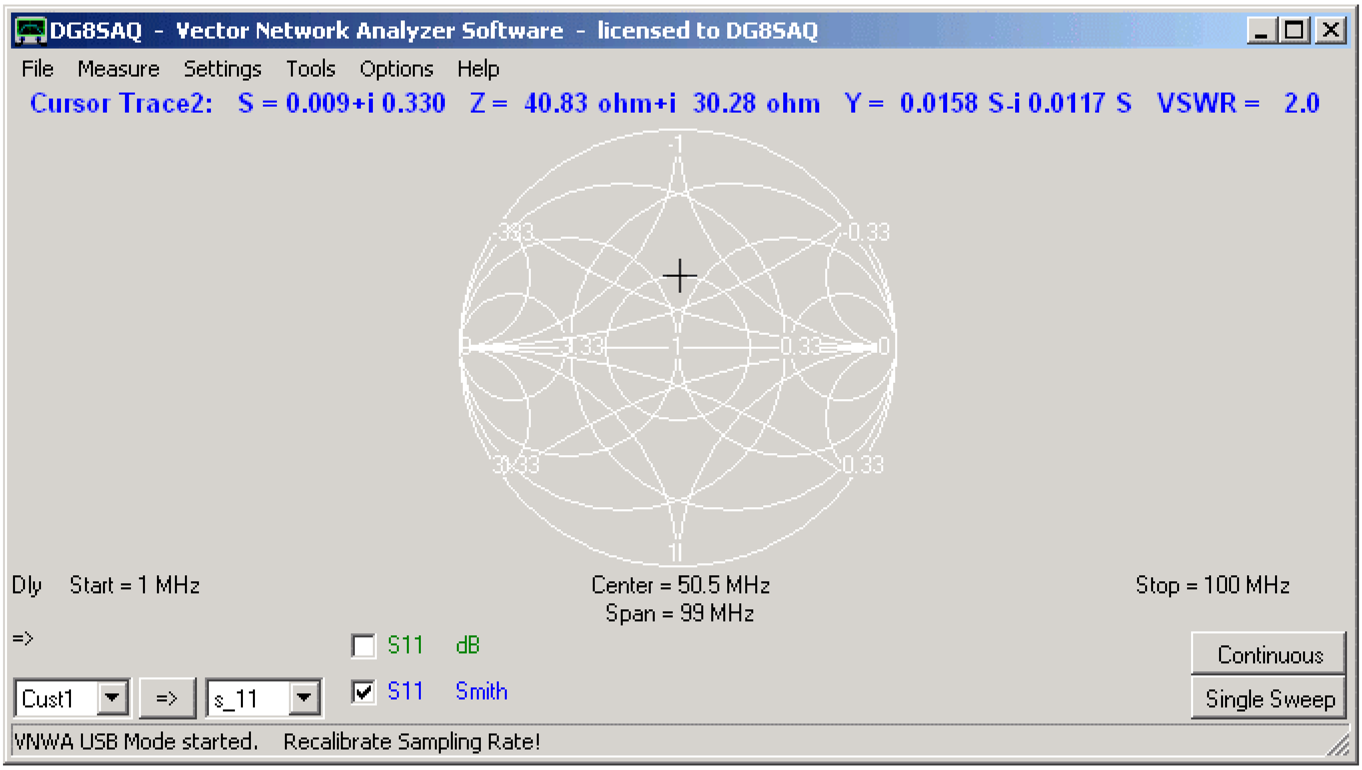


Extended information is available in a Smith grid:

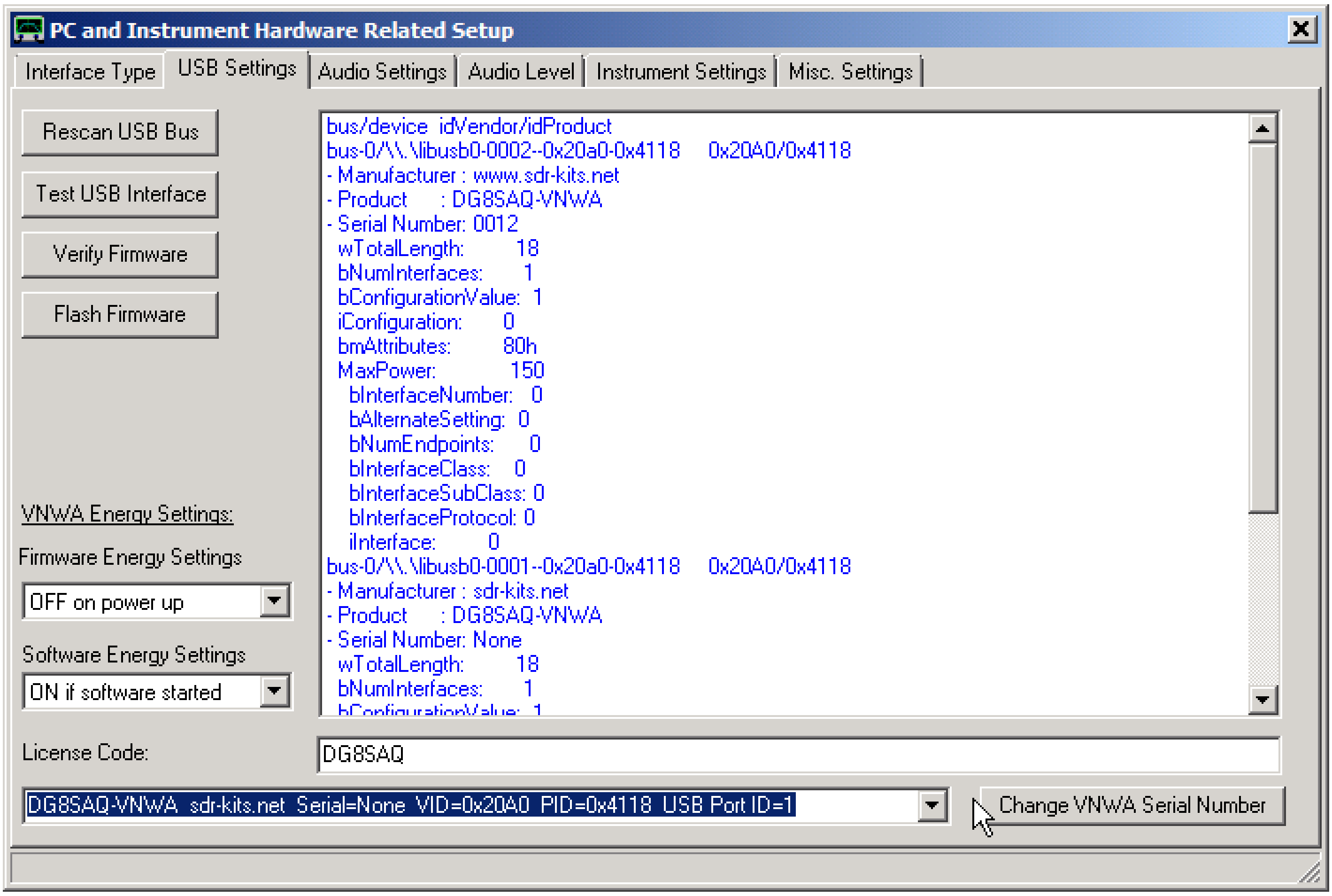


▸▸▸ **Note:**
Displayed coordinate data depends on the selected Smith grid options:
- **Impedance** information is only shown if the impedance grid is activated.
- **Admittance** information is only shown if the admittance grid is activated.
- **VSWR** information is only shown if VSWR circles are activated.

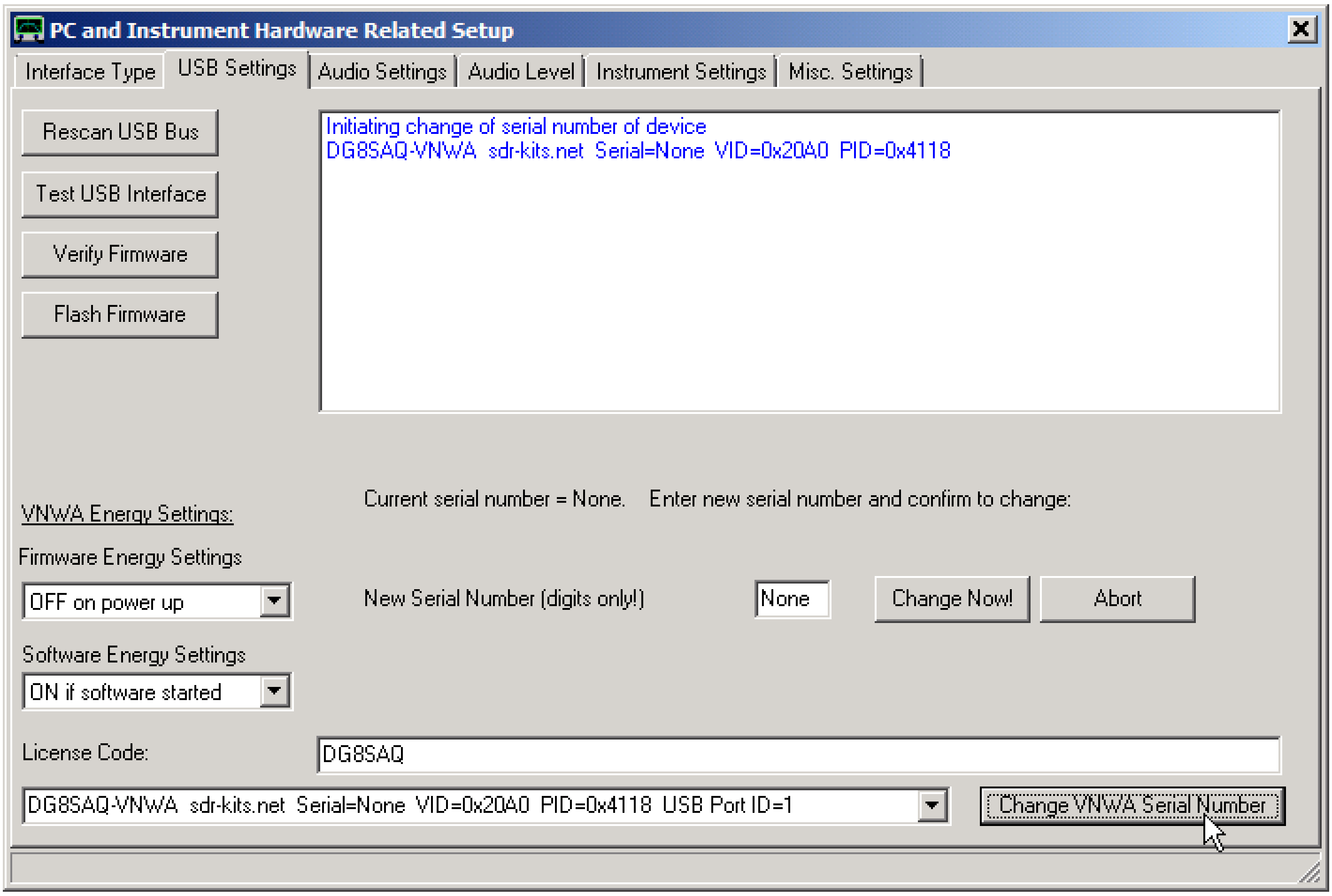
Here, all Smith grid options are activated:



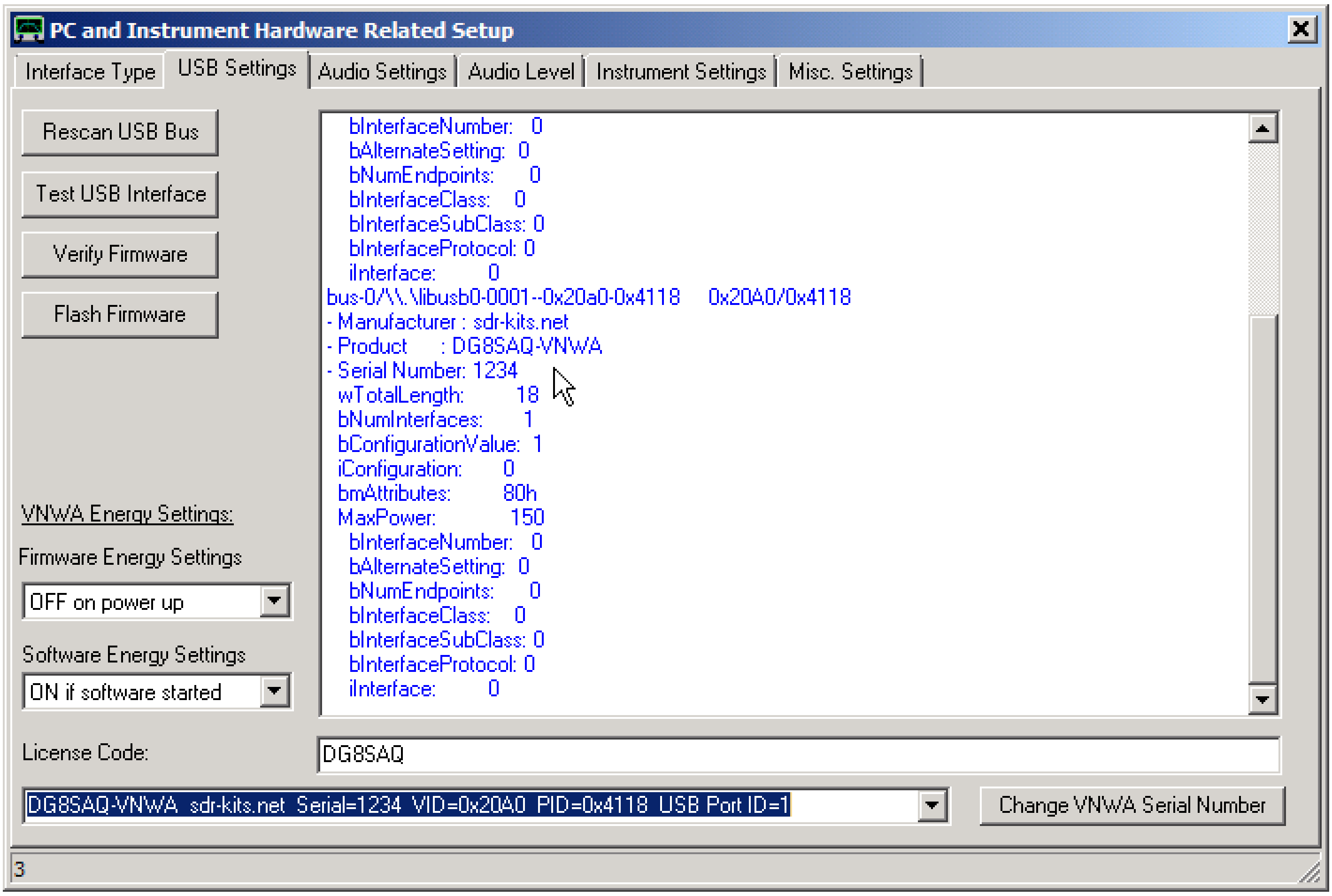
If you have more than one VNWA, you can have all VNWAs connected to your PC simultaneously. If **more than one VNWA is detected** upon program start or upon **"Rescan USB Bus"**, the "Setup"-"USB Settings" menu will show a combo box which allows the user to select one of the connected VNWAs (very bottom, next to the mouse pointer):



Note, that identical VNWAs can be distinguished only by their USB port ID, which is basically the order of device detection. This order might change upon reboot or replugging.
Therefore it is useful to modify the VNWAs' serial numbers, which are factory preset to "NONE". If more than one VNWA is connected, it is recommended to use consequtive serial numbers like 0001, 0002, ...
In order to **change a VNWA's serial number**, select the VNWA which you want to change and press the according button:



Enter a new 4 -digit serial number (e.g. 1234 in below example, you are only allowed to enter digits, no letters!) and press the **"Change Now!"** button:



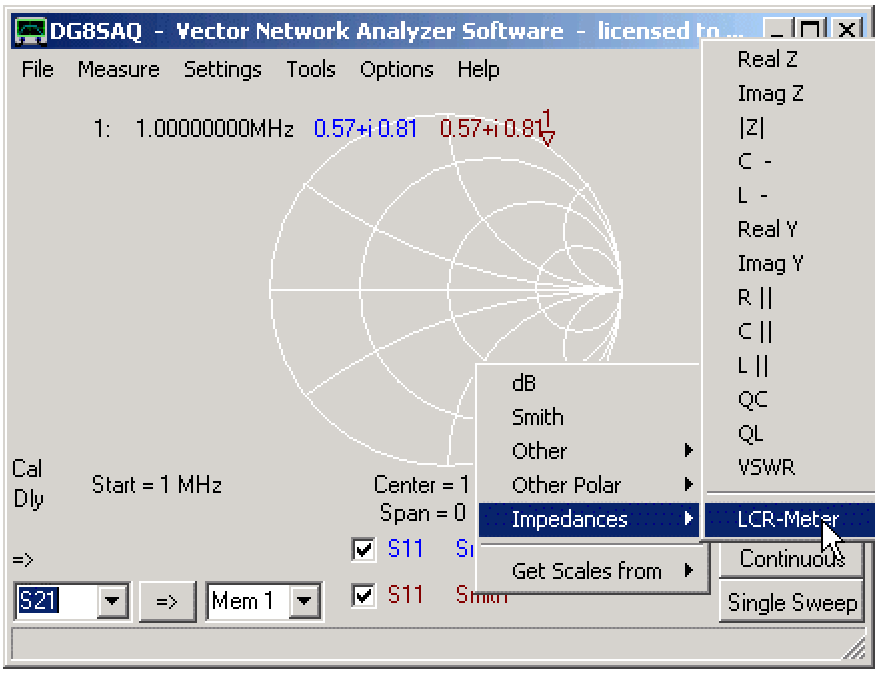
Note, that the VNWA serial has been changed to 1234 now (see entry next to mouse pointer above and also in the selection box).
From now on, this VNWA is uniquely determined and selected by its unique serial number 1234.

The VNWA (starting VNWA V33.z) software can be **controlled with the keyboard** to some extent.
For this purpose, some **keyboard shortcuts** are available **to initiate frequently used tasks**:

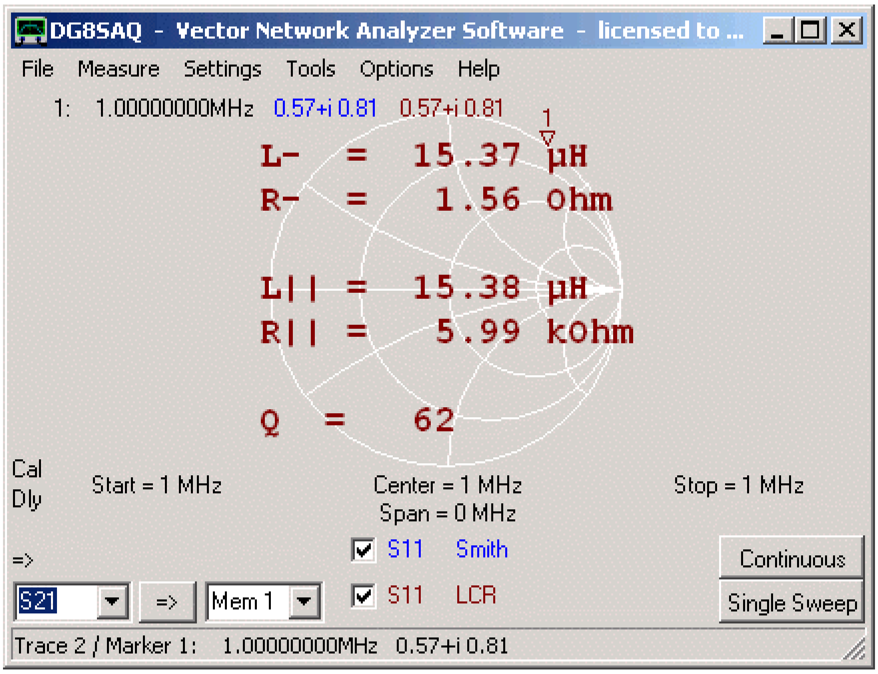
key:	**action:**
F1	open help file
F2	measure 2 port S-parameters
F3	measure 3-port S-parameters
F5	save screen to file
F6	print screen
spacebar	start/stop single sweep
return	start/stop continuous sweep
right arrow	measurement direction forward
left arrow	measurement direction reverse
+	add a trace
-	remove last trace
0	cursor off
1	cursor trace 1 on
2	cursor trace 2 on
3	cursor trace 3 on
4	cursor trace 4 on
5	cursor trace 5 on
6	cursor trace 6 on
a	open **a**verage and autosave
c	open **c**alibrate menu
d	open **d**isplay-trace menu
g	open **g**rid options menu
h	open **h**elp file
k	open cal **k**it settings menu
m	open **m**atching tool
n	open complex calculator tool (**n**umerics)
p	open **p**ort extensions menu
s	open **s**weep settings menu
t	open **t**hree-port analyzer tool
x	open **c**rystal analyzer tool

The VNWA + Software can be configured such, that it behaves like a standard LCR-meter being able to measure a complex impedance at a fixed frequency, decompose the result to parallel and series equivalent circuit and display the result numerically.

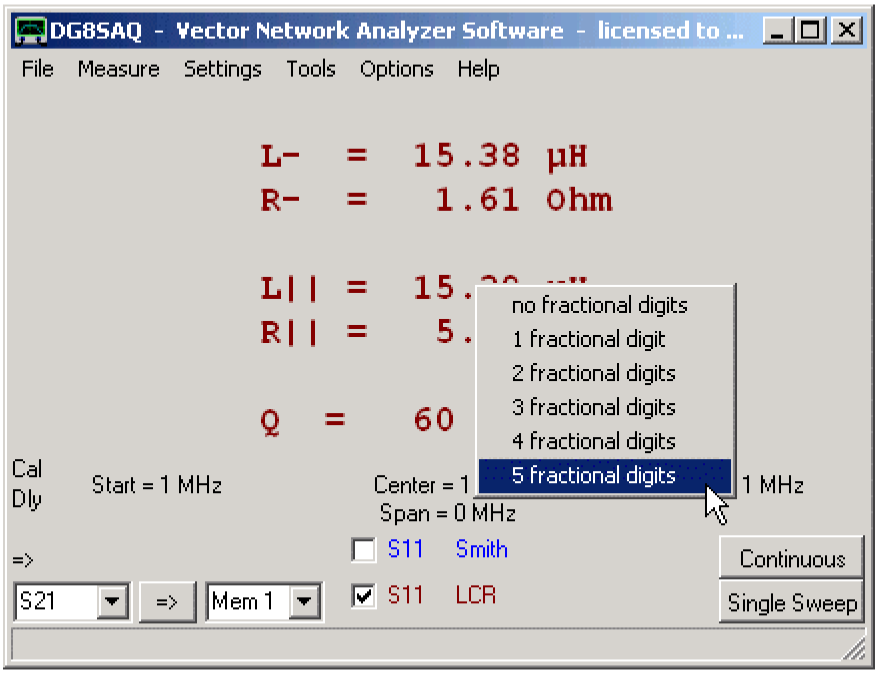
- To do so, first set the VNWA center **frequency** to the desired value and set the span to zero. Any other span will work, too, but as the evaluated numbers will be averages over the full sweep, these will also be frequency averages if a non-zero span is used.
- Set the number of data points and time per data point such, that a **sweep rate** of 1...10 sweeps per second will be performed. This will be the rate, at which the LCR and Q-values will be updated.
- Next perform a **SOL-calibration** in order to accurately measure S11, and if desired, move the calibration plane by means of a **port extension** to where you would like it to be, e.g. to the tip of a test fixture.
- Next, select **display mode LCR-Meter** for an S11 trace as seen below.



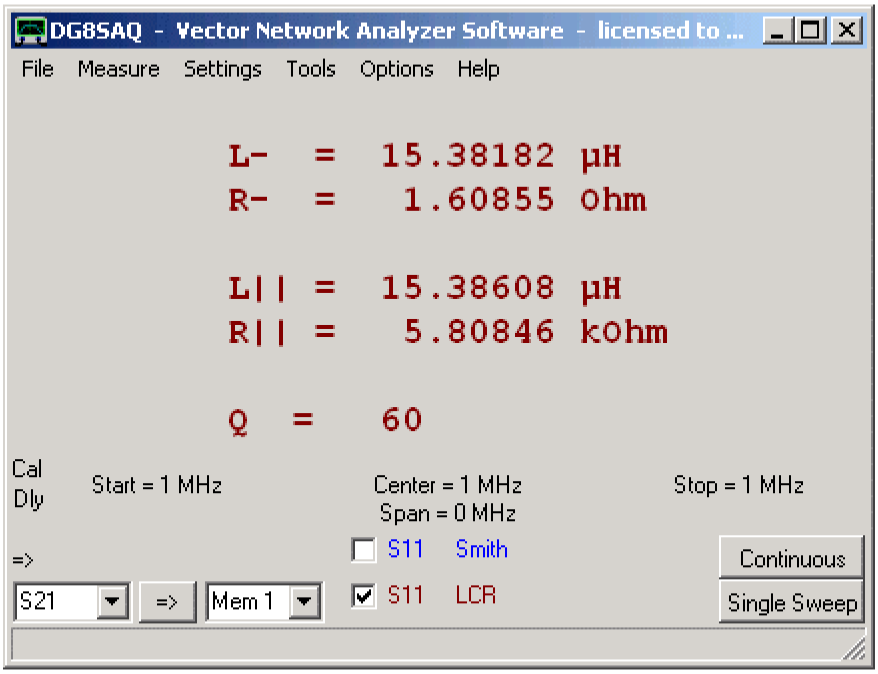
As can be seen below, numerical values appear on the main window. They display **L,C,R and Q-values** for parallel and serial equivalent circuit of the impedance to be connected to the TX port and to be measured into S11. Sweep continuously and change the test object on the TX port. You will see how the displayed values change after every sweep. The software will automatically detect if the test object is capacitive or inductive and chose to display Henries or Farads accordingly.



The number of displayed **fractional digits** can be controled by **right-clicking onto the displayed numbers** as can be seen below. Note, that all other diagrams must be disabled in order to do so.



The above action will cause 5 fractional digits to be shown (below) instead of 2 fractional digits (above).



▶▶▶ **Note:** As inductivity and capacity are frequency dependent effects, that tend to zero as the frequency approaches zero, L and C measurements will become more and more inaccurate the lower the measurement frequency.

The VNWA software allows to perform a realtime **FFT** on measured or imported data from frequency to time domain, display data in time domain, manipulate data in time domain by **gating** and perform an **inverse FFT** back to frequency domain.

A special topics section on time domain measurements demonstrates examples of both usages:

Time domain reflectometry is a useful technique to search defects in transmission lines.

The step response is useful to determine impedance variations along a coaxial cable.

Gating is used to separate responses depending on their arrival times, e.g. to separate the slow mechanical response of a crystal filter from the fast electromagnetic feedthrough of the test board.

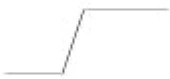
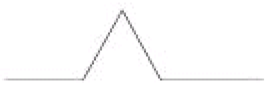
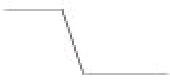
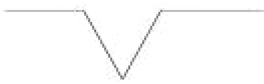
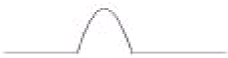
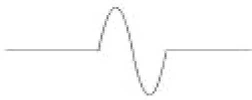
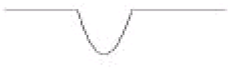
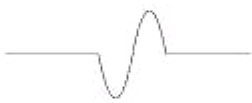
Time Domain Reflectometry

Time domain reflectometry (TDR) is a straight forward and widely used technique to find faults in cables and transmission lines. The idea is to **send a short voltage impulse** into the line and **detect the reflected impulse(s)** which is the so called **impules response**.:

Every not correctly terminated end of the line will reflect some of fraction of the incoming impulse. Also, cable damages like breaks or compressions will cause part of the incoming impulse to be reflected. By measuring the time delay of the impulse reflected by the damage one can calculate the location of the damage if the impulses velocity through the cable is known. The impulse in question must travel the distance from cable input to the fault and back.The impulse velocity (=speed of light * velocity factor) can be determined experimentally as well by observing the reflection from the far end of the cable and by measuring the length of the cable.

A **voltage step** is equally well suited as input signal for time domain reflectometry and it is technically simpler to generate. The reflected signal of a voltage step is called the **step response**.

In fact, every impedance change in a transmission line will cause some of the incident signal being reflected in a characteristic way:

Component	Step Response	Impulse Response
$\Gamma = 1$, Open		
$\Gamma = -1$, Short		
Resistor, $r > Z_o$		
Resistor, $r < Z_o$		
Inductor		
Capacitor		

(Image from Anritsu presentation "Time Domain Analysis UsingVector Network Analyzers" by Dr. Martin I. Grace)

Time Domain vs. Frequency Domain

An impulse or a step signal ***g(t)*** consists of a spectrum ***G(f)*** of an infinite number of frequencies ***f***.
The spectrum can be calculated with the **Fourier transform**:

$$G(\omega) = \frac{1}{\sqrt{2\pi}} \int_{-\infty}^{\infty} g(t) \cdot e^{-i \cdot \omega \cdot t} dt$$

with the angular frequency

$$\omega = 2\pi f$$

Example: An ideal Dirac impulse (width zero, infinite amplitude) will produce a constant spectrum which contains all (angular) frequencies from - infinity to +infinity with equal strength.

If the spectrum is known, the impulse shape can be reconstructed using the **inverse Fourier transform**:

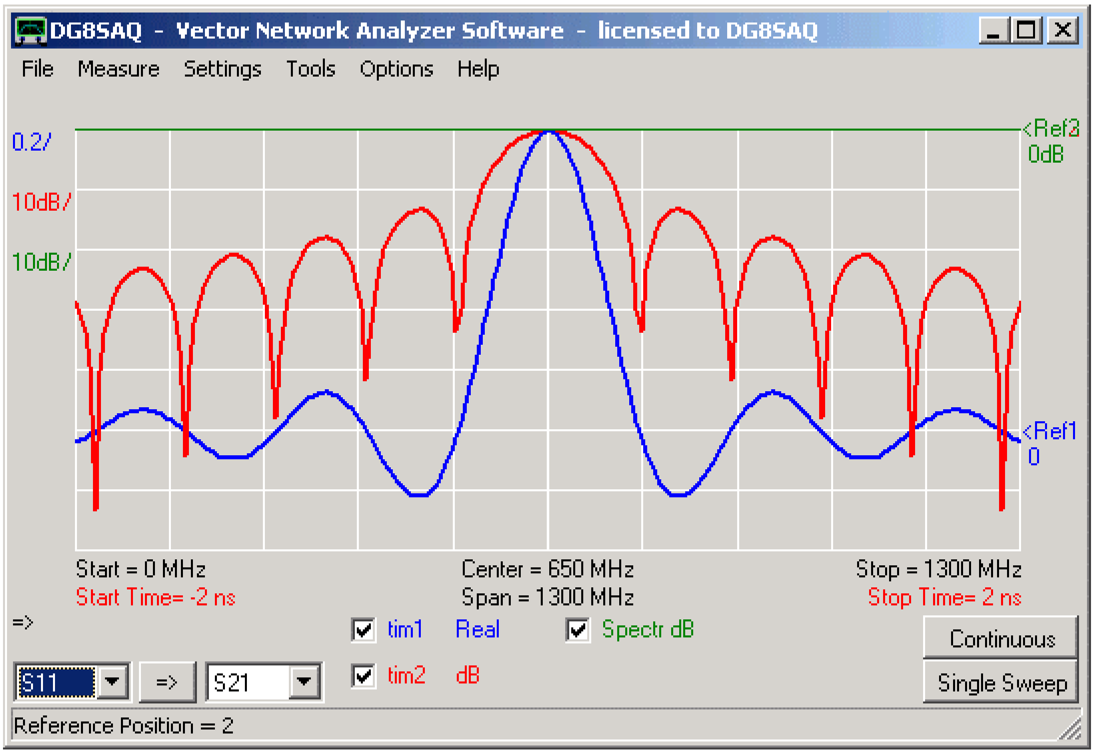
$$g(t) = \frac{1}{\sqrt{2\pi}} \int_{-\infty}^{\infty} G(\omega) \cdot e^{i \cdot \omega \cdot t} d\omega$$

The same holds true for any time signal ***g(t)***. Thus, the time signal and the corresponding spectrum or frequency signal contain the same information.

While a classical time domain reflectometer can generate impulses and measure the impulse response ***g(t)*** in time domain, a VNA cannot do this. On the other hand, a VNA can generate sine waves with almost arbitrary frequencies and thus measure the frequency response ***G(f)***. Since time domain response and frequency domain response are interrelated through the Fourier transform, **the impulse response of a system can basically be reconstructed mathematically from the frequency response measured with a VNA**.

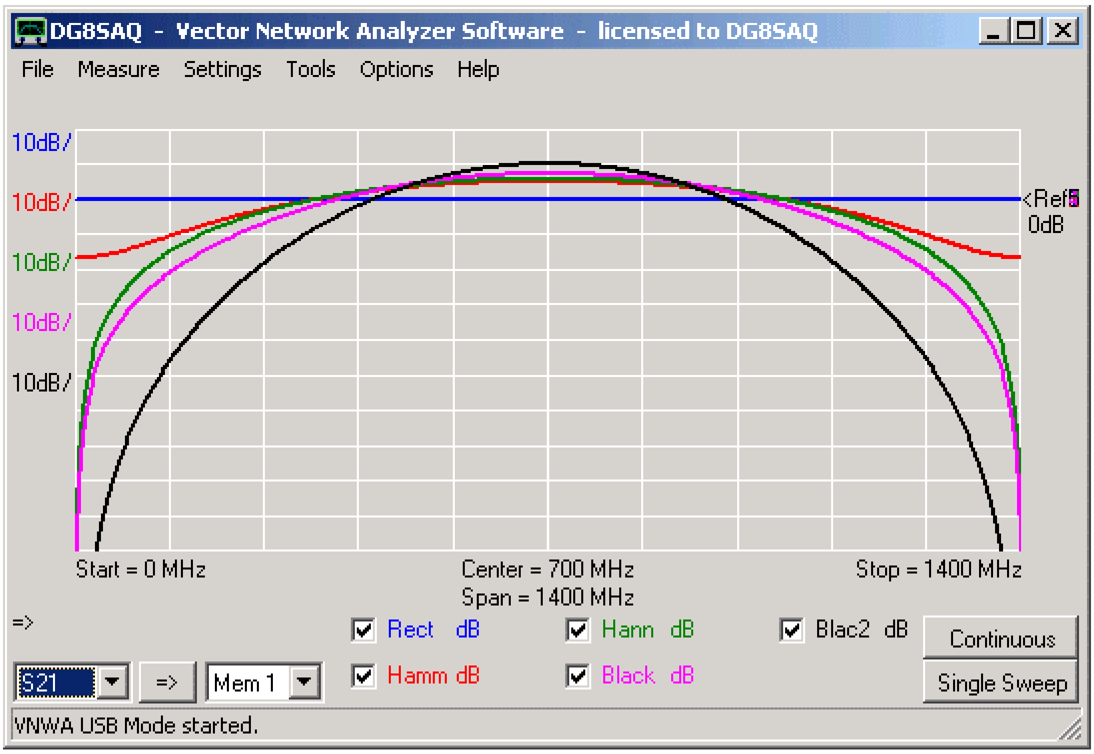
Finite Frequency Range Problem, Windowing

Any existing VNA has only a limited frequency range. Outside this frequency range a system's frequency response cannot be measured. **Reconstructing a time response from such a frequency limited response will generally produce artefacts**. The following graph shows an impulse reconstructed from a spectrum obtained by limiting the spectrum of an ideal impulse to the displayed frequency range:

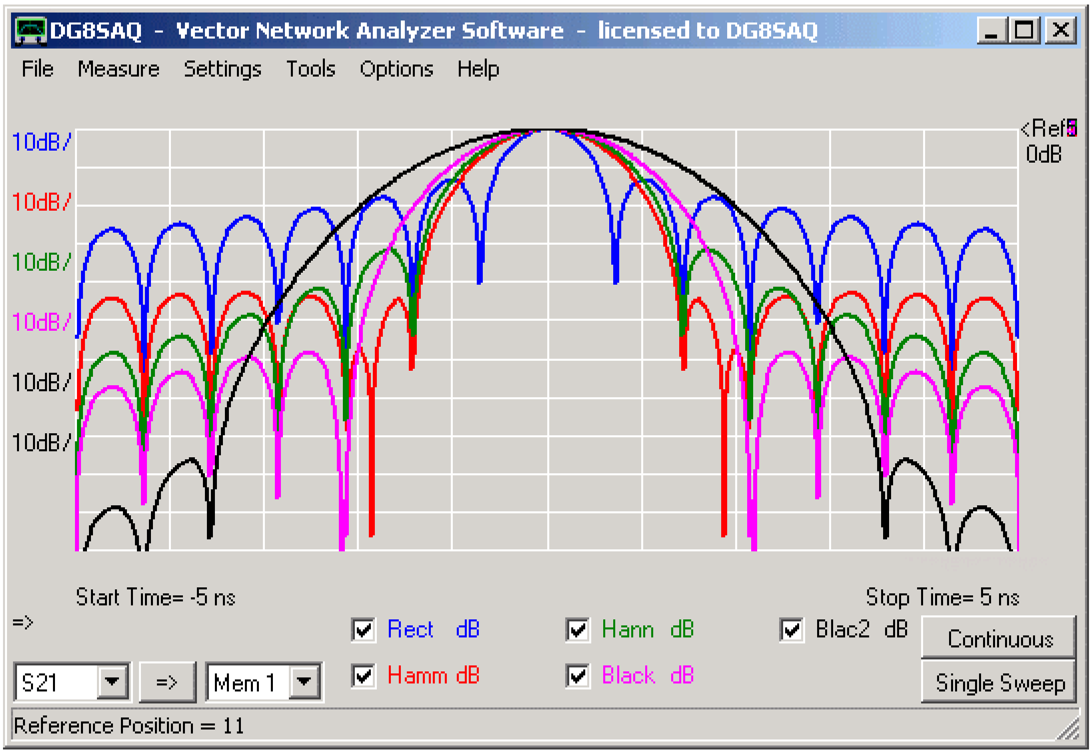


The blue impulse reconstructed from the band limited green spectrum is widened as compared to an ideal Dirac impulse and it shows sidelobes according to the law sin(t)/t. The red trace shows the same reconstructed impulse in dB scale. Note that the zeros of the blue reconstructed impulse produce very distinct notches in the blue dB scale. **These notces and sidelobes are not related to any test object but are merely a consequence of the limited bandwith.** These sidelobes might well cover up some real lower level DUT response. In order to detect such covered low level response the windowing technique was invented. By weighting the frequency response by appropriate windowing functions (e.g. by Hamming, Hanning, Blackman...) the sidelobe level can be considerably reduced. The tradeoff is a reduces time resolution, i.e. the reconstructed impulse becomes wider:

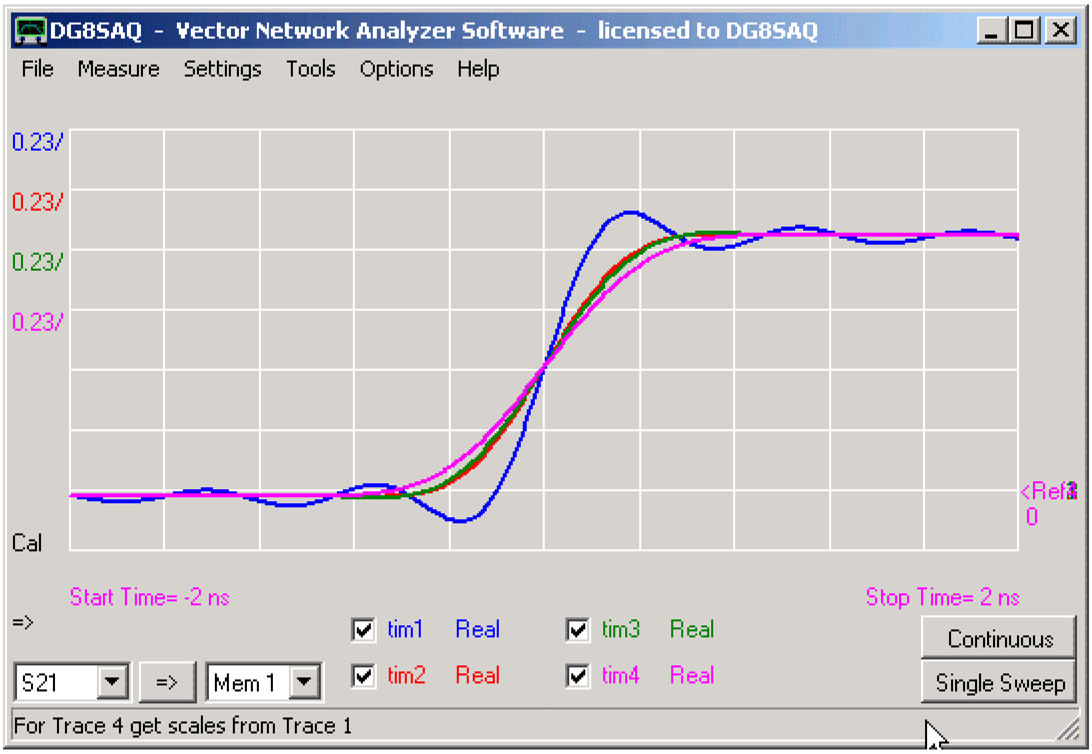
Implemented windowing functions (rectangular=none, Hamming, Hanning, Blackman, Blackman squared):



Effects of applying the above windowing functions:



Note, that the narrower windowing functions lead to wider reconstructed impulses with generally lower sidelobe levels.
The same behavior can also be observed in the step response:



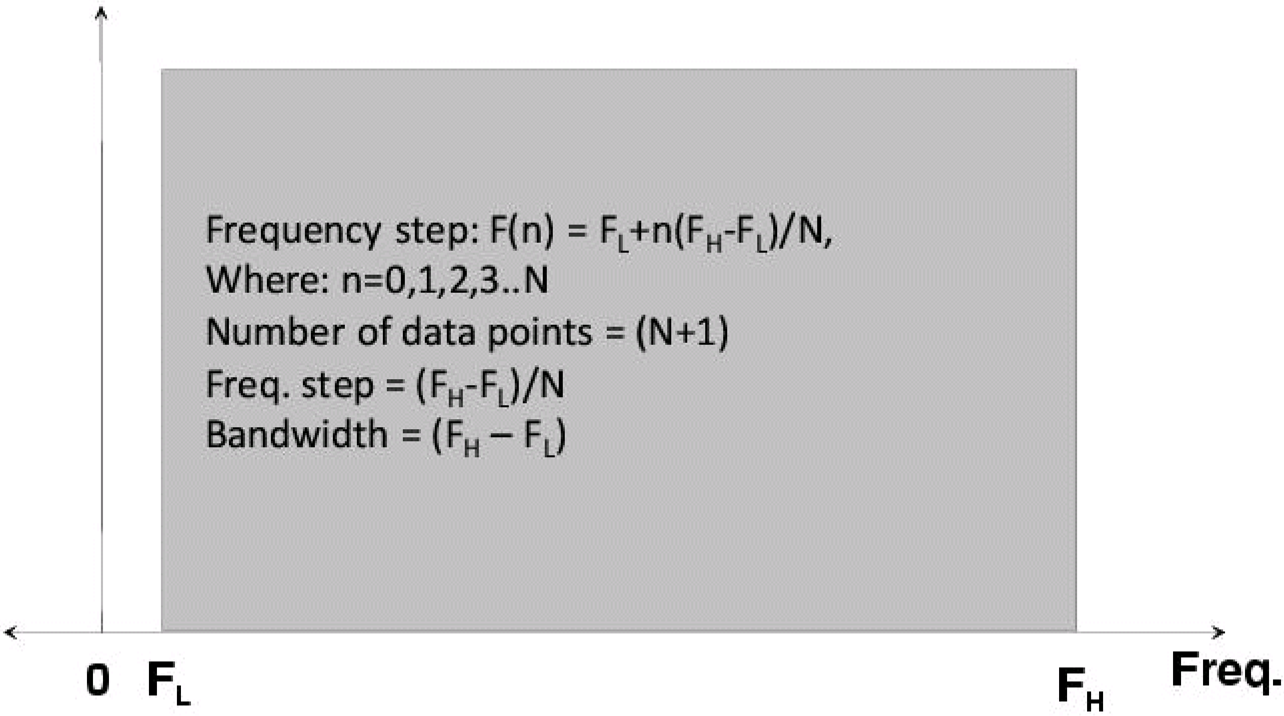
Colors in above diagram correspond to the window functions in the previous diagrams. Sidelobes and impulse width in impulse response correspond with overshoot and step steepness in step response.

Band Pass vs. Low Pass Mode

There are two possible ways to process a measured frequency response in order to obtain a time response, namely band pass and low pass processing.

Band Pass Processing:

In band pass mode, the utilized frequency range is the VNA frequency span.

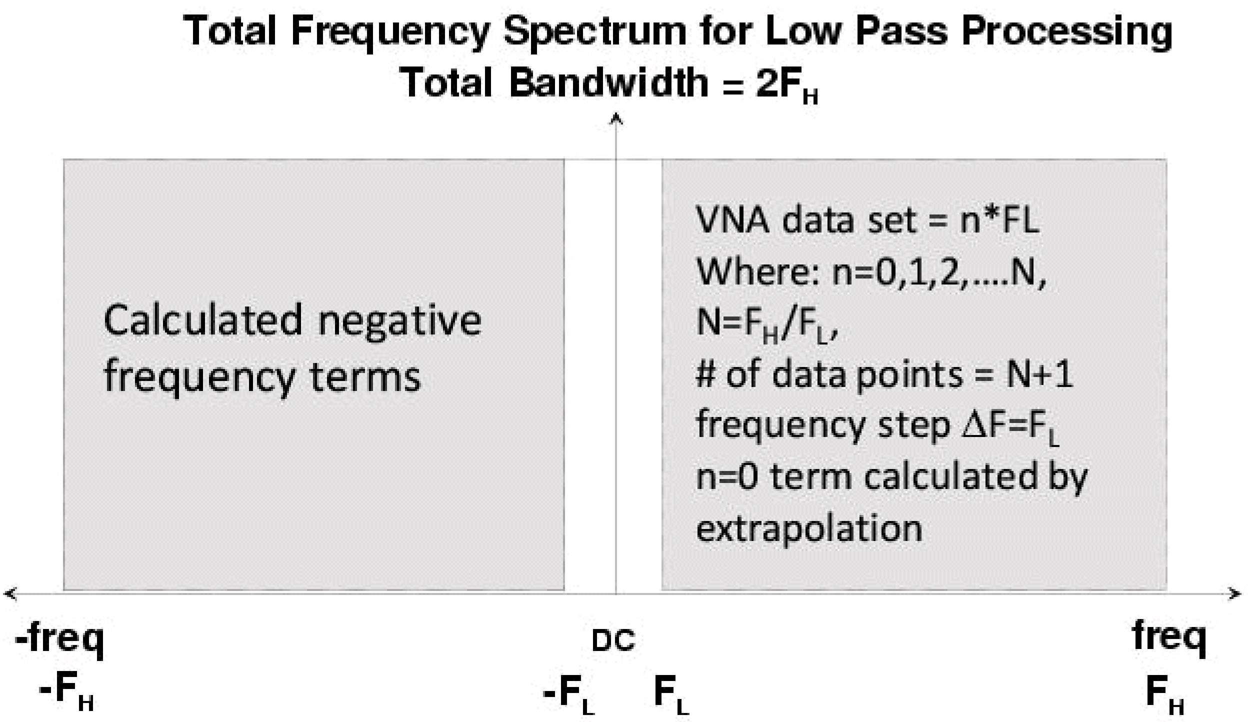


(Image from Anritsu presentation "Time Domain Analysis UsingVector Network Analyzers" by Dr. Martin I. Grace)

This mode ignores negative frequencies. As a consequence of omitting negative frequencies the reconstructed time response is unphysically complex valued. Usually, only the magnitude of the band pass response is of interest. It might give useful information for tuning filter structures, see e.g. Agilent AN 1287-10 "Network Analysis Solutions Advanced Filter Tuning using Time Domain Transforms". Note, that window functions will suppress both the high frequency and the low frequency content of the frequency response in this case. As a voltage step contains strong low frequency components, **band pass mode cannot be used for calculating a step response**.

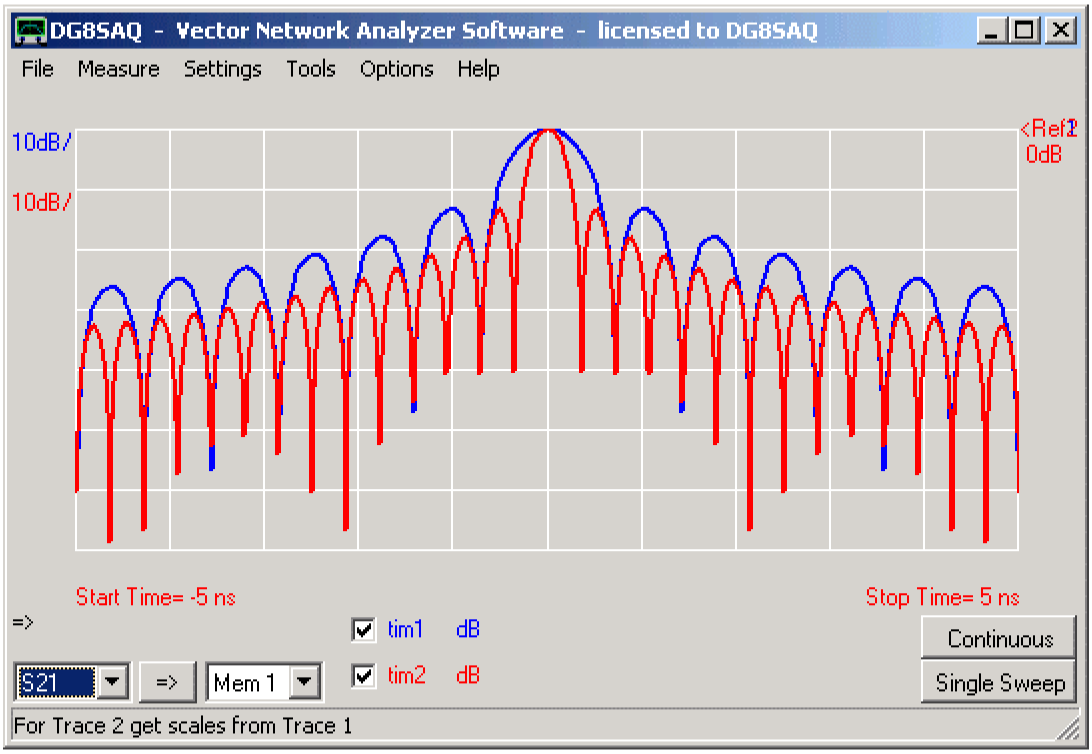
Low Pass Processing:

Low pass mode makes use of the inherent symmetry of the frequency response with respect to frequency zero. Thus, having measured the response for positive frequencies, the response for negative frequencies can be reconstructed mathematically and can be used to calculate a real valued time response, i.e. the imaginary part is zero as expected for a time response. Note that the measured frequency span should extend to (almost) zero in order to obtain good time domain results.



(Image from Anritsu presentation "Time Domain Analysis UsingVector Network Analyzers" by Dr. Martin I. Grace)

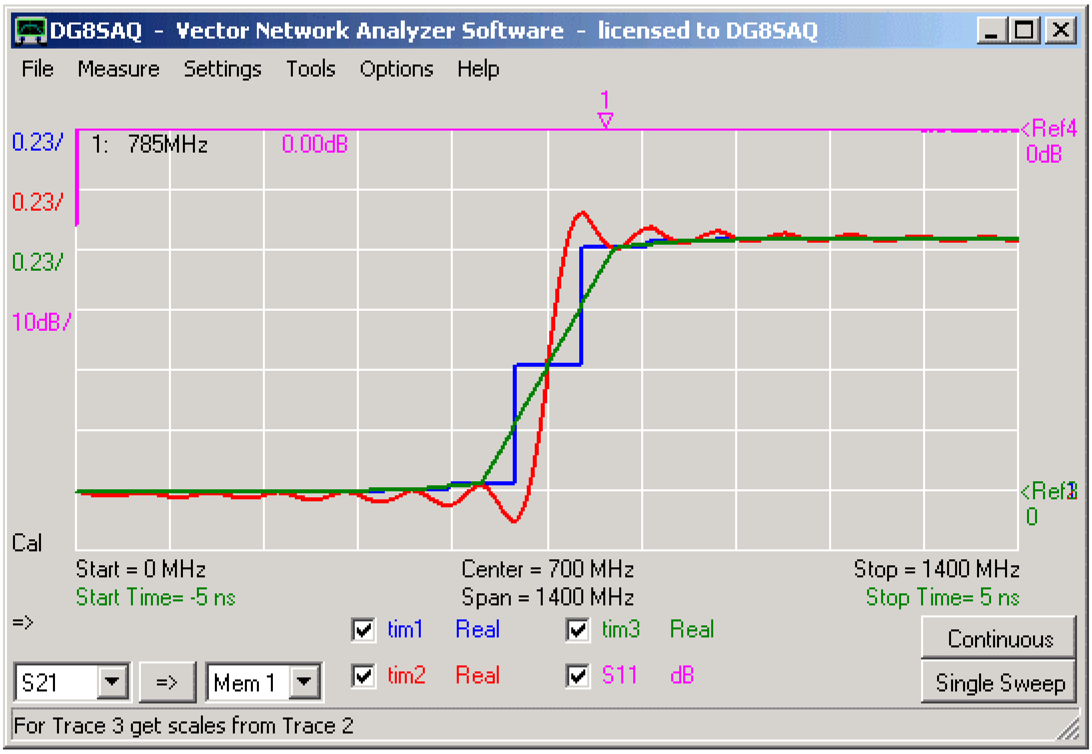
By using this trick, **the utilized frequency bandwith is doubled and thus the time resolution is improved by a factor of two** as can be observed in the following example showing the reconstructed impulse responses of identical frequency data for band pass mode and low pass mode.



Note, that for band pass mode the window functions have their maxima located at zero frequency, which makes this mode particularly useful for reconstructing step responses which contain strong low frequency content.

Fourier Transform Options

There are many algorithms to calculate an inverse Fourier transform for reconstructing a time response from a measured frequency response. The most widely known algorithm is the **FFT or Fast Fourier Transform** algorithm by James Cooley and John W. Tukey. It is very efficient, but it can only calculate a time response at a strictly equidistant grid of times predefined by the frequency span and by the number of data points. If the times of interest lie in between this grid, FFT cannot be used. In such a case, the **Discrete Fourier Transform (DFT)** has to be calculated by some other less CPU efficient algorithm. The VNWA software offers three choices to calculate and display reconstructed time data, namely FFT, FFT interpolated and DFT:



Each FFT data point is displayed as a plateau with the point sitting in the center of the plateau.
In contrast, FFT interpolated linearly interpolated the FFT data points.
DFT also calculates time data in between the fixed FFT grid and uncovers the oscillations caused by the finite frequency span.
Note, that all traces must coincide on the FFT grid, i.e. at the centers of the blue plateaus.
Note, that the above data shows the step response of the open VNWA TX port reconstructed from the displayed S11 measurement without window function, i.e. rectangular window with maximum width.

Time domain reflectometry measures the impulse response of a DUT, e.g. a transmission line. Classicaly, one applies a short impulse to the line input and measures the time dependent voltage at the line input. If the line is perfectly terminated, the impulse will not be reflected back to the line input and no second impulse will be measured after the initial one. If the line is broken somewhere in the middle, some of the input impulse will be reflected there. The position of the defect can then be calculated from the delay of the reflected impulse relative to the initial one, if the velocity factor of the transmission line is known.

The VNWA cannot perform impulse measurements, but it can do a wideband frequency measurement and perform an inverse Fourier transform on it, which is equivalent to an impulse measurement.

VNWA time domain reflectometry basics:

The time (and thus length) resolution depends on the frequency sweep span.
▸▸▸ **Hint:** Do your frequency sweep with the maximum possible span starting at 0 in order to obtain the highest possible temporal and spatial resolution. With the maximum span of 1.5 GHz a spatial resolution up to a millimeter can be achieved.

The longest time (and thus length) you can measure depends on both, the number of measurement points and on the frequency sweep span.
Because of the periodic nature of the Fourier transform the maximum allowed delay you can measure is *number of points * resolution.* The resolution order of magnitude is 1/frequency span.
▸▸▸ **Hint:** Make sure that you have no signals coming in later than the maximum allowed delay, otherwise those will be folded back in time and create ghost signals. Use a larger number of points if necessary.
▸▸▸ **Hint:** If you want to measure really long delays and you have reached the maximum allowed number of points, decrease the frequency span and thus the resolution.
▸▸▸ **Hint:** The FFT responses are computed most efficiently if the number of frequency points is chosen to be a **power of 2**. There is no such efficiency benefit for the DFT modes.

Example:

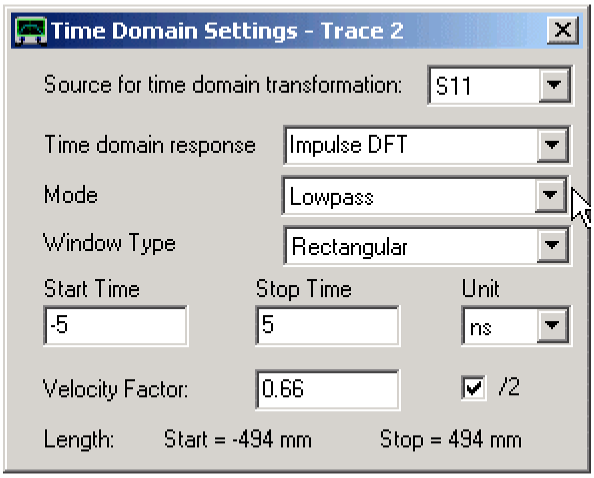
1. Calibrate the VNWA for a reflection measurement (SOL, use auto-clock multipliers). In below example I have used 2000 data points and a 4 milliseconds per data point.

2. Measure the reflection coefficient (S11) of your bare open VNWA TX output and display it in trace1. The result should be one (= 0dB) except for some spurs.

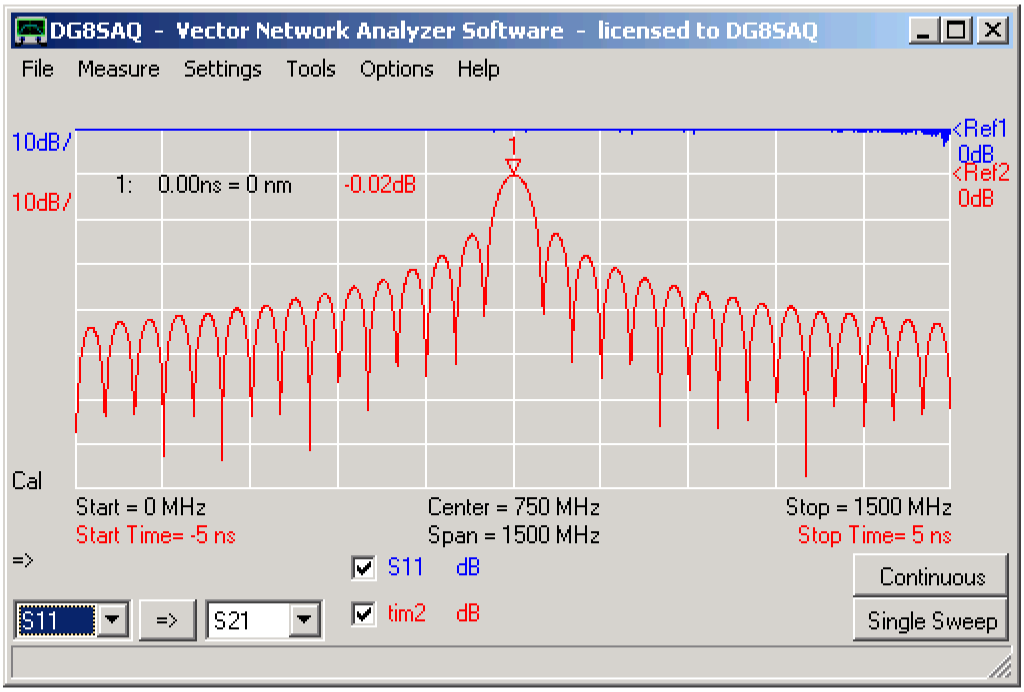
3. Add a second trace with trace type "Time" and open the "Time Domain Settings" via the Settings-Display menu or by the appropriate popup menu on right-clicking the display label.

4. Open the time domain settings window by activating the Settings - Time Domain menu or by double-clicking one of the time labels.

5. Select source **S11**, select **"Imulse DFT"**, **"Lowpass"** and **"Rectangular"** for maximum time resolution and edit **start and stop times** as seen below:

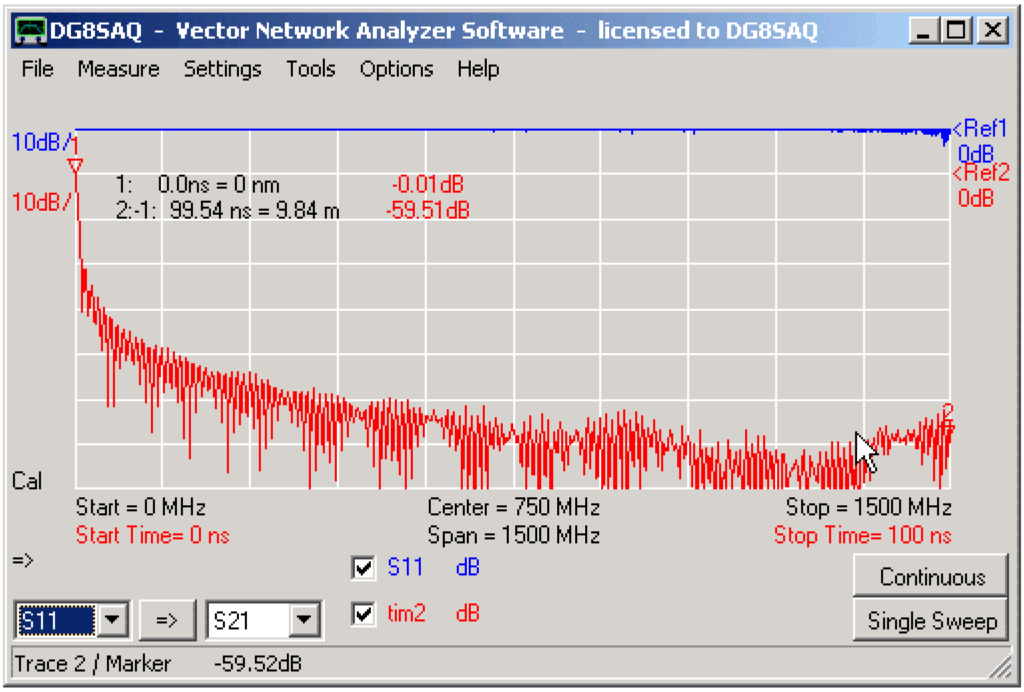


6. Close the time domain settings window. After proper scaling you should see the following result:



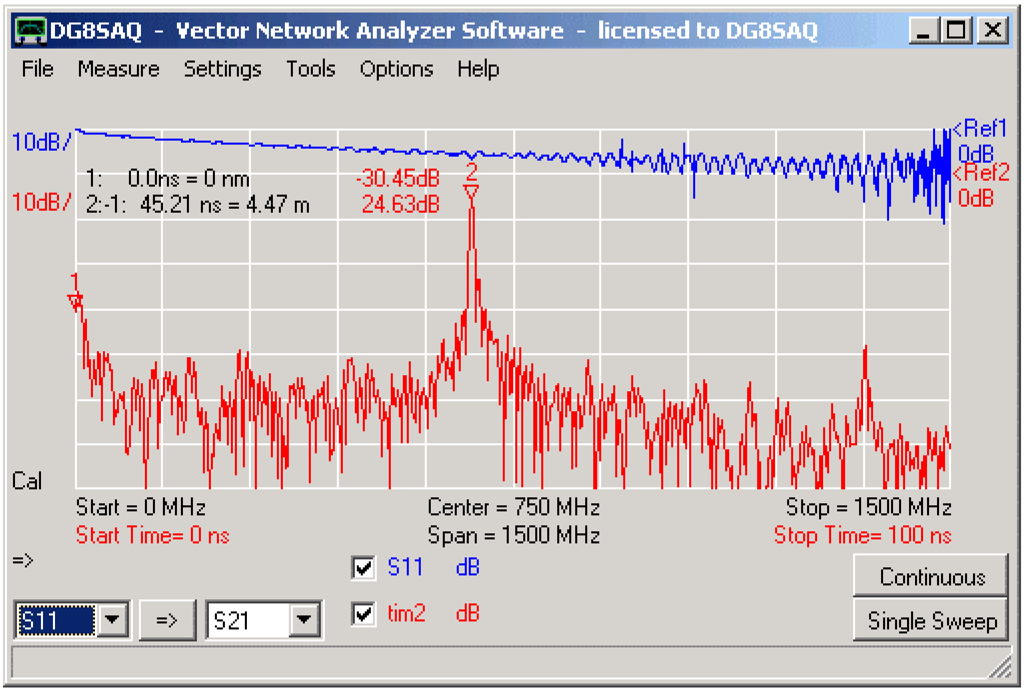
The blue trace shows the constant 0dB reflection of the open TX port in the frequency range 0...1.5GHz. The red trace shows the Fourier transform in the time range of -5ns...5ns, which is equivalent to the length range of about -50cm...50cm. We see a sharp peak at length 0 with level 0dB (see marker caption), which means that all of the TX signal is reflected at the calibration plane. The oscillatory sin(x)/x behavior stems from the fact, that no frequency windowing function is selected.

7. After adjusting for a wider time span, you will see the following.



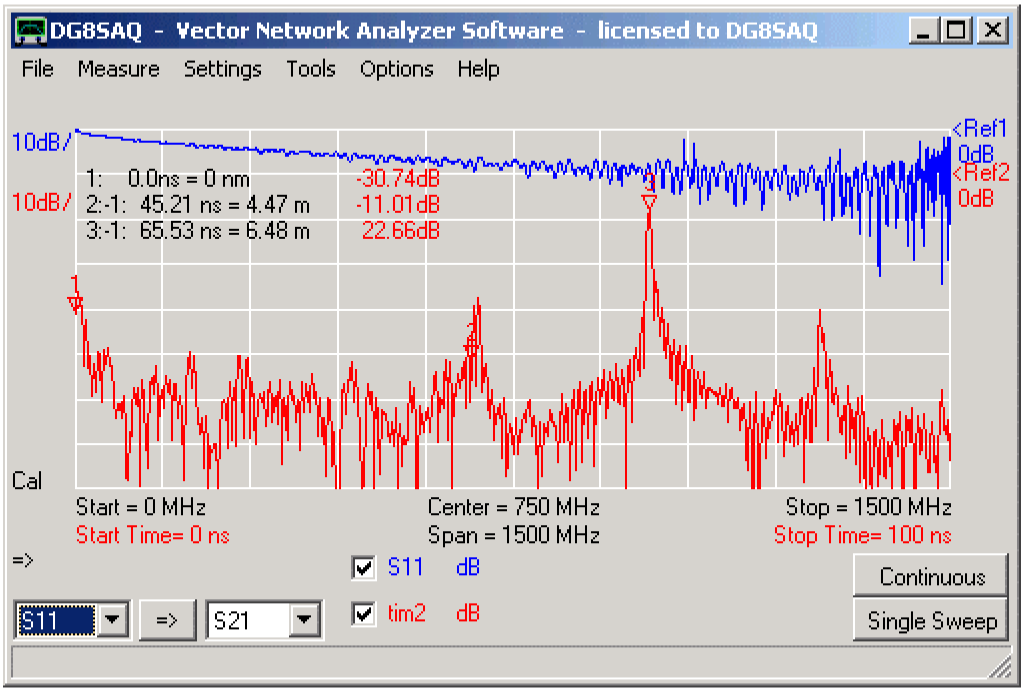
In the above image we look at a length span of about 10 meters. We only see the signal reflected at zero length off the calibration plane.
Note, that with 1.5GHz span and 2000 points a maximum reflection length of 150 meters can be measured. The maximum available 8192 data points at 1.5GHz span allow to measure reflection lengths of around 0.6 kilometers with a resolution of a few millimeters.

8. Connect an open ended coaxial cable to the TX port and sweep.



Note, that the reflected signal now comes in attenuated and with a time delay equivalent to the cable length of 4.5 meters (see time delta marker 2). We have thus measured the length of our coaxial cable.
Note, that there is still weak reflection at zero length coming from the SMA to BNC adapter.
Note, that there is also a signal at 9 meters delay steming from multiple reflection.

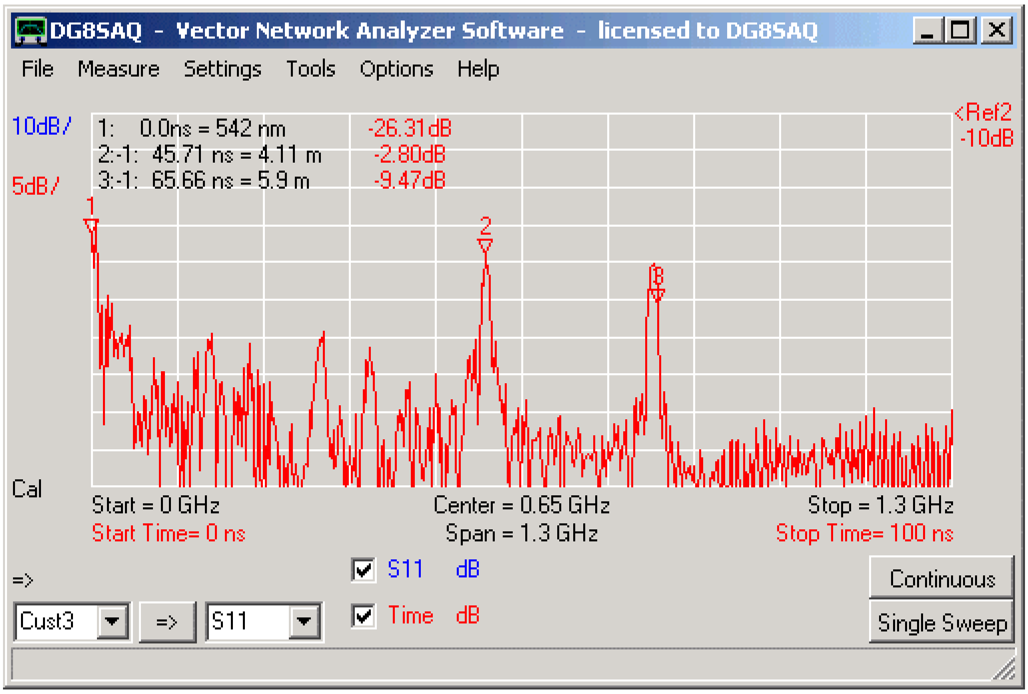
9. Next, I have connected a second open ended coaxial cable to the end of the first one. Note, that the second cable has 90 degree angled BNC connectors:



The reflection is now shifted by the length of the additional cable. The total cable length is now 6.5 meters. Note, that some signal is still reflected at the end of the first cable due to the imperfect wave match of the angled BNC connector.

=> Cable defects can be detected and localized very accurately by this method.

10. Finally, I have connected a BNC load to the end of the second cable:



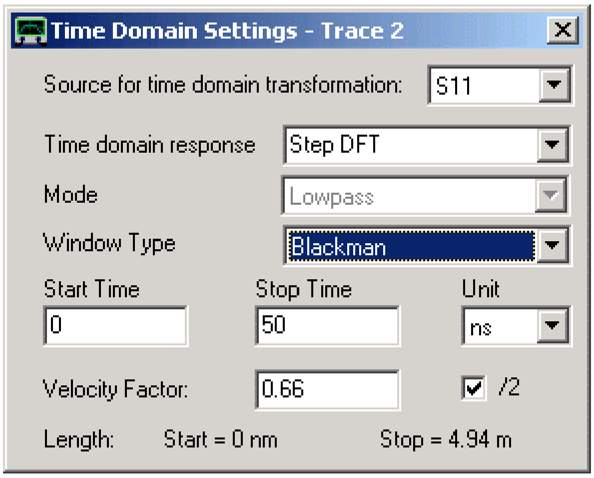
Note, that we still see reflections from the connector bend in the middle and at the end of the cable. Also note, that the overall reflected signal has decreased considerably due to the cable termination.

While the impulse response is the DUT's response to a short voltage impulse, it is sometimes useful to look at the DUT's response to a sharp voltage step which is called the **step response**. The two are closely related. E.g. in time domain, integration of the impulse response basically yields the step response. This means, every impedance change at a certain delay will produce an impulse in the impulse response at a certain delay time, which in turn will produce a step in the step response at the same delay time. The useful feature is that the step height is related to the impedance change. **In other words, the impedance change can be calculated from the step height, thus enabeling one to see varying cable impedances at varying delays or positions.**

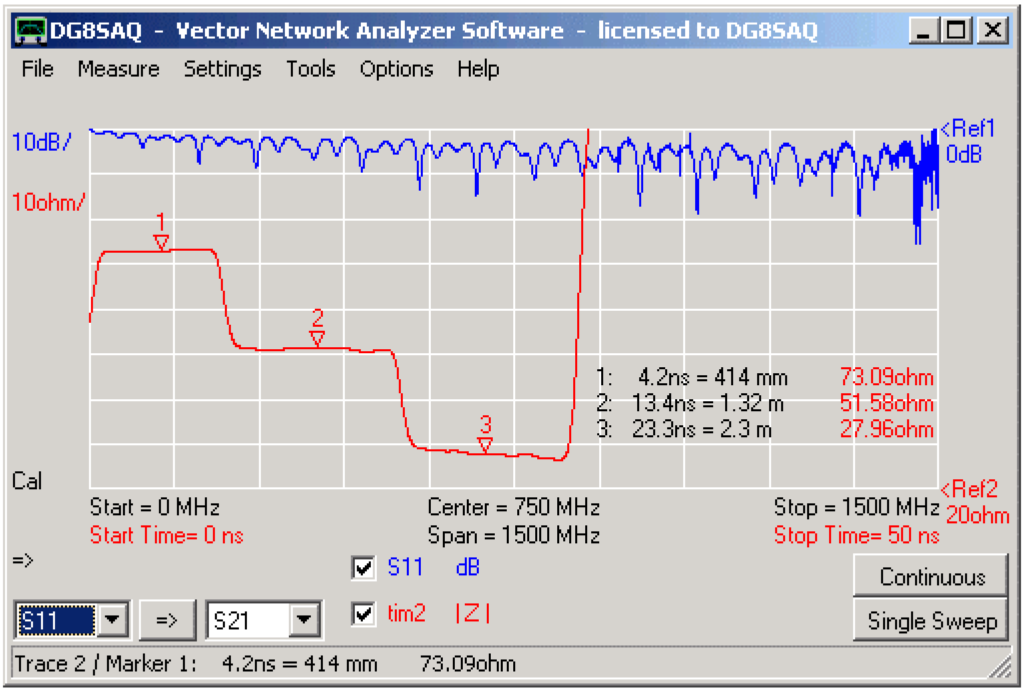
This is demonstrated in the following with a chain of coax cables of different impedances. The DUT below consists of a piece of 75 Ohms coax connected to the VNWA TX port followed by a 50 Ohms coax followed by a 25 Ohms coax the far end of which was left open.

First, calibrate your VNWA in the maximum available frequency range (0-1500 MHz) for maximum spatial resolution with a reasonablly high number of points (2000 points used in this example) to separarate all temporal features. Next, measure the reflection coefficient (=S11) of your DUT, so the result is visible in the S11 data space.

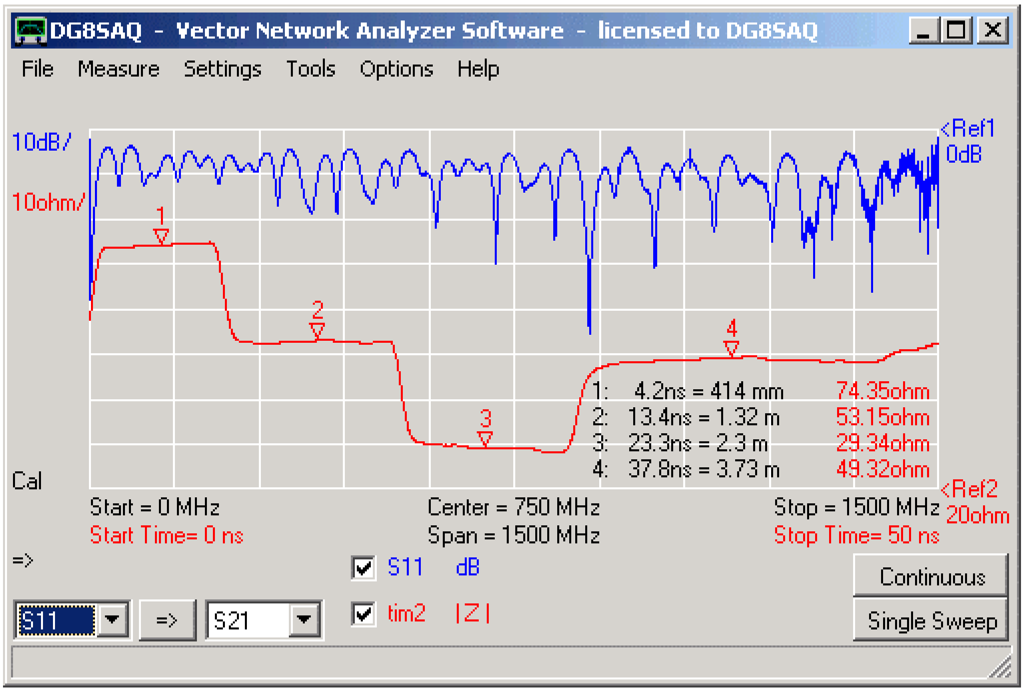
In order to see the step response, select **trace type "Time"** in the main window and select **"Step DFT"** from the "Time Domain Settings" menu. For displaying smooth steps without artificial overshoot, a **Blackman window function** was chosen.



If you select to display the time data in terms of impedances (|Z| in below example), you can immediately read off the varying cable impedance at varying positions:



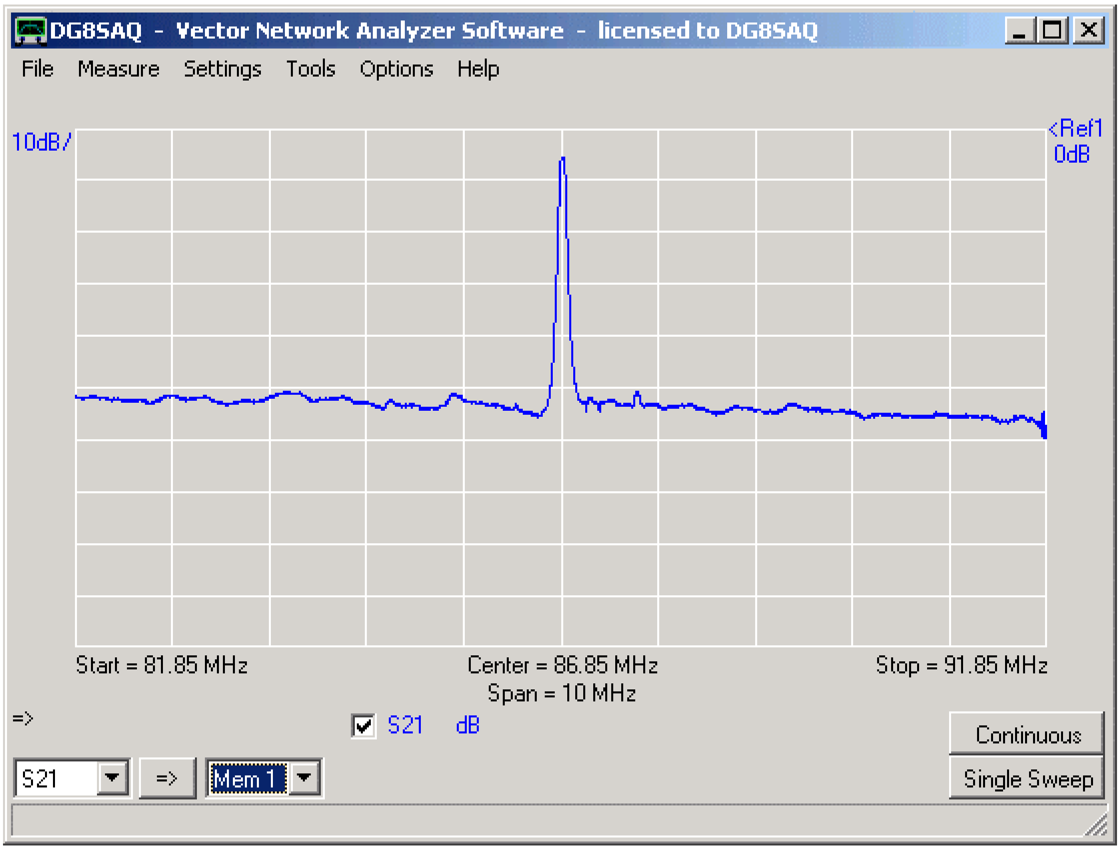
Note, that the impedance jumps from 75 Ohms to 50 Ohms to 25 Ohms are clearly visible. Also, the jump at the open end to (almost) infinite impedance is visible.
If the far cable end is terminated with 50 Ohms instead of leaving it open, the jump to (almost) infinity vanishes and the terminating impedance of 50 Ohms becomes visible.



Bear in mind, that the impedance values determined this way will be influenced by multiple reflections and also by cable losses, e.g. no matter what you connect to the far end of a 75 Ohms transmission line with e.g. 100dB loss, you will always see 75 Ohms impedance at all delays.

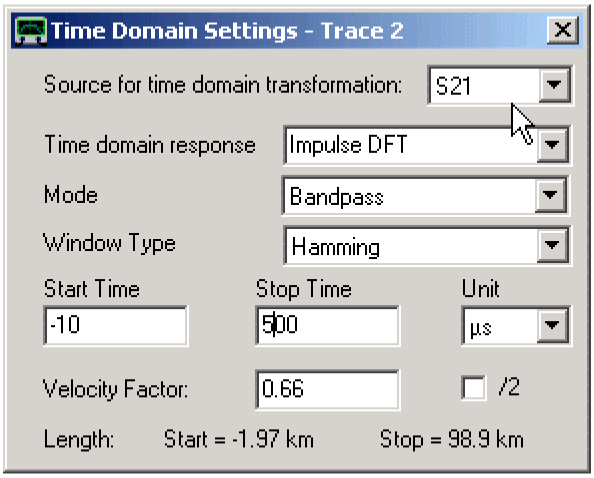
Time domain gating is used to separate responses depending on their arrival times, e.g. to separate the slow mechanical response of a crystal filter from the fast electromagnetic feedthrough of the test board.The following example shows how to use this feature.

1. Measure or import a trace to any memory space.



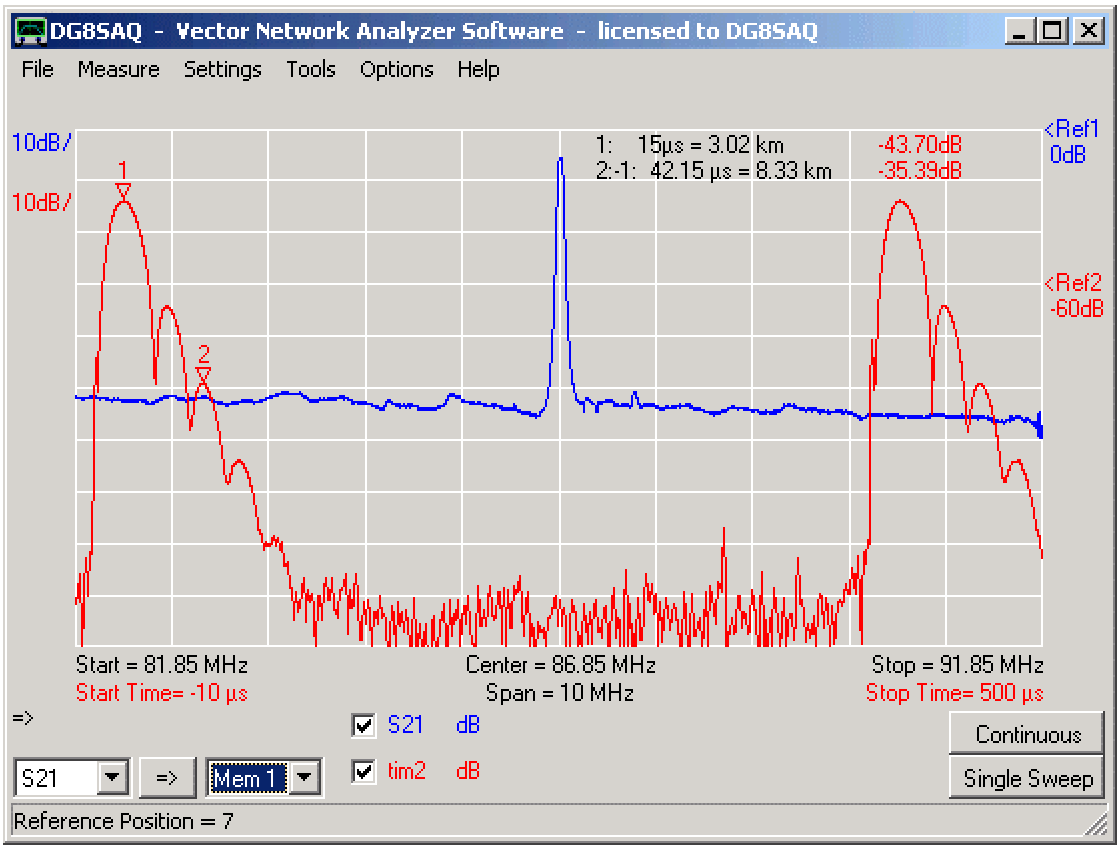
As an example you see a wideband measurement of a narrowband SAW filter as DUT with artificially increased electrical feedthrough from input to output.

2. To transform and view this data into time domain, select the main menu item **"Settings"-"Time Domain Settings"-"Trace 2"**::



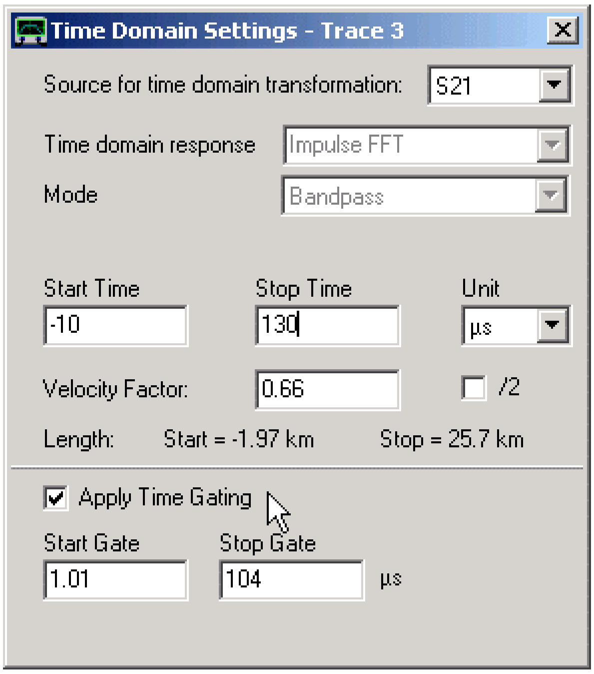
Here you can specify the signal source which is to be transformed to the time domain (here S21) and also the type and parameters of the transformation to be used. Start and stop times for the time domain diagram can be entered.

3. Close the window when done and do select on the main window a second trace with trace type **"Time Domain"**. You can add **time markers** the same way as adding frequency markers. Another way to change the start and stop times is to doubleclick the red **time labels** below the traces of the main window.

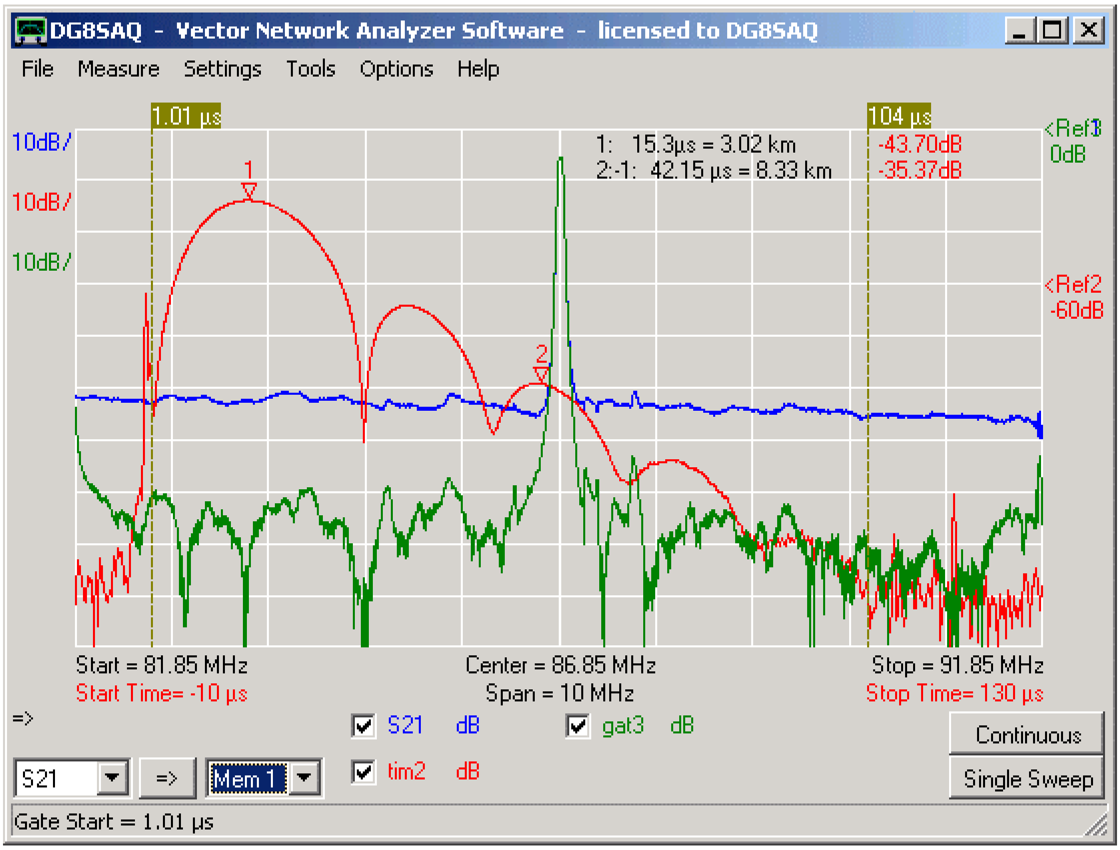


You do now see the inverse Fourier transform of your frequency data (red trace), which is by nature of the discrete Fourier Transform periodic. The part of it starting at time zero is the envelope of the DUT's **impulse response**, which you could also measure by applying a short impulse to the DUT's input. Note, that due to the DUT's high Q-value, it delays and streches an incoming impulse considerably (filter ringing).

4. Add a third trace to the main window with trace type **"Frequency-Domain Gated"** or **Gated** and then select the main menu item **"Settings"-"Time Domain Settings"-"Trace 3"**::



5. Once you have activated **time gating** you will see two dashed vertical lines on the main window with attached labels indicating the time gate window edges (start gate and stop gate). You can move these with the mouse. Now zoom into the time domain so you see only the time range of interest:



6. Move the vertical dashed **gate lines** (or attached labels) with the mouse and observe how the green trace changes. If you look at the red time domain response, you see a sharp peak near time zero. This is the very fast electrical feedthrough signal travelling at speed of light, i.e. the fraction of the input impulse that bypasses the SAW filter due to electrical crosstalk and remains unchanged. The round features at later times is the SAW filter impulse response (=filter ringing), which travels by the nature of the Surface Acoustic Wave (SAW) with the speed of sound on the quartz crystal chip (about 1000m/sec). Due to this speed difference it is possible to zero out the electrical feedthrough in the time domain (=gating) and visualize the transfer function of the filter without electrical feedthrough in the frequency domain (green curve). The green curve shows stopband features which are hidden under the electrical crosstalk in the blue trace.

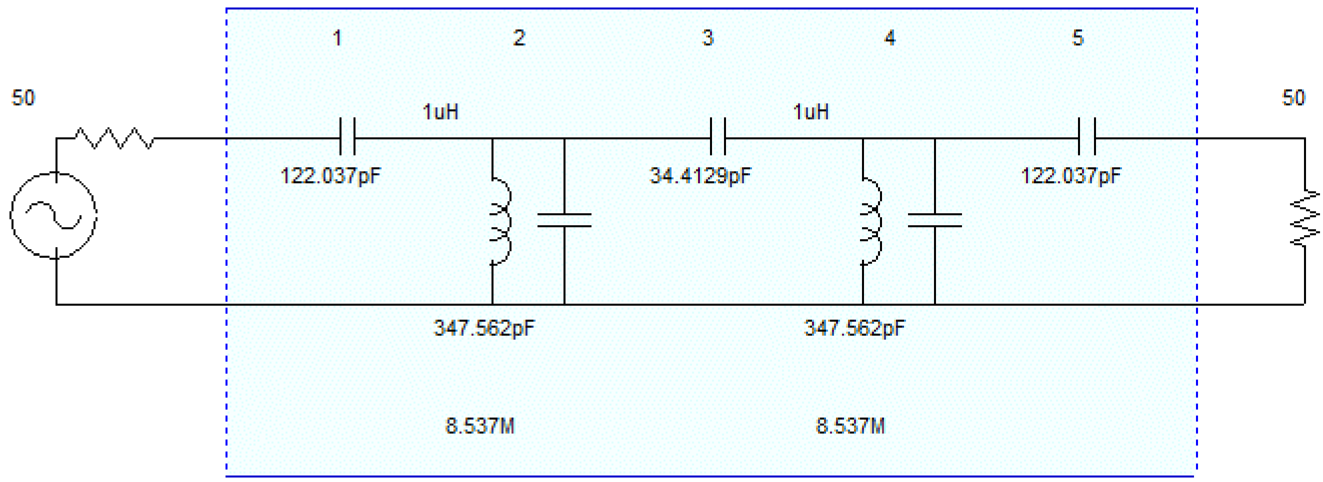
▸▸▸ **Hint:** You can change the color of the start gate and stop gate lines and labels by right-clicking them.

Summary:
This technique called "time domain gating" zeroes out the fractions of the DUT's time domain response outside the vertical gate boundary lines. Transforming the gated impulse response back to the frequency domain might reveal features previously hidden by e.g. electrical feedthrough.

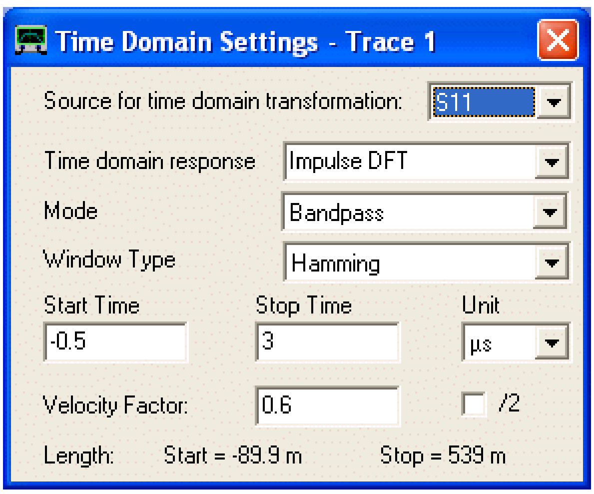
This tutorial was provided by Roderick Wall, VK3YC. Thank you very much, Roderick!

Using Time Domain to tune variable coupling BPFs

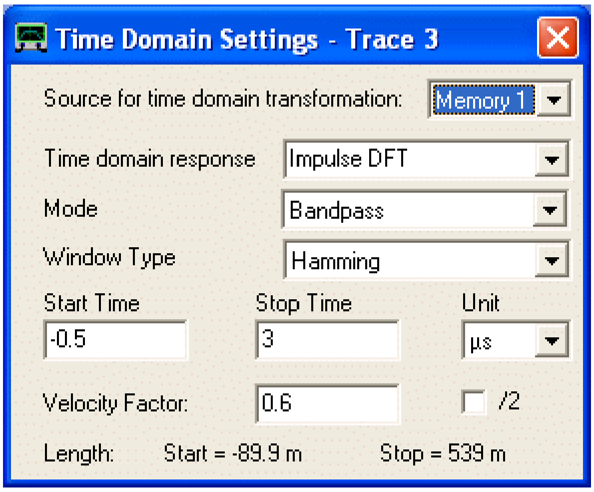
A Template is required to tune variable coupling filters. The Template can be generated from a Gold Standard Engineering filter or simulated in software. This example uses Jim Tonne's Elsie Filter design software to design a BPF and to generate a s2p Touchstone s-parameter file. The s-parameter file is imported into a memory location in VNWA to be used as a Template to tune the filter to. The example 40 meter variable coupling BPF circuit is shown below. Trim capacitors were used for the coupling capacitors. Adjustable inductors were used for the inductors.



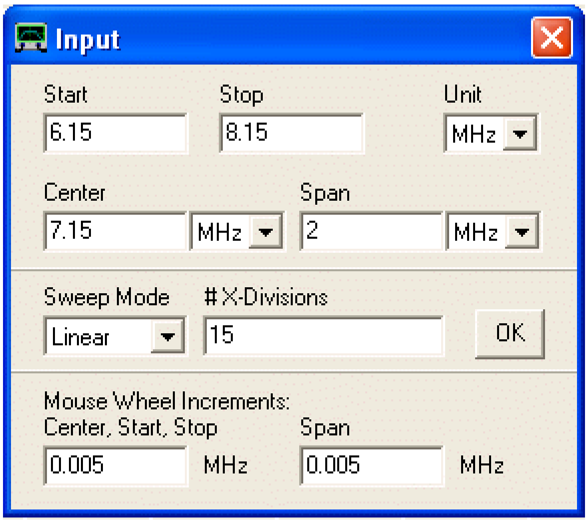
Use Elsie to design the filter. Set the centre frequency to 7.15MHz, frequency span to 2MHz and sweep steps to 500 (also refer to VNWA settings below). Generate a s2p s-parameter Template file. Import the the s2p Template file into VNWA and store S11 into Memory 1. Store S21 into Memory 2. Under "Settings" select "Time Domain" and then select "Trace 1". Set the Time Domain settings as shown below.



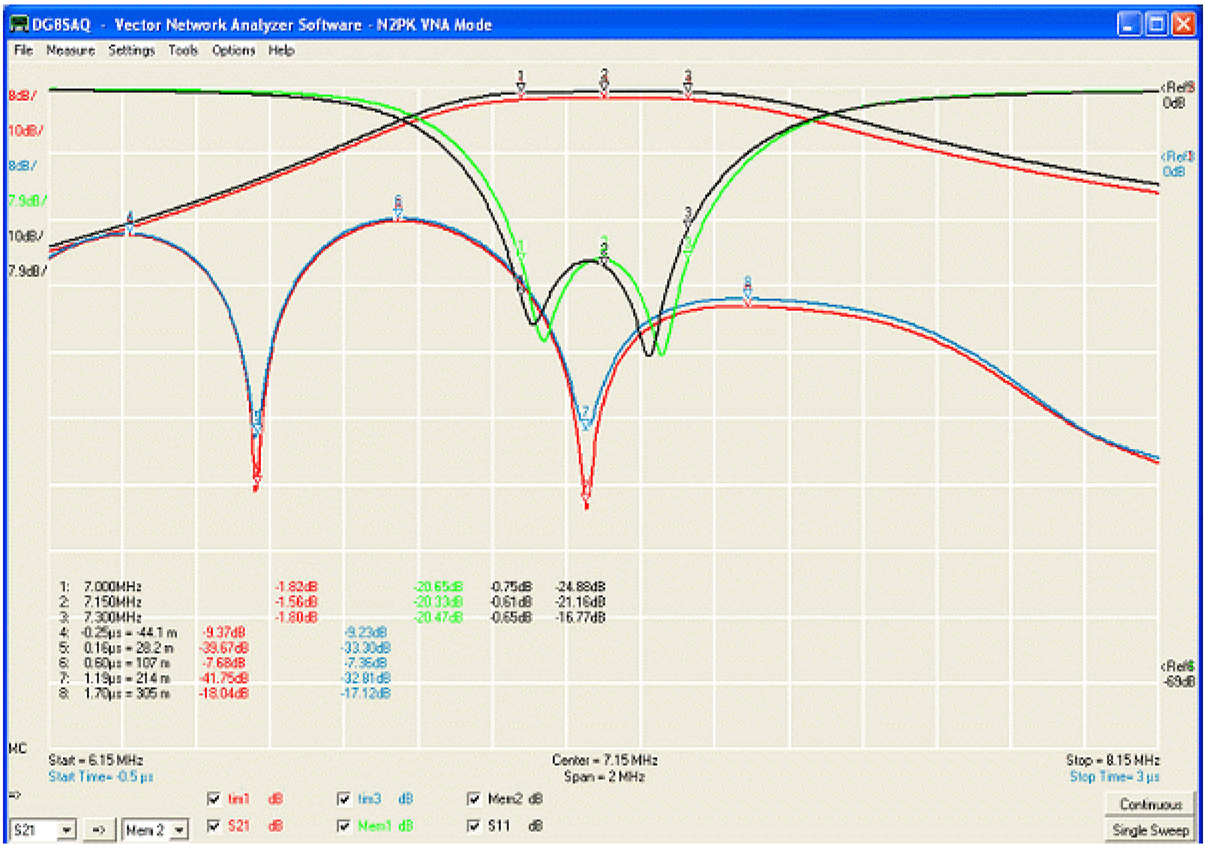
Under "Settings" select "Time Domain" and then select "Trace 3". Set the Time Domain settings as shown below.



Set the center frequency to the resonant frequency that the resonators need to be tuned to. If you don't then you will be tuning the BPF to the wrong frequency. Set the Span to two to five times the bandwidth of the BPF. Set the Mouse Wheel Increments to 5kHz.



Set the number of scan steps to 500. As shown below, set trace 2 to S21 dB, trace 4 to Mem1 dB, trace 5 to Mem2 dB and trace 6 to S11 dB.



Blue Time Domain trace = Elsie Template. Red Time Domain trace = DUT BPF.
Green Return Loss trace = Elsie Template. Black Return Loss trace = DUT BPF.
Black Insertion Loss = Elsie Template. Red Insertion Loss = DUT BPF.

Click "Single Sweep" to scan the DUT 40 meter BPF. Adjust trace scaling to suitable sizes. Your Time Domain trace won't have the deep dips shown above because the BPF has been adjusted using Time Domain. The two Time Domain dips are the return loss from each resonator. The three humps are the three coupling capacitors. Refer to Agilent's application notes AN 1287-8 and AN 1287-10 for the procedure to adjust the filter. Click Continuous scan to adjust the BPF.

Before assembling the BPF you may want to adjust the components to the values shown in the circuit, this will make it easier to adjust the BPF.

The Mouse wheel can be used to adjust the center frequency. Hold the cursor over the center frequency and turn the wheel. Adjusting the center frequency allows you to determine the frequency the resonators are tuned to. Adjust the center frequency for the deepest dip and the center frequency is the resonant frequency. Using the Mouse Frequency wheel makes it easier to adjust the resonators.

Roderick Wall, vk3yc.

The **trace type custom** allows to simulate traces or to manipulate trace data in order to display functions of trace data.

VNWA.exe contains a **compiler**, which can generate fast code to calculate **user defined mathematical complex algebra expressions**. This code can be used for real time data manipulation during the sweep or to generate simulation data.

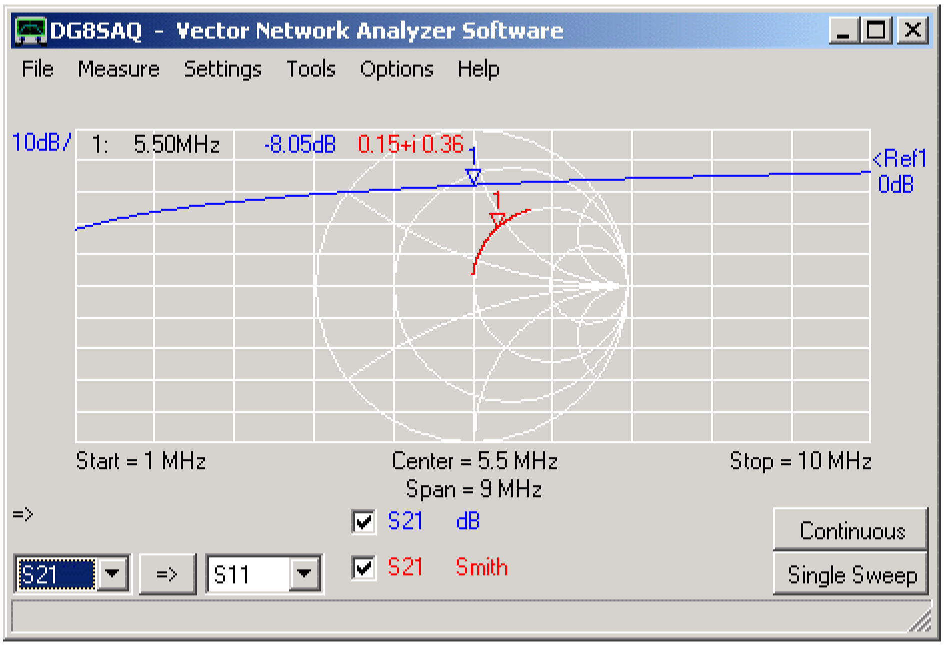
▸▸▸ **Note:** This parser / compiler (own development) is different from that used in complex calculator, thus functionalities might differ slightly.

Example 1: Determining an impedance from a transmission measurement
Example 2: Unitarity relation
Example 3: Subexpressions
Syntax Reference

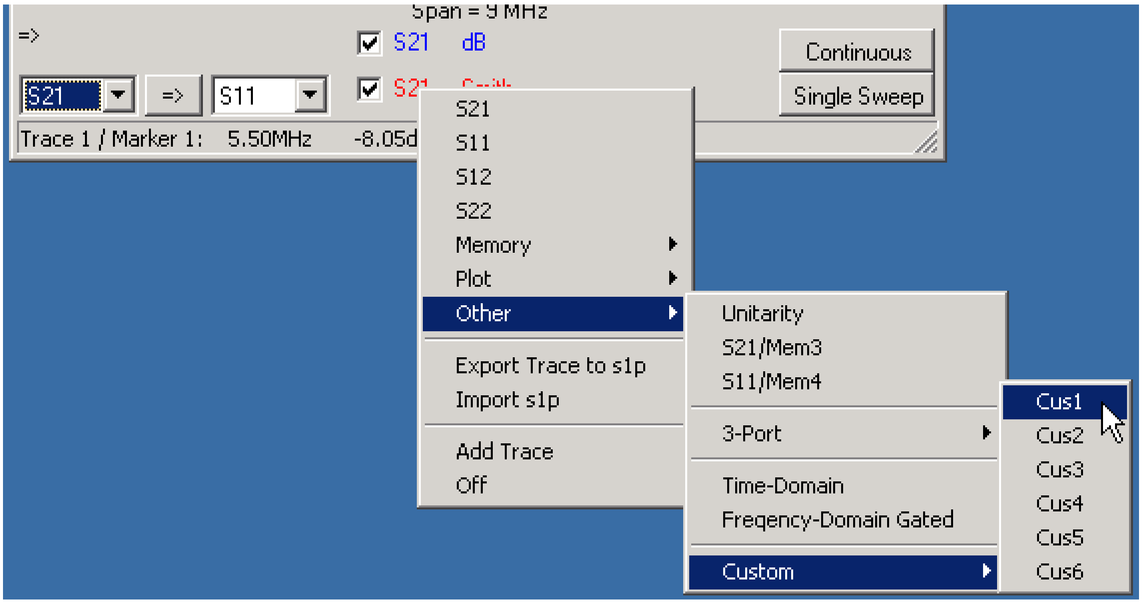
Example 1: Determining an impedance from a transmission measurement

Is it possible to accurately determine an impedance from a transmission measurement with only a thru calibration? The answer is yes and a custom trace greatly simplifies this task.

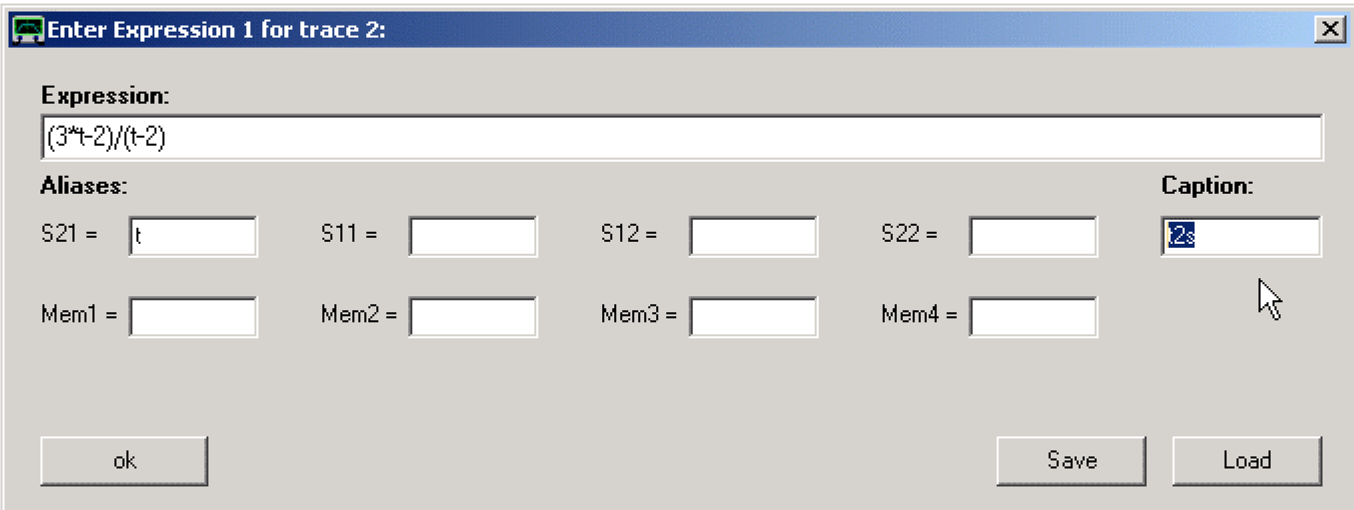
Below is a transmission measurement (S21) of a 120pF capacitor connected between the hot pins of the VNWA TX and RX port after a simple thru calibration (note, that both traces show S21):



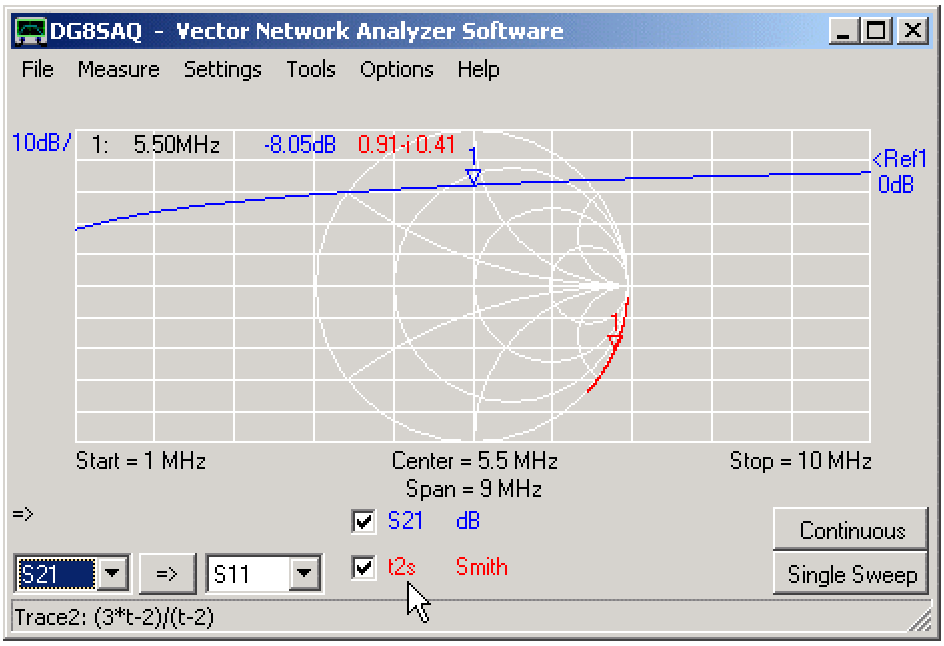
Clearly, the higher the frequency the lower the insertion loss due to the capacitor's decreasing impedance with increasing frequency.
Next, we want to calculate the one port reflection coefficient (S11) of the capacitor from our transmission measurement with the aid of a custom trace. Therefore we **right-click the S21** label and select trace type **"Other-Custom-Cust1"**:



The custom trace editor window opens indicating in its header that we are editing expression 1 (=cus1) which is to be displayed in trace 2:

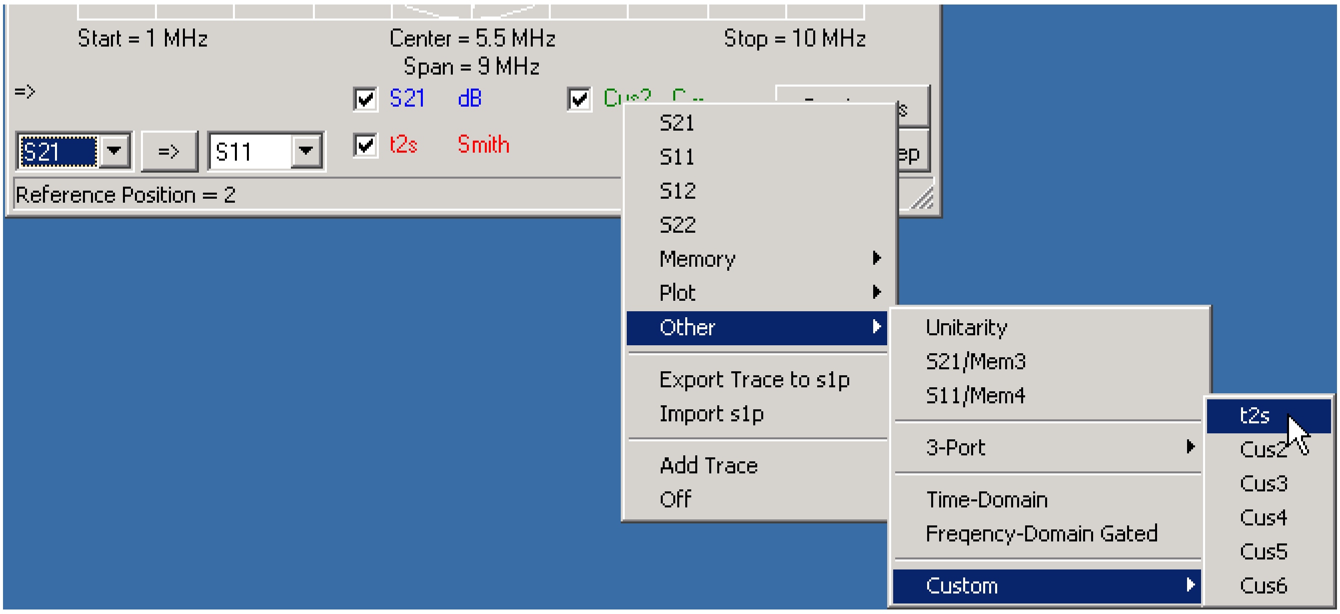


For simplicity, we assign to S21 the shorter **alias name t** (for transmission) and enter the expression (3*t-2)/(t-2), which transforms a thru-measurement result into a reflect measurement result. Also, we give this expression the descriptive name **t2s** (for transmission to scattering parameter, default name is cus1), see highlighted **caption field** above the mouse pointer. Press ok and observe the updated main window:

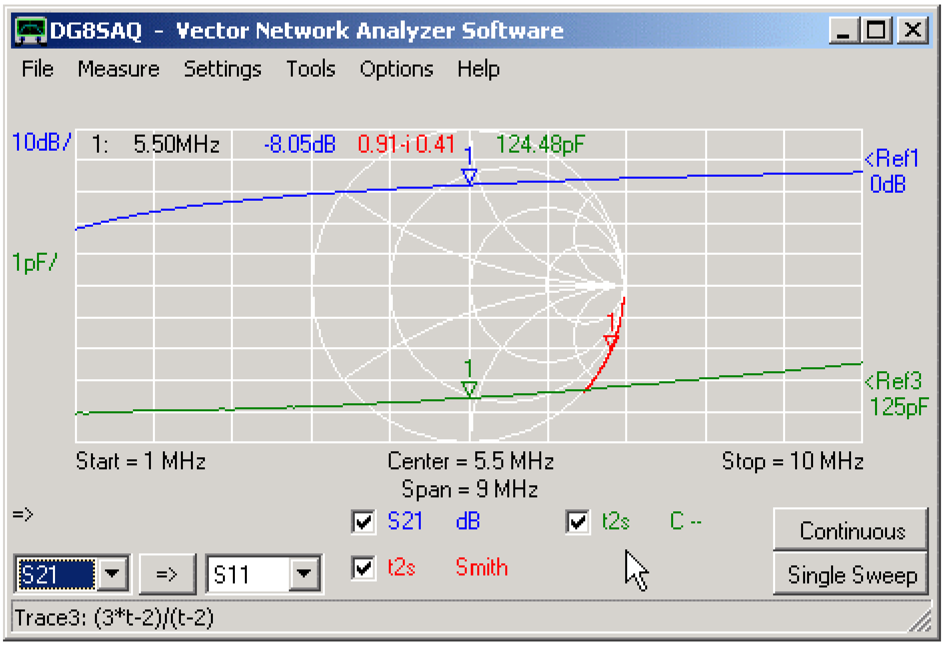


Red trace 2 is now labeled by the assigned name t2s and the data apparently shows the reflection coefficient of a quite ideal capacitor (curve running along the lower edge of the Smith chart).

We now want to determine the capacitance of the capacitor, therefore, we want to see the very same t2s data in a third trace in terms of Picofarads. We add a third trace and select again trace type **"Other-Custom"**:

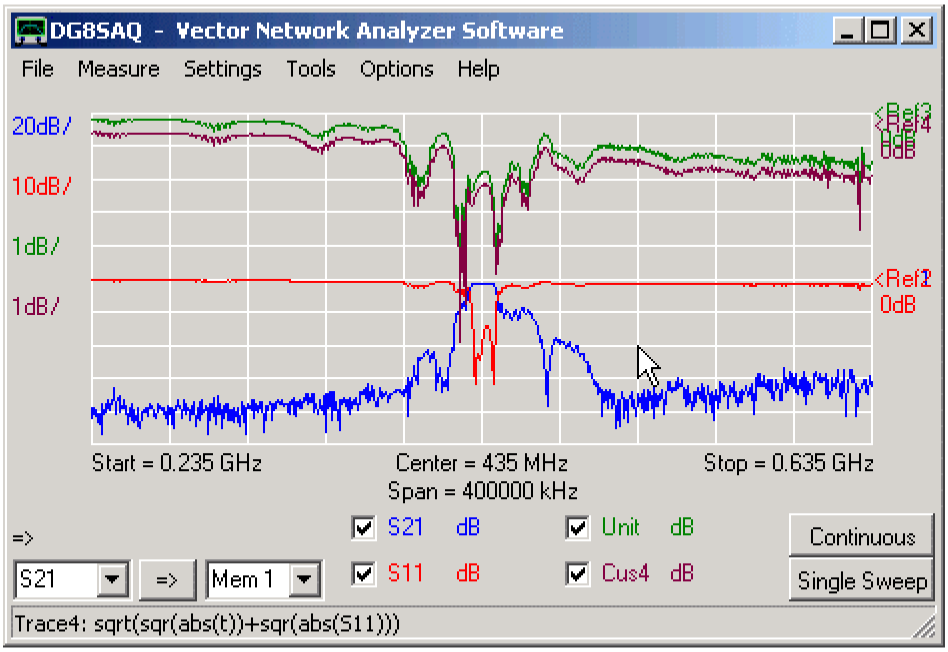


Observe, that the custom drop down menu now shows as first entry **t2s** instead of cus1 because we have assigned this name to expression 1. We select t2s. The custom trace editor opens again, but we do not need to edit anything as we have already entered the required expression before. Therefore we simply close the custom trace editor and select display type **"C--"** for trace 3:



Indeed, our capacitor shows a capacitance of 124pF, which is very close to the nominal capacitance of 120pF printed on it. Note, that the rising capacitance is due to the inductance of the long capacitor leads.

Example 2: Unitarity relation

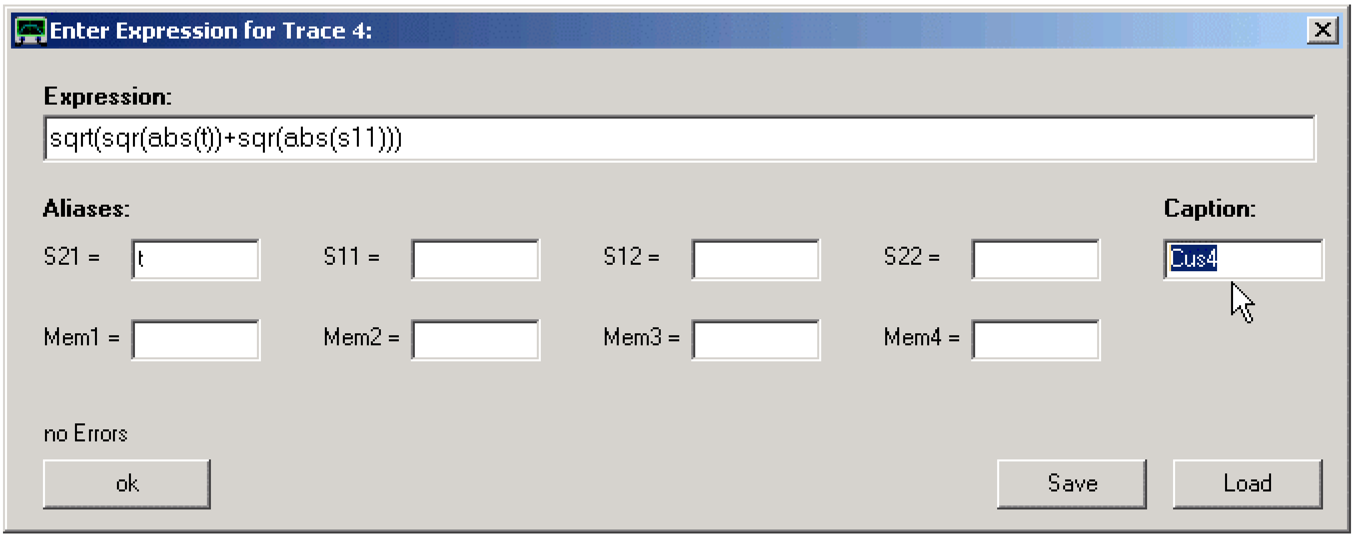


The above example shows measured S11 and S21 data of a SAW filter (red and blue traces). The green trace shows the so called unitarity condition:

$$Unitarity = \sqrt{|S_{11}|^2 + |S_{21}|^2}$$

which, when shown in dB, is the power neither reflected nor transmitted, thus the power dissipated inside the DUT.

The brown "Cus4" trace (= custom trace 4) shows exactly the same as the "Unit" trace, but is user defined.
Double-clicking the Cus4 label will open the custom trace editor window:

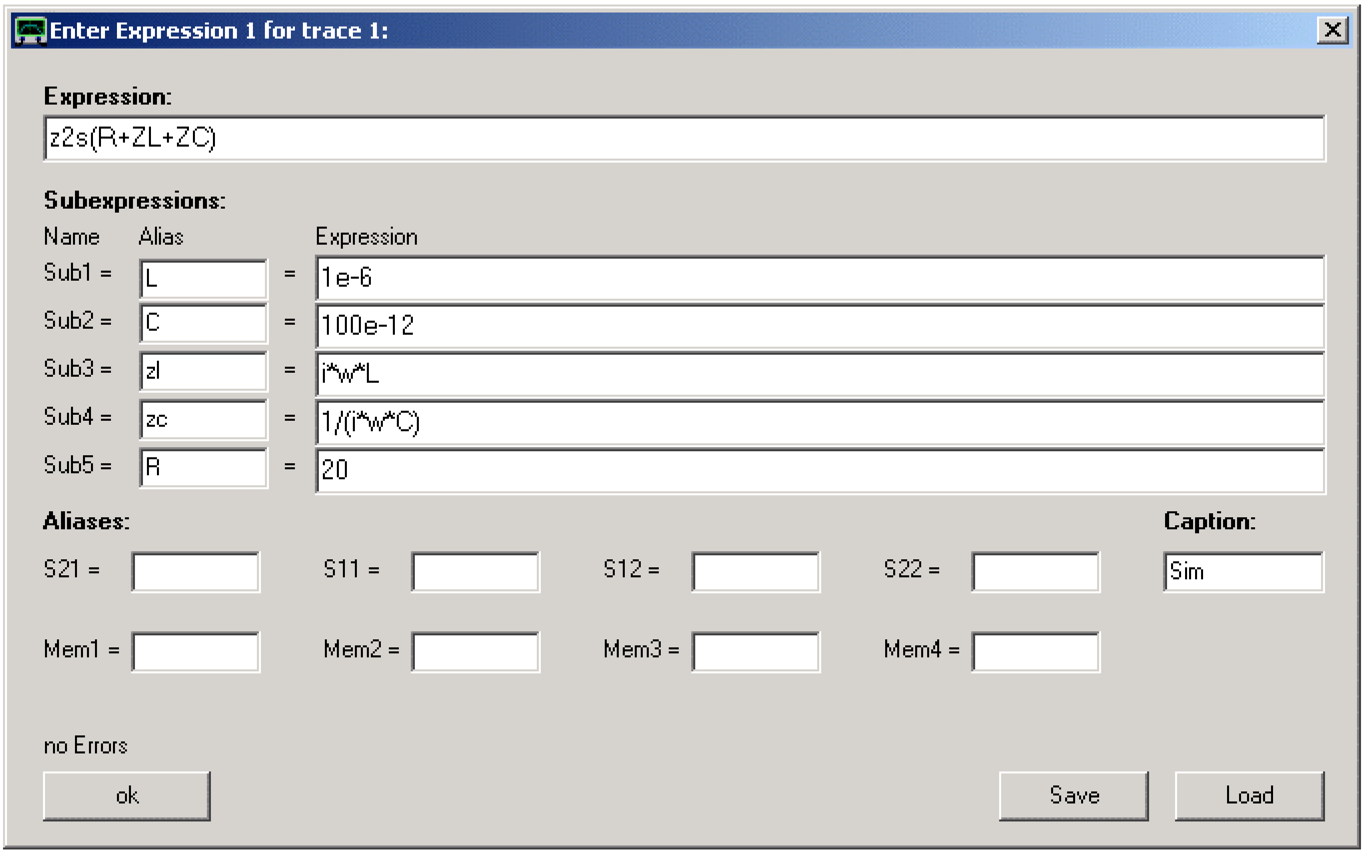


Here, the mathematical **expression** to be displayed can be entered. It is evaluated using complex calculus, thus complex S-parameters can be manipulated in a proper way. In the above example, the unitarity formula has been entered.

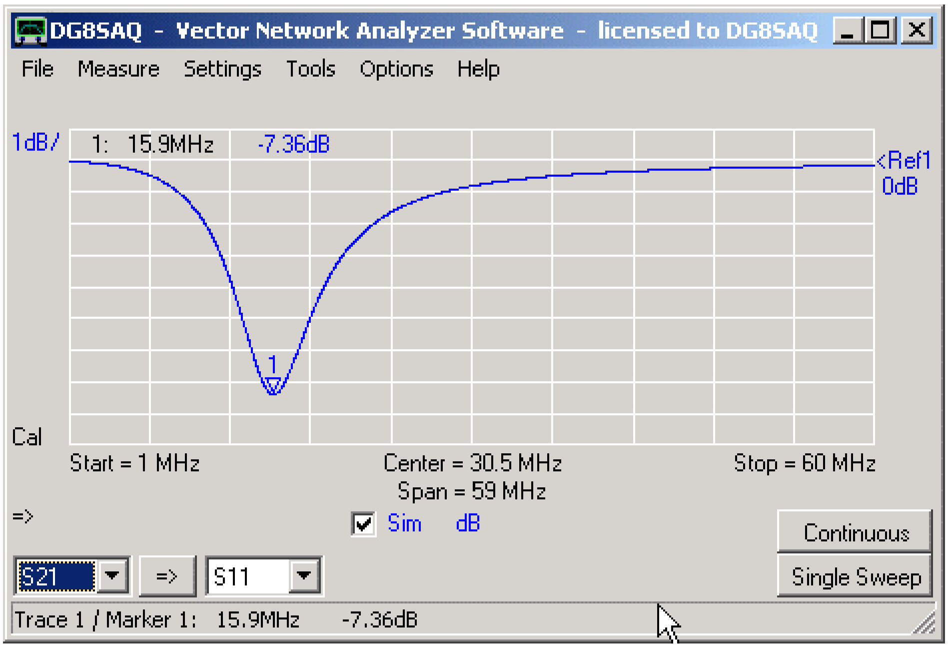
▸▸▸ **Note: Alias variables** can be defined to abbreviate e.g. the rather lengthy variable name S21 by the variable t.
▸▸▸ **Hint:** Typing return upon completion of the expression will invoke a **syntax check** without closing the window.
▸▸▸ **Hint:** Pressing the **"ok button"** will also invoke a syntax check and close the window only if the syntax is correct.
▸▸▸ **Hint:** You can **save and load expressions** to or from a Derive *.mth file.
▸▸▸ **Hint: Expressions will automatically be saved** in the VNWA ini-file upon program termination for reuse after program restart.
▸▸▸ **Hint:** By changing the **Caption** field in above window (see mouse arrow), you can modify the trace identifier label K on the main VNWA window showing up below the grid.

Example 3: Subexpressions

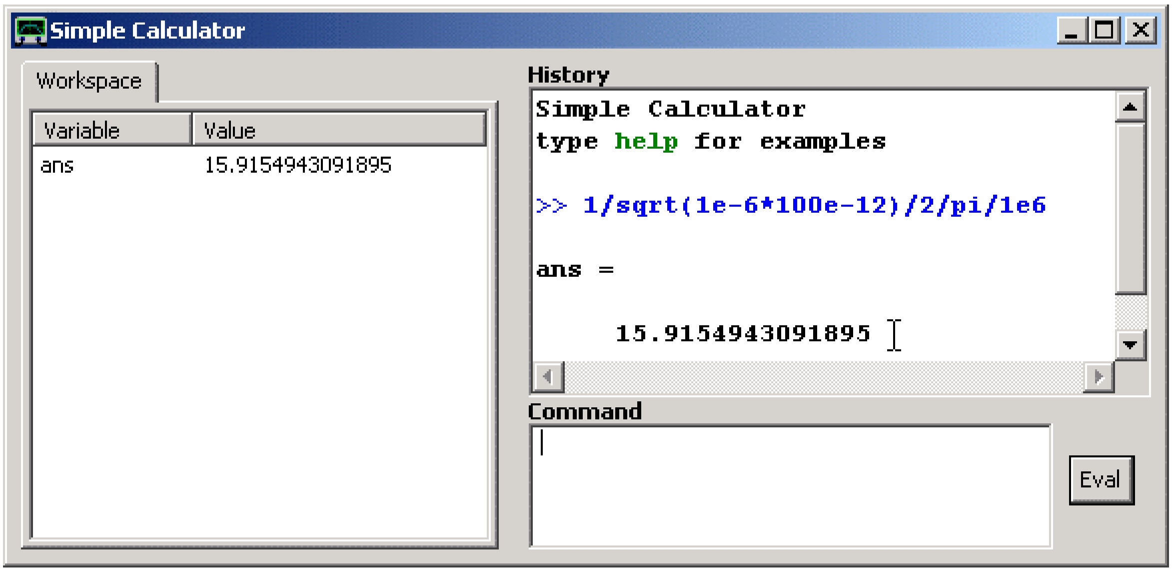
When the custom trace editor window height is increased, up to 6 input fields for **subexpressions** are being revealed:



Subexpressions can be used if a parameter or term is to be reused several times in the same custom expression or simply to improve readibility. In the above example, the reflection coefficient of a series resonat circuit is being calculated using subexpressions. In subexpression 1 an inductance value of 1 microhenry is being defined. Subexpression 2 defines a capacitance value of 100pF. Subexpressions 3 and 4 define the impedances of the inductor and capacitor respectively by making use of subexpressions 1 and 2. Subexpression 5 defines a resistor value of 20 Ohms.
The main expression sums up the impedances defined in subexpressions 3, 4 and 5 and converts it to a reflection coefficient that can be displayed with the standard S11 display modes on the main window:

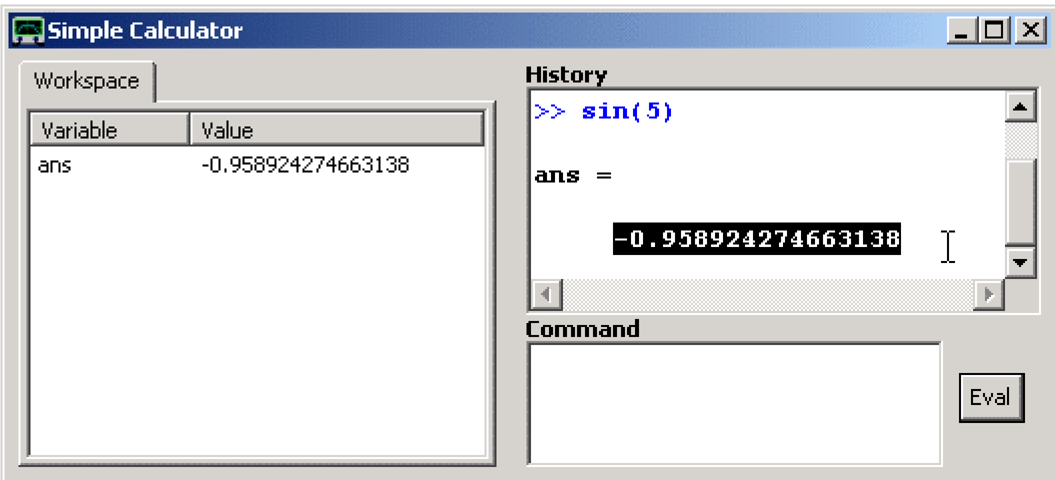


Apparently, we have simulated a series resonant circuit with a resonance frequency of 15,9 MHz. We can use the Complex Calculator tool to check if the resonance is where it would be expected:

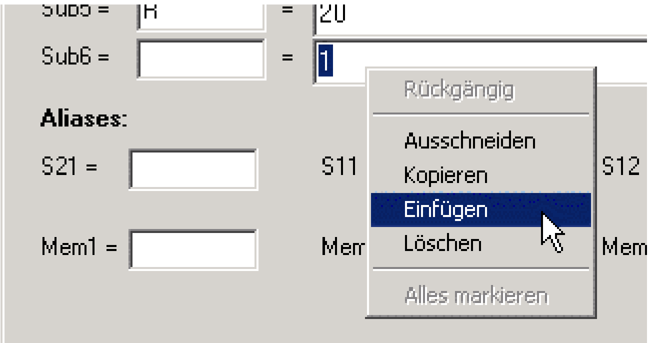


Indeed, the resonance is to be expected at 15.9 MHz.

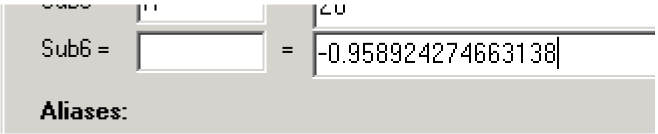
▶▶▶ **Note:** All 6 custom traces share the same subexpressions.
▶▶▶ **Note:** Subexpressions can be accessed either by the names **Sub1**...**Sub6** or by the user defined aliases.
▶▶▶ **Note:** Subexpressions may make use of previous subexpressions but not of subsequent subexpressions. Circular references are forbidden.
▶▶▶ **Note:** Custom traces are only being updated when the return key is pressed after editing any of the subexpressions or expression, or if the ok button is pressed.
▶▶▶ **Note:** You can **transfer precalculated values** from the Complex Calculator tool to one of the subexpressions by marking the result in complex calculator with the mouse ...



..., then pressing **ctrl c** to copy the marked data into the clipboard, then right-clicking into the field where the value is to be transfered to...



... and finally selecting **paste**:



Available Syntax

Available special constants

pi = 3.14...
e = Eulers number 2.718...
i = complex unit (mathematician's choice)
j = i = complex unit (engineer's choice)
clock = DDS input clock [Hz]

Available operators

+ = complex addition
- = complex subtraction
***** = complex multiplication
/ = complex division
^ = complex power *Hint:* exp(x) is more CPU efficient than e^x

Available data functions:

f = frequency
w = 2*pi*frequency
s = complex frequency = i*w
S21 = measured data S21
S11 = measured data S11
S12 = measured data S12
S22 = measured data S22
Mem1 = data space Mem1 data
Mem2 = data space Mem2 data
Mem3 = data space Mem3 data
Mem4 = data space Mem4 data
P1 = data space Plot1 data
P2 = data space Plot2 data
P3 = data space Plot3 data
P4 = data space Plot4 data
s_11...s_33 = 3-port measured data
gated = gated response
a_ = reflect calibration coefficient *a*
b_ = reflect calibration coefficient *b*
c_ = reflect calibration coefficient *c*
M21 = raw, uncorrected measured data for S21

M11 = raw, uncorrected measured data for S11
M12 = raw, uncorrected measured data for S12
M22 = raw, uncorrected measured data for S22
SS = VNWA source reflection coefficient as calculated from calibration data
SL = VNWA load reflection coefficient as calculated from calibration data
n = data point number
delay_s21 = group delay of S21 = -d Arg(S21(w))/d w
delay_s11 = group delay of S11 = -d Arg(S11(w))/d w
delay_s12 = group delay of S12 = -d Arg(S12(w))/d w
delay_s22 = group delay of S22 = -d Arg(S22(w))/d w
delay_mem1 = group delay of Mem1 = -d Arg(Mem1(w))/d w
delay_mem2 = group delay of Mem2 = -d Arg(Mem2(w))/d w
delay_mem3 = group delay of Mem3 = -d Arg(Mem3(w))/d w
delay_mem4 = group delay of Mem4 = -d Arg(Mem4(w))/d w
Sub1...Sub6 = subexpression 1...6

Available complex mathematical functions:

EXP = complex exponential function
IM *example:* im(x+j*y)=j*y
IMAG =imaginary part (standard definition): imag(x+j*y)=y
RE =real part
SIN =complex sine
COS =complex cosine
TAN =complex tangent
ATAN =complex inverse tangent
ABS =complex absolute value or magnitude
ABSSQR =complex square of the absolute value
SQR =complex square
SQRT =complex squareroot
CONJ =complex conjugate
DEG =conversion radiants => degrees: deg(pi)=180
RAD =conversion degrees => radiants: rad(180)=pi
ARG =argument of complex number in radiants: arg(exp(j*x))=x
LN =complex natural logarithm
LG =complex logarithm of base 10
HEAVISIDE=real heaviside function, imaginary part of argument is ignored

Available special functions of S-parameters:

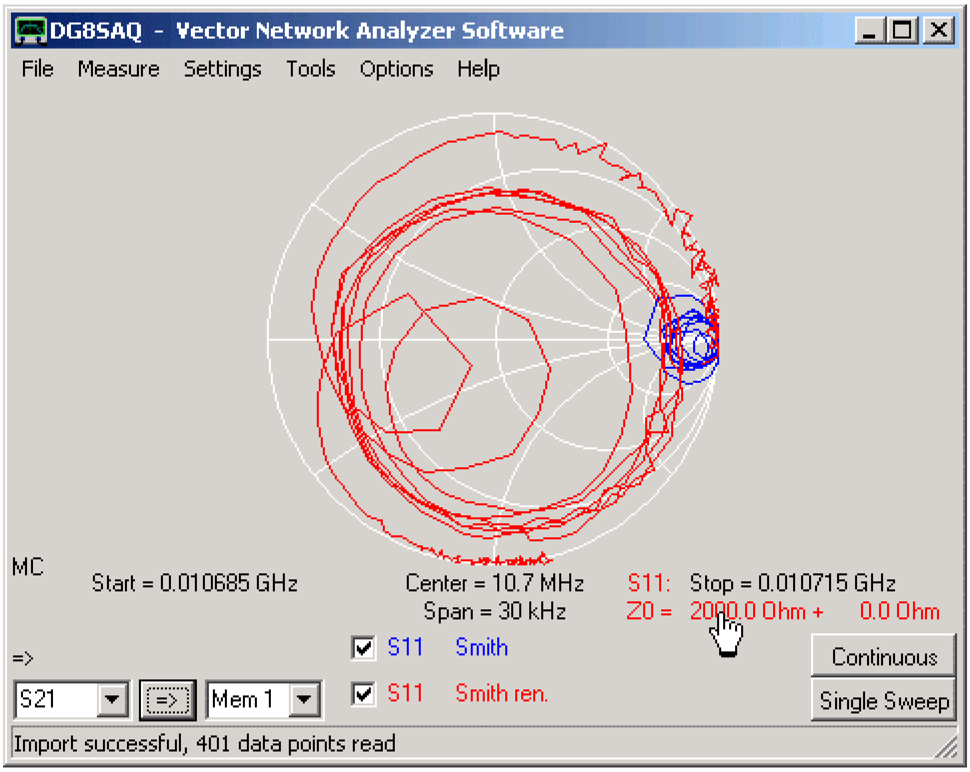
DB =decibels: db(x)=20*lg(abs(x))
Y = S2Y =complex admittance in 1/Ohms Y(s)=(1-s)/((1+s)*50)
Z = S2Z =complex impedance in Ohms Z(s)=1/Y(s)
Y2S =conversion from complex admittance Y(s) to reflection coefficient s, e.g. s=Y2S(Y(s))
Z2S =conversion from complex impedance Z(s) to reflection coefficient s, e.g. s=Z2S(Z(s))
CP =parallel capacitive part of complex admittance in F (as function of reflection coefficient S)
LP =parallel inductive part of complex admittance in H (as function of reflection coefficient S)
RP =parallel resistive part of complex admittance in Ohms (as function of reflection coefficient S)
QC =Q-value of complex admittance
CS =serial capacitive part of complex impedance in F (as function of reflection coefficient S)
LS =serial inductive part of complex impedance in H (as function of reflection coefficient S)
RS =serial inductive part of complex impedance in Ohms (as function of reflection coefficient S)
QL =Q-value of complex impedance (identical with QC, but calculated from Z instead of Y)

▸▸▸ **Note:** The parser is **not case sensitive**!

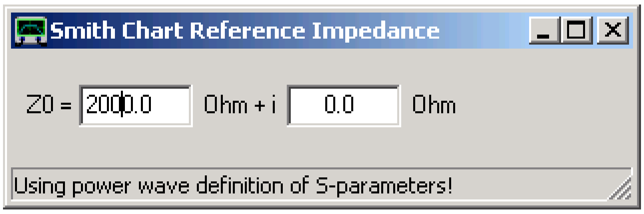
The VNWA software can recalculate **one port reflection coefficients** to an arbitrary **reference impedance** and display it as **renormalized Smith** data.

Most Smith charts display reflection data normalized to a reference impedance of 50 Ohms. If a different reference impedance is chosen, e.g. 75 Ohms, the reflection coefficients will change. For example, a 75 Ohms resistor will have a reflection coefficient of zero if normalized to 75 Ohms reference impedance or in other words in a 75 Ohms system.

An Example is shown below:



S11 is the input reflection coefficient of a crystal filter terminated with 50 Ohms on output. The blue curve near the open circuit point in the Smith chart shows, that the input impedance is much higher than the reference impedance of 50 Ohms. This changes, if a reference impedance of 2000 Ohms (see red **impedance label** under hand cursor). The magnitude reflection coefficient is still quite high in this case, as the power is still reflected at the mismatched 50 Ohms termination on output. To modify the reference impedance, double-click the red impedance label. The reference impedance window will open, where new values can be entered.



▸▸▸ **Hint:** You can also change the real part (imaginary part) of the reference impedance, by holding the mouse over the real part (imaginary part) of the red impedance label in the VNWA main window and turn the **mouse wheel**.

▸▸▸ **Note:** The definition of a reflection coefficient is only unique as long as the reference impedance is purely real. For a complex reference impedance I have chosen to implement the **power wave** formulation, which allows to interpret the reflection coefficient in terms of reflected power like of the real reference impedance case.

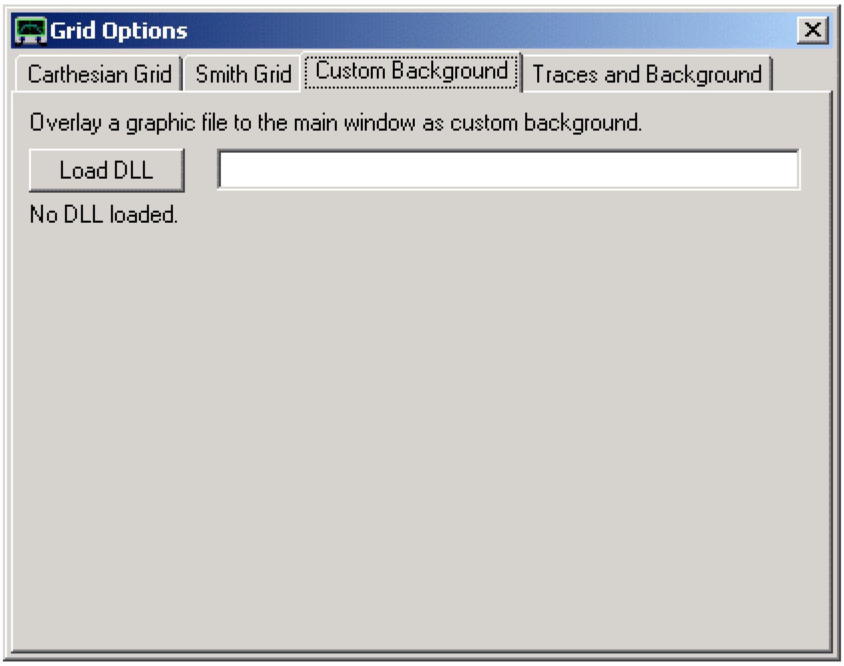
▸▸▸ **Hint:** To renormalize **two port S-parameters** to a new reference impedance, use the matching tool.

The custom background options allow to **load an image to appear as background on the VNWA main window**. Image file import is implemented in an optional external dynamic link library (dll) to avoid incompatibilities with older Windows versions (Windows98 and Windows2000 do not support GDI, which is used to read and display *.svg-files).
Two different dlls are provided for reading image files:

svg_dll.dll supports import of scalable vector graphics files (*.svg)
gfx_dll.dll supports import of most pixel graphics files like *.bmp, *.jpg, *.png... .

Moreover, the source code of a simple dll-example for reading *.bmp files is provided in order to document the programming interface.

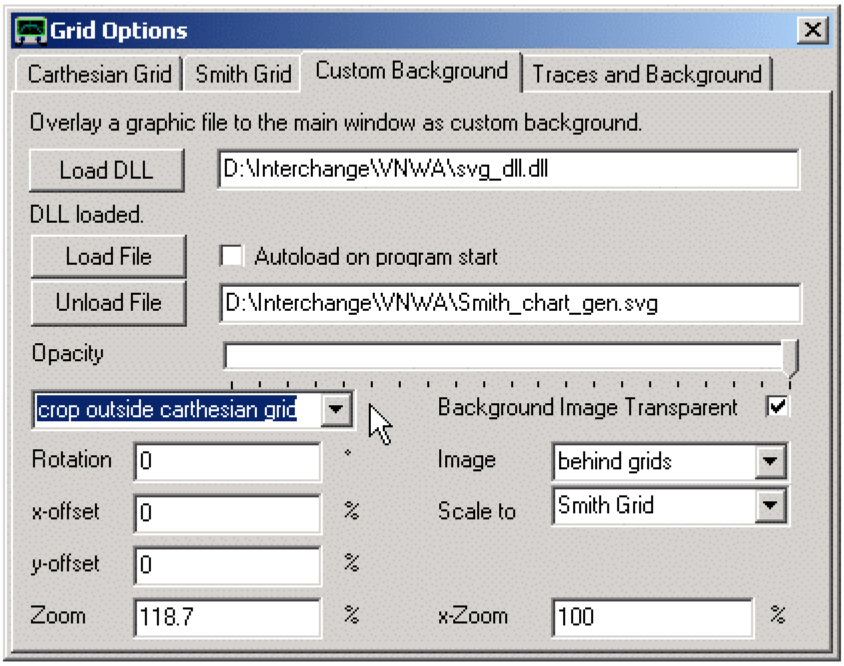
Before importing an image, an appropriate dll-file must be loaded by pressing the "Load DLL" button and selecting the dll-file:



Once the dll is loaded, the controls for loading and manipulating an image appear and an image file can be loaded.

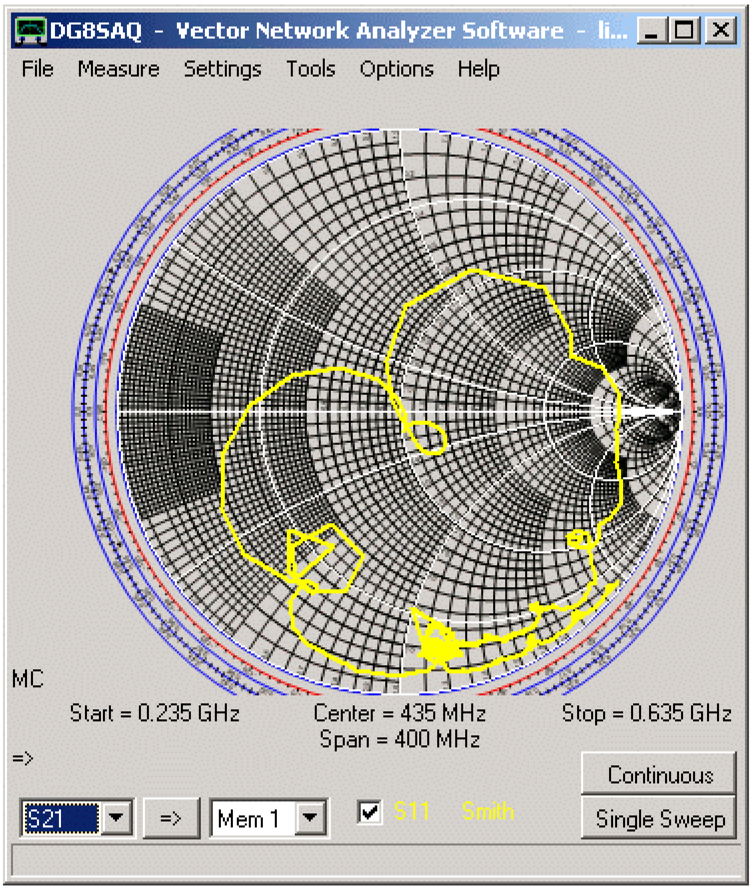
Example 1: Loading a detailled Smith chart on top of the standard one

Wikipedia offers a detailled Smith chart in svg-format:
http://upload.wikimedia.org/wikipedia/commons/7/7a/Smith_chart_gen.svg
The svg or scalable vector graphic format has the advantage of offering constant high display quality on all magnification scales. In order to load this file, svg_dll.dll must be loaded first. Then the image can be loaded by pressing the "Load File" button:

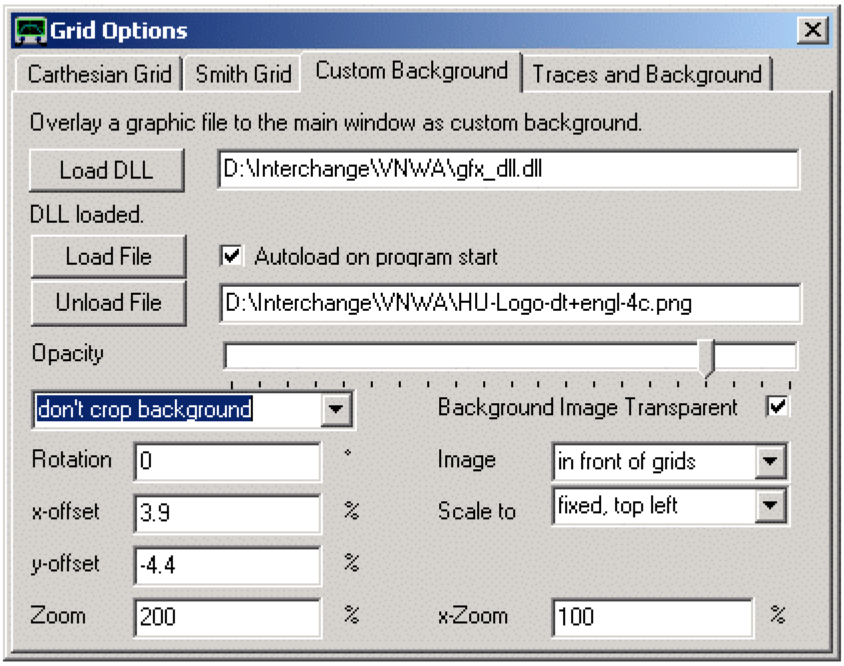


There are several possibilities to manipulate the displayed image:
Opacity ...alows to blend the image more and more into the background until it disappears
cropping ...means cutting off the image portions that are extending outside the selected grid
transparent ...if checked, all portions of the image having exactly the same color as the lower left image pixel become transparent, i.e. invisible. Particularly useful for black and white graphs to make the white portion transparent
The **image** can be placed in foront of or behind the grid lines
It can be **scaled to** one of the grids (i.e. if the grid is resized, the image will be resized accordingly or it can be displayed with fixed size in one of the main window corners
Moreover, the image can be **rotated**, resized by a **zoom**-factor, streched by an **x-zoom**-factor and repositioned by specifying **offsets**.
The image can be chosen to **autoload** upon program start.

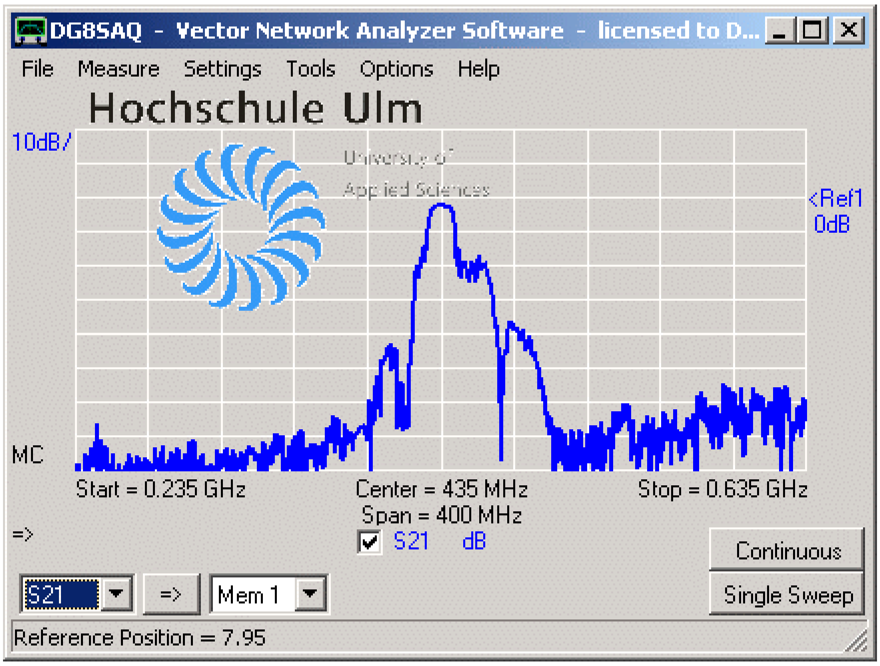
With above settings, the image displays like this:



Example 2: Loading a company logo

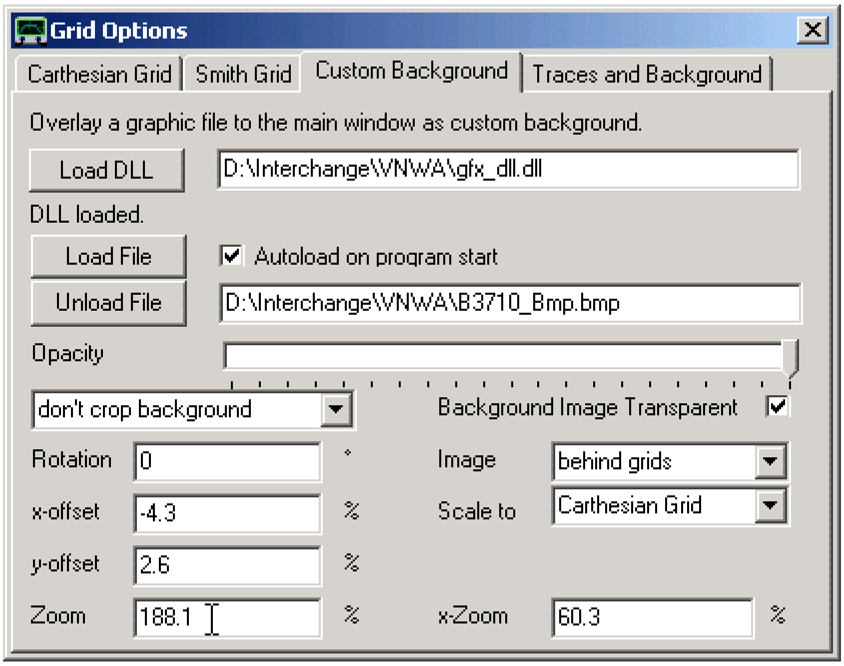


Here a logo in *.png format has been loaded using the gfx_dll.dll and the image has been fixed to the upper left corner of the main window:

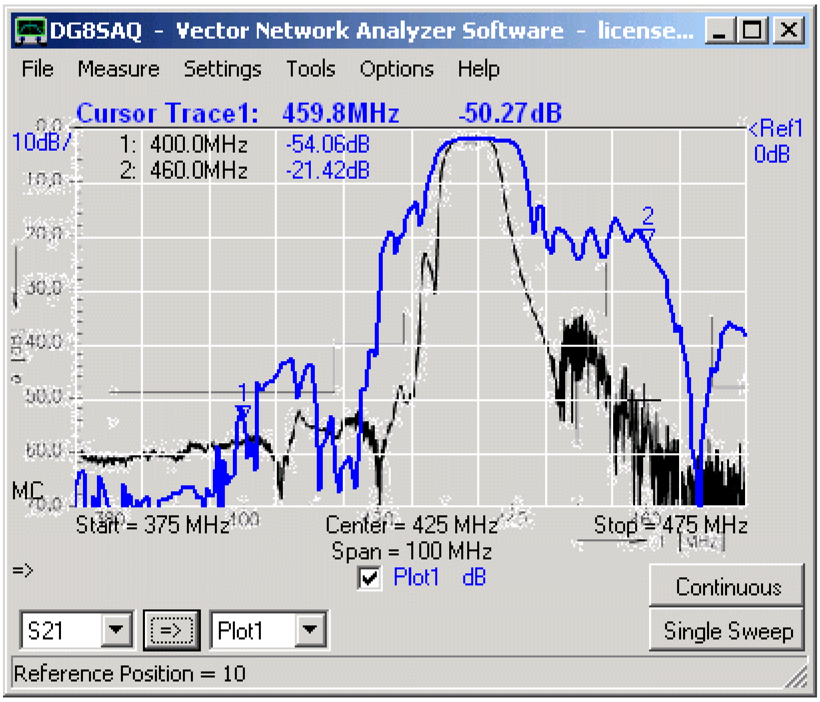


Example 3: Comparing a measurement with a graph from a print or a graphic file

Very often one comes into the situation where one would like to compare an own measurement result with a measurement result published in the literature. Usually, the literature result is not available as S-parameter file, but only as print or pdf-file. In this case, an image file of the literature result can be created by either scanning the paper or extracting the image from the pdf document. In the following, I have extracted an image from a data sheet of a similar filter and loaded it using the gfx_dll.dll:

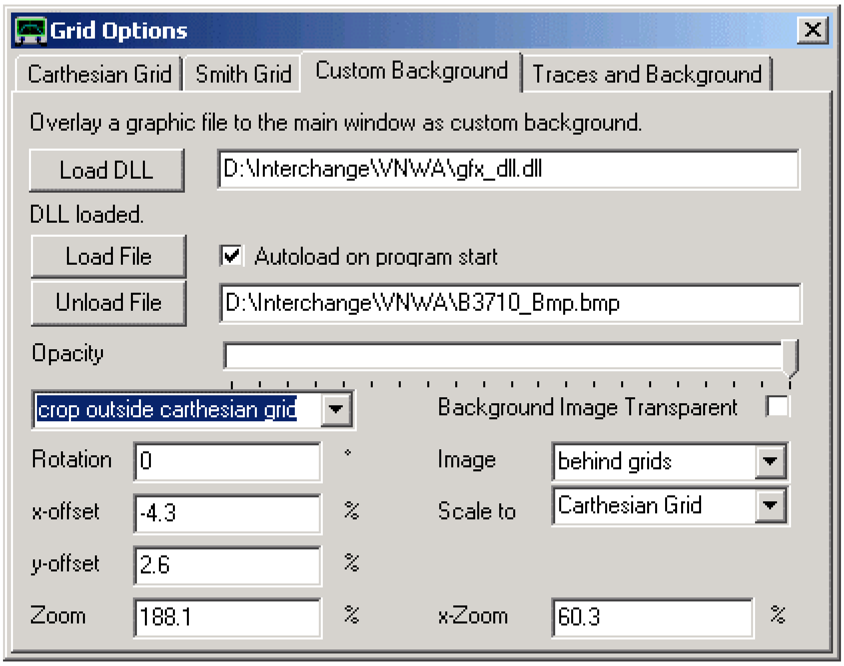


After properly shifting and scaling the image and adjusting the VNWA grid and frequency scale, the image scales match the VNWA scales. As the image is scaled to the carthesian grid, the scale match will be maintained even if the window size and aspect ratio is changed. Now, a measurement can be overlaid. The cursor can be used to compare image values with measurement values:

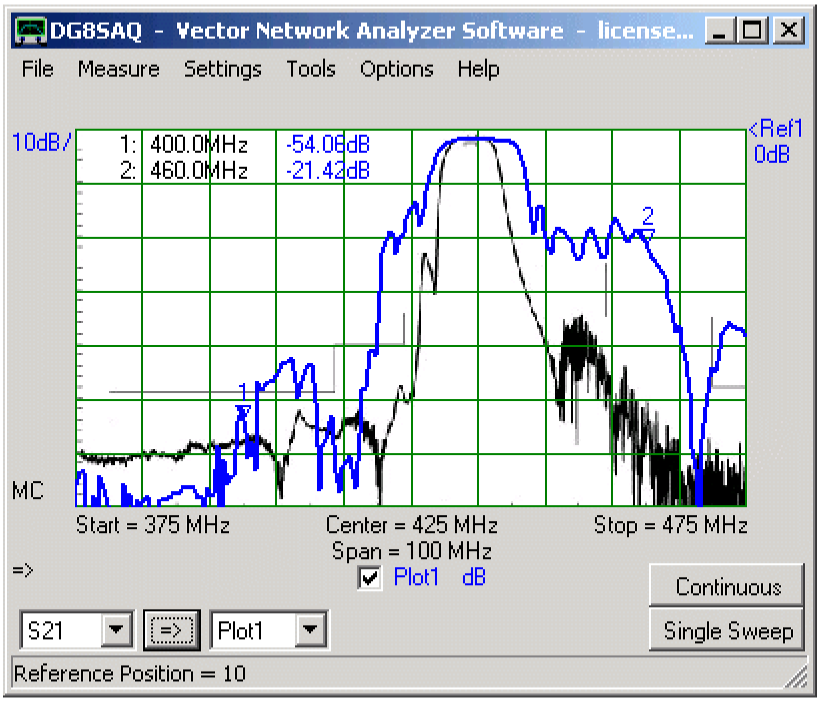


Note, that in above example, the image has some portions that are extending outside the carthesian grid, which might obscure VNWA features. Also, near the image curve transparency is not perfect, which is due to color interpolation in the original image format.

Here, it might be nicer, to make the image non-transparent yielding the white image background and cropping the image parts extending outside the grid:

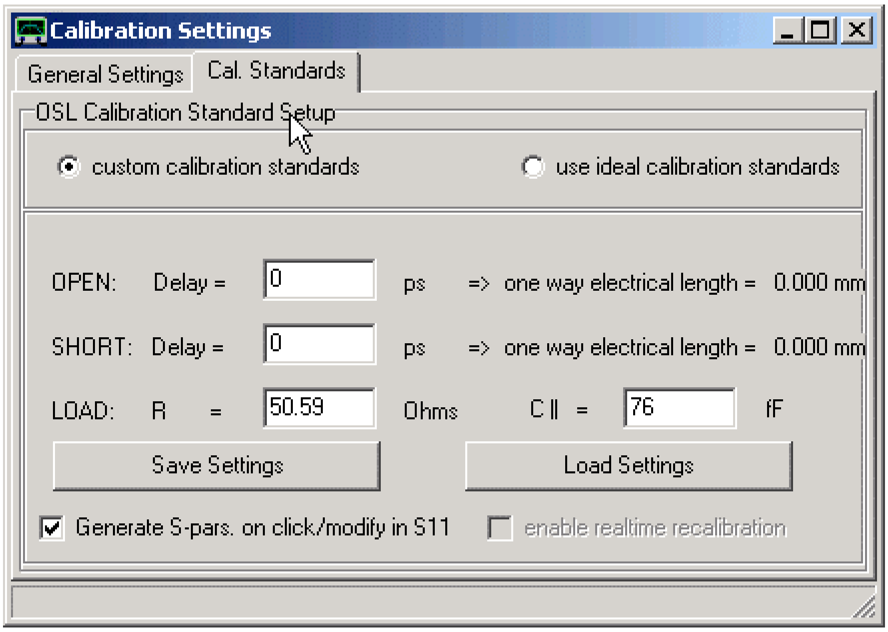


With above settings, the result looks like this:



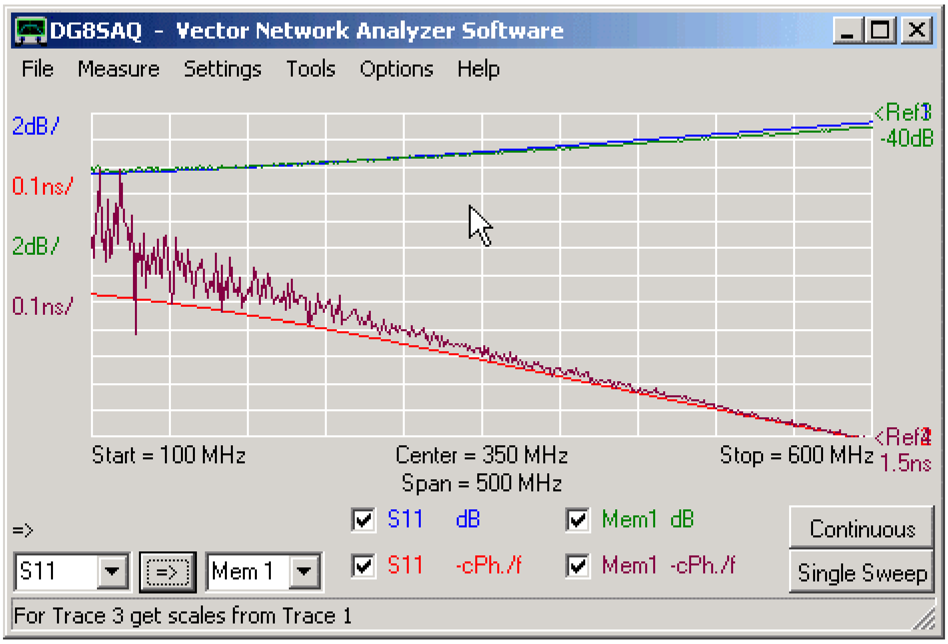
Here you can clearly see that my filter is a lot worse than the literature example.

In order to obtain accurate measurement results, a good knowledge of the used **calibration kit parameters** is mandatory. These must be entered into the "Calibtration Kit Settings" menu:



Extracting parameters from reference measurements

When activating **"Generate S-pars. on click/modify in S11"**, the reflection coefficient of the modified cal standard will be simulated and stored in S11. Note, that the parameters can be "tuned" with the mouse wheel and the changes will be visible on the VNWA main window in real-time. This can be used to fit the calibration kit model to a reference measurement:



The above example shows the reflection coefficient of my load as measured on a R&S ZVM (Mem1) compared to the model generated with above parameters. Note, that the model reacts very sensitive on changes in model parameters.

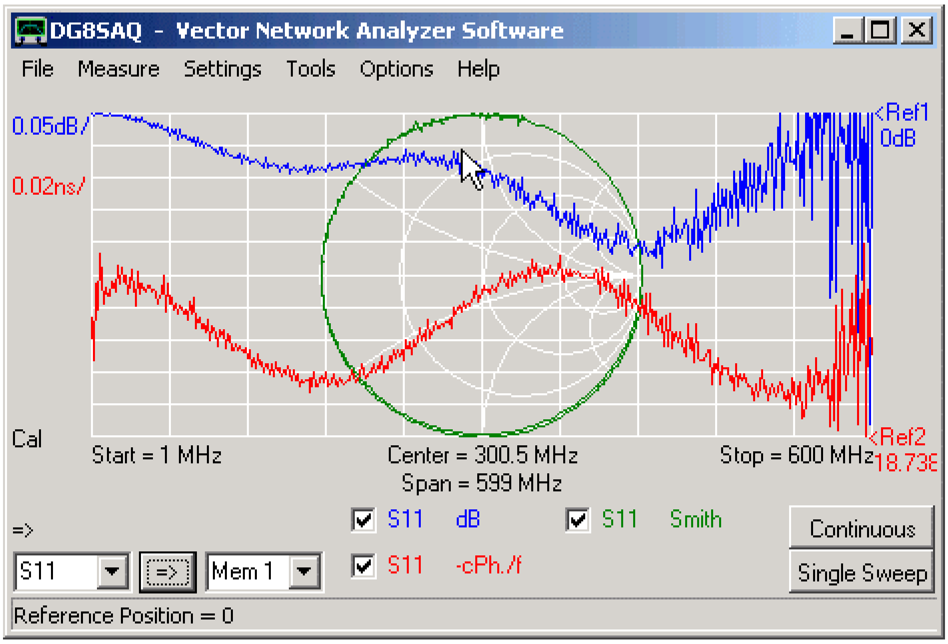
Extracting parameters with the aid of an open transmission line

If no reference VNWA is available, then there is still a possibility to extract model parameters. To do so, do the following:

1) In the "Calibration Settings" window select **custom calibration standards**, but enter the values for ideal calibration kits (Load: 50 Ohms real, all delays zero).

2) Perform a **reflection calibration** (SOL, S11 only) over a frequency range of 1...500 MHz with your unknown calibration kit. Calibrate the VNWA output without a test cable for best accuracy.

3) Measure the input reflection coefficient of 30cm straight high quality semi-rigid 50 Ohms transmission line with its output open or shorted. The result should ideally be a reflection coefficient of magnitude 1 circling around the edge of the Smith chart:

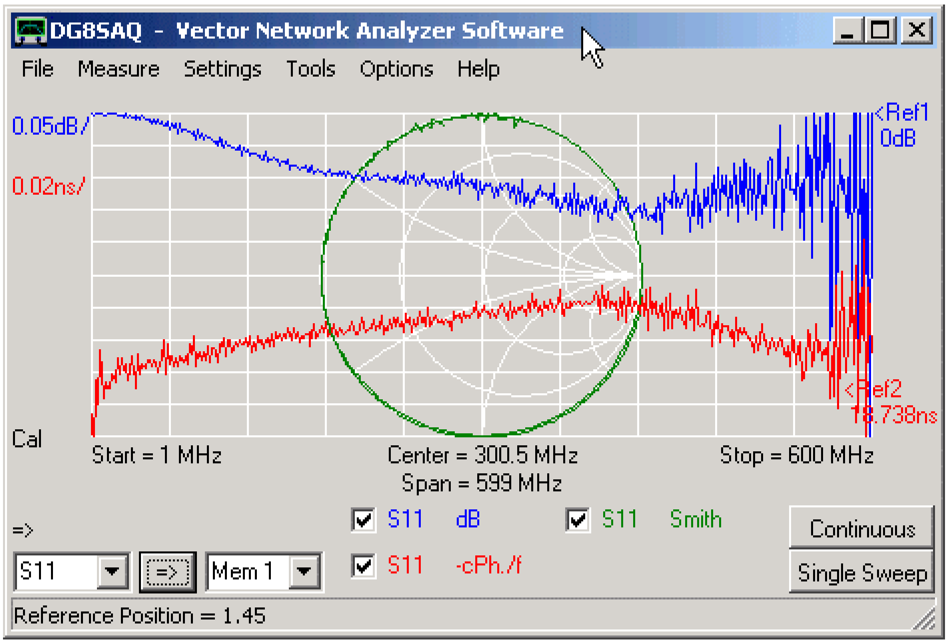


Basically, this is the case. But looking at it a bit closer, the magnitude of the reflection coefficient must decreases with increasing frequency due to the increasing transmission line loss. This is observed, but **superimposed, an oscillatory behavior** with increasing amplitude is visible (blue trace), which may lead to unphysical reflection coefficients greater than unity (outside the Smith chart). Similar oscillations are observed in the phase progression (red trace). These oscillations are not real, but a mathematical artefact arising from the fact, that the real calibration coefficient parameters are different from those used in the correction calculations. This effect is known in the literature as **port mismatch**. Knowing that much, it is straight forward to tune the calibration kit parameters such that the oscillations on both curves vanish in the best possible way and the phase progression becomes as constant as possible (constant "circling speed" around the Smith chart).

4) To do so, activate the **"enable realtime recalibration"** checkbox in the "Calibration Settings" window.

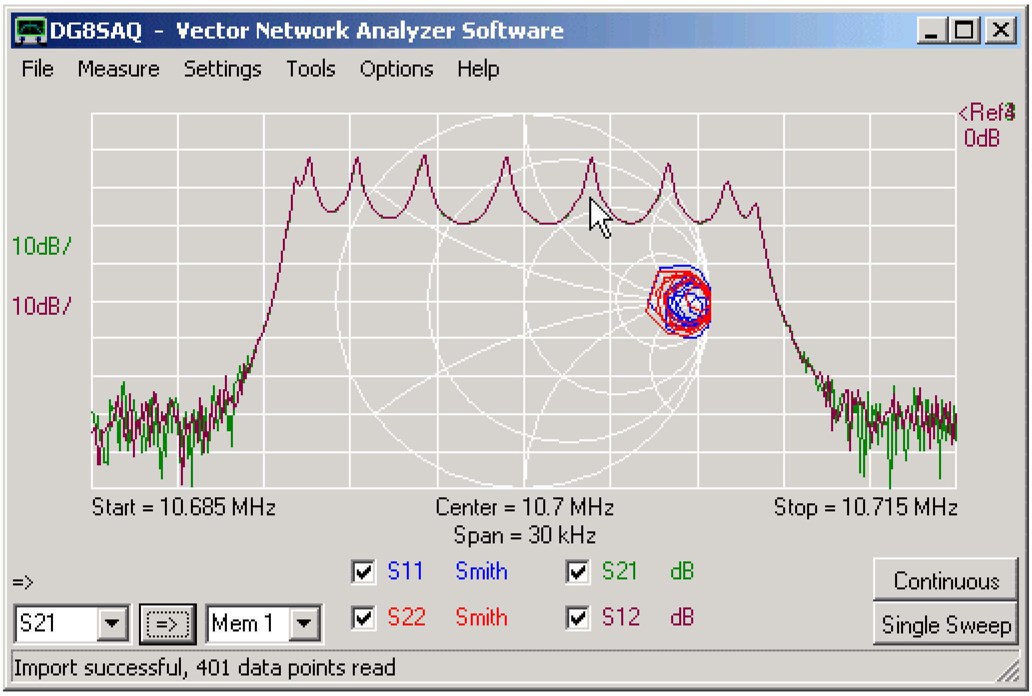
5) Keep the open delay at constant zero value. This will fix the calibration plane.

6) Tune short delay, load R and load C|| with the mouse wheel and observe the change to the recalibrated S11 in the VNWA main window. A good choice with reduced port mismatch is seen below.


DG8SAQ - Vector Network Analyzer Software
File Measure Settings Tools Options Help
0.05dB/
0.02ns/
<Ref1
0dB
<Ref2
18.738ns
Cal
Start = 1 MHz
Center = 300.5 MHz
Span = 599 MHz
Stop = 600 MHz
=>
S11 dB
S11 Smith
S11 -cPh./f
S11
Mem 1
Continuous
Single Sweep
Reference Position = 1.45

Determining the impedance environment for optimum DUT performance

Note, that the VNWA measures **S-parameters**, that are normalized to **50 Ohms source and load impedance!**
This might not be the desired operating conditions for your DUT. E.g. a narrowband crystal filter is usually a high impedance device and performs very poorly, when terminated with 50 Ohms at input and output. The picture below shows a measurement of a 10.7 MHz telecommunication crystal IF-filter.



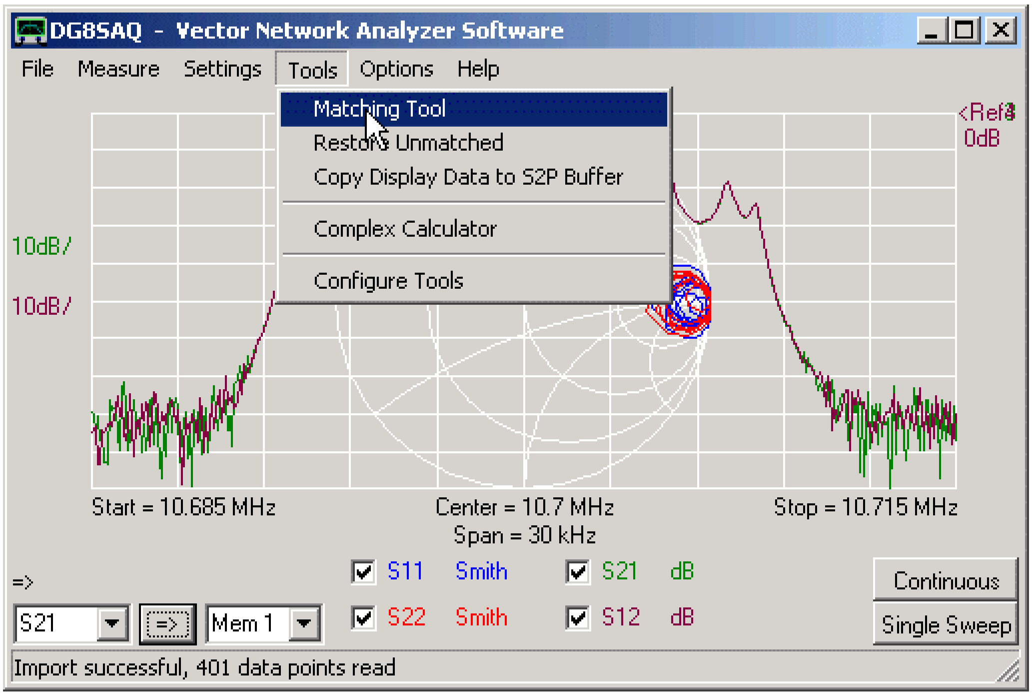
Note, that the filter S21 shows a passband ripple of close to 20dB. Note also, that like for all reciprocal DUTs (all passive devices except isolators and circulators) S21=S12.
As you can see from S11 and S22, the filter is severely mismatched, as its impedances are near the open circuit point in the Smith chart.

When building the filter into a receiver, the high filter impedances must be transformed with inductors and capacitors to match the impedances of the surrounding circuitry. This is called **matching**. Most the time, it is desirable, that the output impedance of one section (e.g. driving transistor) equals the complex conjugate of the following section's input impedance. This condition is called **power match**, as it ensures, that all power is transfered from one stage to the next. Sometimes, different matching conditions are used, e.g. noise match, to optimize the signal to noise ratio.

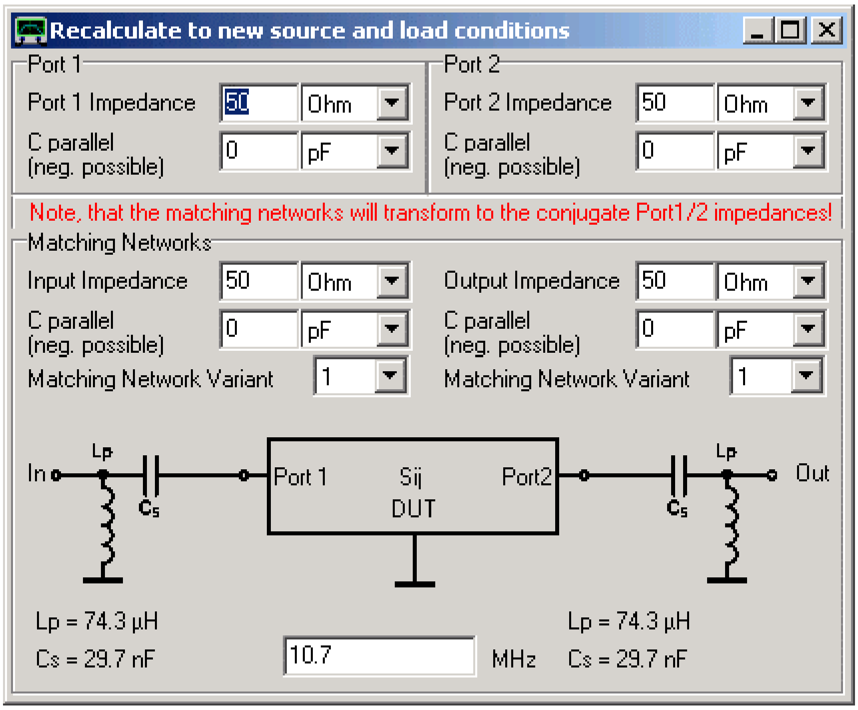
The VNWA's built-in *matching tool* allows to simulate the DUT's transfer characteristics under arbitrary impedance termination conditions.

▸▸▸ **Note:** You must have a complete set of four valid S-parameters (S11,S21,S12,S22) in order to use the matching tool.

To open the matching tool, select **"Tools"-"Matching Tool"**:

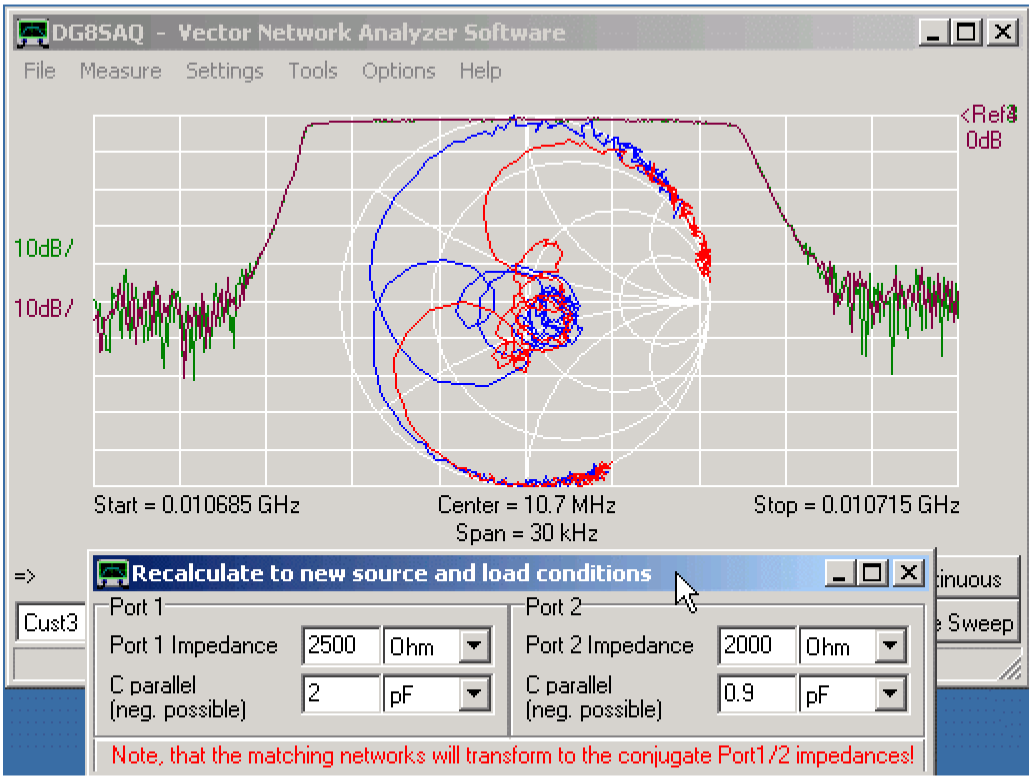


The **matching tool window** will pop up.



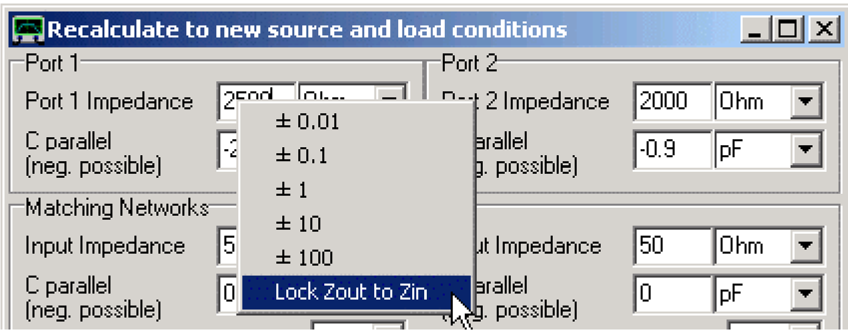
Place this window in such a way on the desktop, that you can see both, the VNWA main window with your measurement curves and the matching tool window.

In the upper part of the matching tool window, you can specify the **your DUT's impedances**. At startup, these are real 50 Ohms on input and output (=impedances of the VNWA). If you modify them, you will find, that the S-parameters on the main window change. The crystal filter of our example has impedances of about 2000 Ohms resistance parallel to about 1pF capacitance (thus impedances represent slightly capacitive loads) on input and output. Entering these impedances yields a perfectly flat passband and a good power match:



▸▸▸ **Note:** The mathematical transformation from the measured 50 Ohms impedance level to calculated 2 kOhms imposes a crucial test on the accuracy and consistency of the four S-parameters (S11,S21,S12,S22). If the passband shows spikes, this is a sign that there was some measurement error, e.g. wrong calibration, inconsistent reference planes, no 6 term correction... .

▸▸▸ **Hint:** You can also modify the impedances with the **mouse-wheel**. To change the increment, right-click onto the according input field. Here, you can also **lock input and output impedances.** This might be useful, if you have a symmetric DUT and you want to change input and output impedances synchronously:

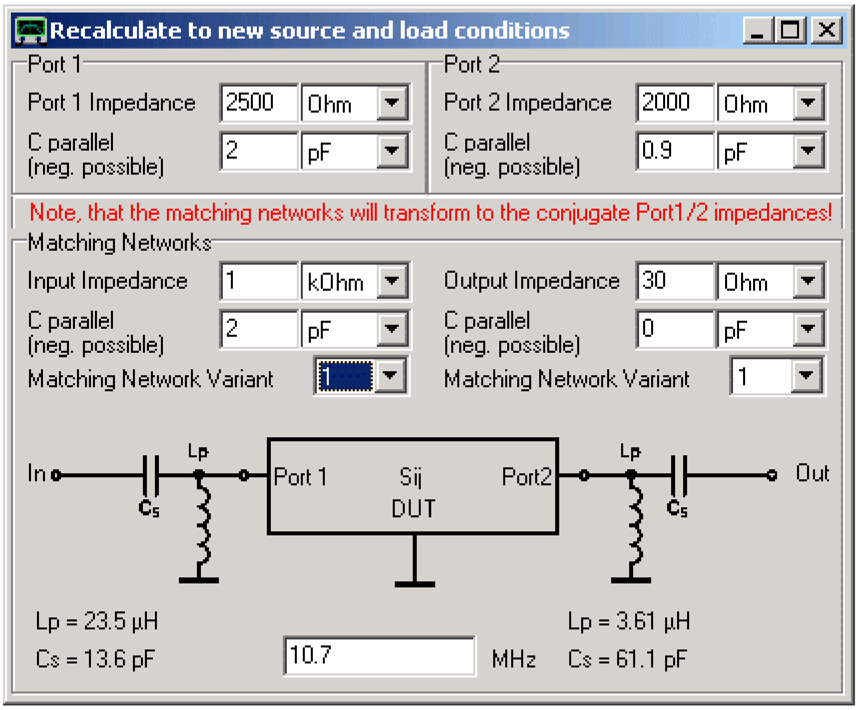


By now, we have determined the impedance environment, our IF filter (=the DUT) wants to see for best performance, namely the complex conjugates of the port 1 and port 2 impedances.

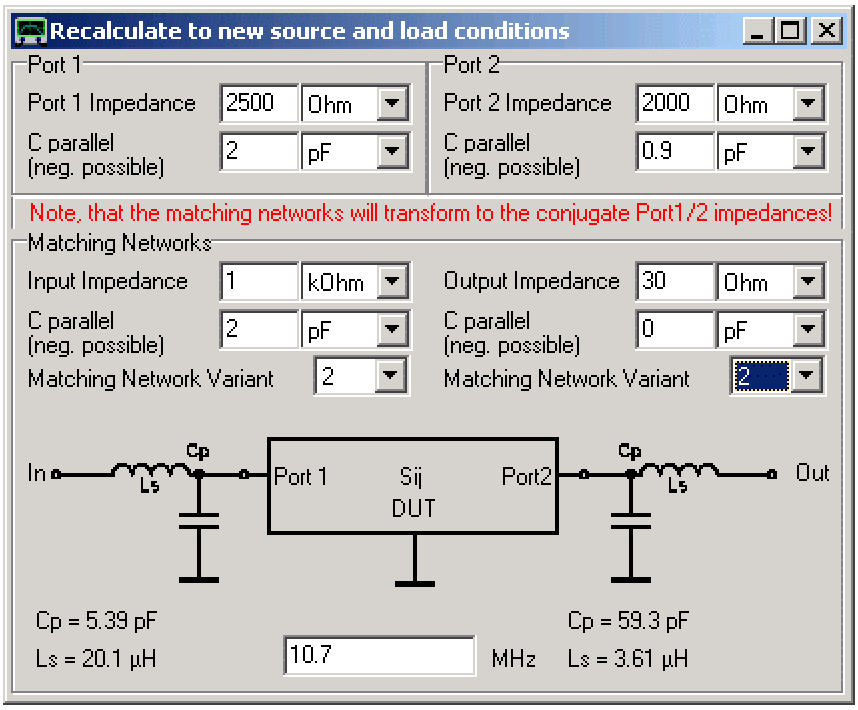
Determining the matching networks

If we want to actually place the above filter in between a mixer output and an amplifier input, we need to **match the mixer output to the filter input and the filter output to the amplifier input with two matching networks**.

The matching networks can be calculated in the lower part of the matching tool window. Assume, the mixer output has an impedance equivalent to 1 kOhm || 2 pF and the subsequent IF amplifier stage has an input impedance of real 30 Ohms (i.e. || 0 pF). Enter these impenances into the lower part of the matching tool window:



The calculated matching networks are shown in a little schematic. With the displayed matching networks, the filter will yield optimum performance now, if the **In-terminal** sees 1 kOhm || 2 pF and the **Out-terminal** sees 30 Ohms.
▸▸▸ **Note:** Calculations are performed with ideal components with infinite Q-values.
▸▸▸ **Hint:** Identical impedance transformations can be obtained with up to 4 different variants of matching network topologies. You can select the one you like with the combo-boxes:



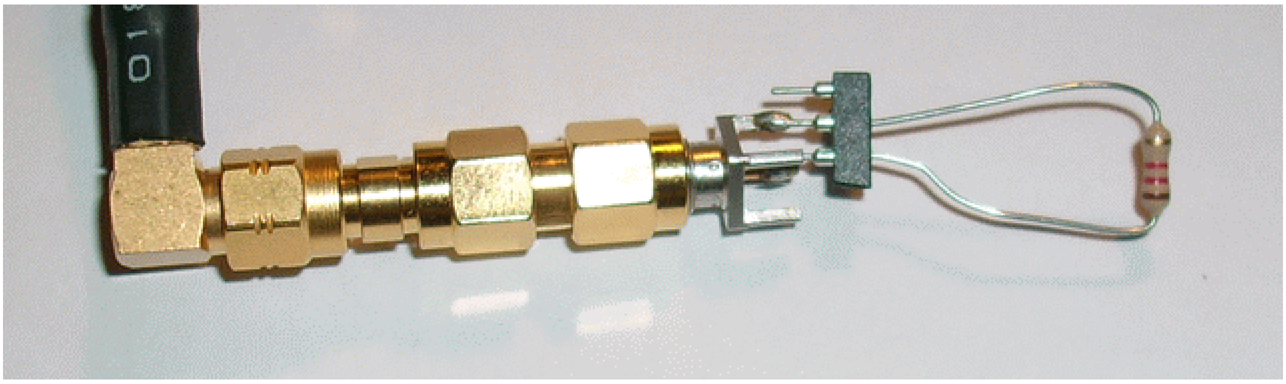
▸▸▸ **Note:** The impedance transformations are **calculated at a single frequency**, by default the center frequency. If this doesn't coincide with your filter's passband, you can change the frequency used by the simulation in the edit field at the very bottom.

This tutorial describes, how the VNWA can be used to **generate and optimize an impedance matching network**, that transforms a given source impedance (in our example 50 Ohms) to a given load impedance.

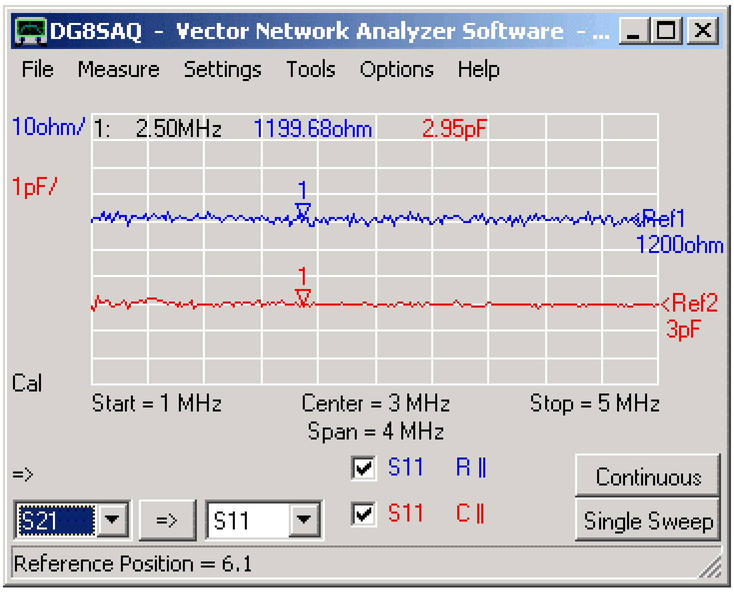
▸▸▸ **Note:** For all measurements, the **VNWA must be properly calibrated**. Some of below screenshots show the VNWA in uncalibrated state. These screenshots have been created from earlier recorded S-parameters. All S-parameters have been measured with the calibrated instrument.

▸▸▸ **Note:** If you repeat below experiments in the VHF/UHF range, a proper setting of the calibration plane using port extensions will be most important. Since below example is performed at 2.5MHz (Wavelength >100m), a few mm shift in the calibration plane will hardly be detectable.

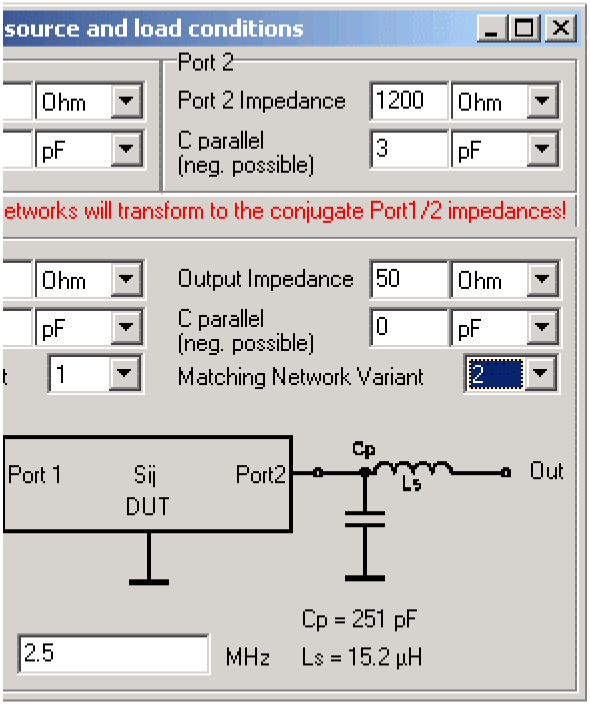
1. First, we need to know the source and load impedance between which we want to achieve a power match, i.e. a matching condition with minimum power loss.
For simplicity, we use 50 Ohms as the impedance of our source. This could be the output of a transmitter.
Our load impedance in this example will be a 1,2kOhm wired carbon resistor:



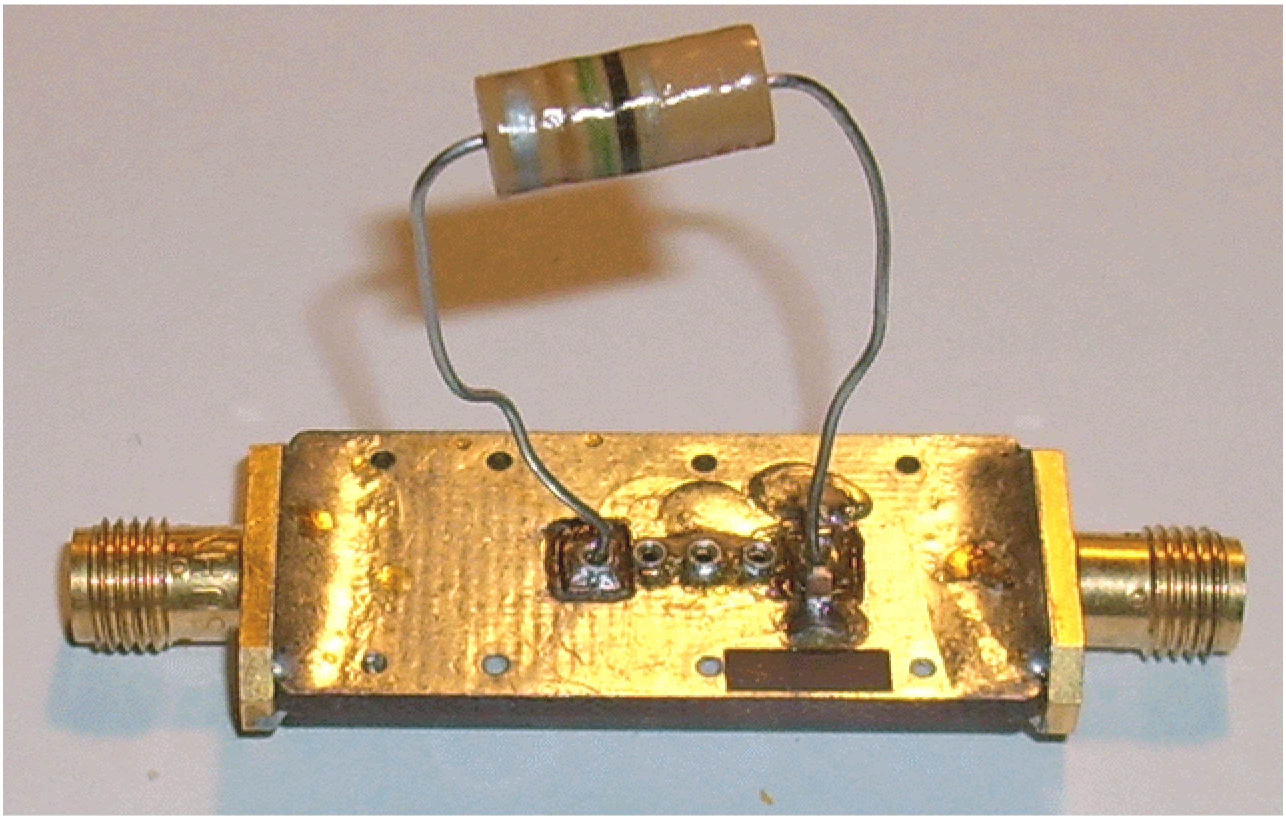
This could be a model of a high impedance antenna. To be on the safe side, we measure the impedance of our load in the frequency range of interest:



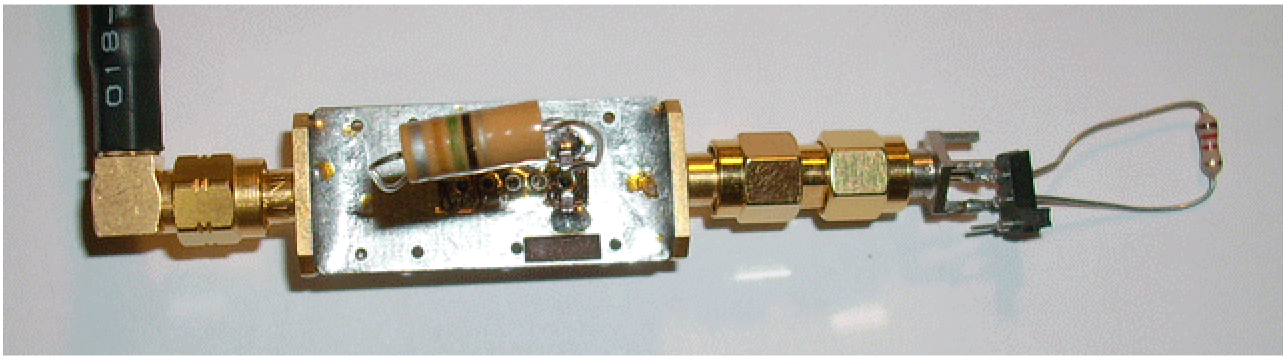
2. We want to achieve a power match at the marker frequency of 2.5MHz, therefore we calculate a matching network for this frequency. This can be done with the VNWA matching tool. We only need half of it. We enter our source impedance (50 Ohms) and the measured load impedance (1200Ohms || 3pF), the frequency where the matching condition is to be achieved (2.5MHz) and select a matching topology (#2):



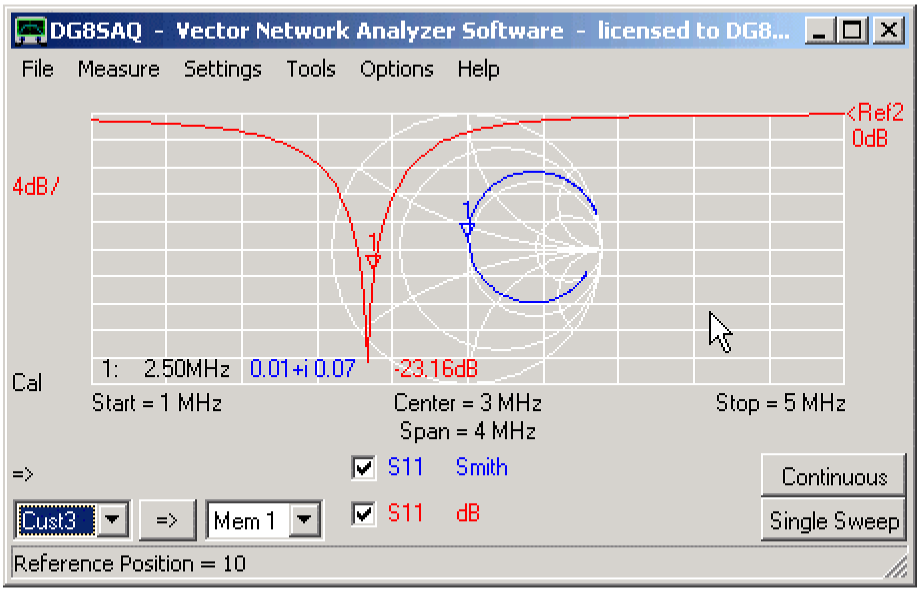
In order to build up the matching network we will need a 15uH inductor (standard value) and a 251pF capacitor which we will approximately create by connecting a 150pF and a 100pF capacitor in parallel. All components are connected on a test board:



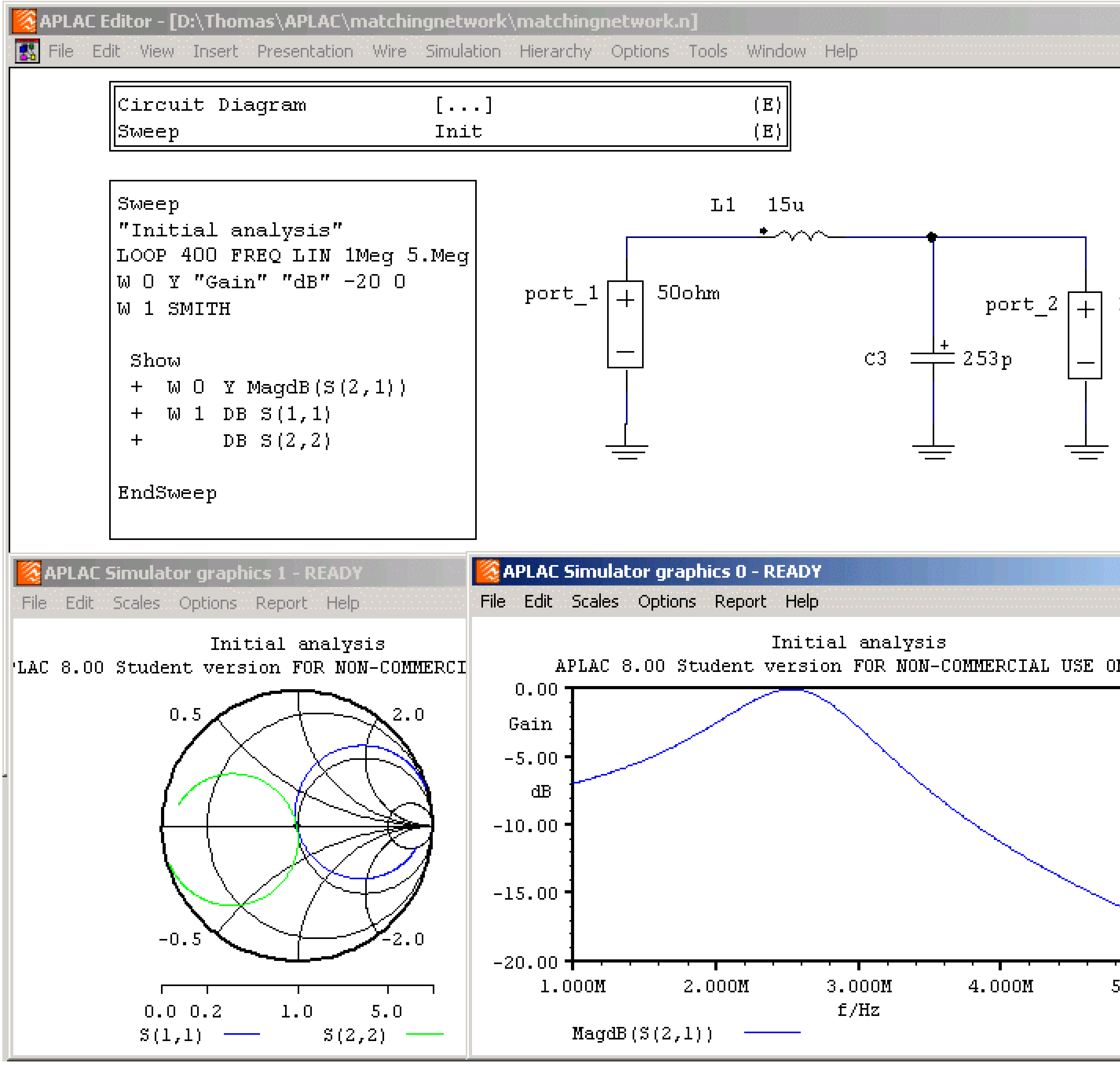
3. We connect our high impedance load to the matching network output ...



... and measure the input reflection coefficient:

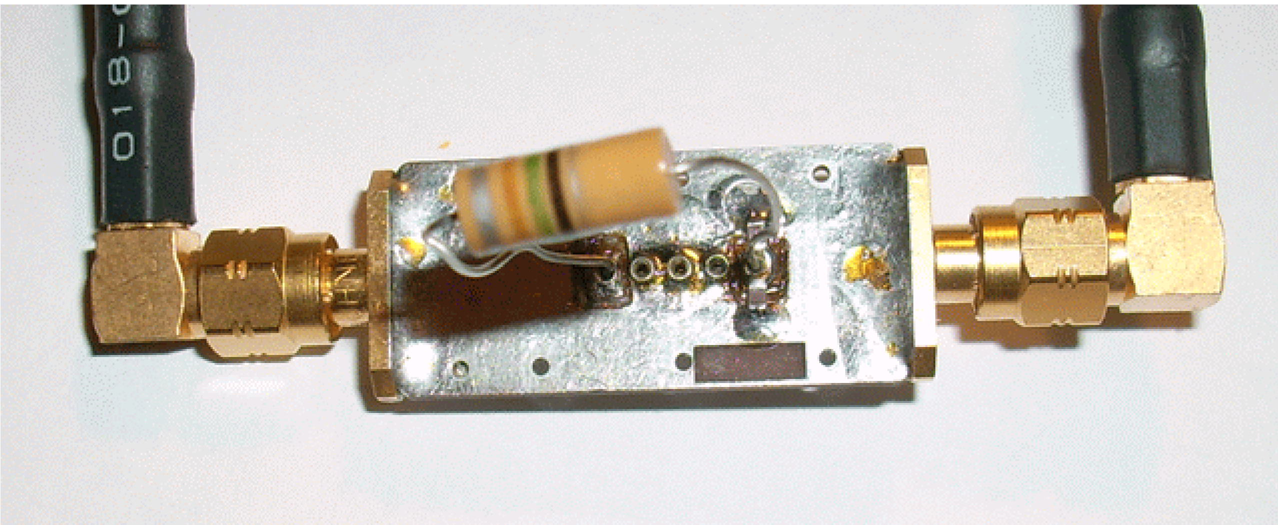


We observe that matching works. We achieve a return loss on input which is better than 20dB. Now the question arises, what is the insertion loss of our matching network.
We could do a simulation e.g. with APLAC:

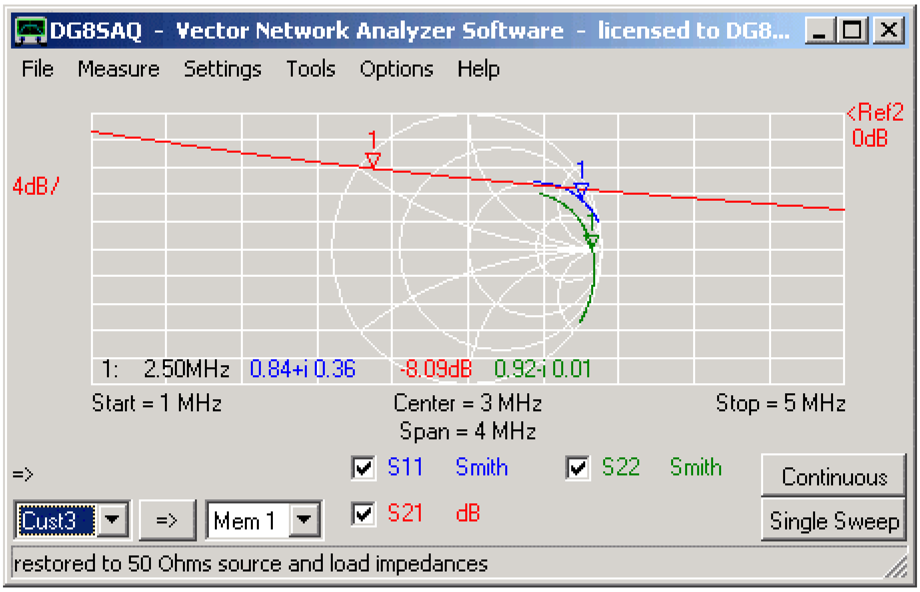


We see that S11 looks like the measurement and S21 becomes nearly 0dB at 2.5MHz because we have simulated with lossless ideal components.

4. In order to find the real insertion attenuation of the matching network, we must perform a full 2-port measurement of the matching network (S11, S21, S12, S22):

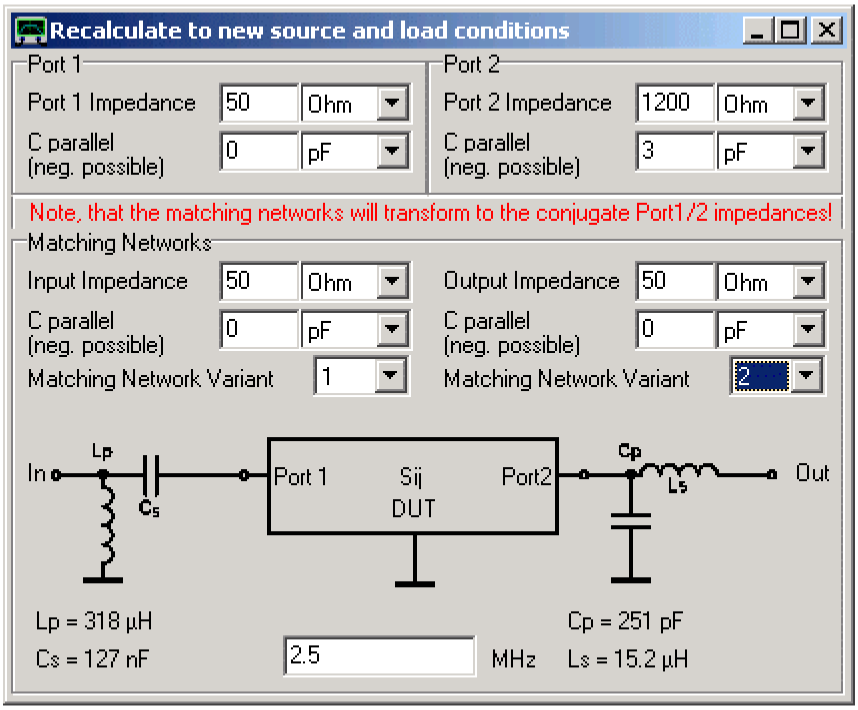


We will see an insertion loss of about 8dB and a severe mismatch on input and output:

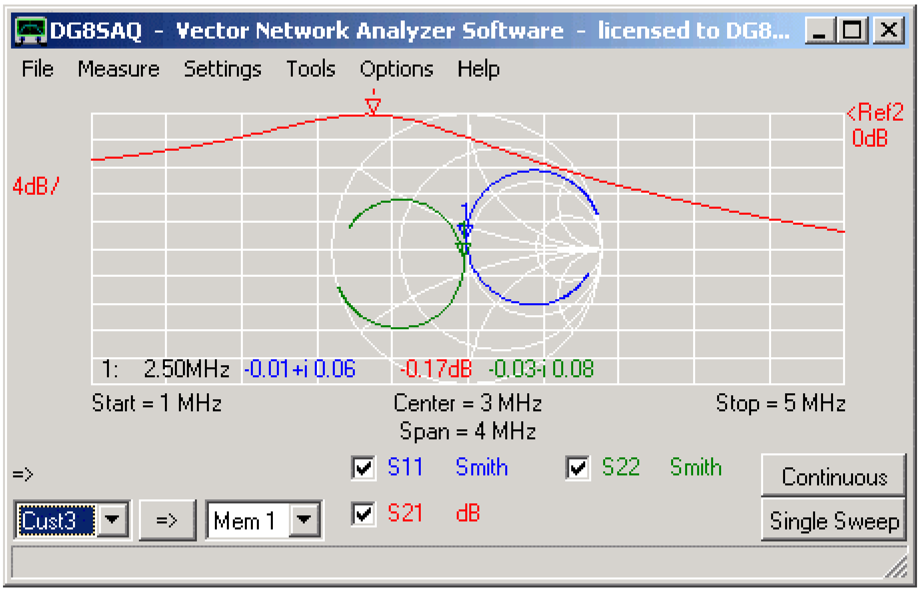


This is because the matching network is now terminated with 50 Ohms of the VNWA.

5. Using the matching tool we can recalculate the S-parameters to what they would be like if the load impedance were the one of our hardware load. We enter the source and load impedance and the matching frequency:

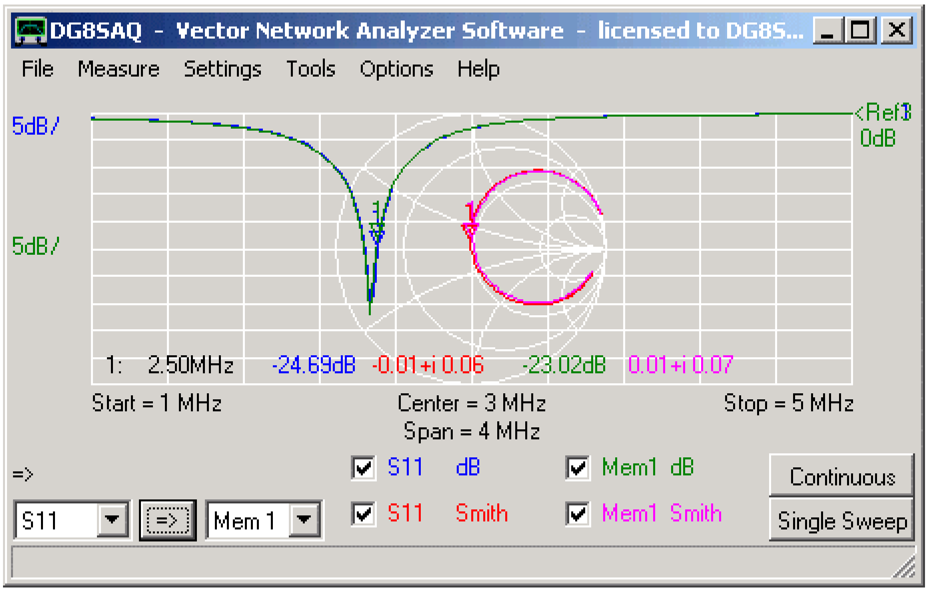


We observe how the S-parameters change under the new load condition:



Now we can see that our matching network introduces a transmission loss of 0.17dB, which is very good. Observe how closely this recalculated measurement resembles the APLAC simulation.

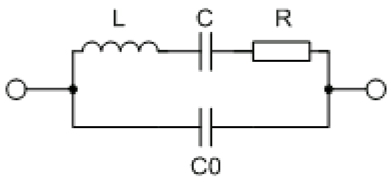
6. The renormalized S11 for the new load condition should be identical to the measured S11 when our hardware load is connected to the matching network output. This can be verified by a measurement:



S11 contains the renormalized 50Ohm measurement while Mem1 contains the input reflection coefficient of the matching network with the hardware load connected to the output. As expected the two are identical.

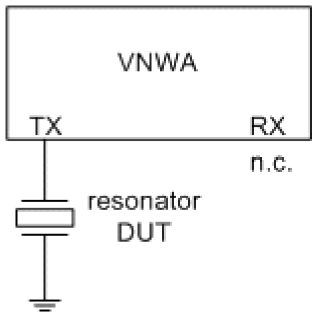
General

A crystal or any other electrical resonator can be described reasonably accurate by the following **equivalent circuit model** :

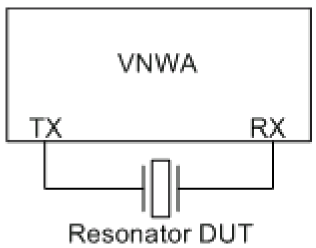


Knowledge of the model parameters is useful as input to circuit simulators for e.g. synthesizing crystal filters or modeling crystal oscillators. The **VNWA crystal analyzer tool** allows to **extract these model parameters from a measurement**.

A crystal or one port resonator can be measured in **reflection** (S11)



or in **transmission** (S21)



Often low impedance (e.g. 12.5 Ohms) test fixtures are available for crystal transmission measurements. Therefore the **fixture impedance must be taken into account** for transmission measurements. **You should NOT use a low impedance fixture for reflection measurements, unless you exactly know how to calibrate it properly.**

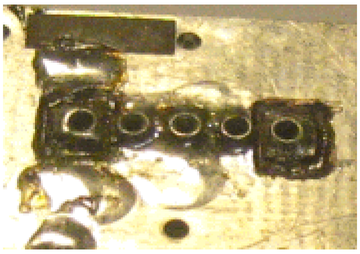
The following examples show several methods to measure crystal resonators and to extract model parameters. In all measurements 3000 data points and a sweep time of 2ms/data point have been used for measurement. This is not a recommendation but simply the first and only settings I have tried. Feel free to experiment. But keep in mind that a crystal is an extremely narrow band device and as such requires some time for settling. So, do not sweep too fast. Also, the accuracy of your calibration will improve with slower sweep rates.

Example 1: Reflection Measurement

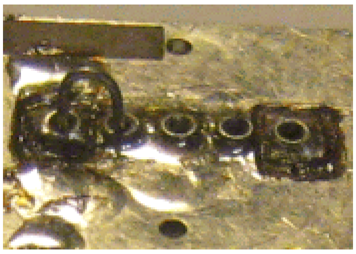
1. Do a proper SOL calibration of your VNWA + test fixture:

You might want to use customized calibration standards that plug straight into your test fixture for this purpose, e.g.

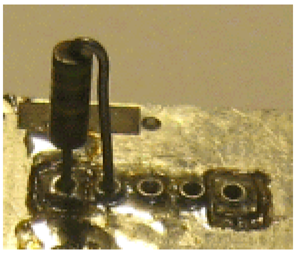
OPEN:



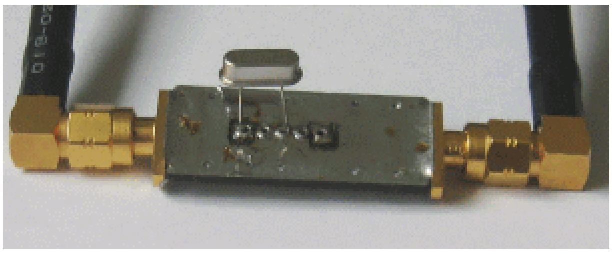
SHORT:



LOAD:

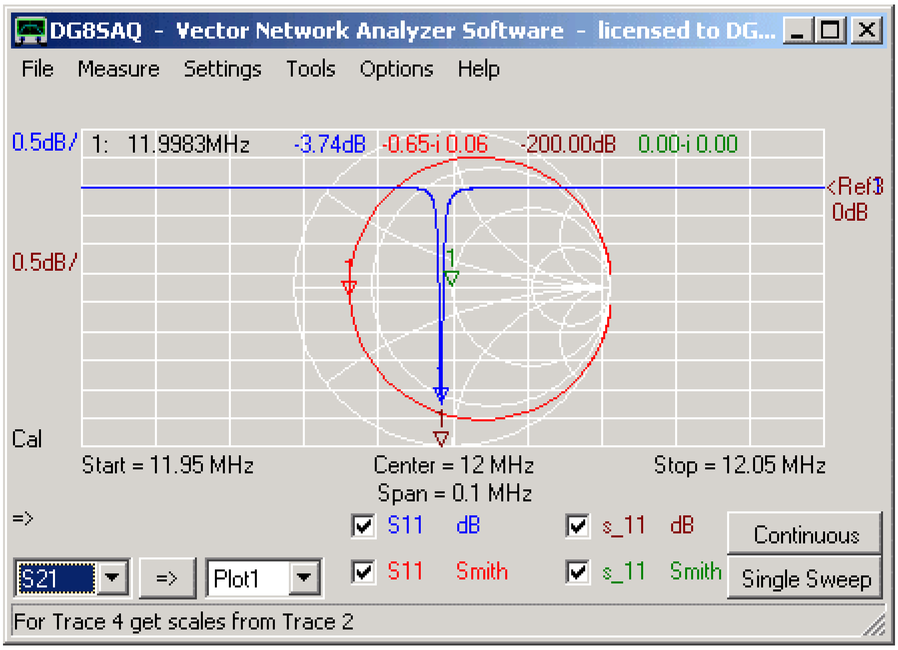


2. Perform a reflection (S11) measurement of your crystal with one crystal pin grounded.



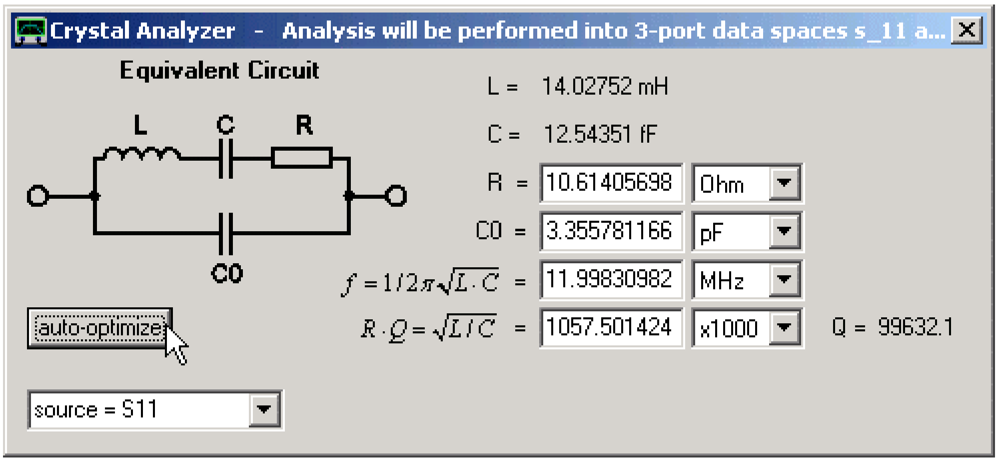
By doing so, you measure the reflection coefficient of your resonator as 1-port device. Make sure, the crystal is connected as close as possible to the calibration plane.

This is what the measurement result should look like for a 12 MHz crystal:

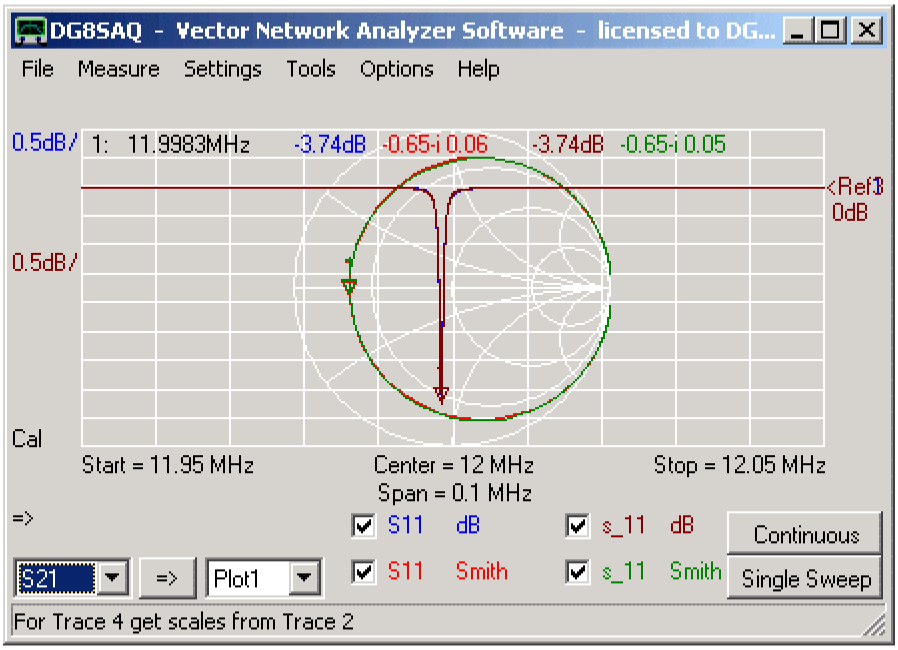


Make sure, to display 3-port memory space s_11 before moving to the next step. 3-port memory spaces s_11 and s_21 will be used to store and allow to display the model simulation in reflection and transmission.

3. Start the "Crystal Analyzer" tool via the main tools menu.



4. The above is what you see a short moment **after having pressed the auto optimize** button. Make sure that **source = S11** is selected before auto-optimizing. The displayed equivalent circuit model is fitted to the measurement data. The simulated data is stored in s_11 (and s_21) for comparison. Note that there is a perfect match between measurement and simulation:



▸▸▸ **Hint:** After auto-optimization, the fitted model parameters are automatically stored in the Windows cut-and-paste buffer and can be pasted into any text document in the following form:

F = 11998309.8281313
Q = 99632.1600629463
L = 0.0140275240017728
C = 1.25435105103075E-14
R = 10.6140569892503
C0= 3.35578116630557E-12
F1= 12020712.3
F2= 12020712.9
F3= 11998309.2
F4= 12020632.9
F5= 12020772.2
F6= 11998312.5

Units are Hz, none, Hy, F, Ohms and F.

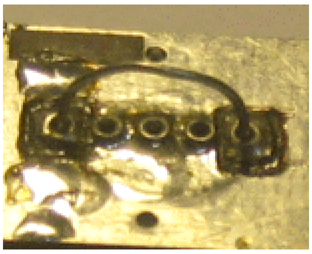
Note: Frequencies F1...F6 will be explained in the next section about measuring the crystal in transmission.

Example 2: Transmission Measurement

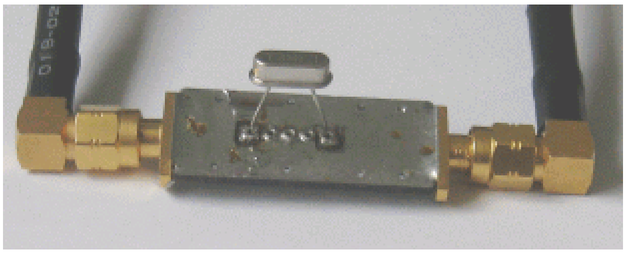
1. Do a Thru calibration of your VNWA + test fixture:

You might want to use a customized calibration standard that plugs straight into your test fixture for this purpose, e.g.

THRU:

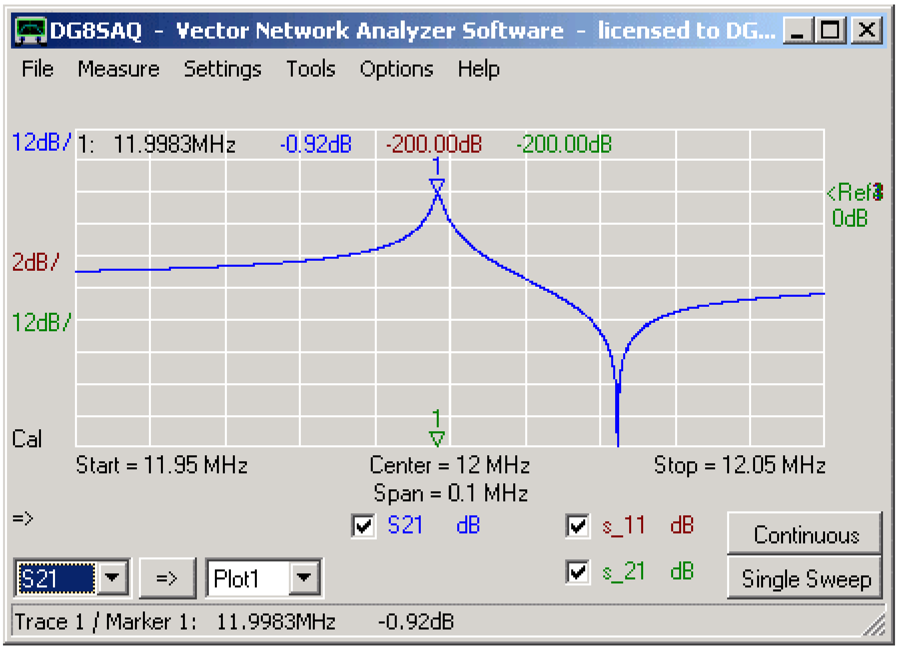


2. Perform a transmission (S21) measurement of your crystal with one crystal terminal being input and the other being output.



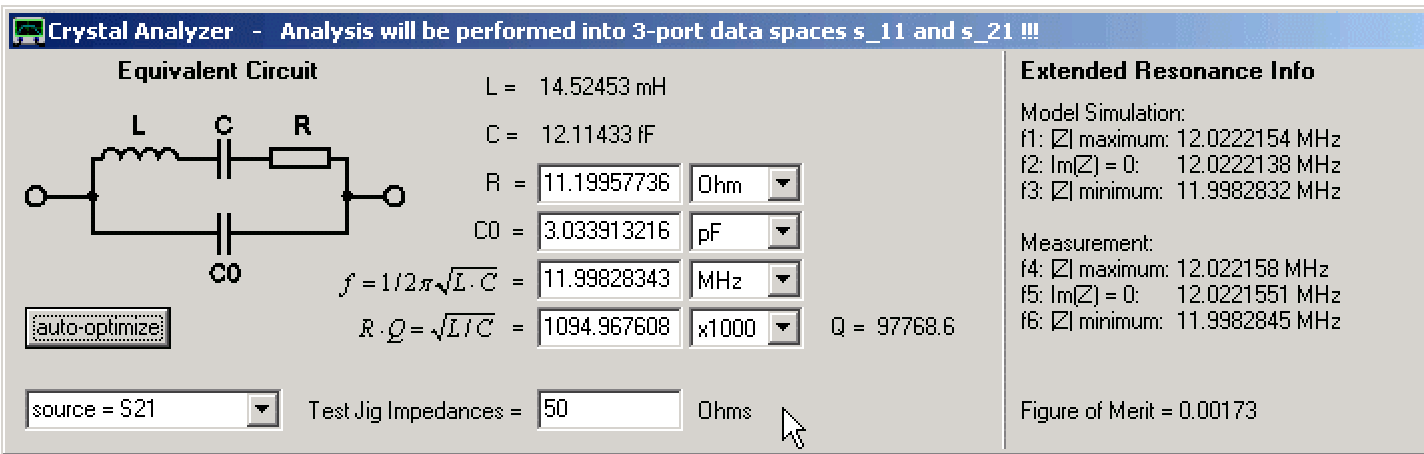
By doing so, you measure your resonator as 2-port device. Make sure, the crystal is connected as close as possible to the calibration plane.

This is what the measurement result should look like for a 12 MHz crystal:



In transmission one can clearly see the serial resonance (marker) as well as the parallel antiresonance.
Make sure, to display 3-port memory space s_21 before moving to the next step. 3-port memory spaces s_11 and s_21 will be used to store and allow to display the model simulation in reflection and transmission.

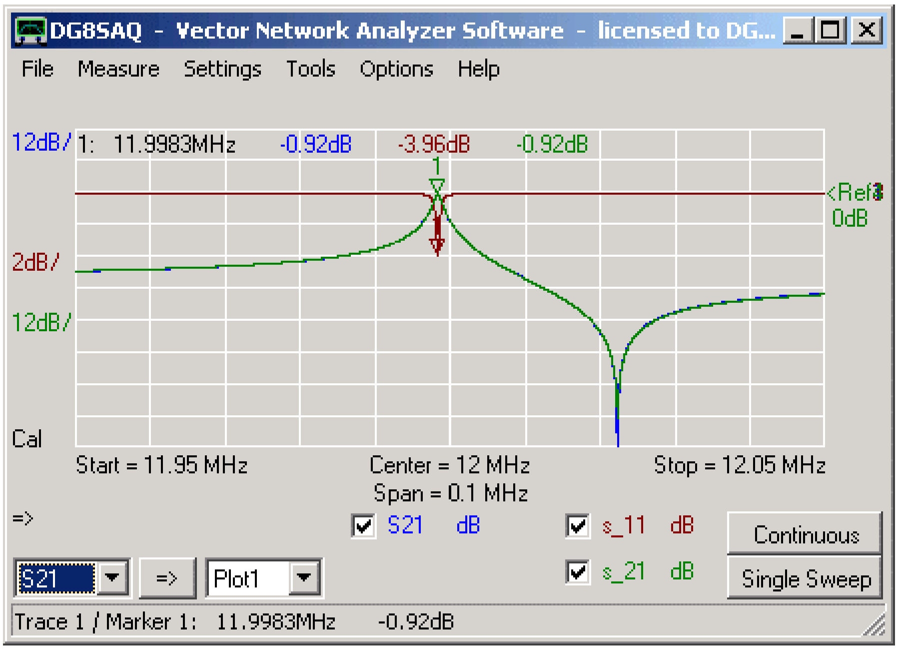
3. Start the "Crystal Analyzer" tool via the main tools menu and increase the width of the crystal analyzer window:



4. Make sure that **source = S21** and the correct **test fixture impedance** (must be identical on input and output) is selected before auto-optimizing.The above is what you see a short moment **after having pressed the auto optimize** button. The displayed equivalent circuit model is fitted to the measurement data.

Note that on the right hand side of the crystal tool, various frequencies are extracted from the simulation as well as from the measurement data. f1/f4 amd f2/f5 are two different definitions of the parallel antiresonance frequency. f3/f6 is a definition of the serial resonance frequency differing slightly from f on the left hand side. This data is also calculated for reflection measurement fits.

The simulated data is stored in s_21 (and s_11) for comparison. Note that there is a perfect match between measurement and simulation:



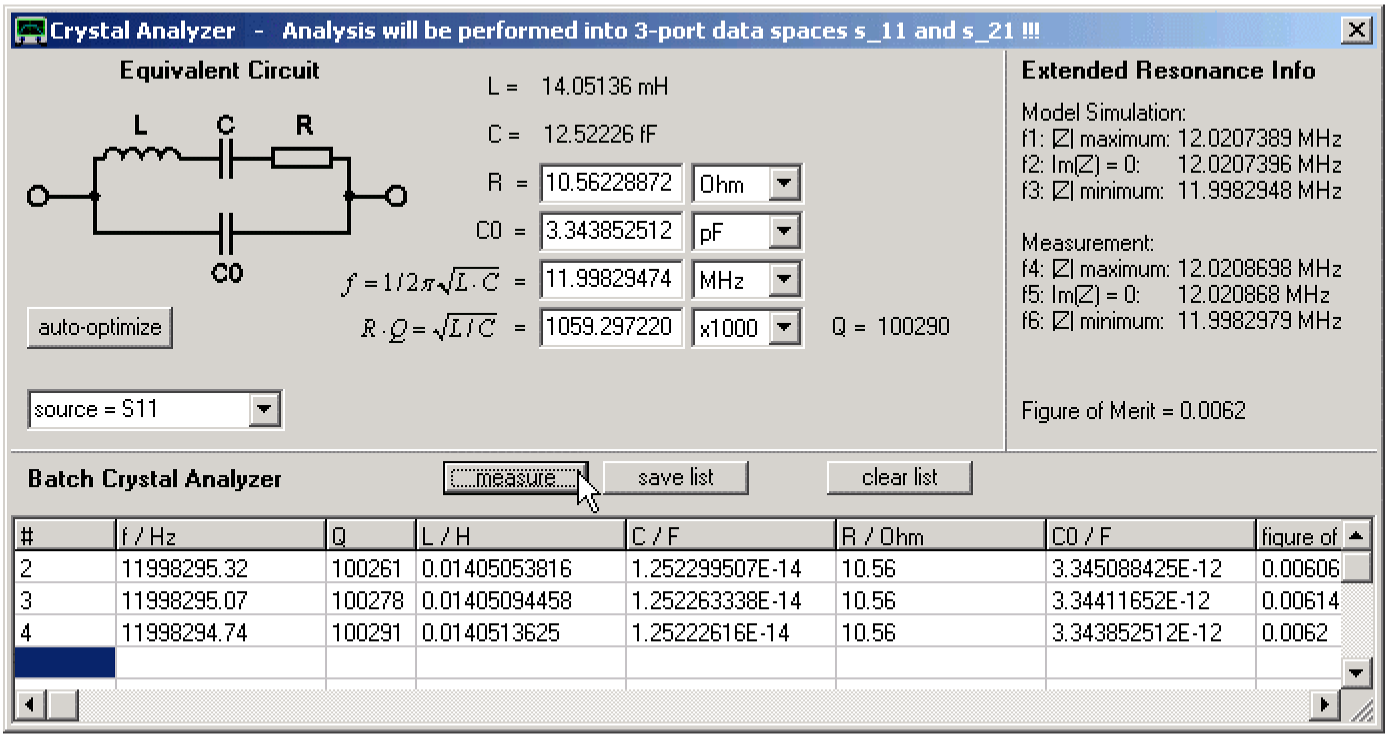
Note, that after auto-optimization, the fitted model parameters are automatically stored in the Windows cut-and-paste buffer and can be pasted into any text document like in the reflection example.

General Hints

You can also **manually optimize** by changing the data in the edit fields by keyboard or by **mouse-wheel** while observing the changes of s_11 and s_21 on the VNWA main window..
Mouse-wheel increments can be set individually by **right-clicking onto the edit fields**.
The displayed **figure of merit** is a distance between simulation and measurement and should thus be as close to zero as possible.

Analyzing a batch of crystals

If you want to analyze a whole batch of more or less identical crystals, you can increase the size of the crystal analyzer window downwards in order to access the batch table:



On pressing the "measure" button, a measurement sweep will be started and at its end, the data will be analyzed and the values will be entered into the table at the cursor position. You can remeasure a line, by selecting it with the mouse prior to pressing measure. You can save the list into a csv-file, which can be read by Excel. You can also erase the list.
The line numbering is assigned automatically, but you can manually change it if desired. The above picture shows three consecutive measurements on the same crystal.

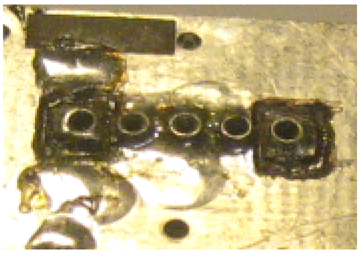
▸▸▸ **Hint:** The fitted model parameters of the last sweep will also be automatically stored in the Windows cut-and-paste buffer and can be pasted into any text document.

Example 3: Mixed measurements with virtual ground calibration

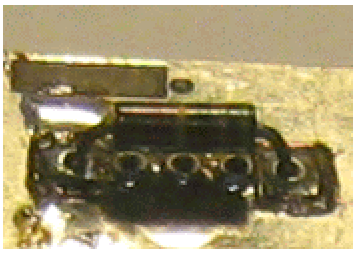
Kurt Poulsen OZ7OU has come up with the idea of calibrating the VNWA+test fixture such that the hot RX pin serves as a virtual ground for the reflection measurement. In the following this type of calibration is refered to as virtual ground calibration. It is an SOLT calibration where the real ground is replaced by a virtual ground for reflection measurements only. The advantage of this type of calibration is that you can measure your crystal in transmission and in reflection without having to change connections in the test fixture. So, the crystal stays in the fixture in the same way (between hot TX pin and hot RX pin) no matter if you measure it in reflection or in transmission.

1. Do a virtual ground calibration of your VNWA + test fixture:

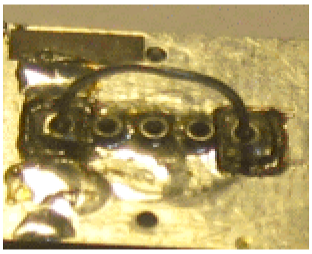
OPEN:



LOAD:

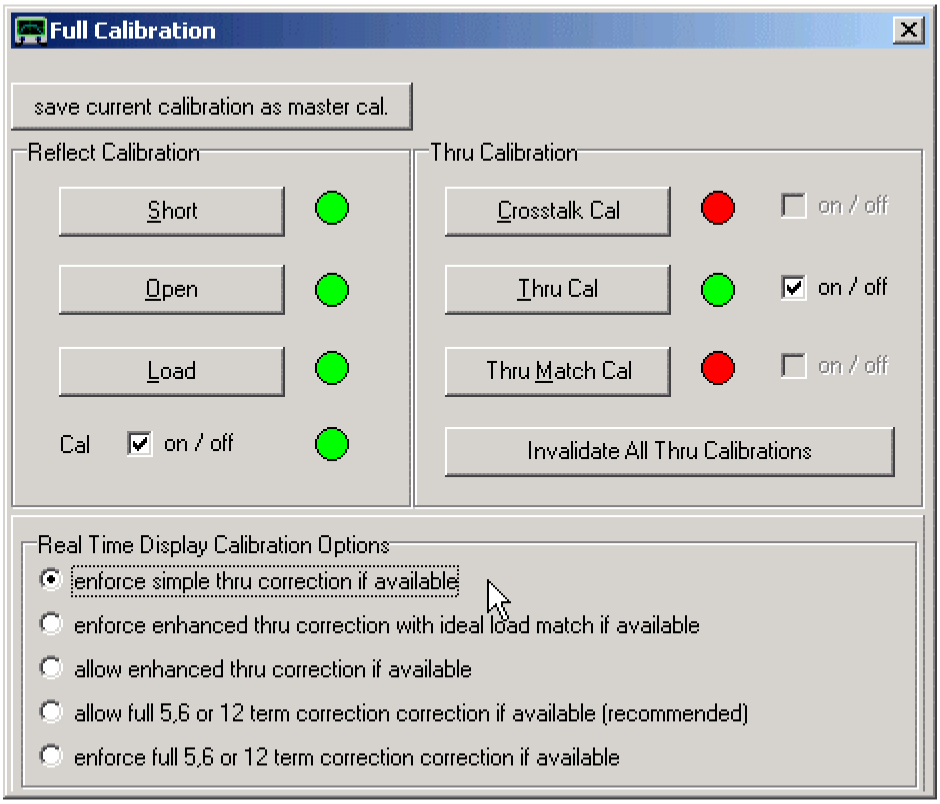


SHORT and THRU:



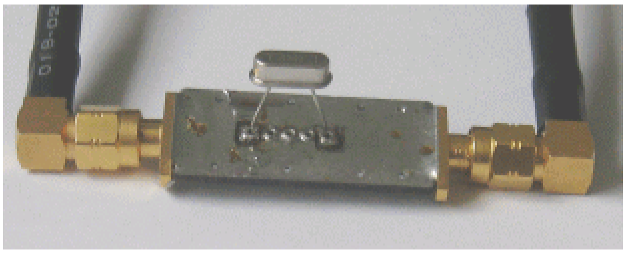
Don't perform a thru match calibration.
▸▸▸ **IMPORTANT:** Select **"enforce simple thru correction..."** as real time display calibration option (see mouse pointer below).

▸▸▸ **Remark:** The parallel capacitance C0 that will be calculated from the measurement, will contain also the capacitance of the test fixture from input to output. This test fixture capacitance can be effectively removed by performing an **isolation calibration** with the empty test fixture in place.

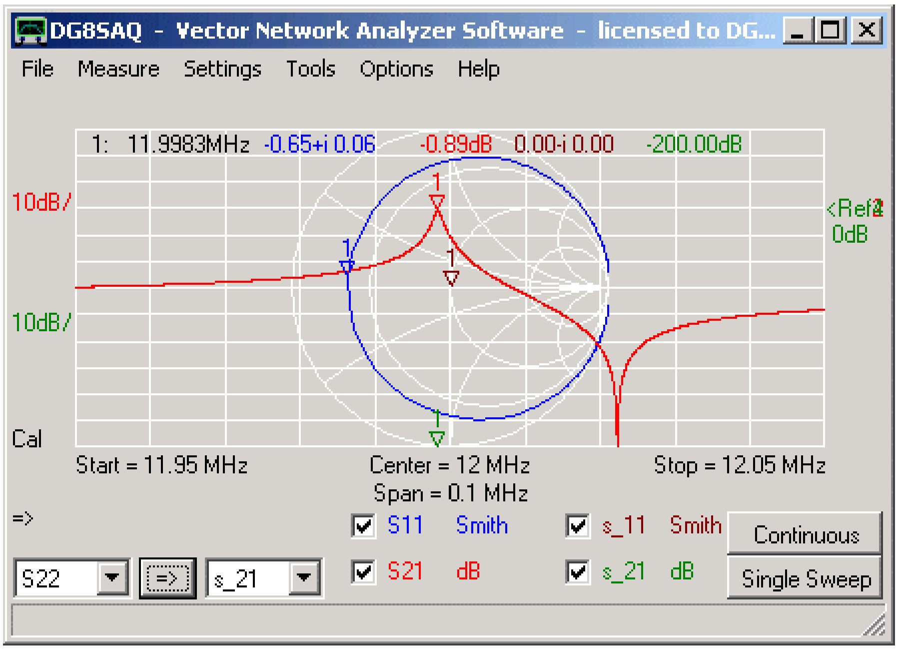


Any correction scheme other than the one shown above will produce wrong transmission measurement results! This is due to the fact that transmission and reflection do NOT share the same ground.

2. Perform a transmission (S21) and a reflection (S11) measurement of your crystal with one crystal terminal being input and the other being output.

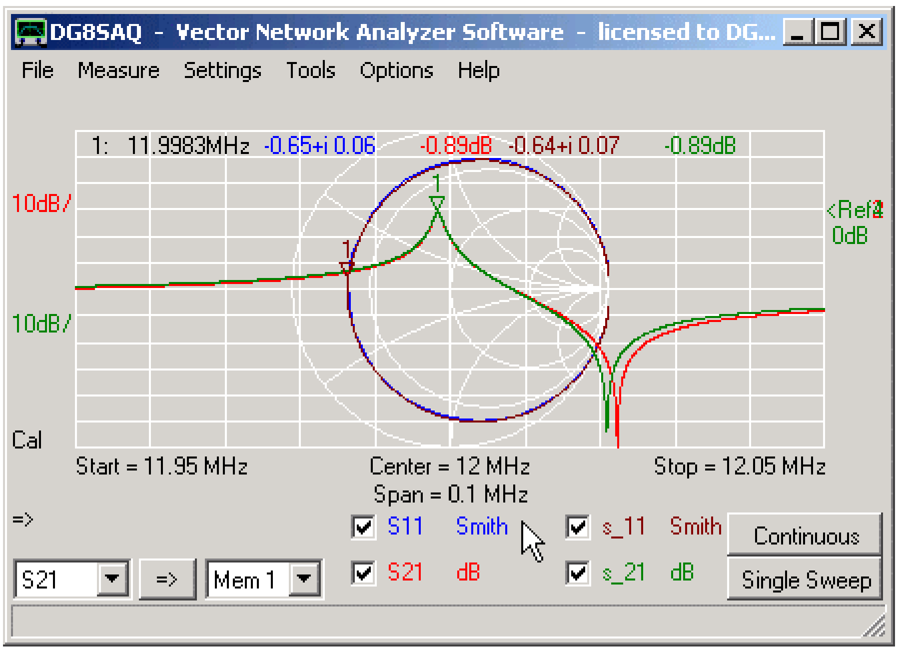


This is what the measurement result should look like for a 12 MHz crystal:



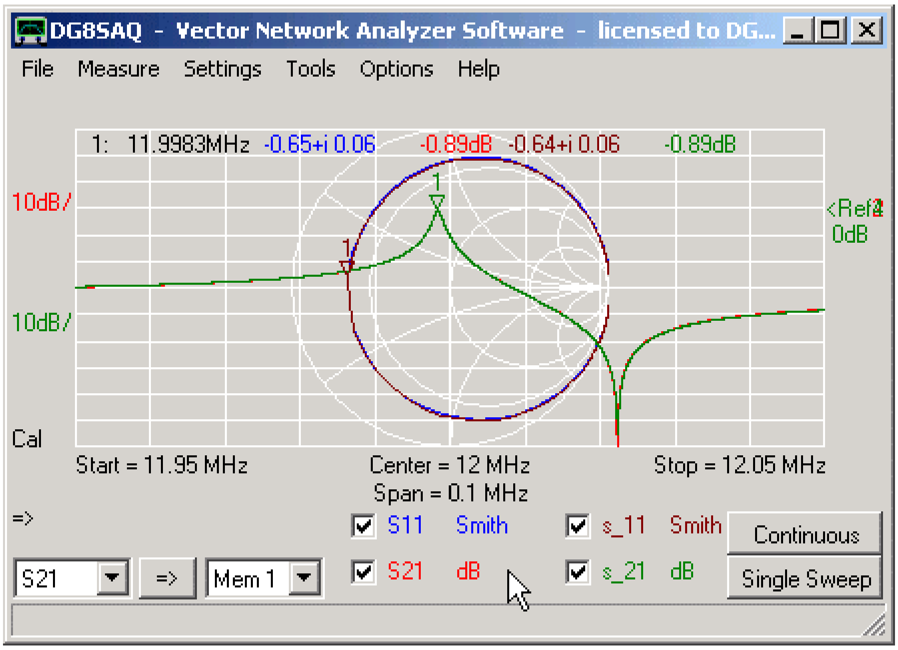
In transmission one can clearly see the serial resonance (marker) as well as the parallel antiresonance.
Make sure, to display 3-port memory space s_21 and s_11 before moving to the next step. 3-port memory spaces s_11 and s_21 will be used to store and allow to display the model simulation in reflection and transmission.

3. Start the "Crystal Analyzer" tool via the main tools menu, select **source = S11** and press the **auto-optimize** button. When the tool is finished you will see this simulation result:



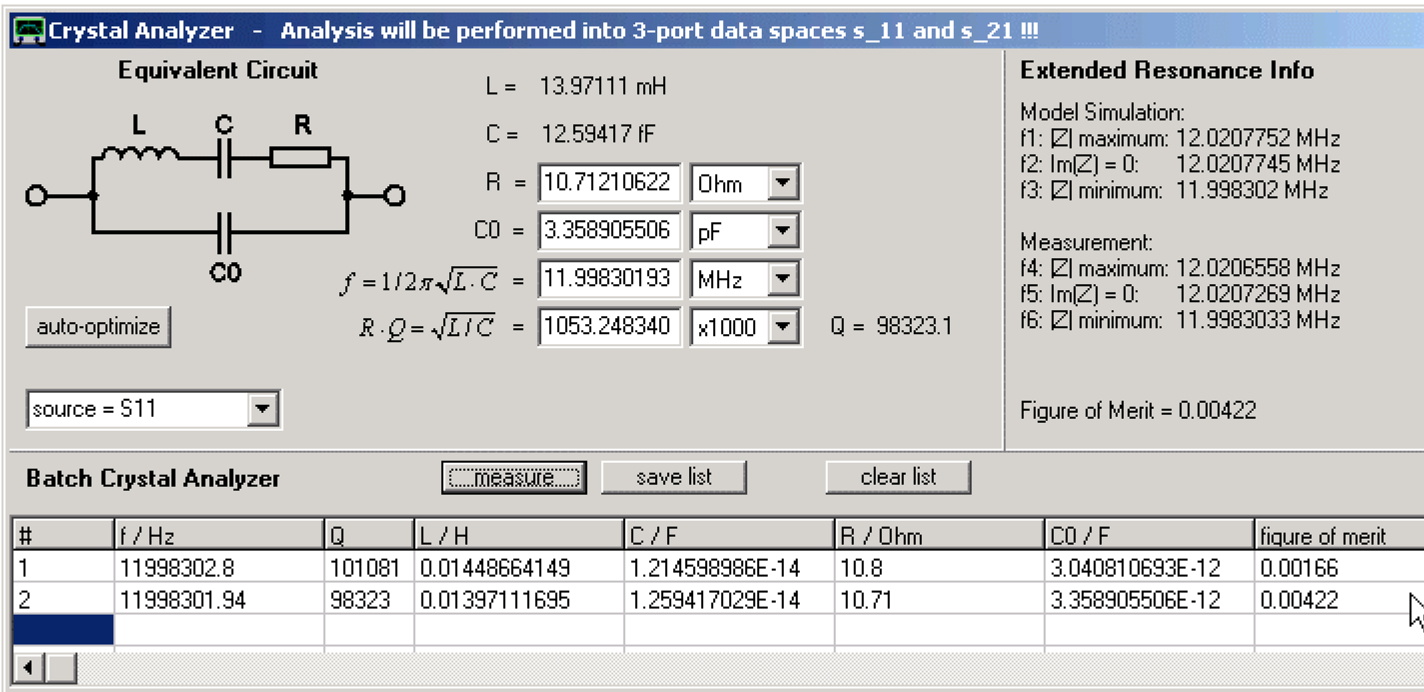
The model perfectly matches the reflection measurement S11. Model parameters can be read off the Crystal Tool window. In transmission (S21) there is a mismatch visible for the parallel antiresonance. Note, that the transmission simulation is mathematically derived from the reflection fit. The reason for this deviation in transmission will be discussed below.

4. Make sure that now **source = S21** and the correct **test fixture impedance** (must be identical on input and output) is selected before auto-optimizing. Now, the model simulation looks like this:

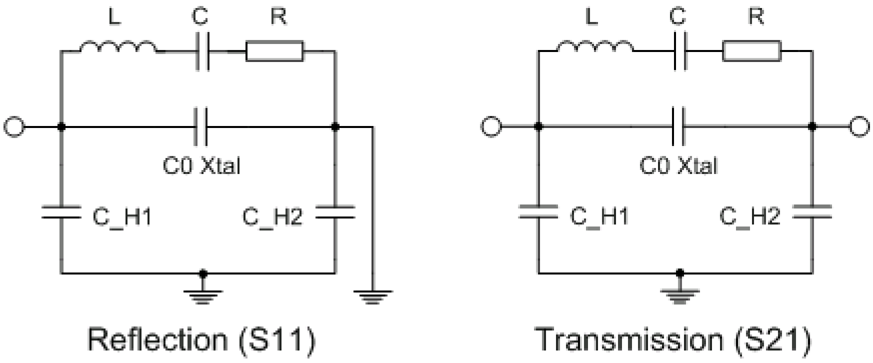


Now, the parallel antiresonance of the transmission (S21) measurement is perfectly modeled, while that of the reflection measurement is off (what you cannot really see in above Smith chart, though, since we missed to place a marker onto the parallel antiresonance).

Next, we compare the extracted model parameters of above two optimizations:



Above, line #1 shows the model parameters for the transmission (S21) fit, while line #2 shows the reflection (S11) fit parameters. The most apparent difference is that of the parallel capacitance C0. While in transmission C0=3.04pF, in reflection C0=3.36pF, which is 0.32pF higher. The increased parallel capacitance shifts the parallel antiresonance to lower frequencies. But what's the reason for this C0 increase? It lies in the fact that the model for a real crystal is a bit more complex than assumed above:

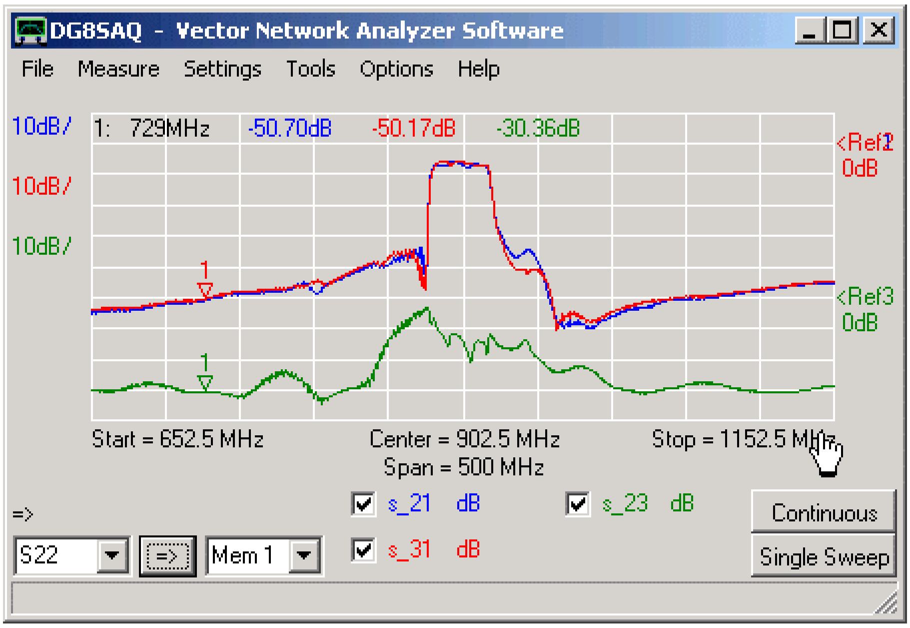


As shown above, a real crystal consists of the crystal disk and a holder. The holder pins each have a capacitance to ground (C_H). While in transmission the series circuit of the holder capacitances add to the crystal parallel capacitance (C0_Xtal), C_H2 is effectively shorted in a reflection measurement and C_H1 alone adds to C0_XTal to yield an effective C0. Assuming that C_H1 and C_H2 are approximately identical, C0 for a reflection measurement should be 0.5*C_H larger than for a transmission measurement. We can thus interpret our above experiment such that the holder capacitances are C_H=2*0.32pF=0.64pF and C0_Xtal=3.04pF-0.32pF=2.72pF.

Three port RF devices like baluns or SAW-filters with e.g. balanced outputs confront the user with a complex characterization task:
On the one hand, one would like to know e.g. the insertion attenuation from single ended input to differential outputs and the complex differential output impedance (**differential mode**). On the other hand, the **common mode attenuation** is of interest.

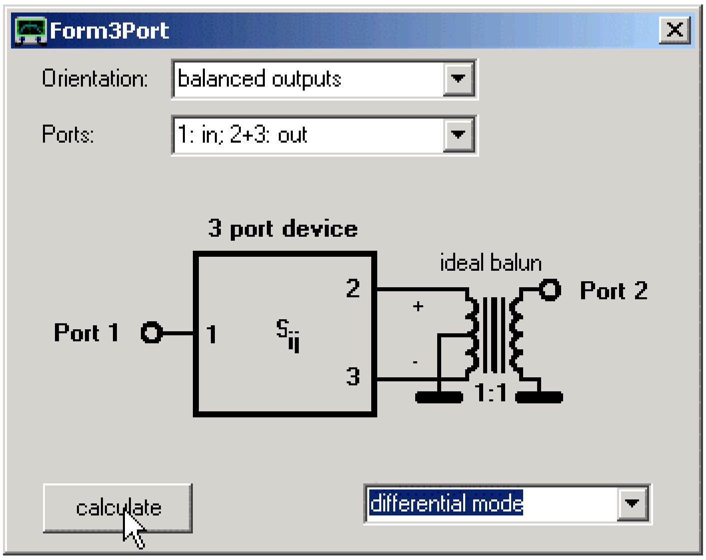
The 3-Port analyzer tool allows to perform both characterizations on a set of 3-port S-parameters.
The following is a step by step example of performing a 3-port analysis:

1. Import or measure the 3-port S-Parameters of your DUT.:



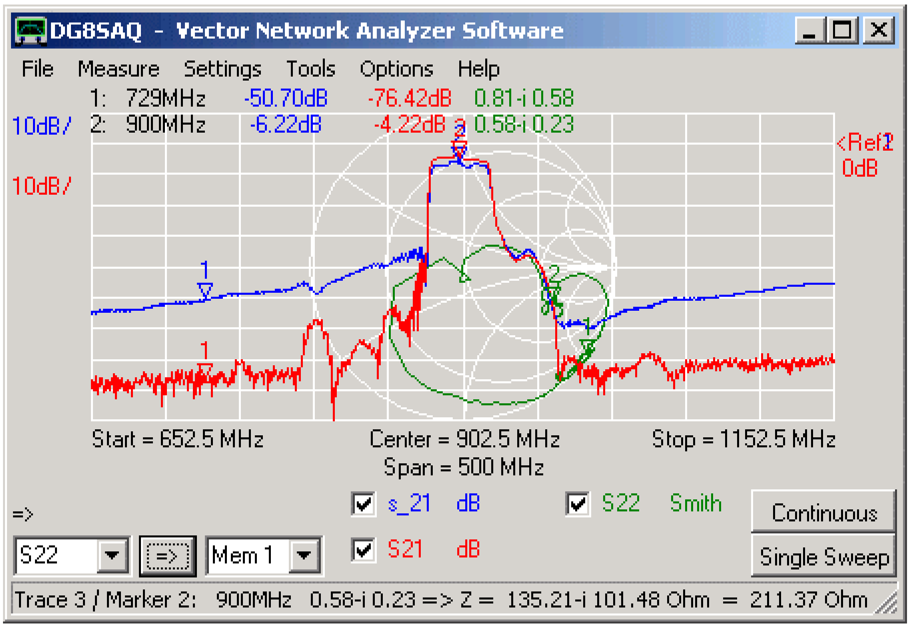
Note, that all signal paths from any DUT port to any other DUT port including all reflections must be characterized. The above imported 3-port S-parameters of a GSM RF filter are courtesy of EPCOS. Filter port 1 is the single ended input and ports 2 and 3 are the differential filter outputs. The above plot shows highly symmetric filter transmissions from port 1 to 2 and from port 1 to 3 (s_12,s_13, note the underscore!).

2. Start the 3-port analyzer



Select the desired mode of operation (1 = in, 2&3 = balanced out, **differential mode**) and press calculate. The result of this simulation will be stored as 2-port S-parameters into the 2-port memory spaces S11,S21,S12 and S22 and into the 2-port S-parameter buffer (useful for later matching analysis). This simulation is equivalent to combining the two output signals with an ideal 1:1 balun and thus turning the 3-port into a 2-port.

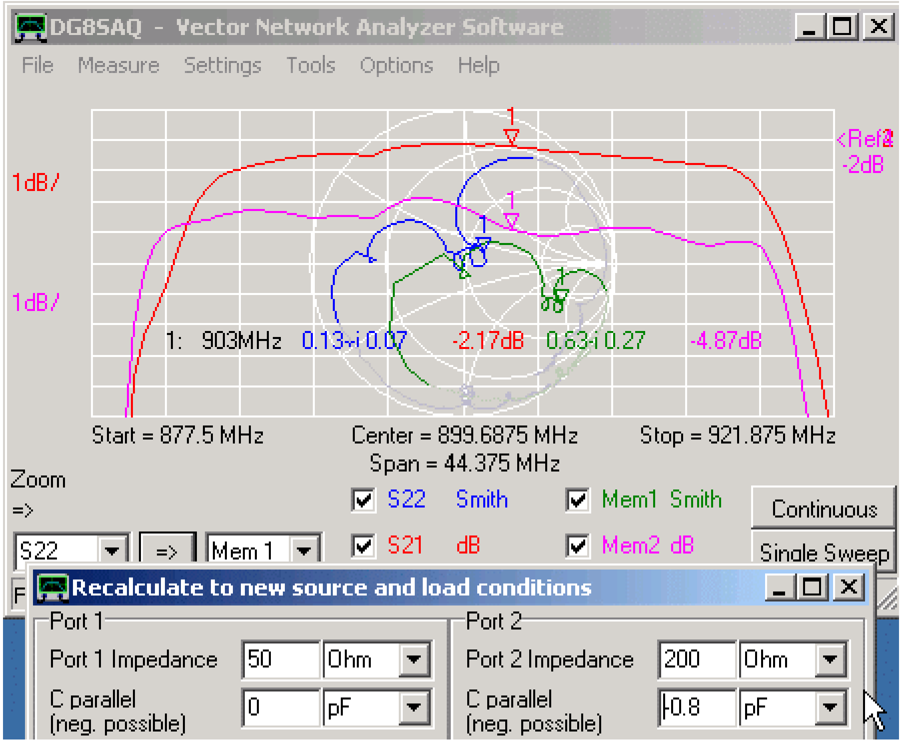
3. Observe the result in the main window.



Note that the simulated 2-port insertion attenuation S21 is almost 3dB lower than the 3-port one s_21, because the signals of DUT ports 2 and 3 are being combined now.
Note, that the attenuation level improves dramatically, since the almost identical feedthrough signals 1-2 and 1-3 cancel each other at the balanced port.
Also note, that the differential output impedance (parallel equivalent circuit) of the filter is 200 Ohms (see bottom status line with marker info).

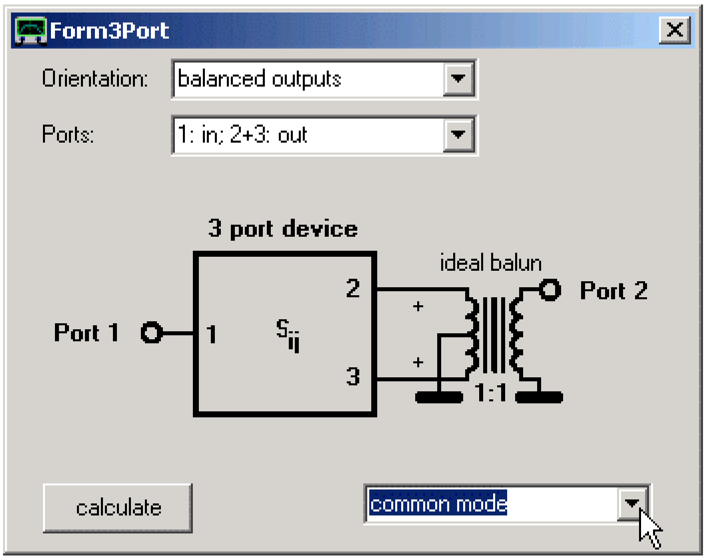
4. Perform a matching analysis

By using the matching tool, the simulated 2-port transfer characteristic can be recalculated to what it would look like if the filter was terminated with 200 Ohms instead of the 50 Ohms of the measurement system:

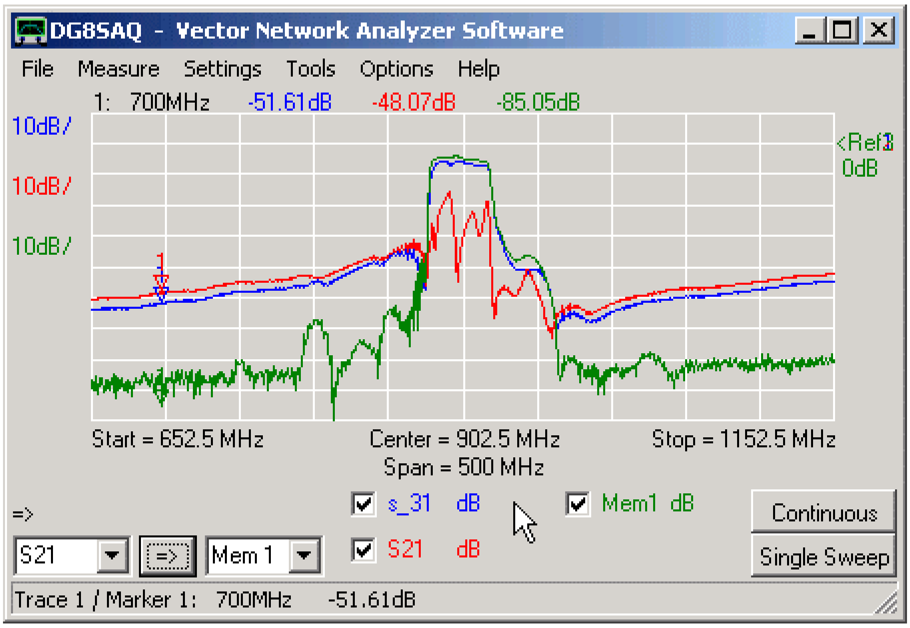


For comparison, the original 50 Ohm terminated data has been stored to Mem1 and Mem2 respectively. Note, how by proper termination the insertion attenuation can be further reduced.

5. Perform a common mode analysis:

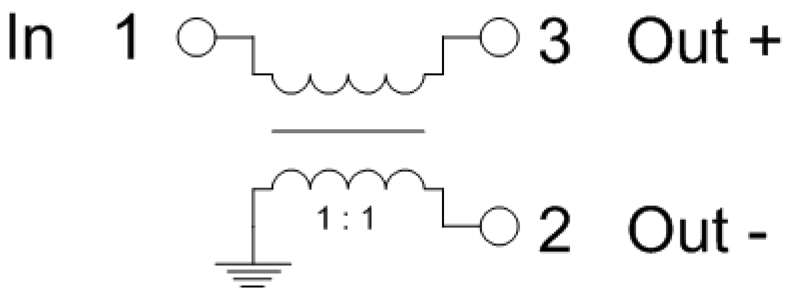


Select the desired mode of operation (1 = in, 2&3 = balanced out, **common mode**) and press calculate. The result of this simulation will again be stored as 2-port S-parameters into the 2-port memory spaces S11,S21,S12 and S22 and into the 2-port S-parameter buffer. The common mode analysis is equivalent to connecting the normally differential outputs in parallel, thus turning the 3-port into a 2-port with the two output ports connected in parallel.



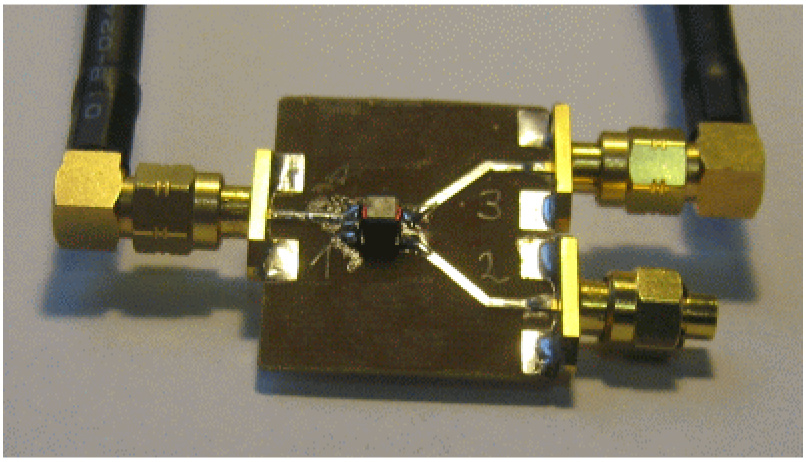
Note, that the electrical feedthrough of the common mode (S21) is 3dB higher than the s_31 signal of the original 3-port, since s_31 and s_21 are being added.
Also note, that there is signal cancellation in the filter passband now, as in the passband region s_21 and s_31 exhibit a 180° phase offset to each other.
For comparison, Mem1 shows the differential mode signal.

A BALUN is generally a passive device with 3 hot terminals and a ground terminal, which transforms an unbalanced signal relative to ground to a push-pull signal pair:

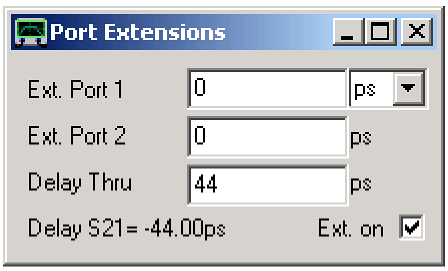


1:1 BALUN built from a 3dB coupler

Once a BALUN is built, the question arises how to characterize it, e.g. what is its impedance transformation ratio, insertion loss, balance and common mode suppression.
All these questions can be answered once the **3-port S-matrix** of the BALUN is known. This can be measured with the VNWA 3-port S-parameter acquisition menu. Before measuring the 3-port S-parameters of the BALUN, clearly mark the BALUN ports with **unique port numbers** (e.g. 1,2,3):

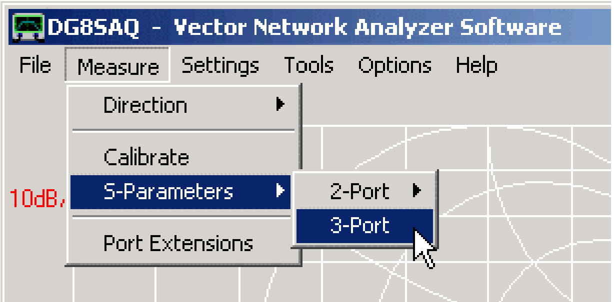


Next, after proper calibration of your VNWA, you have to determine the electrical length of evey line from the calibration plane to the BALUN terminal on the test board. To do so, switch on the port extensions and enter the delay of your thru calibration standard:

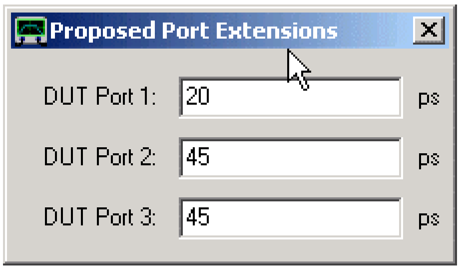


Next, measure the reflection coefficient of every port while generating a short circuit at the BALUN terminal and then adjust the port extension = electrical delay such, that the short shows up as a short in the Smith chart (dot on the left edge. Write down the values for all three ports as these will be needed for the 3-port measurement.

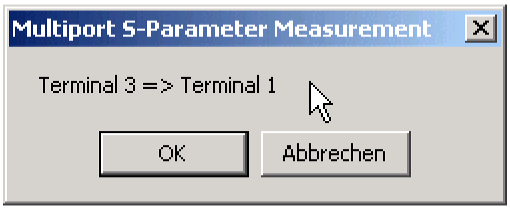
Next start a 3-port measurement:



A popup window will ask for the delays you have just determined. Enter those...

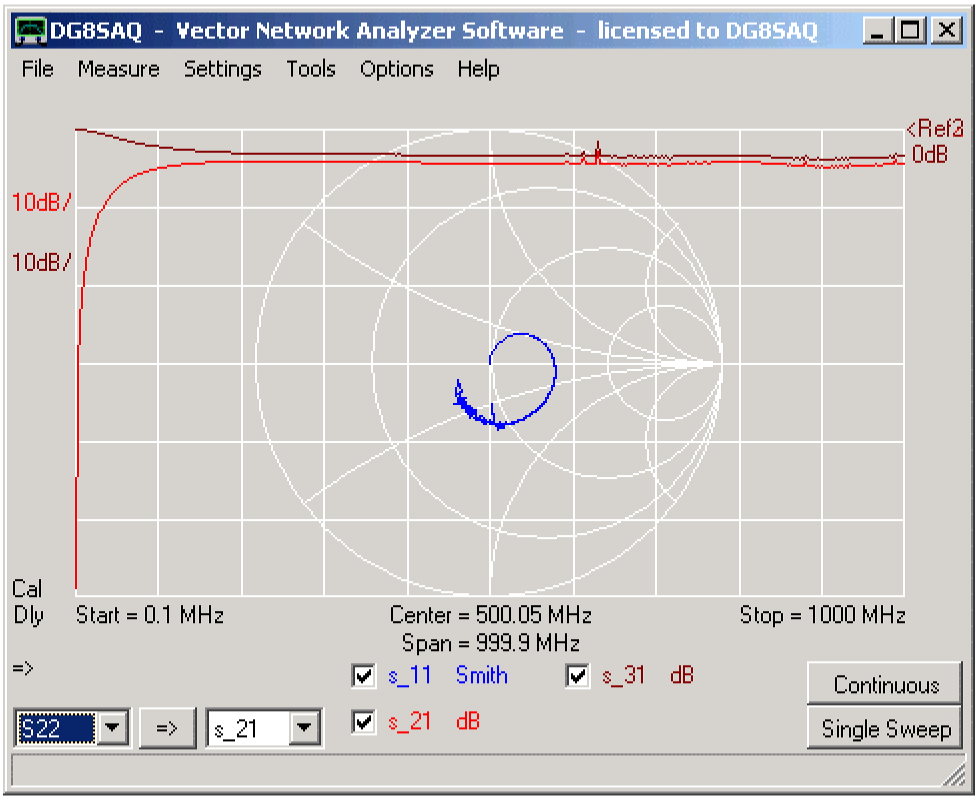


... and close the window. Now you will be asked to connect your 3-port DUT to the VNWA in various ways, e.g.



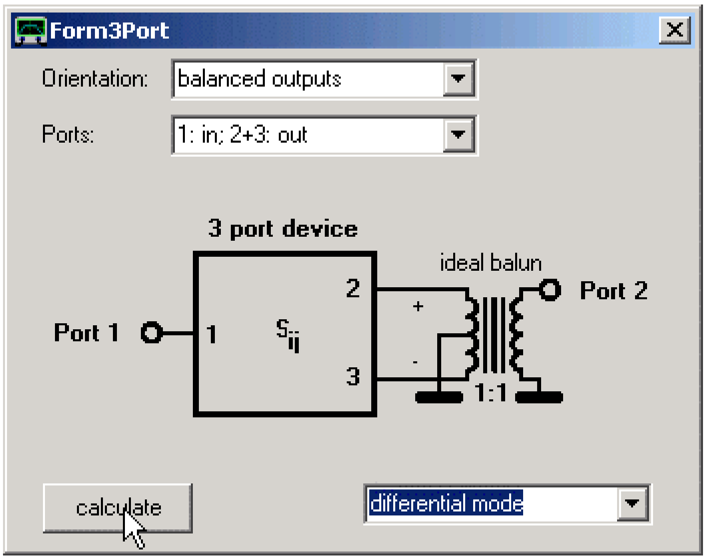
The above window asks to connect the VNWA TX port to DUT port 3 and the RX port to DUT port 1.
The unused DUT port (here port 2) must always be terminated with a 50 Ohms load.
In the course of a 3-port measurement, you will have to measure from every port to every other port of your DUT which involves a lot of reconnecting. The reconnecting effort can be reduced by means of an S-parameter test set.

Once the measurement is done, the DUT 3-port S-matrix resides in the 3-port memory spaces s_11, s_21,...,s_33:

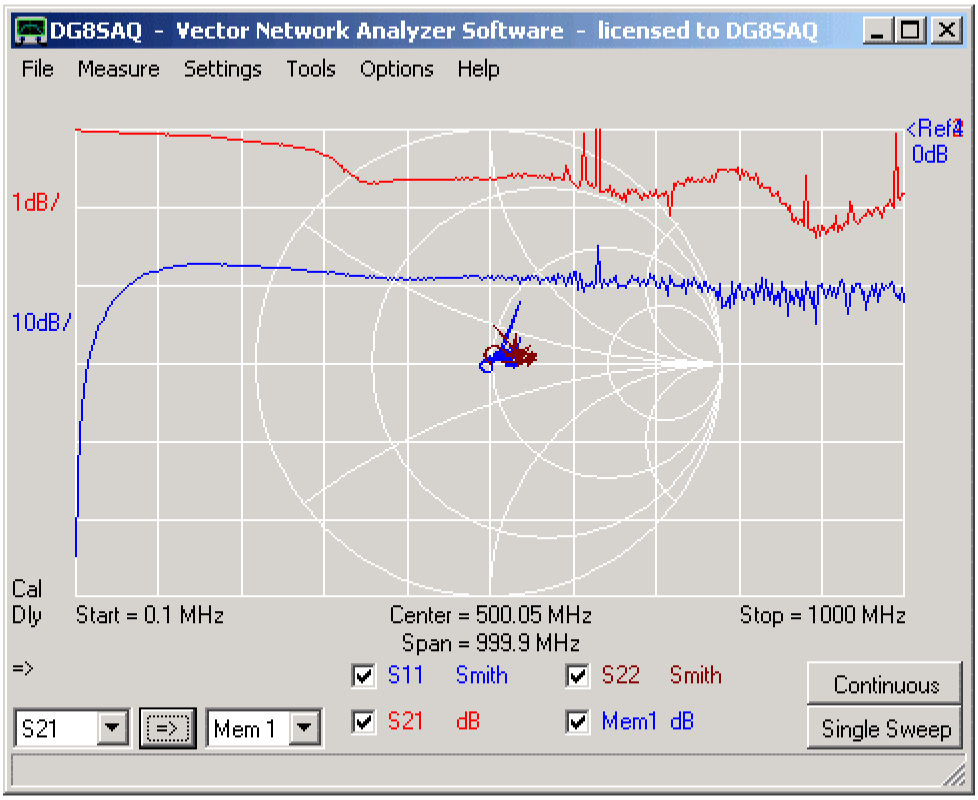


The S-matrix may be exported to an s3p file for later analysis.

In order to analyze the BALUN open the 3-port analyzer tool, select proper port topology (e.g. 1 in, 2+3 balanced out), select **differential mode** ...

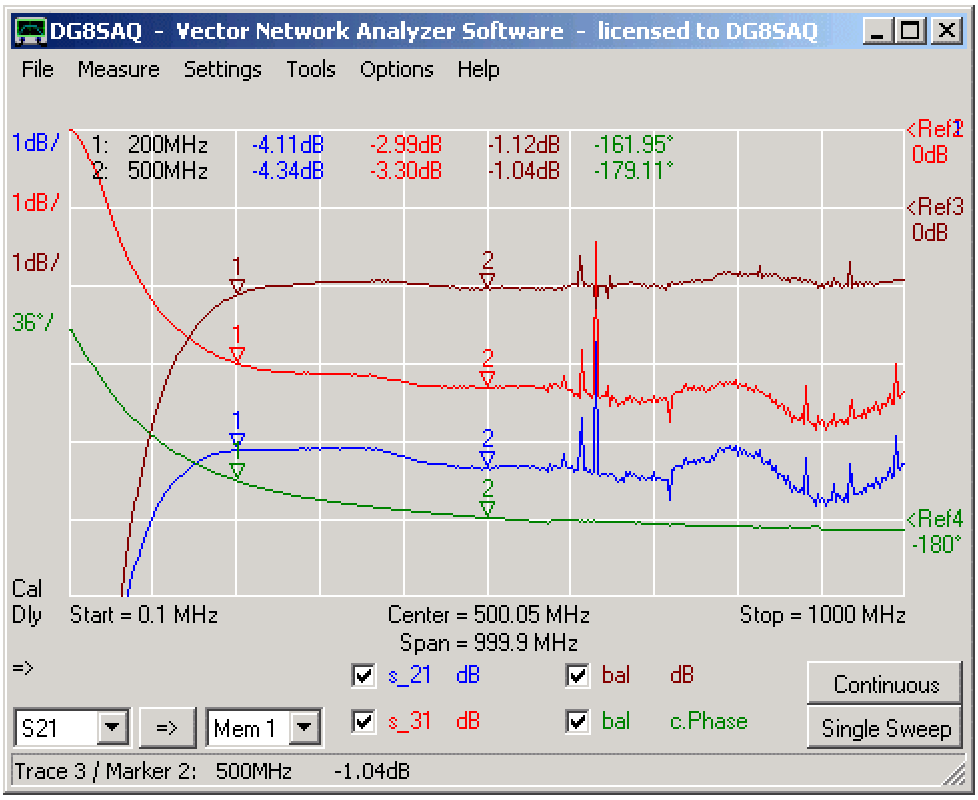


and press **analyze**. The tool will calculate the differential mode response and store in into memory spaces S11, S21, S12,S22:



It appears that the BALUN works with very low loss drom 0.1MHz to 1GHz (see S21 trace) and there is an almost perfect match to 50 Ohms on both ports, the single ended input (S11 trace) and the balanced output (S22 trace). Mem1 in above screenshot contains the S21 result when chosing **common mode** instead of differential mode. This means our BALUN has a common mode suppression of around 20dB over the entire frequency span.

Now, does this mean our BALUN shows perfect BALUN action over the entire measured frequency span? No, we haven't looked at the balance yet, i.e. we haven't looked if the port 2 and port 3 signals actually exhibit identical amplitudes and a +-180° phase shift. We can investigate this by comparing s_21 and s_31 either by inspection or by generating a custom trace with the function s_21/s_31 as is shown below for the two custom traces named bal. For a perfect push-pull operation s_21/s_31 should be -1.



Here we see that even though, the BALUN exhibits a very low total loss down to 0.1MHz, it is only usable down to about 200MHz as below this frequency the amplitude and phase balance degrades rapidly.

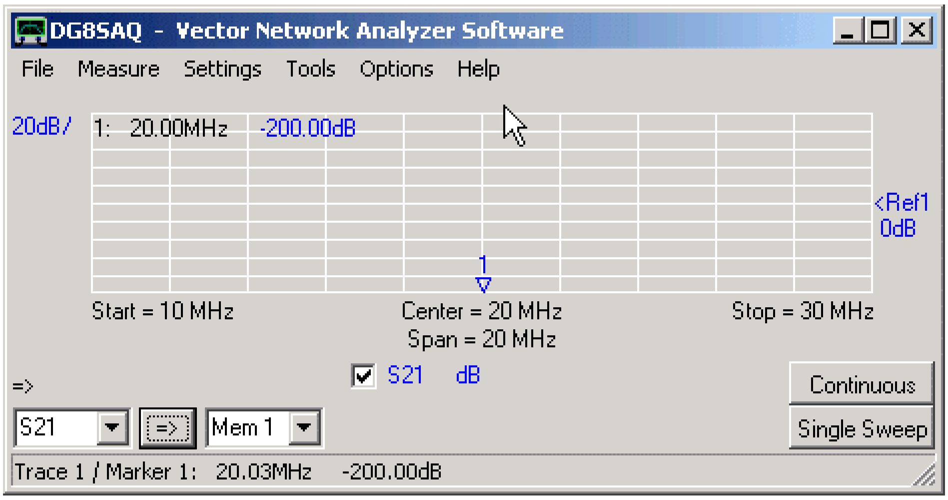
The VNWA software allows to do frequency sweeps in the following sweep modes:

1) **Linear sweep**: The frequency grid spacing is equidistant.
2) **Log sweep**: The grid spacing increases from point to point such that a logarithmic frequency axis is created. Useful for Bode plots.
3) **Listed sweep** The user can specify up to 20 different linear sweep segments which build up the frequency grid.

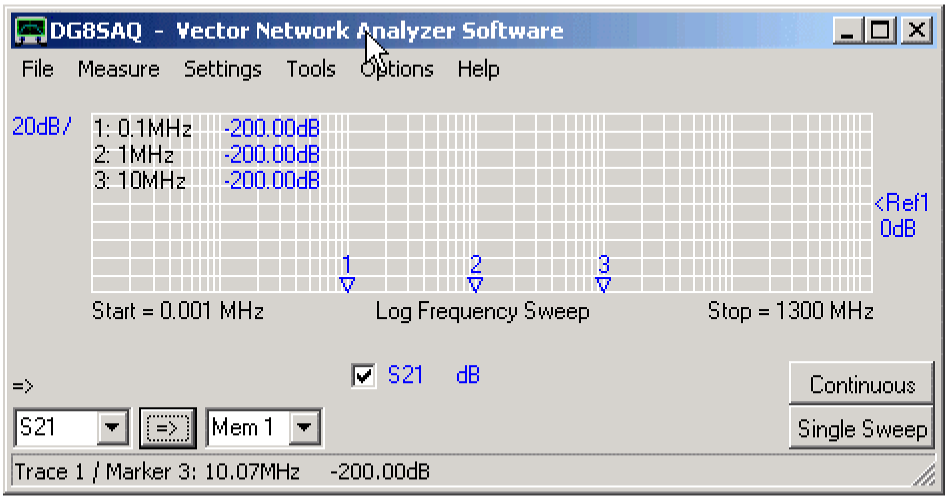
The **"sweep mode"** is either selected in the frequency input window opening on double-clicking the main window's frequency labels or by right-clicking the center or span label.

Examples:

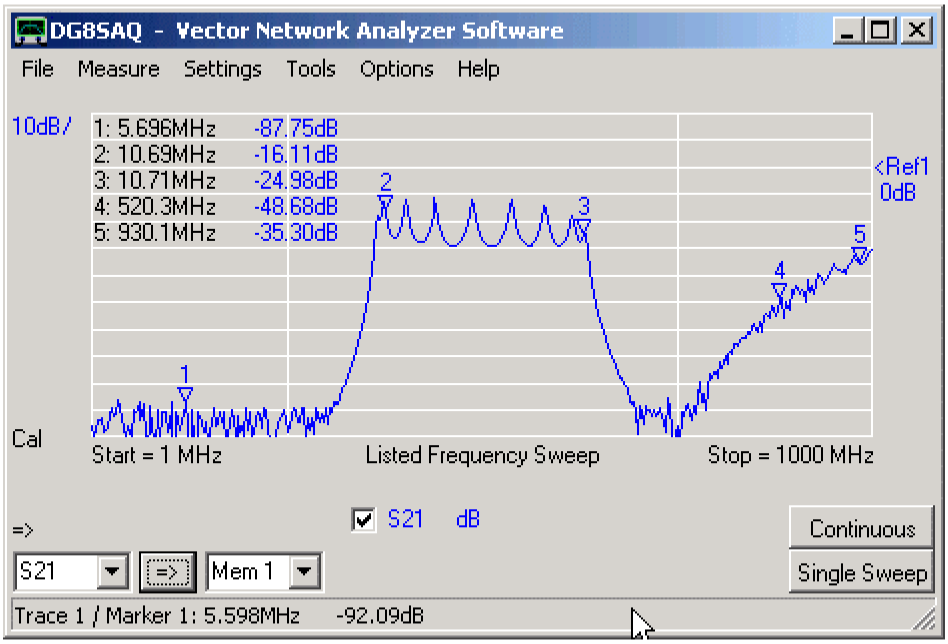
The screenshot below shows **linear sweep** settings, where the center frequeny is in the middle of the screen.



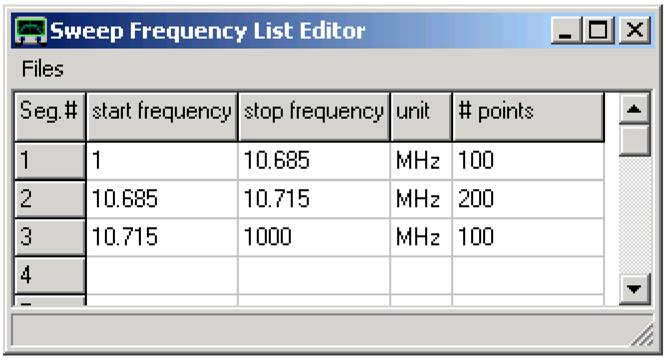
The screenshot below shows a Bode type plot with **logarithmic frequency sweep** settings. Looking at the markes, it is seen that with every full frequency grid progression, the frequency increases by a factor of 10. Observe, that instead of the center frequency, "Log Frequency Sweep" is displayed. The sweep mode can be changed by right-clicking this label.



The screenshot below shows an example of a **listed sweep**. The DUT (a narrow band crystal flter) is swept from 1 MHz to 1000 MHz using 400 data points, yet the 20 kHz wide filter passband is clearly resolved (see markers). For a linear sweep with the same point number, the spacing between frequency points would be 2.5 MHz and the filter passband would not be seen.



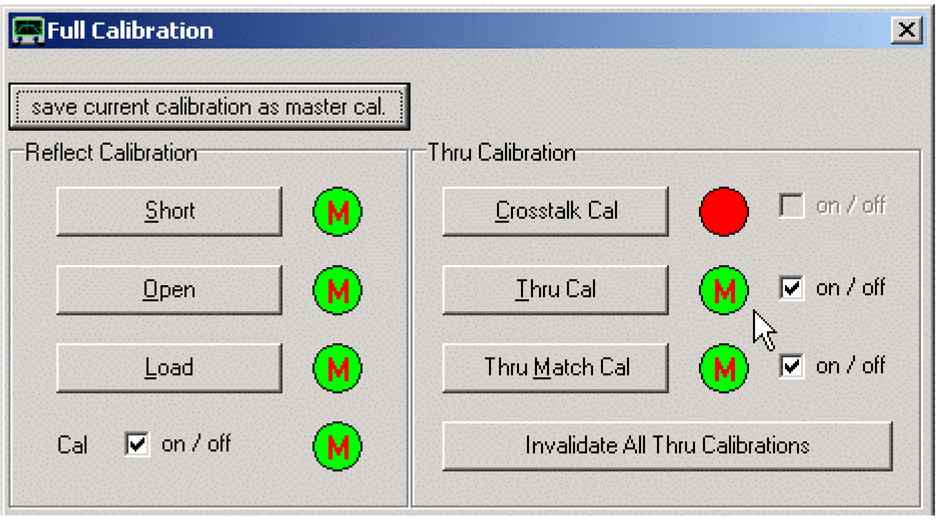
This is achieved with the following frequency list:



The frequency list specifies 3 frequency segments, which are swept consecutively and displayed simultaneously. Note, that in the VNWA main window segment boundaries are marked with vertical grid lines.

▸▸▸ **Hint:** The sweep **"sweep frequency list editor"** opens automatically, when "listed frequency sweep" is selected for the first time. Right-click a line to insert or delete a segment.
▸▸▸ **Hint:** The sweep **"sweep frequency list editor"** can be reopened by right-clicking or double-clicking the "Listed Frequency Sweep" label below the main graphics.
▸▸▸ **Hint:** Frequency lists can be stored and reloaded to/from ***.csv files**, which can also be processed with Excel.

The VNWA software has implemented several error correction models, which can be selected in the lower part of the calibration window.



Real Time Display Calibration Options determine the error correction model to be used for **real time sweeps** continuously displayed in the main window.

S-Parameter Acquisition Calibration Options determine the error model used for the main menu command "measure" "2-port S-parameters".

▸▸▸ **Note:** For ***one port*** measurements these options are of **no importance**.

For ***two port measurements***, the following **error correction models** are available:

- simple thru correction
The transmission data is simply corrected by division with the thru calibration measurement.

- enhanced thru correction with ideal load match
The transmission data is additionally approximately corrected for errors arising from the TX port not having 50 Ohms source impedance. Note, that a reflection calibration as well as the measurement of S11 is necessary to apply this correction to S21.

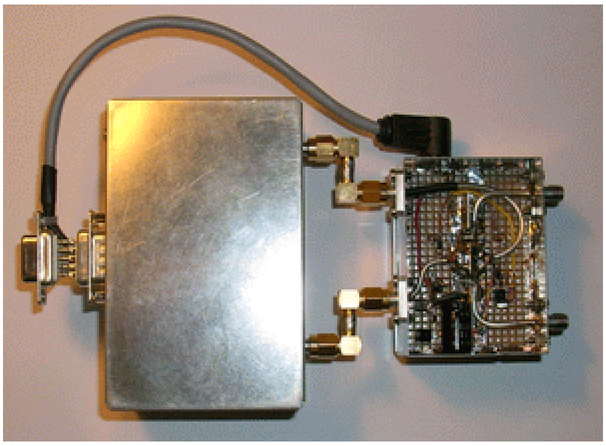
- enhanced thru correction
The transmission data is additionally approximately corrected for errors arising from the RX port not having 50 Ohms load impedance. Note, that a reflection calibration, a thru match calibration and the measurement of S11 are necessary to apply this correction to S21.

- 5,6 or 12 term correction
This is the best available error correction scheme. All transmission and reflection data are exactly corrected for imperfect TX source and RX load impedances. A full reflect calibration as well as a thru and a thru match calibration are required (5 term correction). If an isolation calibration is done, it will be extended to a 6 term correction. If using an S-parameter test set and both measurement directions are calibrated, it will be extended to a 12 term correction model.

▸▸▸ **IMPORTANT:** In order to apply this error correction, all four S-parameters (S11,S21,S12,S22) must be measured.
- If **"*allow* full 5,6 or 12 term correction"** is selected, it is only applied if the user decides to display and thus measure all 4 S-parameters. Note, that VNWA usually measures only those S-parameters which are displayed in order to save measurement time.
- If **"*enforce* full 5,6 or 12 term correction"** is selected, VNWA will always measure all 4 S-parameters, even though they might not be displayed.

▸▸▸ **Note:** The term **"if available"** in above menu denotes the circumstance, that the error correction model might not be available if the user hasn't performed all necessary calibration measurements or if not all necessary

S-parameters are measured. In such a case, the software will fall back onto the best *available* correction.

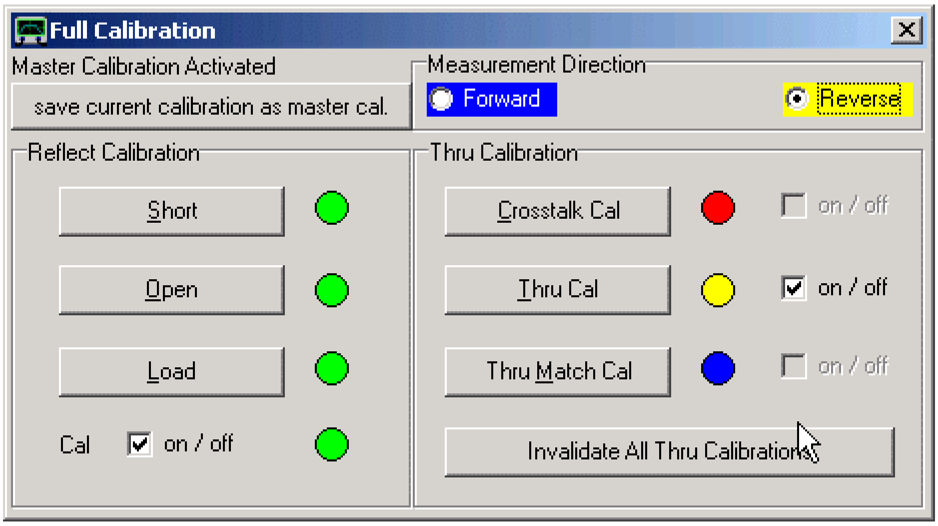


The VNWA software supports usage of an **S-parameter test set**. The test set contains several switches, which interchange output and input of the VNWA. This actually allows for automatic change of the signal flow direction, which is useful for measuring S-parameters of two port devices. In order to measure S12 and S22, the DUT doesn't need to be turned around manually any longer, but the test set simply exchanges input and output of the VNWA instead.

In LPT mode, the test set is controlled via the parallel printer port terminal **1/"Strobe"** of the Sub D-25 connector. In USB mode, the control signal for the S-parameter test set is available on the USB_VNWA Interface at pin 3 / J26 or alternatively pin 7 / J5 (Sub-D9) provided the appropriate resistors are in place.

Test set support is activated in the instrument settings tab of the "options"-"setup" main menu.

If activated, **two independent calibrations** for the two measurement directions will be required. This is indicated in the calibration menu window by the additional choice of **"Forward"** and **"Reverse"** measurement direction.

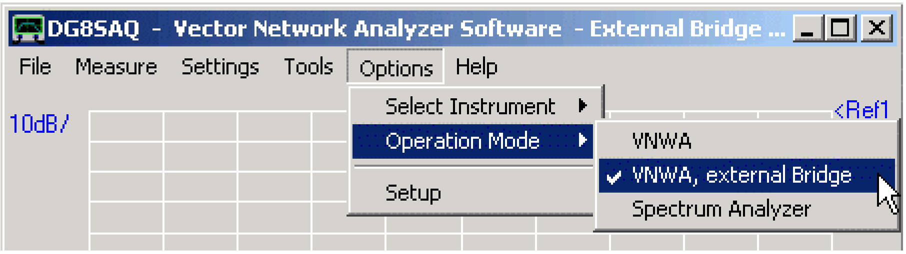


The two calibration directions are indicated by the colors blue (= forward) and yellow (= reverse). The calibration indicator lamps can show four colors:

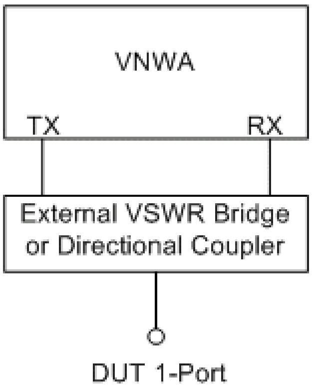
red: no calibration done in any direction
blue: calibration done in forward direction only
yellow: calibration done in reverse direction only
green: calibration done in both directions

Using a test set will also allow convenient usage of a 12 term error correction model in realtime.

The VNWA hardware can also be operated with an **external reflection bridge**. To do so, select **Operation Mode - VNWA, external bridge**.



The "bridge" can be as simple as an **SMA-Tee**, or as sophisticated as a **directional coupler** or a **Wheatstone bridge**. The bridge is connected with its signal input to the VNWA TX port and the bridge voltage output is connected to the RX port as can be seen below.



Clearly, **only reflection measurements are possible** in this setup.

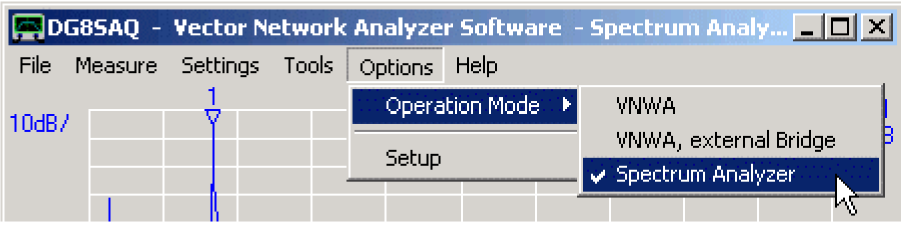
The setup can be SOL calibrated in the normal manner. The calibration standards are to be connected to the DUT port of the bridge.
Measurement results are **displayed in the S11** memory space. Note, that the memory spaces **S21 and S12 contain meaningless results** in this case and should not be displayed.

▸▸▸ **Note:** The advantage of this setup lies in the fact, that an **attenuator pad** can be inserted between bridge and RX port. This might be necessary, when measuring **antenna impedances in strong BCI environments**. In such environments the BCI signals might well saturate the internal VNWA bridge mixer. An external bridge followed by e.g. a 20dB attenuator in front of the RX port avoids mixer saturation. Of course, the desired reflect signal is also attenuated by 20dB, but there is still enough dynamic range margin to obtain reflection data with reasonable accuracy. You could aslo use the internal bridge, connect your 20dB attenuator to the TX port and calibrate at the attenuator. This would also attenuate the BCI signals by 20dB, but the desired reflection signal would be attenuated by 40dB, since it travels through the attenuator twice (forth and back).

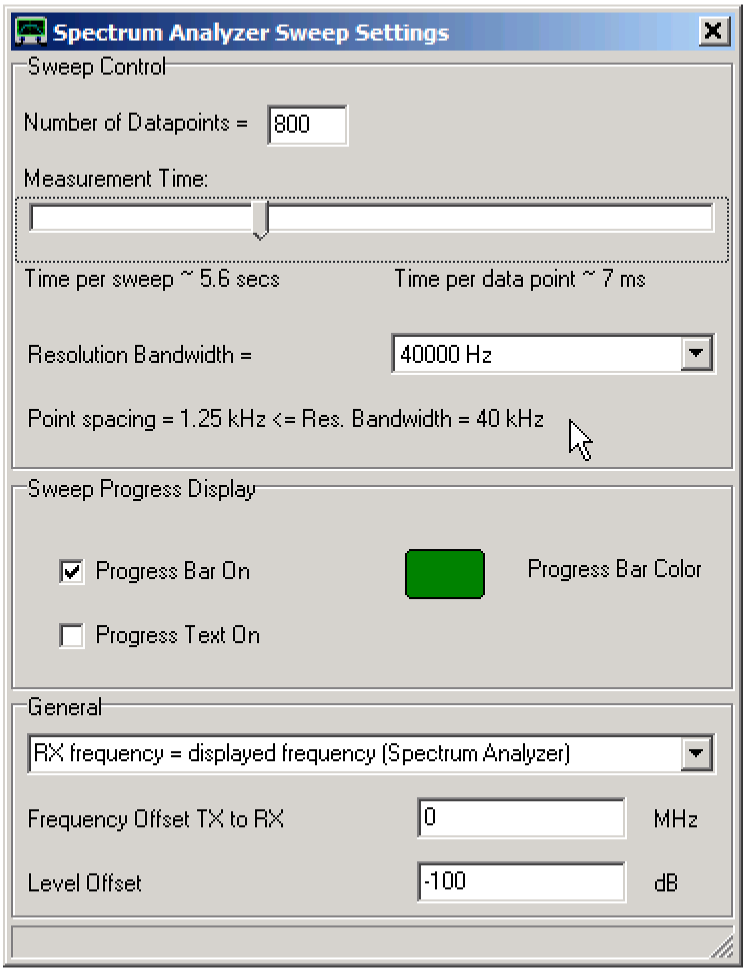
Spectrum Analyzer

The VNWA can also be used as a rudimentary **spectrum analyzer**.

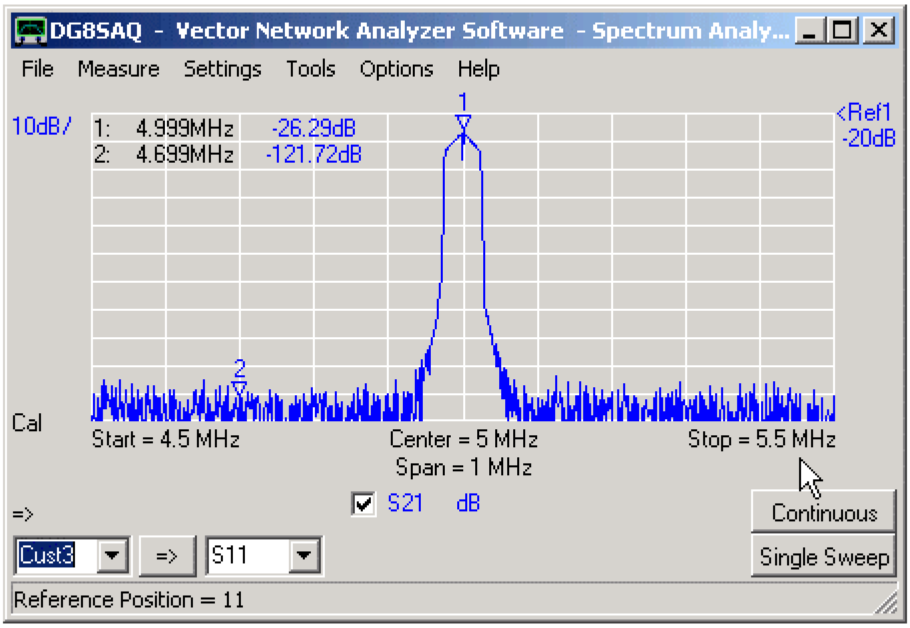
In order to activate this mode, select **"Options"-"Operation Mode"-"Spectrum Analyzer"**.



Next, you need to specify the sweep parameters. The following picture shows the settings used in the following example.
▸▸▸ **Warning:** Note, that in SA mode (unless used with tracking generator) the **frequency point spacing must be smaller than the resolution bandwidth**! If this is not the case, spectrum gets lost, e.g. there might be spectral lines but you won't see them. If settings are inappropriate, the **status text** (left of the mouse arrow in above screen shot) will turn red.

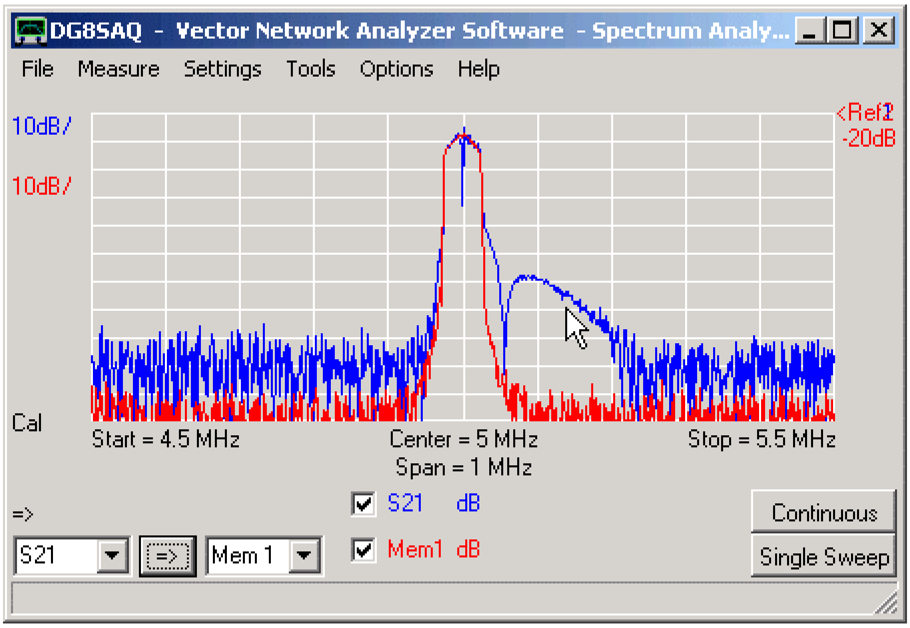


A 30mVpp sine wave signal (= maximum level, that doesn't create overload yet = 30mVpp = -26.5dBm =2.25uW) is fed into the VNWA RX port.



Note, that by choice of the level offset, **signal power** can be read off directly in **dBm**, e.g. from marker 1.
Note, that a dynamic range better than 90dB is achieved.
▸▸▸ **Note:** For most sound cards, the dynamic range is limited by the DC offset of the sound card ADCs. This DC offset can be efficiently removed by performing a **crosstalk thru calibration**, which has ben done in all displayed measurements.
The **shape of the spectral line** is determined by the frequency response of the sound card. As zero frequency is in the center of the spectral line and sound cards are AC coupled, there is a small notch in the center of any spectral line, which might or might not be seen dependent on choice of the frequency grid.

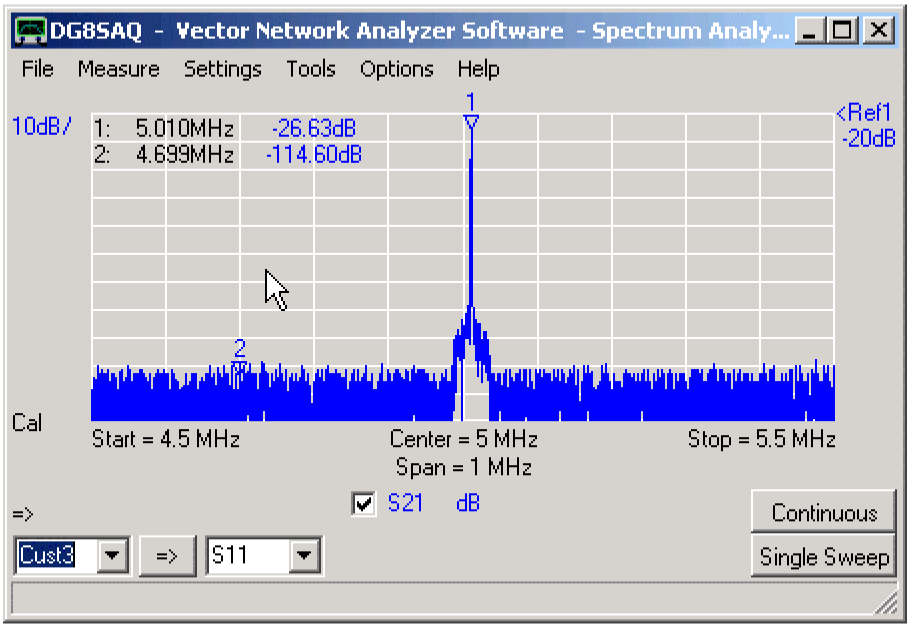
The picture below, compares the measurement results for two different **sweep time rates**. The red trace is identical with the above picture.
The blue trace was measured with the fastest available sweep rate (0.2ms / frequency point)



Note, that decreasing the sweep rate increases the noise floor.
Also note, that if you sweep too fast, the IF filter (=sound card) starts "ringing" (feature above the mouse pointer) due to the the limited bandwidth.

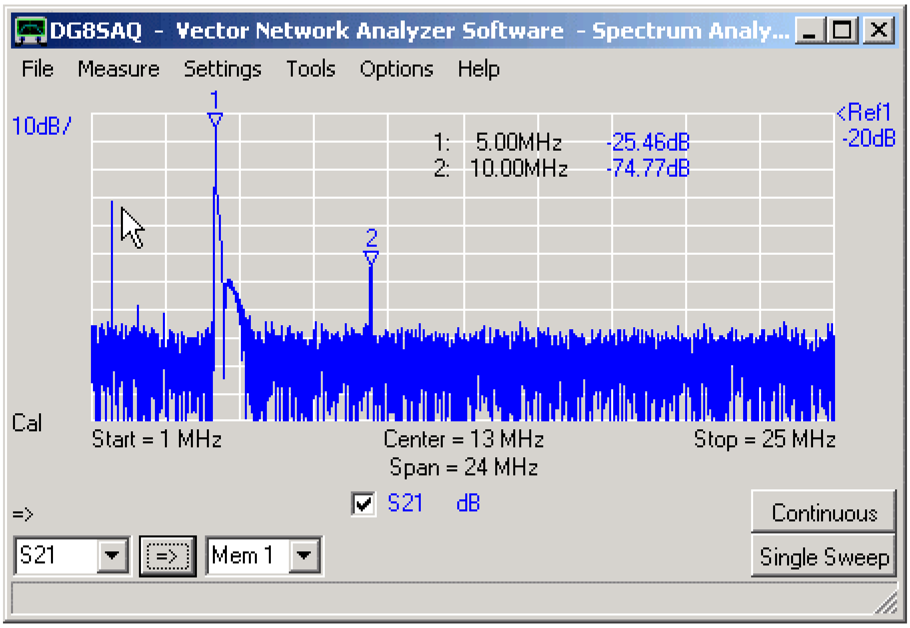
The picture below shows again the same signal with a lower resolution bandwidth and slow sweep rate.
Resolution bandwidth control is achieved by changing the sample rate and utilizing the fact, that the resampler of

the Windows sound system provides a special anti aliasing filter fitting for every sampling rate.



Note, that the noise side humps are likely artifacts of the Windows anti-aliasing filter algorithm.

Below, again the same signal is swept in a wider frequency span.



Due to the fast sweep rate, ringing is visible on the right skirt of the main peak at 5 MHz. The first harmonic is clearly visible at 10 MHz. At lower frequencies, spurious signals are being seen, which come from unfiltered aliasing frequencies of the DDSes. Some of them can be identified by their width, which is half of the regular peak width.

Tracking Generator

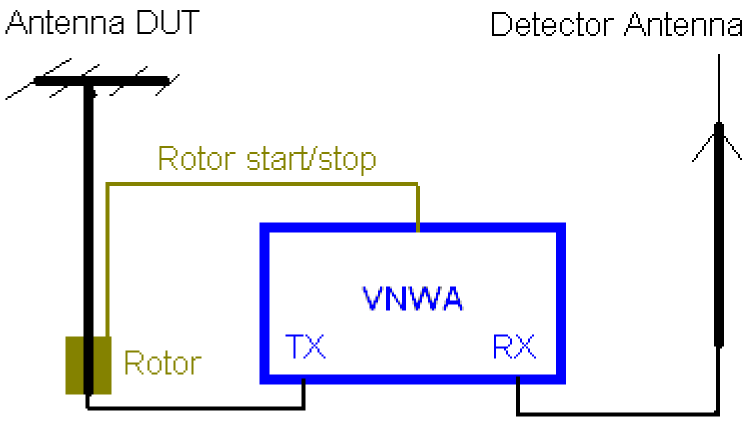
Note, that in spectrum analyzer mode, the TX oscillator is running all the time as a tracking generator. Thus, in this mode, the VNWA can be operated as a **scalar network analyzer**.
One can still perform a SOLT calibration, but the pase information will not be available.

The benefit of using this scalar mode lies in the fact, that the TX frequency can be offset by a value specified in the sweep menu (available in LPT mode only) and the analysis bandwidth is rather large (20kHz). This allows to

measure transfer characteristics of frequency mixers and converters by using an external local oscillator, which frequency must coincide with the specified frequency offset. Its stability must coincide with the resolution bandwith.

This extension to the VNWA measurement capabilities evolved from experiments Erik ON8DC, who also betatested the described features. It allows to measure antenna radiation patterns and display and analyze them with the VNWA software.

The following image shows the basic test setup:



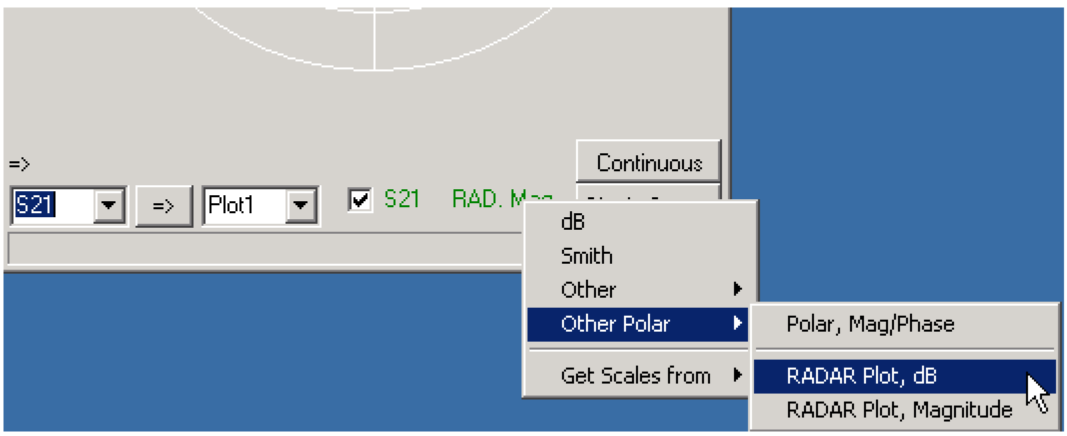
The VNWA TX output feeds the DUT antenna mounted on a rotor, while the RX input detects the radiation power in some distance. DUT antenna and detector antenna may be swapped.
The VNWA is set to a fixed frequency (span = zero). The VNWA sweep and the rotor engine are started simultaneously, such that the radiated power depending on turning angle is measured. The rotor start/stop signal is only implemented in LPT mode to date. The rotor may also be started manually.

If the distance between DUT antenna and Detector antenna becomes large (far field measurements), it is not feasible to feed the DUT antenna from the VNWA TX because of excessive cable length and/or insufficient TX power. In this case, a seperate highly frequency cw-transmitter may be used to feed the DUT antenna. As the transmitter signal and the VNWA LO are not phase locked any longer, the measurements have to be performed in Spectrum Analyzer mode in this case.

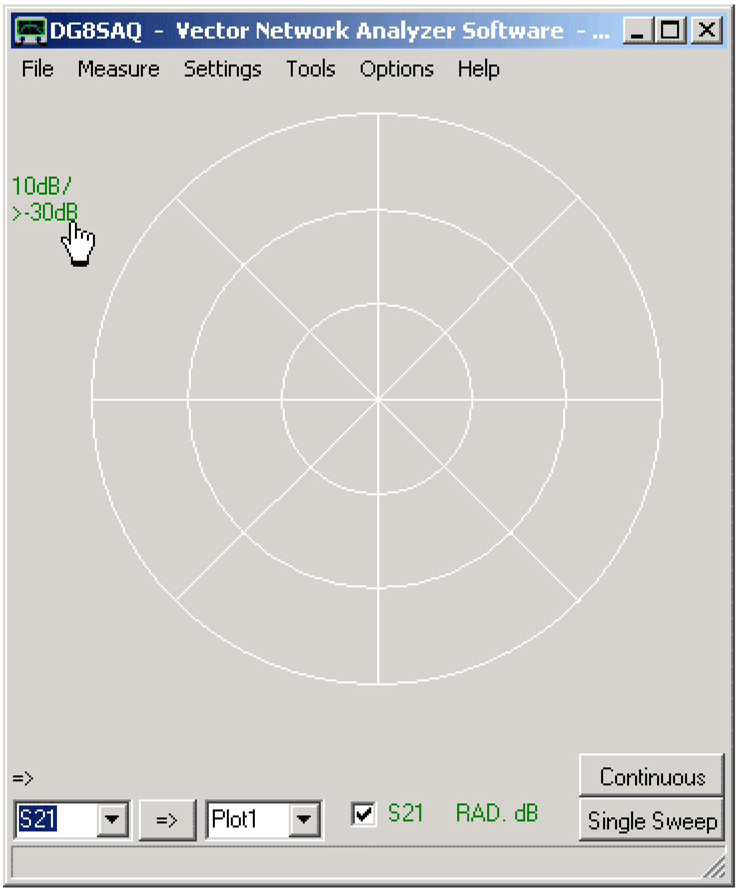
The following shows how to set up the VNWA software for this measurement:

1. Set the proper measurement frequency (range) with span zero.

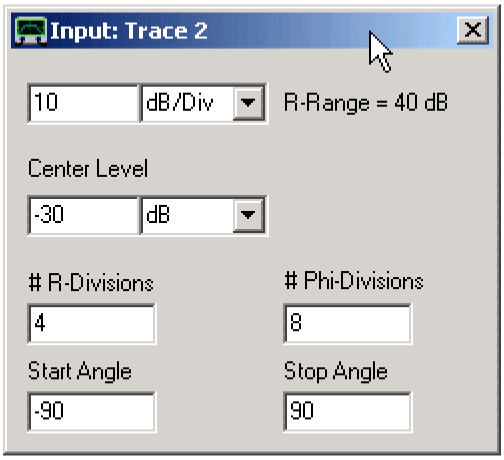
2. Select to display S21 in a polar RADAR diagram:



3. Set the display range to the desired values by doubleclicking the the label with the handpointer below. The displayed values indicate a center level of -30dB and 10dB per radial unit.

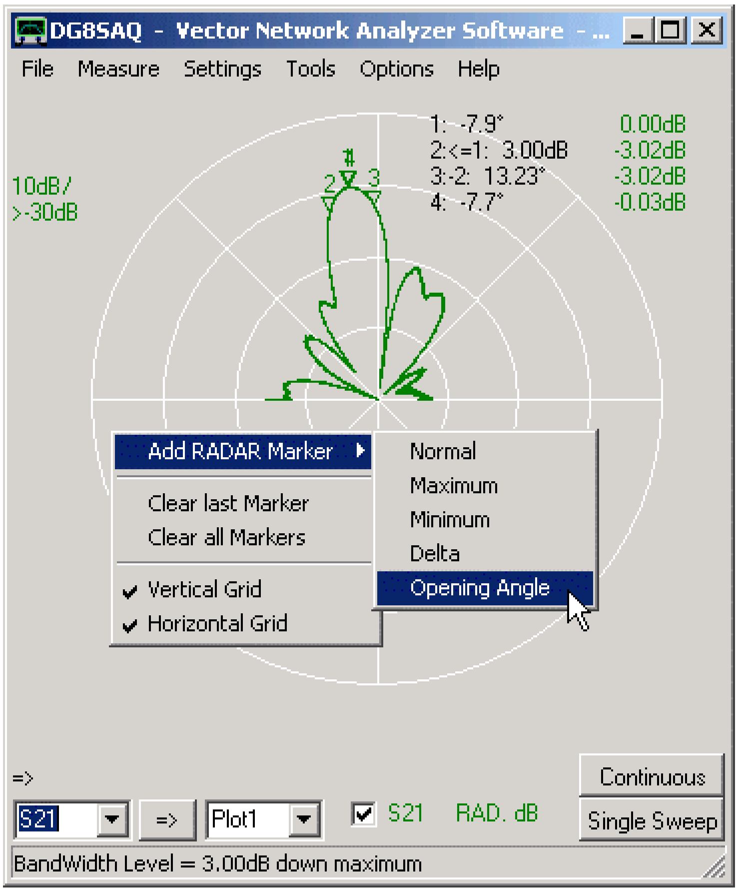


Upon doubleclicking the above mentioned label, the following parameter entry window pops up:



With the above settings the sweep data is interpreted as streching from -90° to 90°. Note, that the angle settings can also be corrected after the measurement to account for angle shifts.

4. Perform your measurement by pressing **single sweep** and simultaneously starting the rotor engine. After the sweep you may add RADAR markers, e.g. to determine the antenna 3dB opening angle as seen below.



The above displayed data was provided by Erik ON8DC. It was measured on a 23cm band loop antenna at 1.25 GHz.

Here is Erik's description how he performed the measurement:

- DUT Ant needs a stable freq source.(It could be a generator, QRP TX) or if enough gain the TX output from the VNWA. Power is depending on distance and gain of antennas. (A far end test must be possible)

- At RX side use an antenna connected to RX VNWA. Again, depending on distance and power used at TX side, a gain antenna may be needed.

- Set VNWA to SA mode.

- Verify with a sweep (S21) if you can receive the DUT. Lower the span to lowest value 0Hz. The RX is a receiver with a narrow filter.

- Adjust the level offset in the sweep menu such that your RX receives 0dB at max position.

- Verify that in the min signal received you are above noise level of SA. (linearity verify with attenuators)

- Measure the time your antenna rotor needs for a full turn and adjust sweep-time until sweep time=rotor time.

- Select S21 and S21 RAD dB (other polar) as traces.

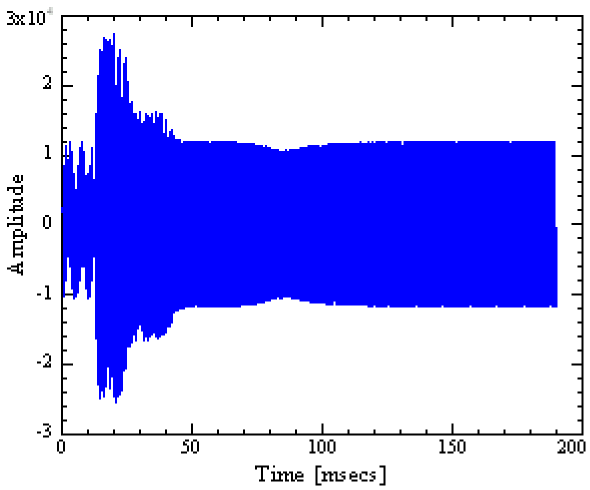
- Start manually or automatically synchronic the rotor and the sweep. You will see now the signal variation with changing angle at the screen.
For the antenna plot you can change the trace parameters start/stop angle divisions and center level.

- For automatic rotor start you can use the high signal on D7 Lpt (pin9) to drive a relay, it stays on until the sweep is finished.

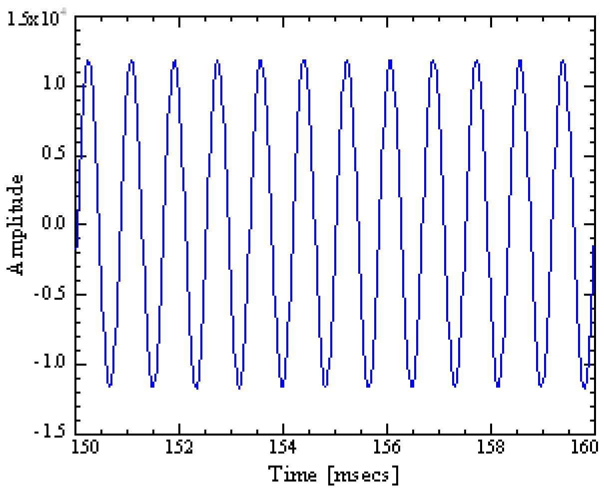
Special care must be taken, when the VNWA is used at audio frequencies **below 20 kHz**.

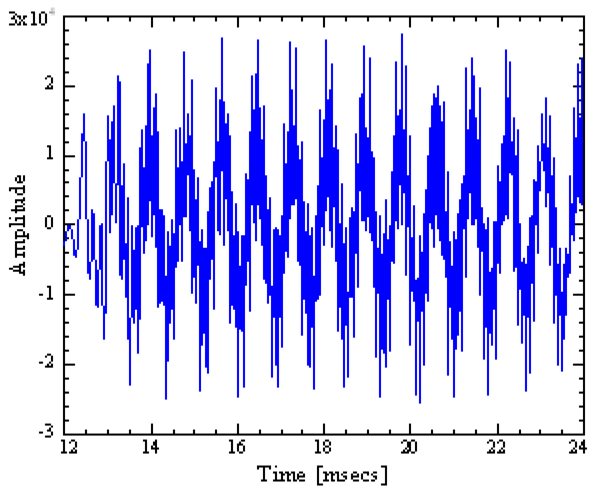
The reason is that the mixers used show some "RF" feed through from the RF input to the IF output and from the LO input to the RF output. Thus, when measuring in the audio range the RF feedthrough signal will be superimposed on the IF signal leading to **signal distortion** and **ADC overloads** as it falls inside the sound card bandwidth.

The following image shows the raw audio stream captured with a 16 bit audio card for a sweep from 1 kHz to 100 kHz. Note, that for a 16 bit card the maximum amplitude range is -32767...32768.



For frequencies outside the sound card bandwidth (>20 kHz, beyond 50 msecs in the plot) the amplitude of the sound stream is about 50% of the allowed maximum amplitude. This will also be seen with the Test Audio function, as it performs the test in the MHz range. At low frequencies the amplitude approximately doubles with the risk of overloading the sound ADC. The mechanism leading to this amplitude increase can be seen when zooming into a high frequency and a low frequency region for comparison:

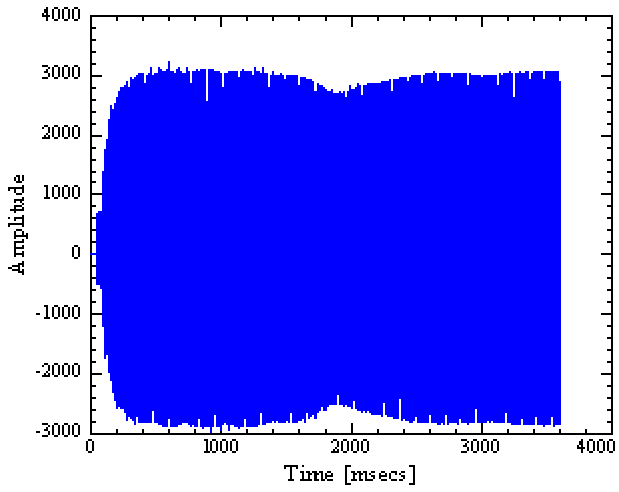




The left image shows a clean IF signal corresponding to a 100 kHz input frequency, while the right image shows an apparently "noisy" IF signal corresponding to an input frequency in the 1kHz range. Here, the superimposed "noise" is actually a superposition of RF, LO and all kinds of mixing products.

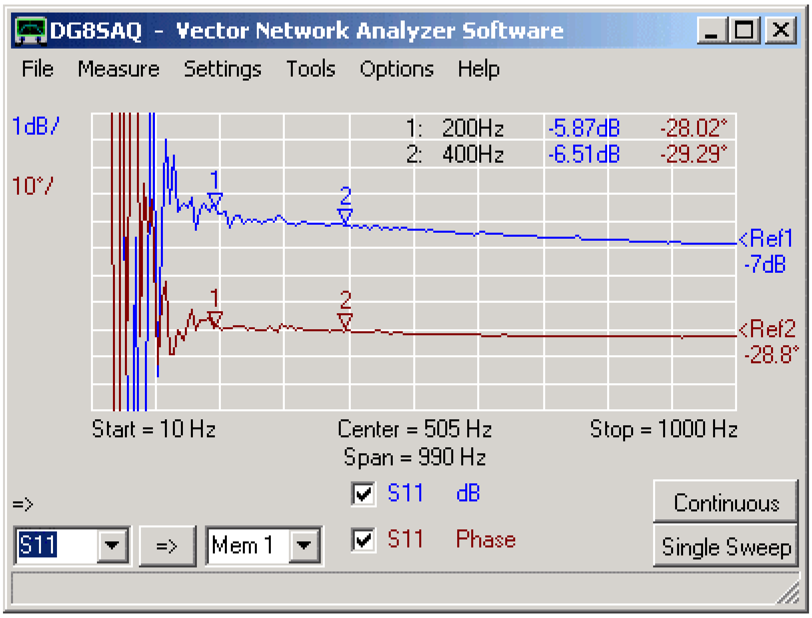
Thus, with default settings, measurements in the audio range are not possible because of strong interference inside the sound card bandwith!

But there is a simple workaround. All sound cards I know of have a built-in anti-aliasing low pass filter, which adapts according to the Shannon sampling theorem to the selected sample rate. Reducing the sampling rate from 48 kHz to 900 Hz will reduce the upper cutoff frequency from about 20 kHz to about 400 Hz. The effect is shown in the following image, which shows the audio stream for the same frequency range as above, but with a capture rate of 900 Samples per second. Note, that when lowering the sample rate also the number of samples per IF period must be reduced and the IF in turn raised, otherwise the IF might fall out of the lower end of the sound card bandwidth.



The overall amplitude is a lot smaller, as the response at very low IFs is dropping. But note, that the amplitude is now approximately constant with no overshoot over the whole frequency range.

Now, what's the practical lower frequency limit of the VNWA? This is best answered with a test measurement. The following extreme low frequency sweep is performed with a sample rate of 900 samples per second and the highest possible IF of 75 Hz:



Es expected, the sound device anti-aliasing filter cleans the respone above 400 Hz. **200 Hz seems the practical lower frequency limit of the instrument.**

Summary:

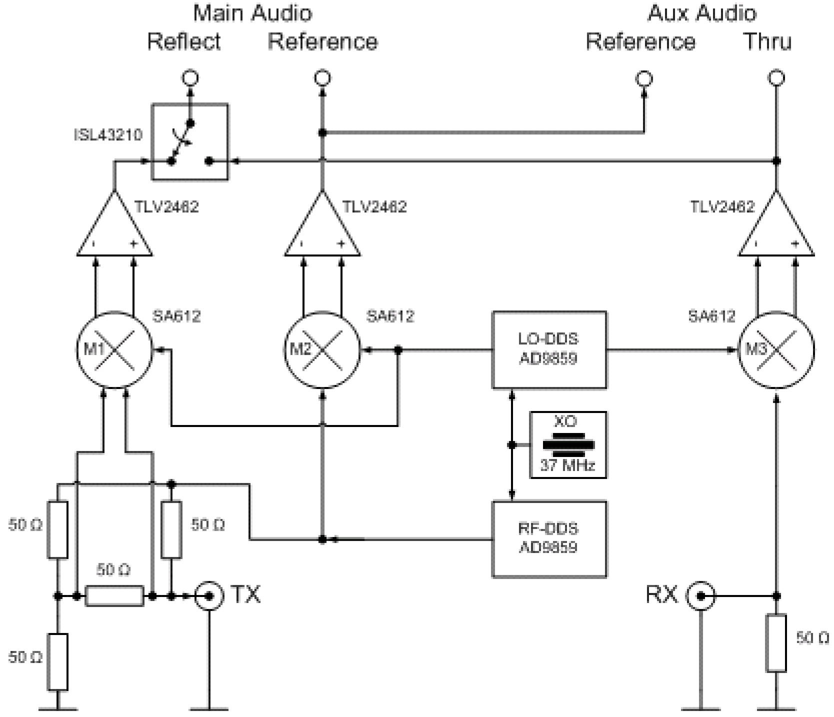
- The VNWA can work down to 200 Hz.
- Below the sound card cutoff frequency (typically 20 kHz, on some sound cards up to 100 kHz) the sample rate must be decreased.
- **A wideband mastercalibration with standard settings should not extend below the sound card cutoff frequency**. Note, that changing the sample rate will invalidate any calibration including the master calibration.

▸▸▸ **Note:** This feature is only available in USB-mode!

In its original design, the VNWA cannot measure S11 and S21 simultaneously, but with the aid of a multiplexing switch only alternately.
The reason for this lies in the fact, that three signals must be captured for a simultaneous measurement (Reference, Reflect, Thru), but a standard stereo sound card has only two input channels.

This restriction can be overcome by **adding a second sound card** to the measurement system, thus adding two audio channels. As the sound cards are not running synchronously in any way, both need to receive the Reference signal. This still leaves two free channels for the Reference and Reflect signal to be captured simultaneously:

Hardware Modification

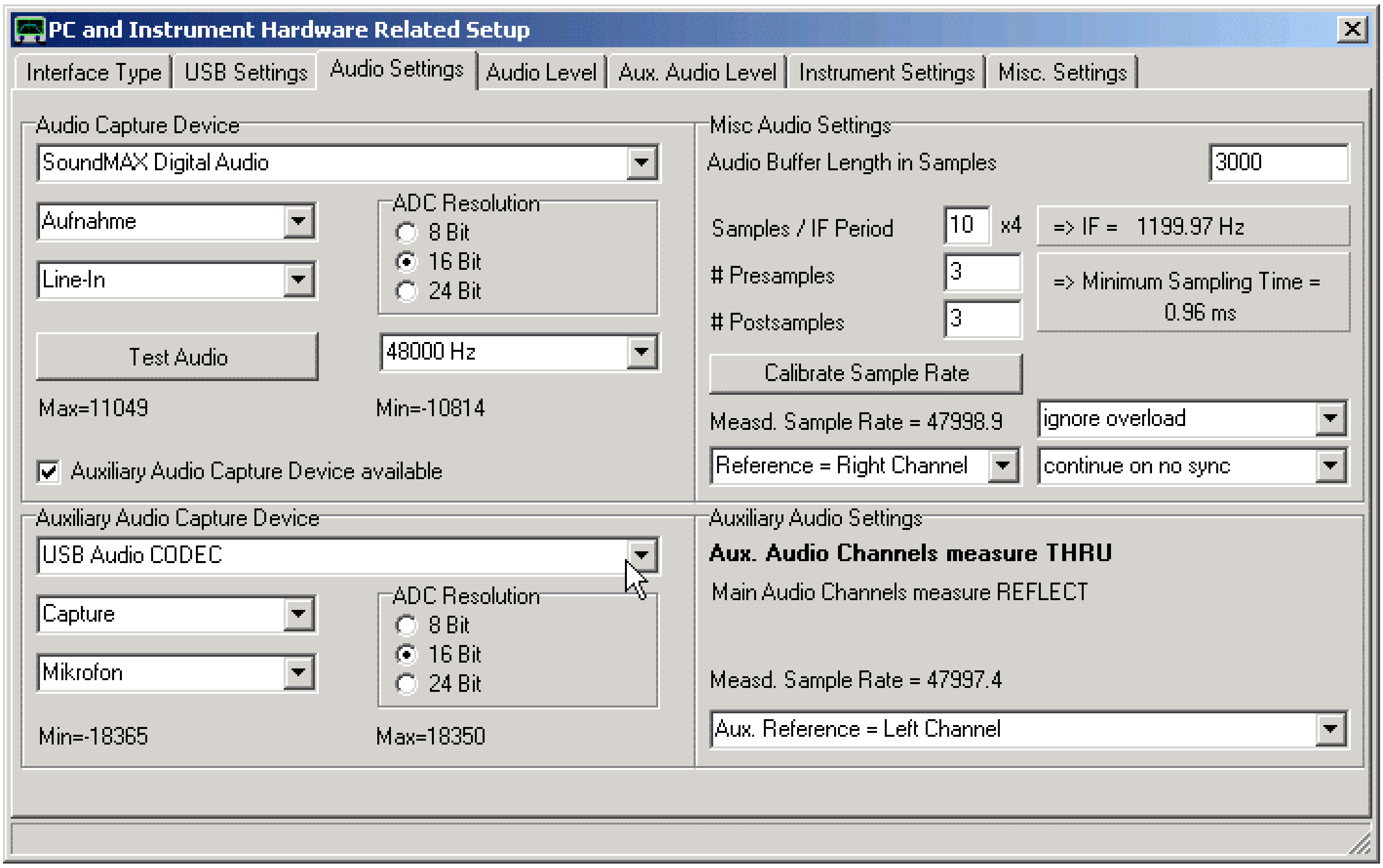


As can be seen above, the main audio capture device remains unchanged and is used to measure S11. The multiplexing switch will remain fixed in the position shown above at all times.
In order to measure S21, the RX OpAmp (behind M3) must be tapped to provide the Thru signal for the second, auxiliary sound card. Make sure to properly DC decouple the OpAmp output to the aux audio input by means of a capacitor (e.g. 10uF) if it is not DC decoupled by itself. Note, that the auxiliary sound card will also need the reference signal again, which can be tapped at the main audio connector.

Activating and Configuring the Auxiliary Sound Device

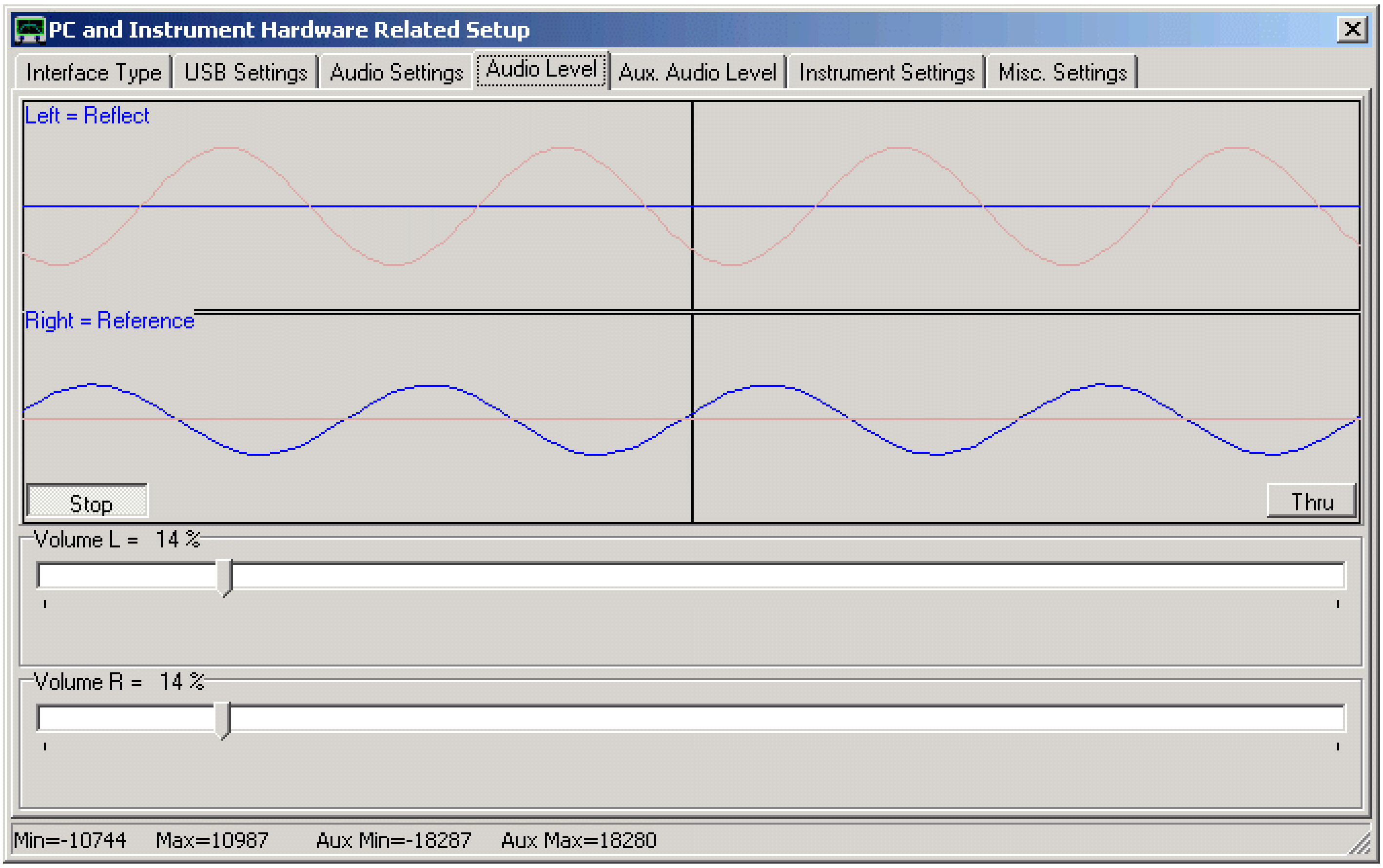
Usage of an auxiliary sound device is activated by using a text editor and adding the line "**AuxAudio=1**" to the file **VNWA.ini** to be found in the VNWA program directory. VNWA must not be running while doing this modification.

Next, start VNWA. A warning will pup up to tell you, that there is missing setup information. Confirm it. The setup window will open. Go to "Audio Settings" and update the settings according to your modified hardware. Don't worry about the settings on the right hand side, yet.



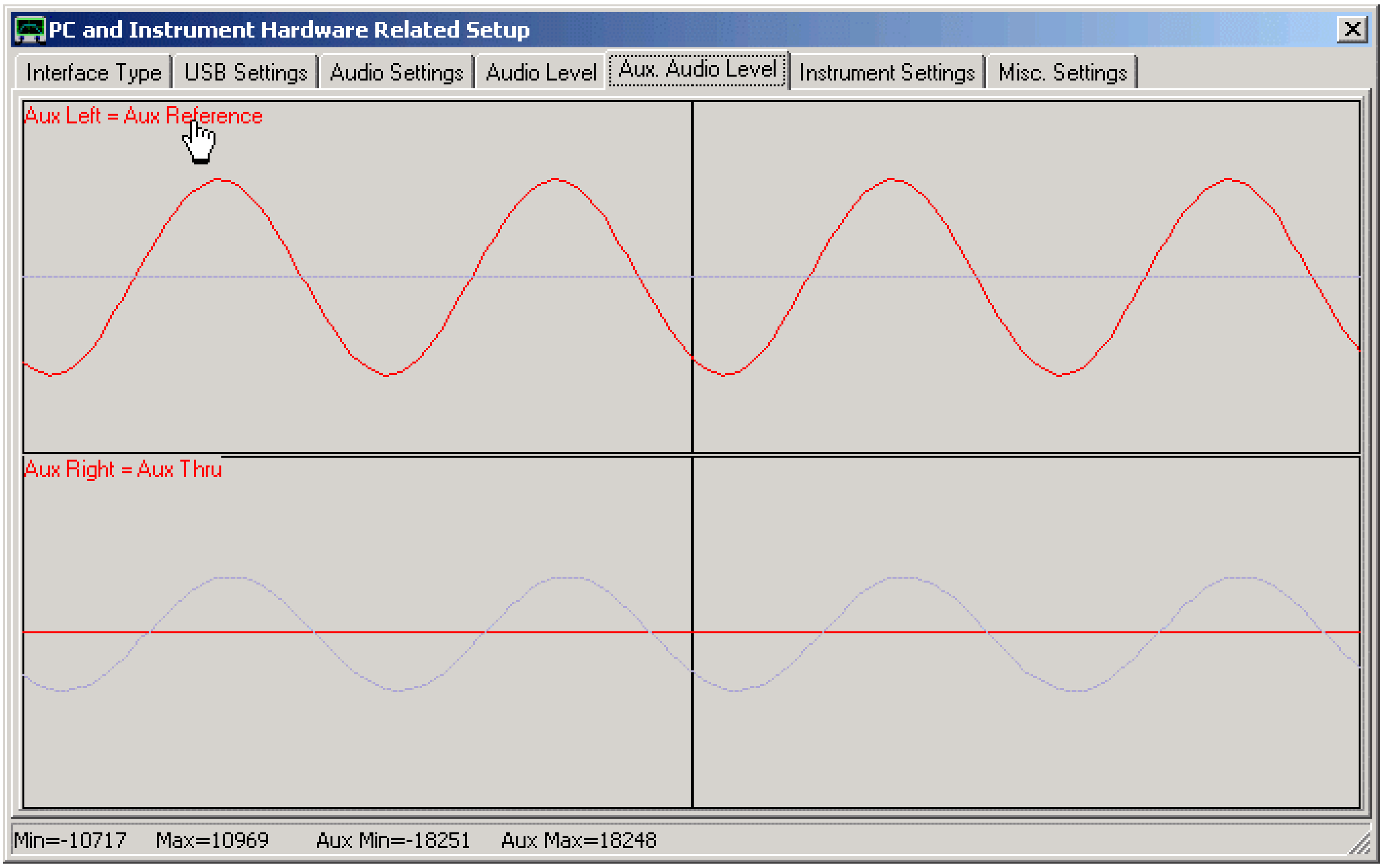
Note, that generally, unlike shown above, the built-in USB codec must be the main capture device, as it is hard wired to the multiplexing switch.

Next go to the "Audio Level" tab and press "Test Audio".



In Thru-mode with no cable connection between TX-port and RX port, you should see the **blue main reference** signal and a **flat blue line**. Don't worry about the lighter aux audio signals, which are also visible. If the blue sinewave signal is not marked as "Reference" like seen above, right-click onto the blue label denoting the sinewave signal. It will then change to "Reference". Thus you have selected the appropriate channel as main reference channel. Press the Thru-button to check if the reflect signal is working properly.

Next go to the "Aux Audio Level" tab.



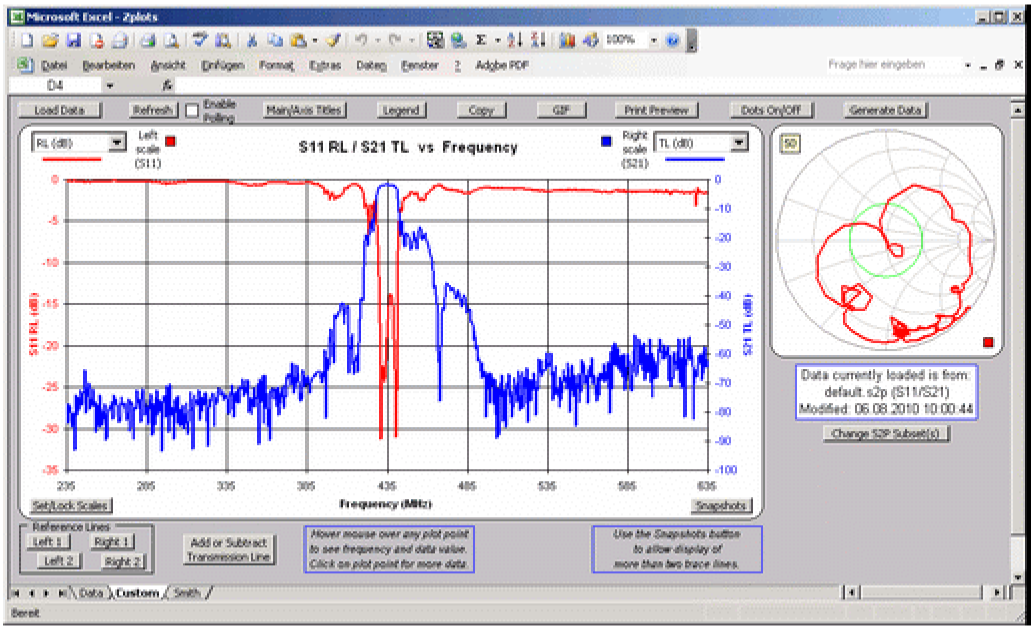
Without TX to RX through connection, you should see the red aux reference sinewave signal only. Make sure that this is selected as aux reference signal as seen above (if necessary) by clicking onto the red label denoting the sinewave signal and observing the correct description. Next connect a coax thru connection from TX to RX port and make sure you do see an unclipped thru signal as well.

Next, go back to the "Audio Settings" tab and perform a sample rate calibration. Note, that both sound cards must be calibrated as they might have slightly differing sample rates as can be seen above.

Finally, close the setup.

Perform calibrations and measurements in the normal fashion. From now on S21 and S11 traces will update with new data simultaneously, thus cutting the measurement time down by a factor of two.

Dan, AC6LA, has written **Zplots** (http://ac6la.com/zplots.html), a wonderful Excel application, that uses the power of the Excel charting engine to neatly plot and analyze S-parameters like the ones generated during a VNWA measurement.



Many thanks to Dan for having modified ZPlots such that it can be easily integrated into the VNWA software.

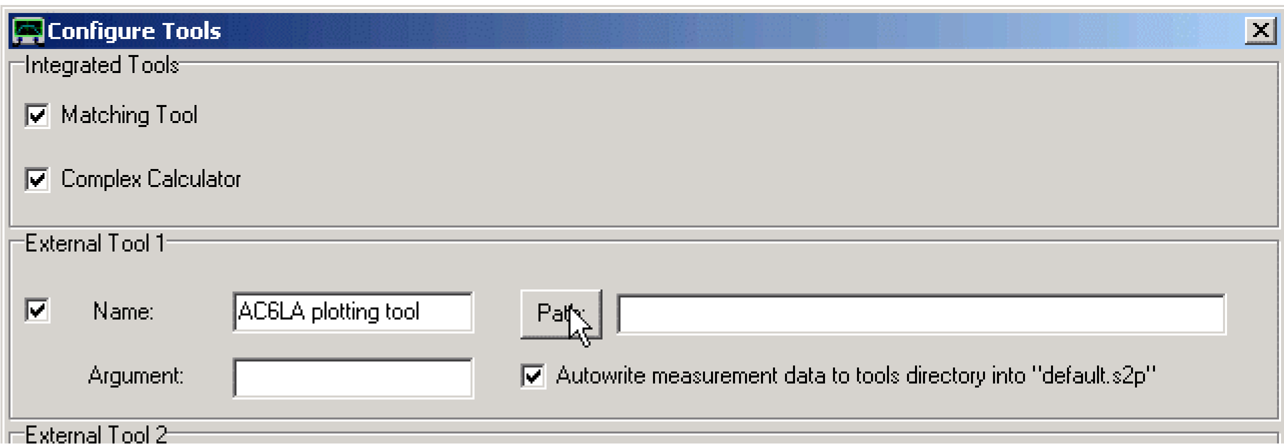
The following interactions between VNWA and Zplotsare available:

- Zplots can be started or reactivated as external tool from within VNWA.
- Upon (re-)activation of Zplots through the tools menu VNWA can automatically transfer measurement data to Zplots.
- Zplots can automatically poll for new measurement data, that VNWA can continuously provide after each completed sweep.

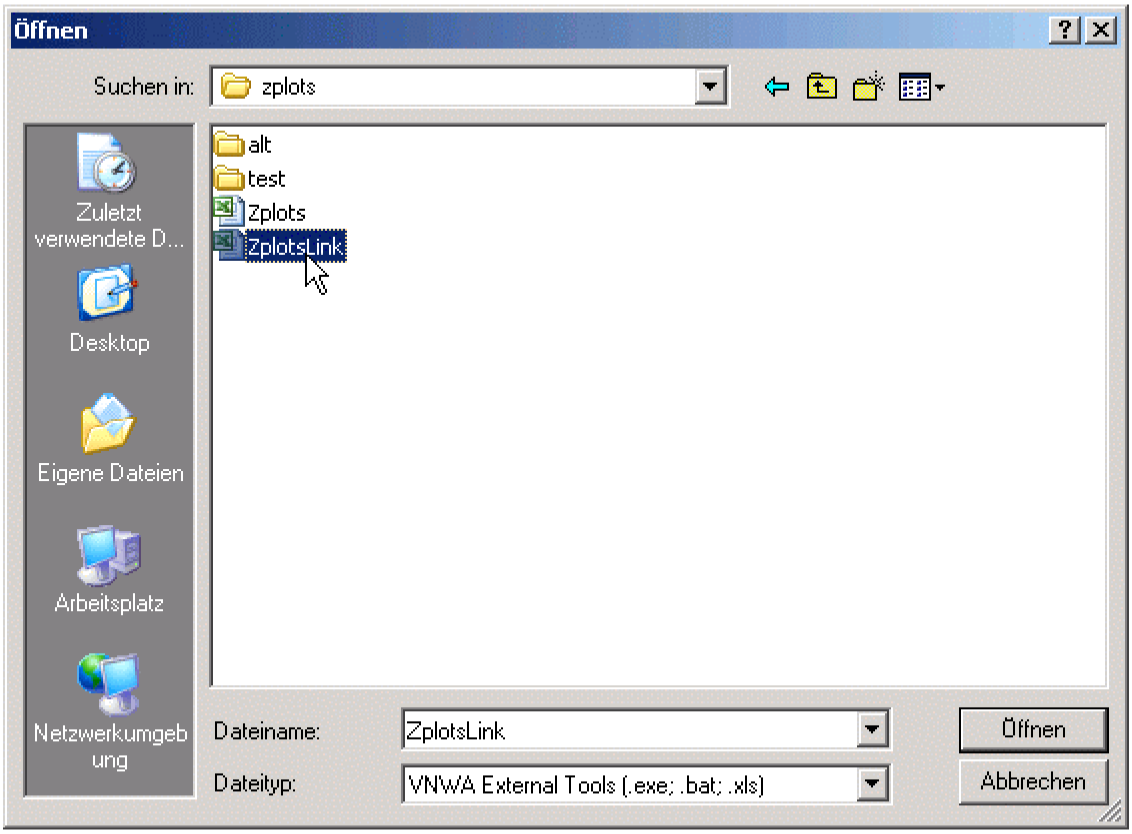
Configuring VNWA for interfacing Zplots:

1. Copy the latest version of Zplots to any directory of your choice. The Zplots package consists of two files, namely the application **Zplots.xls** and the interface **ZplotsLink.xls**. Both files need to reside in the same directory.

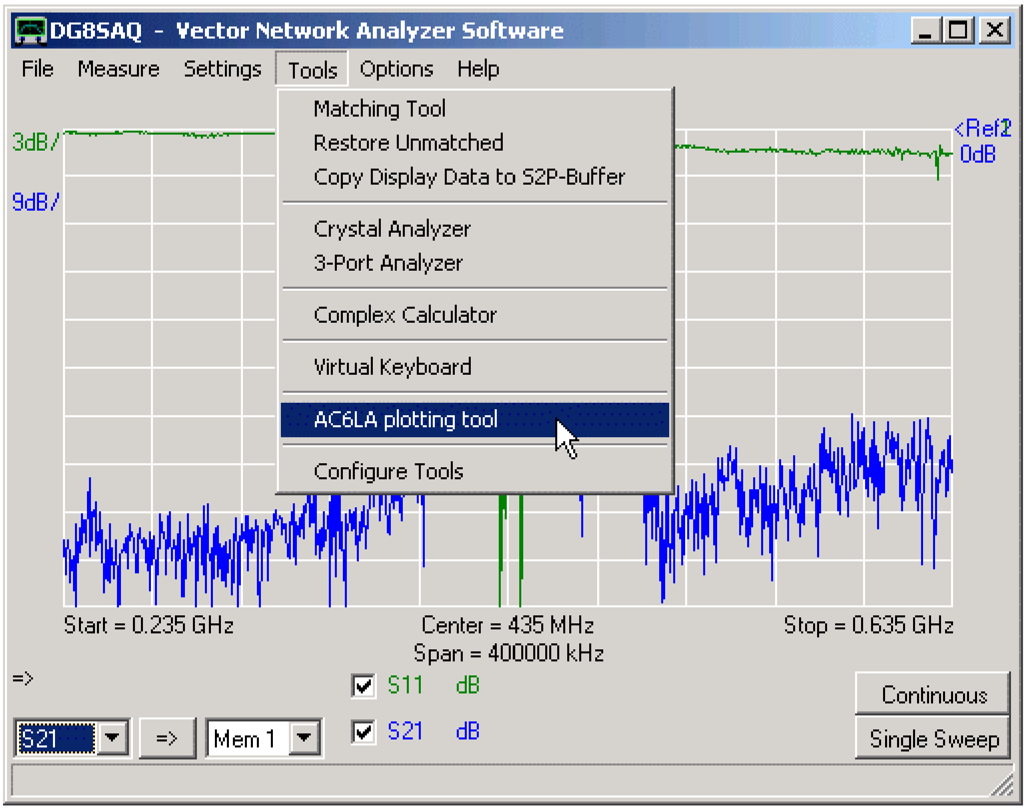
2. In VNWA open the "Tools"-"Configure Tools" menu, activate an external tool, give it a descriptive name e.g. "AC6LA plotting tool" and select "autowrite measurement data..." in order to give Zplots access to the VNWA measurement data. The "Argument" field should remain blank.



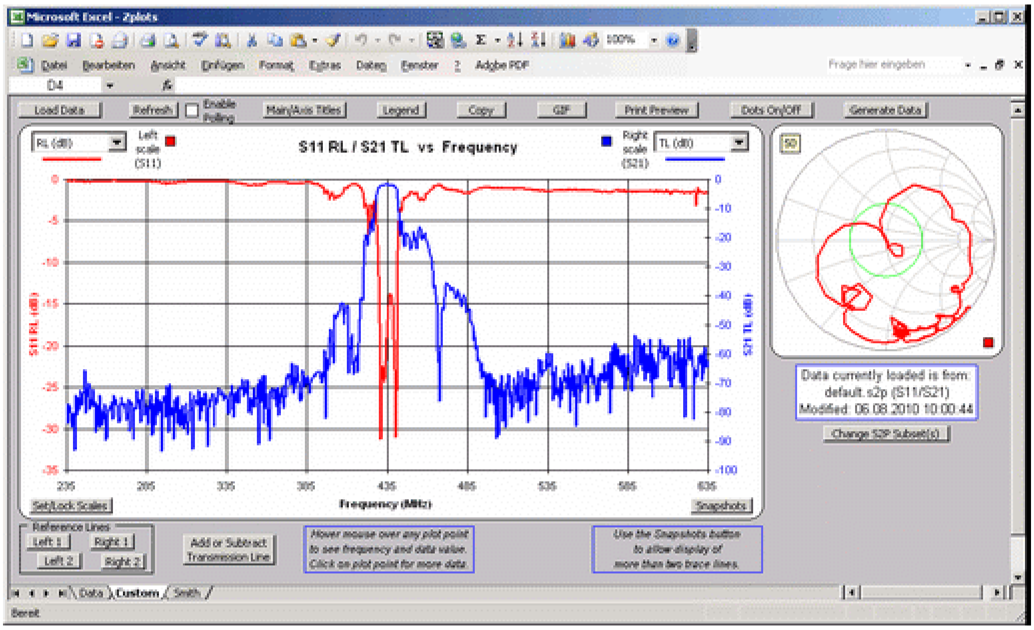
3. Next browse for the Zplots application by pressing the **"Path"** button.
▸▸▸ **Note: You must select the interface file ZplotsLink.xls** and open it.



4. Close the **"Configure Tools"** menu. Observe, that your new Zplots tool is now visible in the VNWA "Tools" menu as "AC6LA plotting tool".



Selecting the just generated new tool menu "AC6LA plotting tool" will open Zplots and transfer the current measurement data to Zplots. If Zplots has previously been started already, only new data will be transfered:



For more information on Zplots, please consult the author's documentation, e.g. on http://ac6la.com/zplots.html.

For automated measurements the VNWA software can be remote-controlled via **Windows messages** or via a **script file**. Under normal conditions the remote control interface in VNWA is disabled. In order to enable it, the VNWA software must be started with certain runtime arguments:

Starting VNWA for accepting Windows messaging commands:

Syntax:

VNWA.exe -remote -callback ownhandle ownmessage [-silent] [-debug]

[...] is an optional argument, multiple arguments must be separated with blank characters.

-remote activates the VNWA remote control interface
-callback allows to specify a handle and a message number which VNWA will use to return its own handle and remote control message number and notify end of processing of a remote command via a Windows message.
-silent makes the VNWA main Window invisible. Note, that this will also hide the VNWA application from the task bar and the task manager application tab.
-debug opens a debug Window (also in silent mode), where all incoming remote commands are being listed.

Example:

VNWA.exe -remote -callback 2622836 1024 -debug

VNWA will be started in remote control mode with the main window and debug window visible. The ownhandle and ownmessage parameters must be generated at runtime by the calling program and might change from program run to program run.

For details on the remote control interface see page Controlling the VNWA by Windows Messages.

Starting VNWA for processing a script file:

Syntax:

VNWA.exe filename.scr [-silent] [-debug]

[...] is an optional argument, multiple arguments must be separated with blank characters.

filename.scr is the name of the script file to be processed. Note, that the file ending MUST be **.scr**.
-silent makes the VNWA main Window invisible. Note, that this will also hide the VNWA application from the task bar and the task manager application tab.
-debug opens a debug (also in silent mode) Window, where all processed commands and error messages are being listed.

Example:

VNWA.exe test.scr -debug opens the VNWA main window and the debug window and executes the script file test.scr.

For details on the script file syntax see page Controlling the VNWA by a Script File.

The Windows Message Interface:

Basic Concept of Windows Messages
The Windows command "PostMessage" allows to send a short message to any specific visible or invisile Window:

```
BOOL WINAPI PostMessage(
  __in_opt  HWND hWnd,
  __in      UINT Msg,
  __in      WPARAM wParam,
  __in      LPARAM lParam
);
```

Parameter description:

- **hWnd** is the window handle (or address) so the specific window can be reached.
- **Msg** is a unique message number upon which VNWA is prepared react.
- wParam and lParam are two 4 byte integer constants, that can be sent to VNWA:
  wParam is interpreted as VNWA command.
  lParam is interpreted as an optional parameter for the VNWA command.

Implemented VNWA remote commands:

wParam:	Command:	lParam:
0	terminate VNWA	n.a.
1	sweep	bits:select 0:S21, 1:S11, 2:S12, 3:S22
2	load cal [rfile]	n.a.
3	load mastercal [rfile]	n.a.
4	write s2p [wfile]	n.a.
5	write s1p [wfile]	value:selects 0:S21, 1:S11, 2:S12, 3:S22
6	change rfile string	256: clear string 0...255: add char(lParam) to string
7	change wfile string	256: clear string 0...255: add char(lParam) to string
8	set start frequency	start frequency [Hz]
9	set stop frequency	stop frequency [Hz]
10	set number of points	number of points
11	set sweep mode	0: Lin, 1: Log
12	set time per data point	time [microseconds]
13	read and execute [rfile] script file (*)	
255	echo (wParam & 0xFFFF) | 0x10000 and lParam by PostMessage back to sender (for debugging while software development)	

(*) For details on the script file syntax see page Controlling the VNWA by a Script File.

wfile is a write file name including path held inside the VNWA application while it is running. This is the file, where measured S-parameters can be stored to. The filename can be modified via remote command 4.

rfile is a read file name including path held inside the VNWA application while it is running. This is the file, where a calibration or mastercalibration is read from. The filename can be modified via remote command 5.

Software Implementation

The following Delphi/Pascal example code shows how to remotely start the VNWA software and remotely control it:

```
unit URemote;

interface

uses
  Windows, Messages, SysUtils, Variants, Classes, Graphics, Controls, Forms,
```

```
  Dialogs, StdCtrls, ShellAPI, Menus;

type
  TForm1 = class(TForm)
    BStartVNWA: TButton;
    ListBox: TListBox;
    EPath: TEdit;
    ERuntimeArgument: TEdit;
    BSendMessage: TButton;
    ECommand: TEdit;
    EParameter: TEdit;
    Label1: TLabel;
    Label2: TLabel;
    PopupMenu1: TPopupMenu;
    ClearScreen1: TMenuItem;
    Edit5: TEdit;
    BSendWName: TButton;
    Label3: TLabel;
    Label4: TLabel;
    Edit6: TEdit;
    BSendRName: TButton;
    procedure BStartVNWAClick(Sender: TObject);
    procedure FormCreate(Sender: TObject);
    procedure BSendMessageClick(Sender: TObject);
    procedure ClearScreen1Click(Sender: TObject);
    procedure BSendWNameClick(Sender: TObject);
    procedure BSendRNameClick(Sender: TObject);
  private
     Private declarations
  public
     Public declarations
    remotehandle: hwnd;
    WM_REMOTE: WORD;
    procedure Receiver(var Msg: TMessage); message WM_USER;
  end;

var
  Form1: TForm1;

implementation

$R *.dfm

procedure TForm1.BStartVNWAClick(Sender: TObject);
var path, argument: string;
begin
PostMessage(remotehandle, WM_REMOTE, 0, 0);     //try to terminate VNWA if it is still running
sleep(500);                                     //wait for VNWA to terminate
path:=EPath.Text;
argument:=ERuntimeArgument.Text;
ListBox.Clear;
ListBox.AddItem('ShellExecute return code = '+
inttostr(ShellExecute(Handle,'open', PChar( Path),PChar( Argument), nil, SW_SHOWNORMAL)),nil  );
end;

procedure TForm1.FormCreate(Sender: TObject);
begin
EPath.Text:='D:.exe';
ERuntimeArgument.Text:='-remote -silent -debug -callback '+inttostr(self.handle)+' '+inttostr(WM_USER);
end;

procedure TForm1.Receiver(var Msg: TMessage);
var cmd: integer;
begin
```

```
cmd:=Msg.WParam shr 16;
ListBox.AddItem('Message received LParam='+inttostr(Msg.LParam)+'   WParam='+inttostr(Msg.WParam),nil);
case cmd of
  0: begin
      WM_REMOTE:=Msg.WParam;
      remotehandle:=Msg.LParam;
      ListBox.AddItem('remote handle received',nil);
    end;
  1: begin
      ListBox.AddItem('remote command executed successfully',nil);
    end;
  else ListBox.AddItem('remote command executed with ERROR! Error code = '
    +inttostr(cmd-1),nil);
  end;
end;

procedure TForm1.BSendMessageClick(Sender: TObject);
begin
PostMessage(remotehandle, WM_REMOTE, strtoint(ECommand.Text), strtoint(EParameter.Text));
end;

procedure TForm1.ClearScreen1Click(Sender: TObject);
begin
ListBox.Clear;
end;

procedure TForm1.BSendWNameClick(Sender: TObject);
var s: string;
   i: integer;
begin
s:=Edit5.Text;
PostMessage(remotehandle, WM_REMOTE, 7, 0);
for i:=1 to length(s) do
  PostMessage(remotehandle, WM_REMOTE, 7, byte(s[i]));
end;

procedure TForm1.BSendRNameClick(Sender: TObject);
var s: string;
   i: integer;
begin
s:=Edit6.Text;
PostMessage(remotehandle, WM_REMOTE, 6, 0);
for i:=1 to length(s) do
  PostMessage(remotehandle, WM_REMOTE, 6, byte(s[i]));
end;

end.
```

The VNWA Script File Syntax:

<u>Implemented VNWA script file commands:</u>

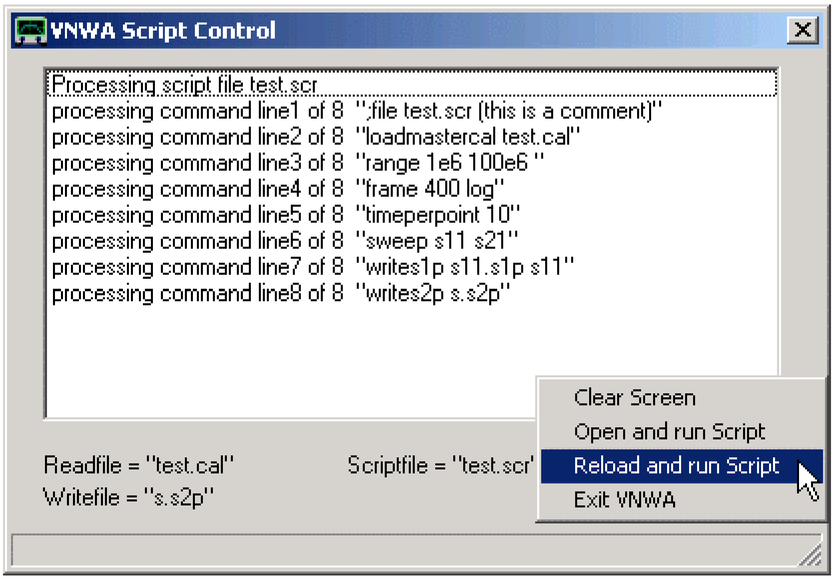
; comment a line starting with ";" is interpreted as comment.
loadmastercal cal-filename load a calibration file as master-calibration
loadcal cal-filename load a calibration file
timeperpoint <t> sets the measurement time per data point. The number t denotes the time per data point in milliseconds
range <start> <stop> sets the measurement frequency range. start=start frequency [Hz], stop=stop frequency [Hz]
frame <n> <sweeptype> sets the sweep frame data. n=number of data points, sweeptype=lin or sweeptype=log sets a linear or logarithmic frequency grid.
sweep [s11] [s21] [s12] [s22] starts a sweep for the specified data fields
writes1p filename <trace> writes measurement data of trace to an s1p file. trace can by any of s11, s21, s12, s22
writes2p filename writes the full measurement data (s11, s21, s12 and s22) to an s2p file
exitVNWA terminates the VNWA software

▸▸▸ **Note:** VNWA is automatically terminated if an error occurs and the debug-option is not activated.

Script File Example:

```
;file test.scr (this is a comment)
loadmastercal test.cal
range 1e6 100e6
frame 400 log
timeperpoint 10
sweep s11 s21
writes1p s11.s1p s11
writes2p s.s2p
```

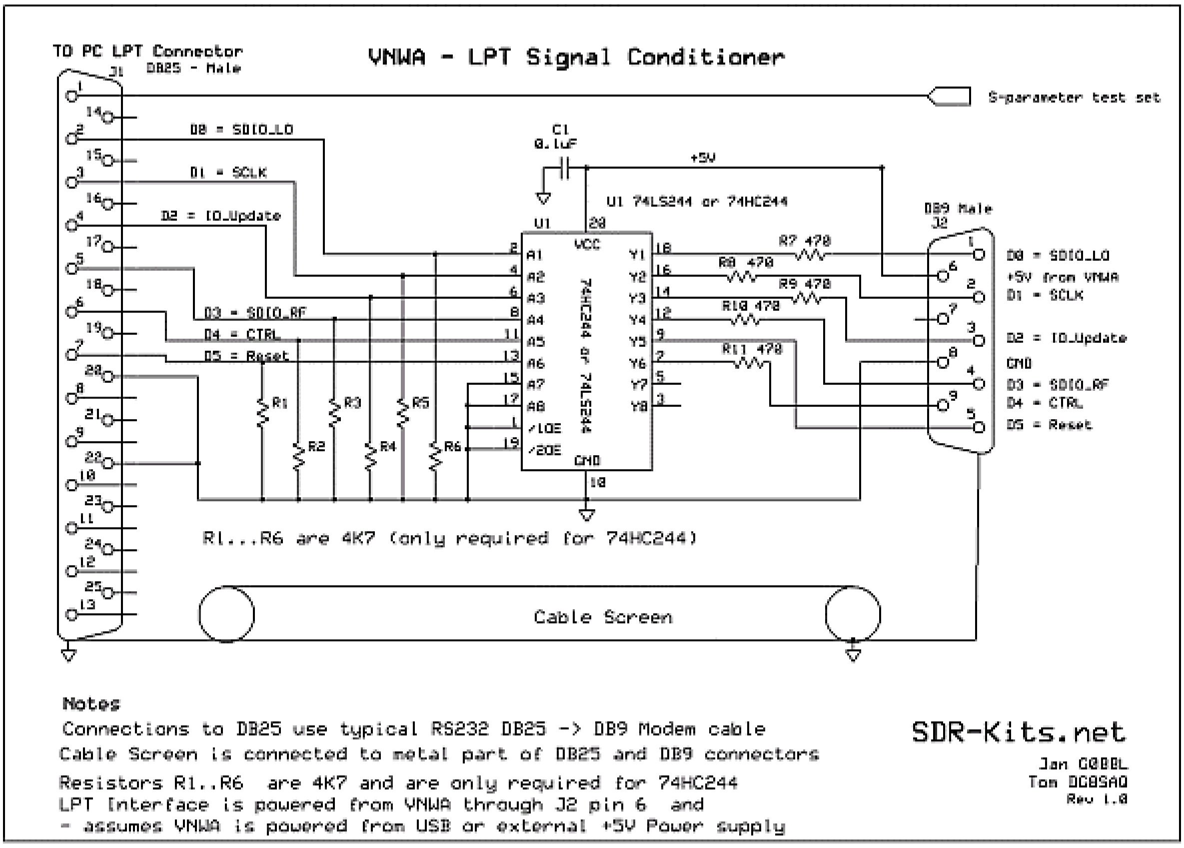
Starting VNWA with the command **VNWA.exe test.scr -debug** will open the VNWA window and the debug or **script control window**:



The script control window shows the progress of script execution and occuring errors.
Right-clicking onto the script control window will open the **script control drop-down menu** seen above. It allows to:
- Clear Screen = clear the white message box

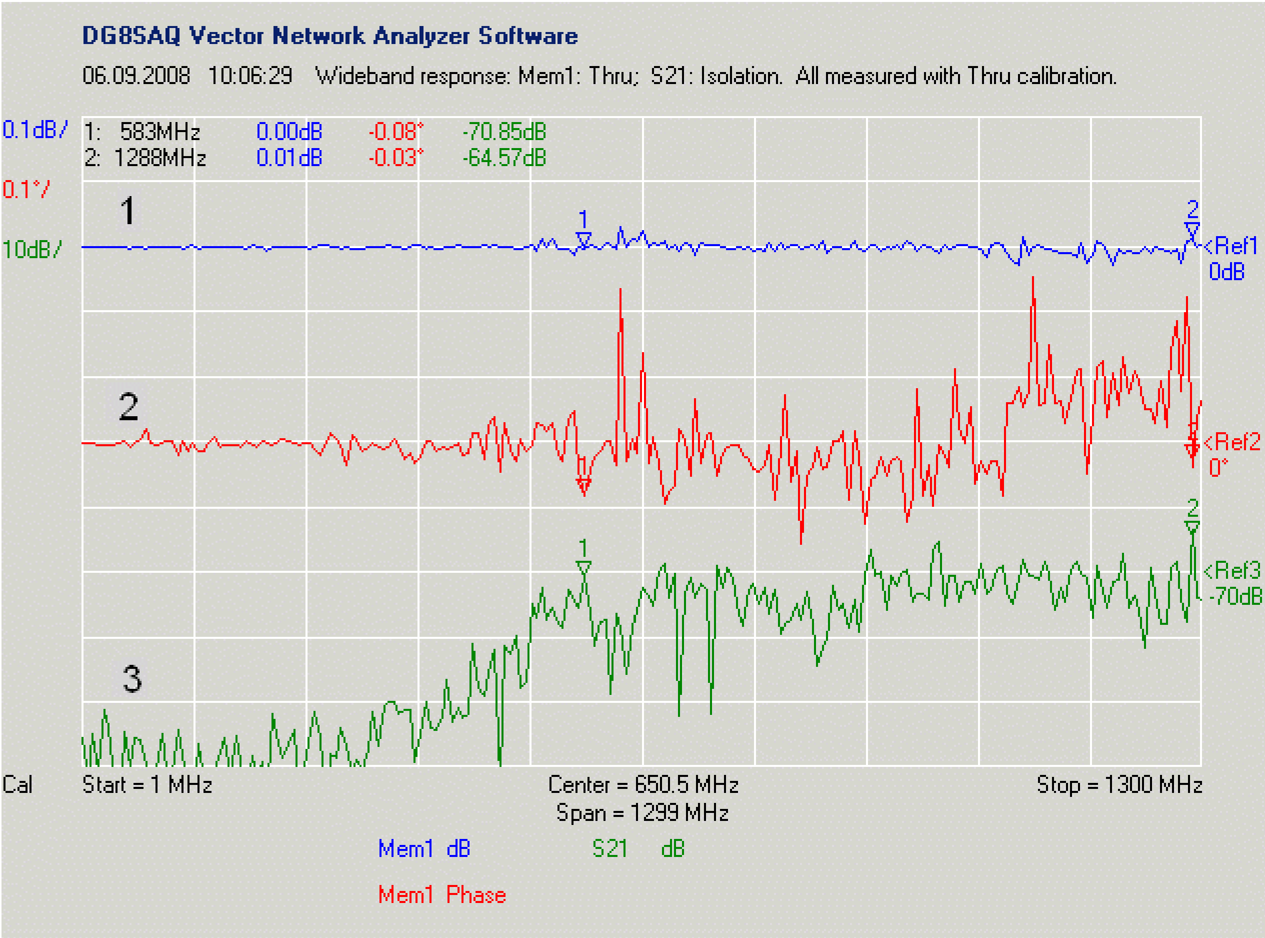
- Open and run a script file
- Reload (the previously opened) script file and run it (useful if the file has changed)
- Exit the VNWA application.

▸▸▸ **Note:** I have observed extremely slow signal rise times on some new PCI LPT interface cards. This is a problem when controlling the VNWA. Should you encounter unreliable communtication between PC and VNWA in LPT mode, you might need the following signal conditioner, which sharpens the digital pulses.



A question often posed, is what is the achievable **dynamic range and accuracy** of the VNWA on transmission measurements.

The following picture taken from one of my QEX publications may serve as a reference:



The blue and red traces show the transmission characteristics of the Thru Calibration Standard. The green trace shows the measured noise floor with open RX input.
All measurements were done after a thru calibration (no crosstalk calibration was done!). The highest possible sweep time of 100ms/frequency point was used. The VNWA was powered via USB, controled via LPT, the on board switching power supply of the VNWA was active and the on-board 16 bit SoundMAX sound card of my IBM R52 notebook with line-in exposed by a docking station was used. I used auto-clock rate to access the frequency range beyond 500 MHz.

If you can't reproduce the above displayed dynamic range, you may check the following items:

1. Have you selected proper **clock multipliers** to cover your frequency range? If you are not afraid of overclocking, use auto.

2. Calibration measurements must be performed with the lowest sweep rate (=longest time per data point) you ever want to use, otherwise the noise of the calibration measurement will dominate your later measurement and you won't see any noise reduction when further increasing the sweep time.

3. To obtain the highest possible dynamic range use the **slowest possible sweep time**.

4. With average 16 bit sound cards the dynamic range is limited to about 90dB. Somebody has demonstrated >100dB dynamic range measurements on the Yahoo VNWA reflector using a decent **24bit sound card**.

5. Your **VNWA must be boxed and properly soldered to the box walls** to achieve the highest possible dynamic range.

6. Poor solder joints on the VNWA board might degrade the dynamic range.

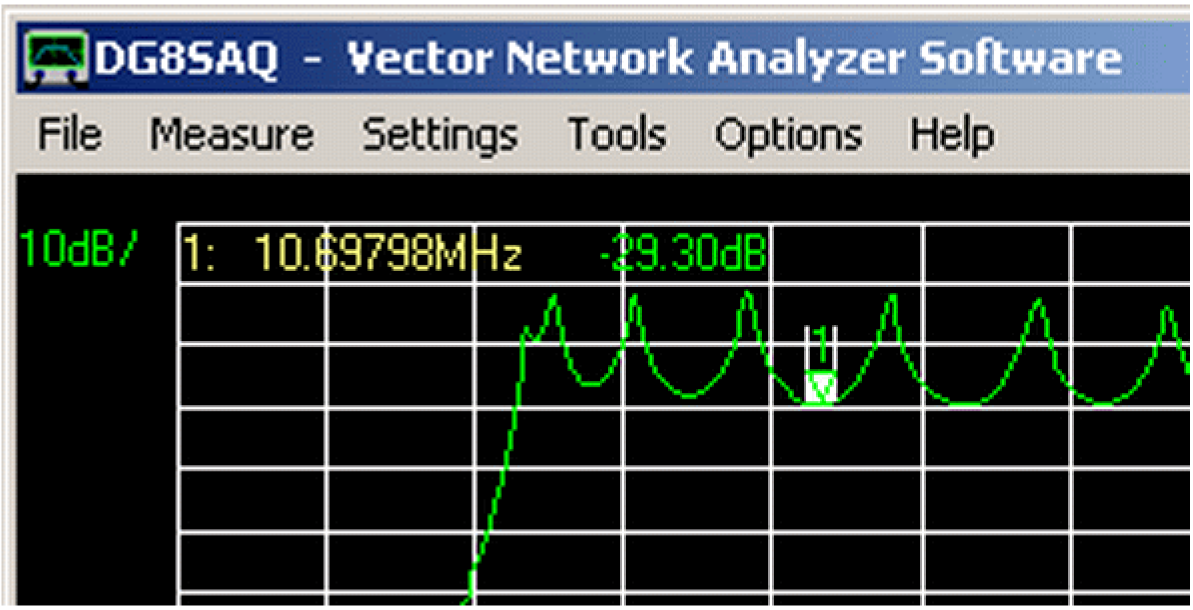
7. If you want to perform high attenuation level transmission measurements, you may consider adding a buffer amplifier to the TX port. Of course, this will cause overload at low attenuation levels.

8. Audio crosstalk between left and right audio channel on your sound card my also degrade your dynamic range. The reason is, that there is always the very strong reference signal on one channel, while the RX signal might well be 90+dB down. If the sound card only offers 70dB interchannel isolation, you are stuck to 70dB dynamic range. You can test the interchannel isolations and more with the RightMark Audio Analyzer software from http://audio.rightmark.org and run a loopback test using the same audio cable as used on your VNWA. Also, the software Spectrum is useful for analysis of the VNWA audio signals. Take a look at the audio spectrum from the VNWA. Setup a L+R view side by side, startup the VNWA software and go into the audio test setup screen. Toggle Thru/Reflect mode in the audio level tab and you will see the impact of the stereo seperation.

From Software version V34.0 on below issues should be fixed.

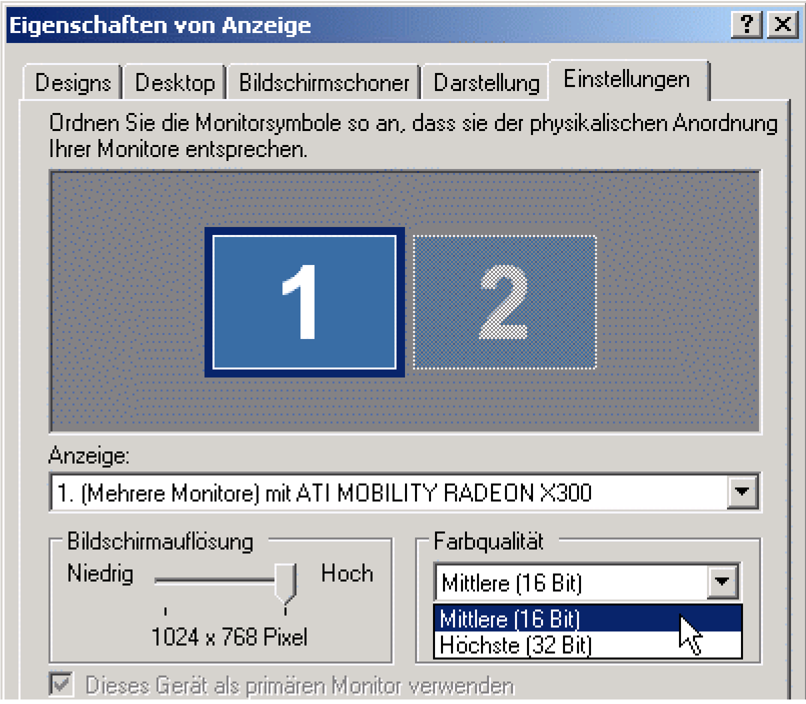
Status prior to software version V34.0

In some cases, the **trace markers are not displayed in a transparent way**, but rather with a white box around them as can be seen below.



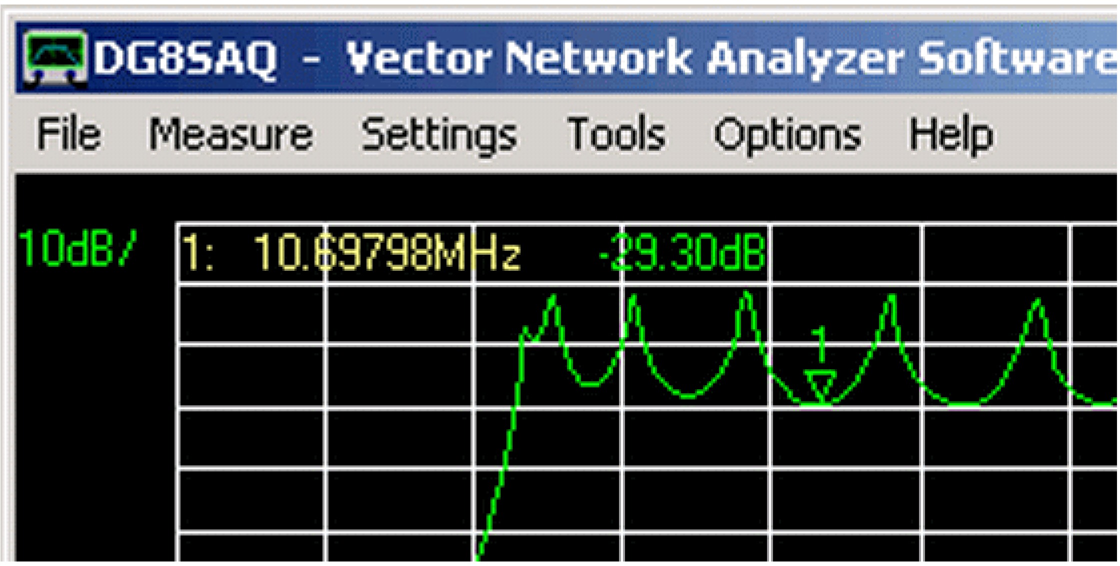
The root cause is not clear to me. One way to remedy this is to reduce the color depth of Windows via the control panel to < 32bit:

Select a color depth of 16 bit:

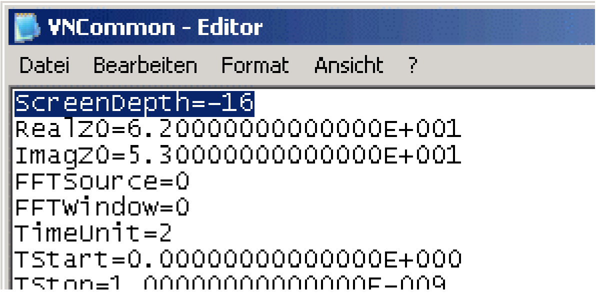


▸▸▸ **Note:** You must **terminate and restart the VNWA software** after this change!

Now, the markers are displayed correctly:



An alternative approach is to let **VNWA automatically switch the color depth** to 16 bit upon program start and restore the original value upon program termination.
To enable this hidden setting, you must manually edit the file **VNCommon.ini** found in the program directory with a text editor, e.g. with the Windows Notepad:



Change the first line to
ScreenDepth=16

▸▸▸ **Note:** No **blank characters are allowed** within the ini-files.

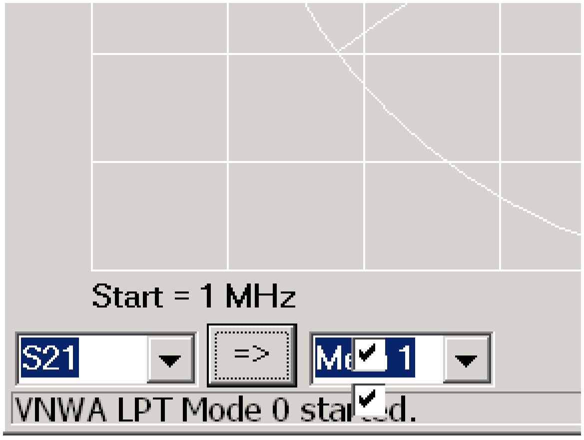
Positive values specify the desired color depth. With negative values, the command will be ignored.

▸▸▸ **Note:** Some monitors require some time for syncing upon program start, if automalical switching is activated.

From Software version V32 on below issues should be fixed. V32 onwards allows for *96dpi fonts (standard)* as well as *120dpi fonts (= big fonts)*.
Also, *arbitrary Windows styles* are supported.

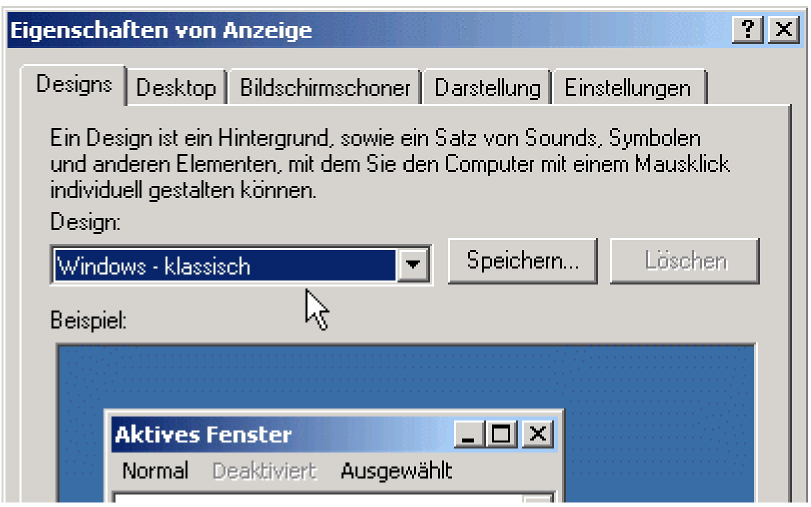
Status prior to software version V32

For some display settings, software controls appear in odd positions or even not at all, as can be seen for the case of the two trace select checkboxes in below example:

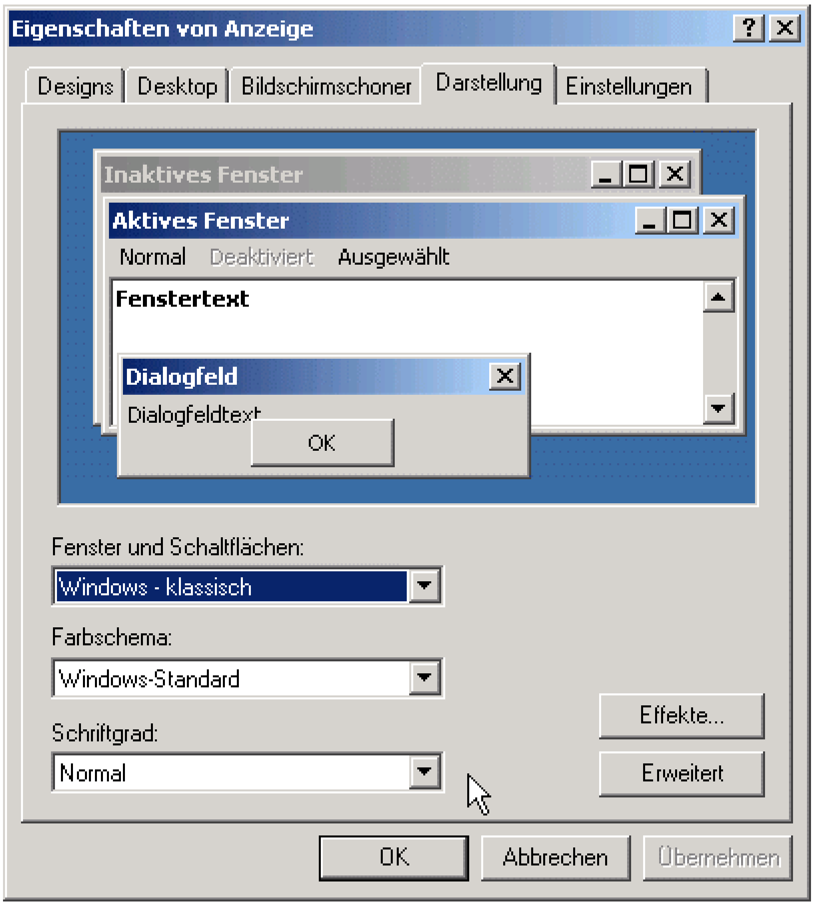


This is caused by the fact, that VNWA was designed for a certain Windows style and VNWA cannot adapt to dramatic style changes like increased font size.

In order to display all VNWA windows in a proper way, it is recommended to choose the following display settings in the Windows control panel:

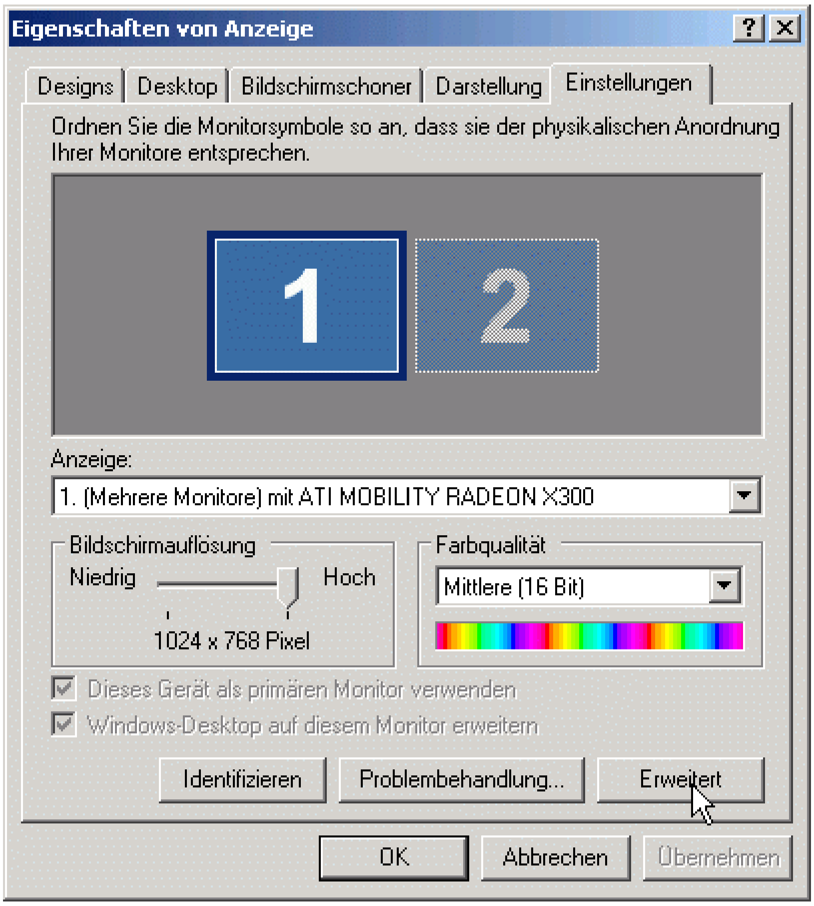


I do use the **Windows - Classic**, but I have verified proper appearance with the **XP-Design** as well.

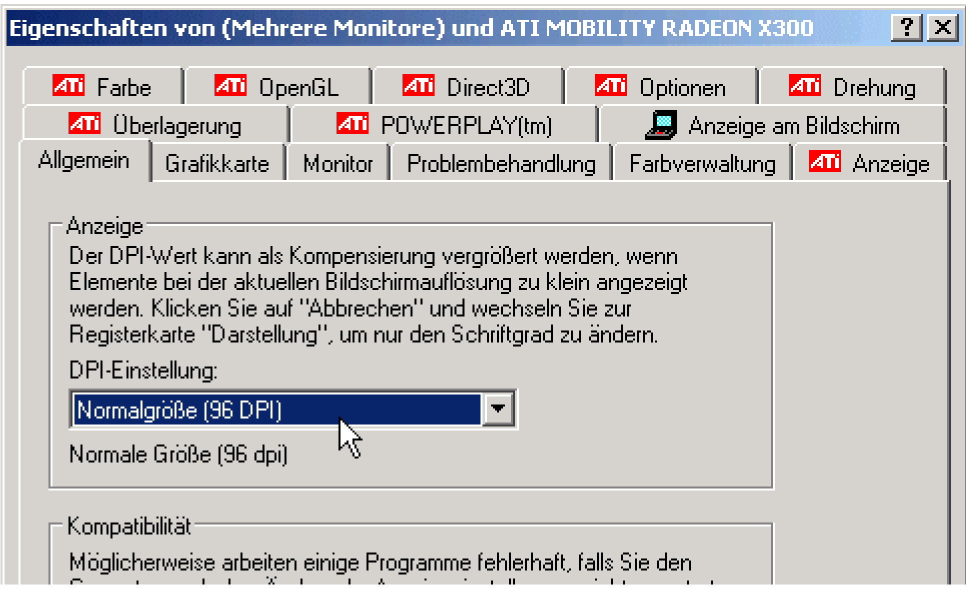


An inportant setting is the **Font Size** setting above, left of the mouse arrow. You must set this to **normal**. If a bigger font size is selected, all controls become bigger and won't properly fit onto the VNWA windows any longer. Todate, VNWA cannot handle different font sizes.

There is another setting, that influences the font size. In order to access it, you must select the extended display settings:



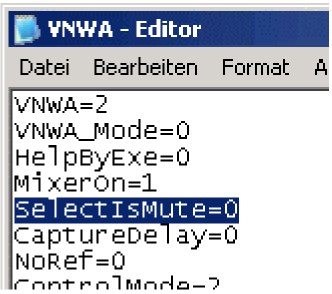
In the general display settings, the **DPI Settings** must be set to **96 DPI**!



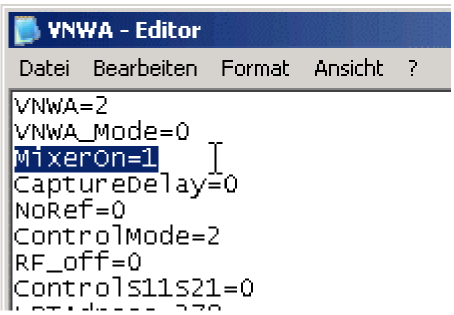
Windows Sound Mixer Issues

Upon program start, VNWA is designed to enable the selected audio capture device and set the volume controls to the previously stored values.
Depending on sound card drivers, this procedure sometimes fails, e.g. for some drivers, the **capture device is *deactivated* instead of activated upon program start**.

One way to cure this is to edit the file **VNWA.ini** found in the program directory with an ASCII editor like the Windows Notepad. Change the line "**SelectIsMute=0**" (see below) to "**SelectIsMute=1**" . The VNWA software must not be running while doing this modification.



An alternative approach is to **disable the Windows sound mixer** altogether.
In order to do so, you must edit the file **VNWA.ini** found in the program directory with an ASCII editor like the Windows Notepad:



Change the line
MixerOn=1
to
MixerOn=0

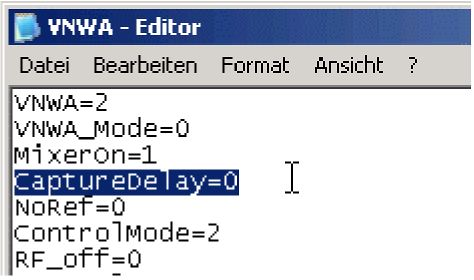
▸▸▸ **Note:** No **blank characters are allowed** within the ini-files.

By doing so, VNWA will not touch any sound settings any longer except for selection of the capture device. In this case, you must activate the capture device manually and use the Windows sound mixer or an equivalent software of your sound card's manufacturer to adjust the recording volumes.
Note, that this switch will have no effect on Windows Vista, as **the mixers are always disabled on Vista**.

Capture Startup Issues

Some sound cards fade in the volume upon capture start. Clearly, this is poison for this accurate measurement system, as the measurement results around the start frequency are questionable in this case.

The hidden Capture Delay software switch is a workaround to bypass this problem. In order to activate it, you must edit the file **VNWA.ini** found in the program directory with an ASCII editor like the Windows Notepad:



Increase the number in the ***CaptureDelay*** line. A value of 100 means, that the acual measurement is delayed by 100 milliseconds after opening the capture device. Of course, this will slow down the sweep rate for continuous sweeps by the specified value.

Here is a systematic step by step guide to verify proper interaction of the VNWA and the corresponding software.

0. Are you running a supported operating system?
- NOT supported: Win95 and older
- partially supported: Win98, Win98SE, WinME (USB mode only, NOT LPT mode)
- Fully supported: Win2000, WinXP, Win Vista, Win 7
- The following OSes require special settings: Windows Vista, Windows7. Consult the Vista/Win7 information in this helpfile.
- Currently it is not possible to install unsigned LPT and LibUSB drivers to Vista 64bit and Windows7 64 bit

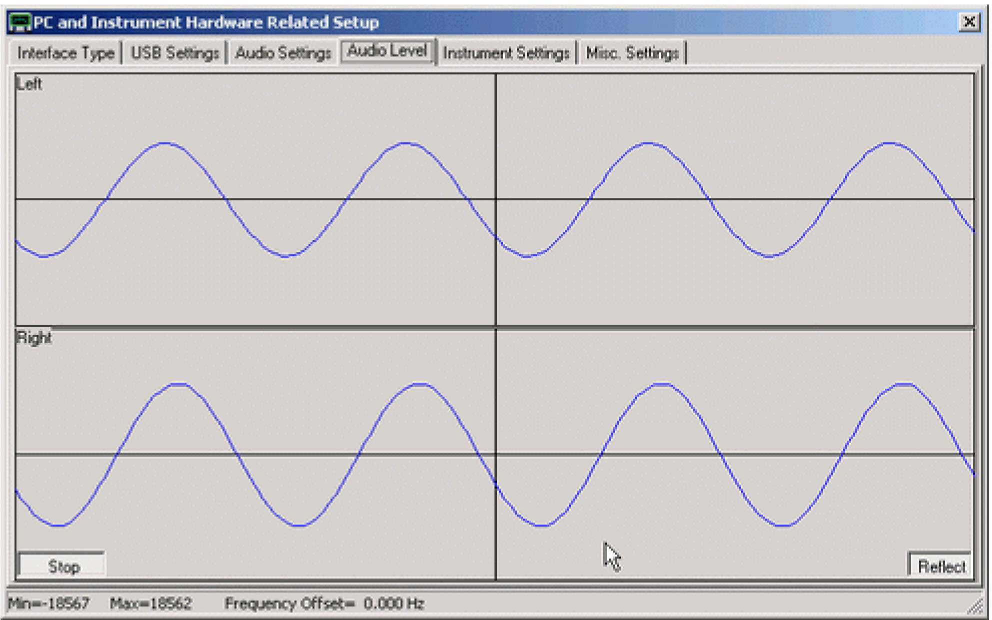
1. Start with a fresh installation of the latest VNWA software release to be found in the VNWA Yahoo forum at http://groups.yahoo.com/group/VNWA/, section "Files", folder "DG8SAQ".

2. Start the software and perform the setup as described in section "Getting Started".
The tests assume, that nothing is connected to the VNWA test ports unless stated otherwise.
Also, all calibrations should be switched off unless stated otherwise.
The software is to be run in VNWA mode, NOT in spectrum analyzer mode.
Averaging must be switched off.
All of the above will automatically be fulfilled for a clean new installation.

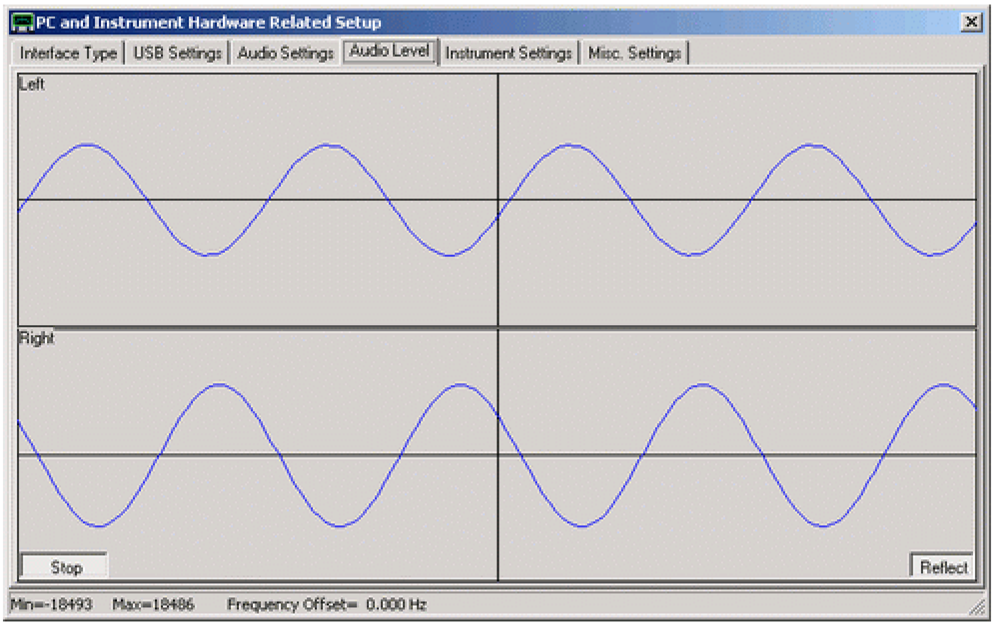
Check the following:
- correct sound card and capture device selected?
- correct reference channel selected?
- audio level set to about 50% of maximum level?
- does the Audio Level screen show two sine-waves for reflect and one sine wave for through?
- auto clock multipliers selected if overclocking is desired?

A first simple functional test at the predefined test frequency of 5MHz can be performed in the setup - **audio level tab** .

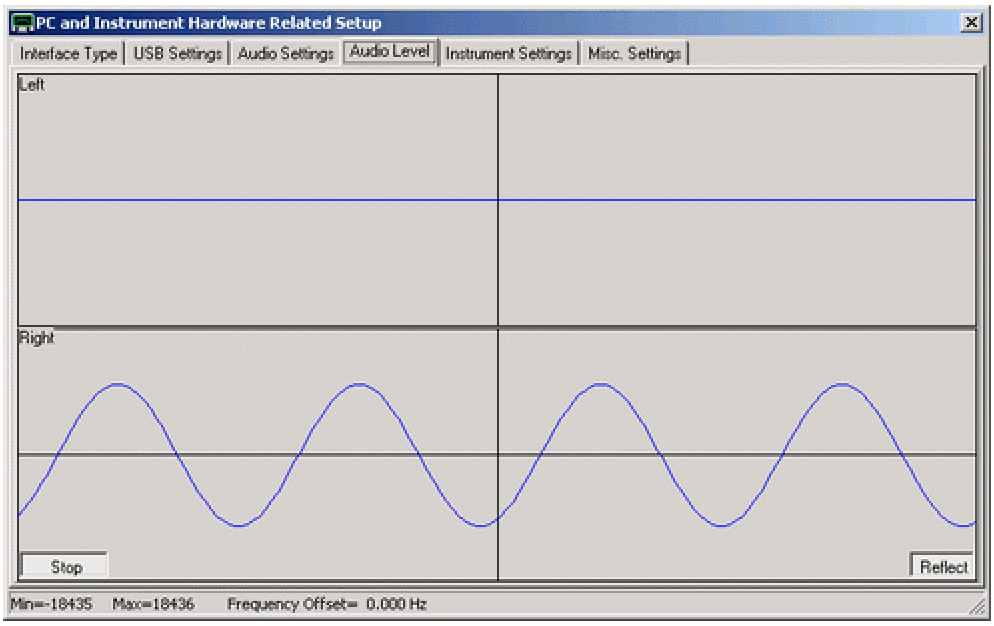
a) Press the **test audio** button and make sure you have selected **reflect** and nothing is connected to the VNWA TX port. You should see two similar sine waves. One sine wave is the Reference signal, the other is the Reflect signal, which is in phase to the reference signal for the open TX port:



b) Next, connect a short to the VNWA TX port. This will cause the Reflect signal to be phase shifted by 180° as can be seen below:



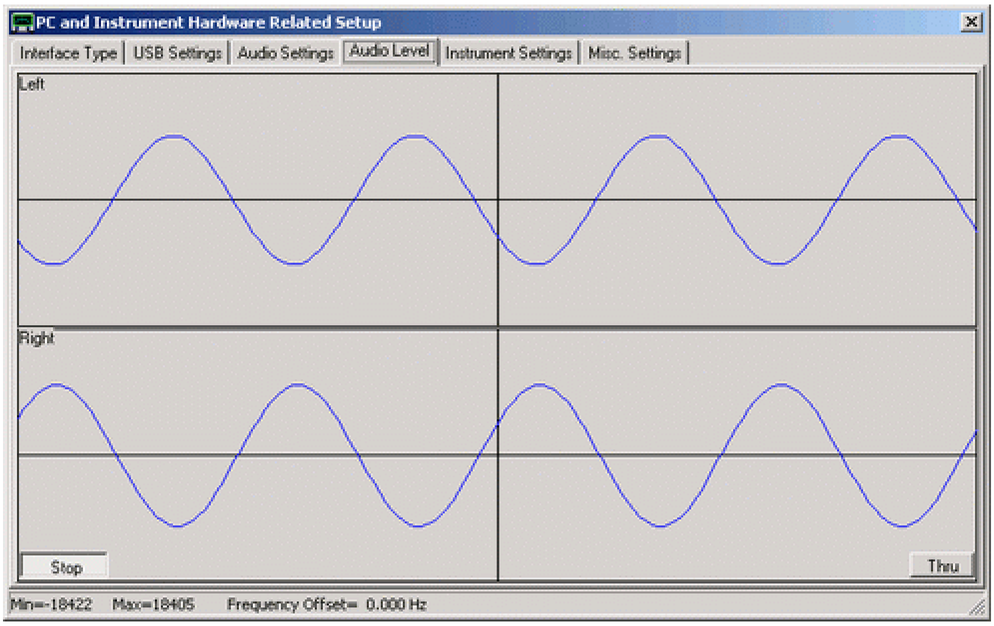
c) Next, connect a 50 Ohms load to the TX port. As the load absorbs all incident power, the reflect signal becomes zero:



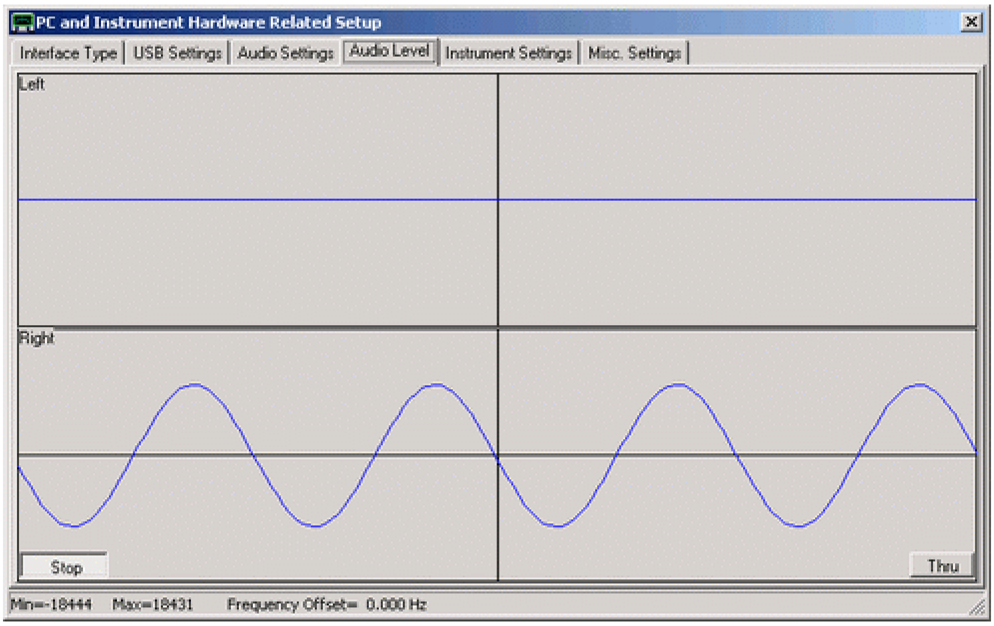
With the above tests you have verified that the reflection bridge works properly. From the last image (load) you can deduce, that the right (bottom) channel is the reference channel as it never changes.

d) Connect TX port with RX port by means of a short coaxial cable. As the RX port is designed to have 50 Ohms impedance, you should obtain the same result as in c) above.

e) Next toggle the button showing reflect to **Thru**. On the top, a sine wave appears again, which is the Thru signal transmitted through the coax cable from d). These happen to be out of phase which is of no relevance for the proper function of the VNWA:



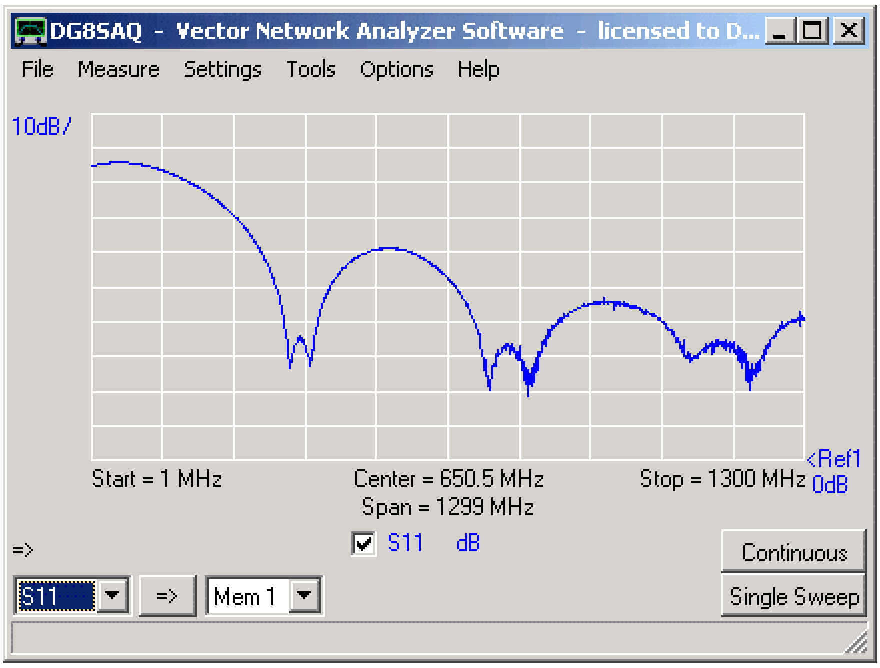
f) Disconnect the coax cable between TX and RX port. You should see the upper thru signal disappear, as no signal is fed into the RX port any longer:



Tests d)-f) prove, that the RX signal chain works properly.

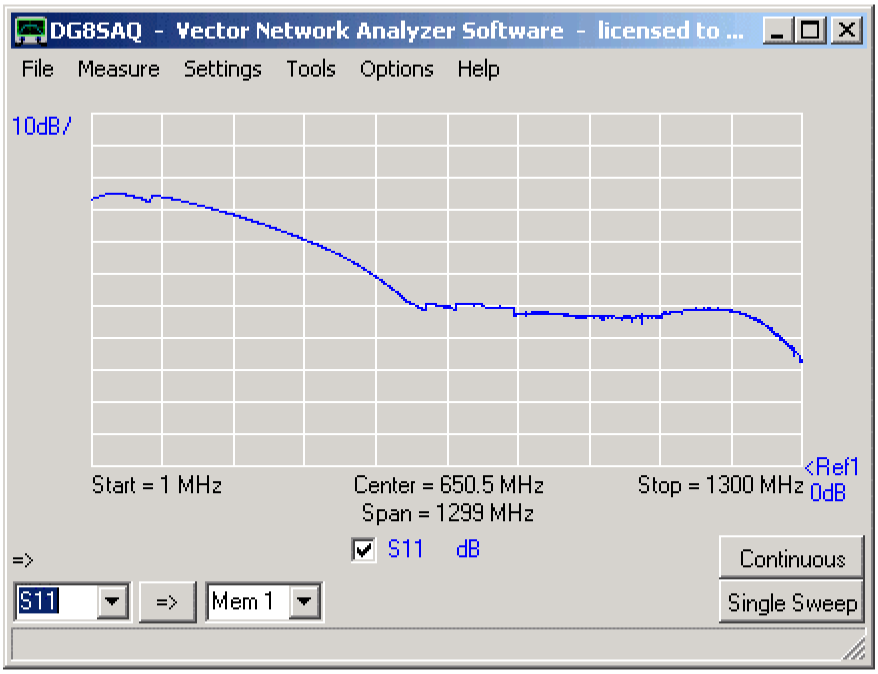
▸▸▸ **Note:** Should you always see the same sine wave on both channels, then your operating system (Vista, Windows7) assumes, that your sound device is mono. You must instruct Windows to open it as a stereo device in this case.

3. Proper operation of audio capture
a1) Go to Options-Setup-Misc Settings and tick "do not normalize to reference channel". Note, that the phase information will be lost with this setting and thus Smith charts do not make sense.
Close the setup, remove all connectors from TX and RX ports, display S11 only in dB mode and perform a single sweep with about 1ms per data point and 1000 data points. Using the **default clock multipliers of 10/11**, the result should look like below:



Dips occur at integer multiples of the DDS clock frequencies, i.e. n*10*36MHz and n*11*36MHz, where the DDS output power drops to zero. To overcome these signal dips, **select auto clock multipliers**.
▶▶▶ **Warning:** You are using **auto clock multipliers** at your own risk. By doing so, **you select to severely overclock** your DDSes from the allowed 400MHz to up to 750MHz.

a2) The same test as in a1), but using auto clock multipliers looks like this:

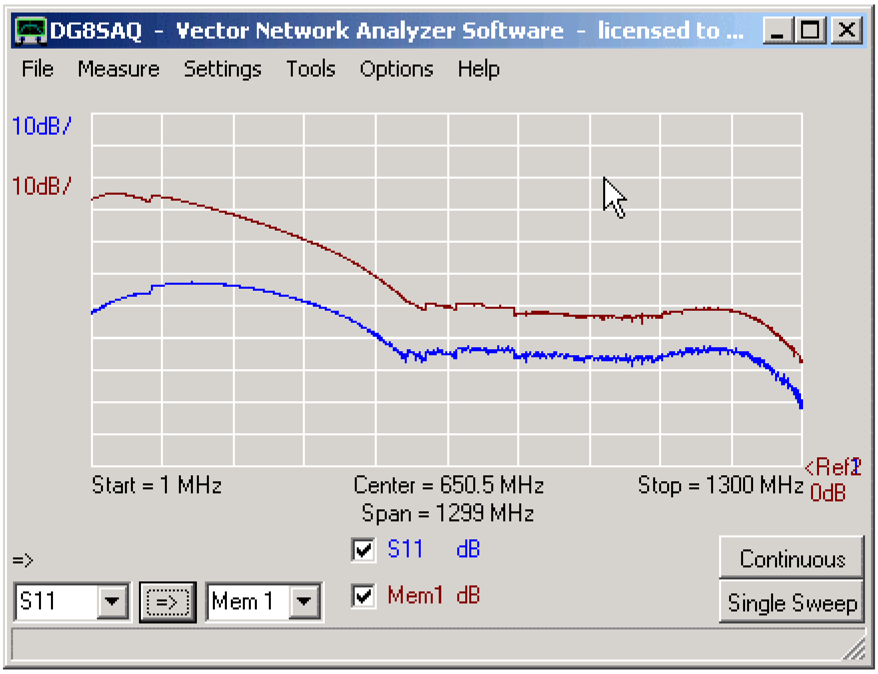


The dips are gone and you see steps, where the clock multiplier values abe being switched.

All the following results are obtained by using auto clock multipliers.

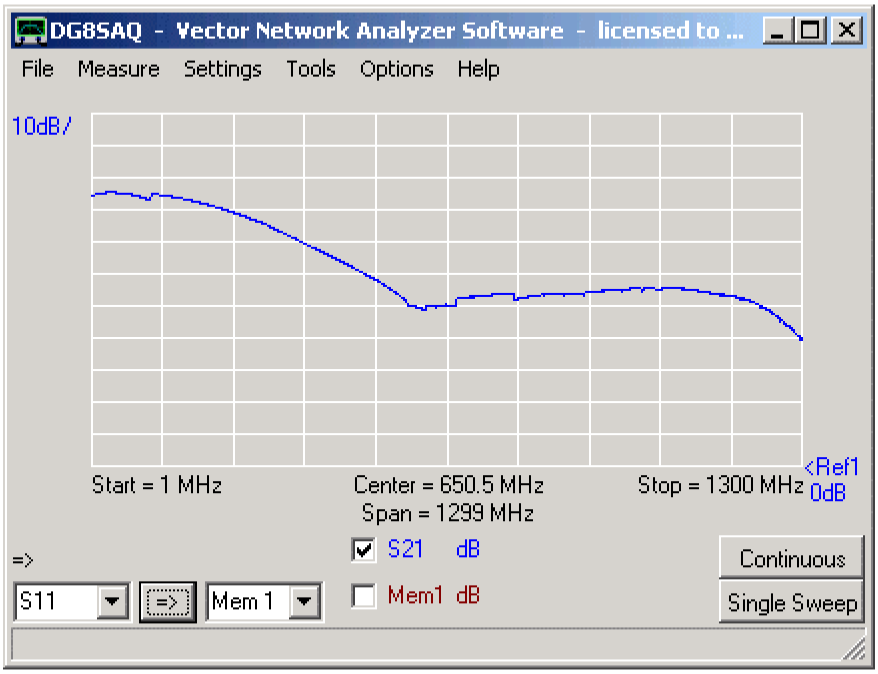
Save the sweep in a2) to Mem1, display Mem1 as well and repeat the sweep several times to verify that the result is stable and repeatable.
This plot directly shows the available signal amplitude in ADC readings. Note, that the plot shown is valid for a 16bit sound card. If a 24bit sound card is used, amplidudes will be higher by a factor of 256. Also note, that the steps in the trace prove proper operation of auto clock multiplier switching if selected.

b) Next, connect a 50 Ohms load to the TX port and repeat the sweep:



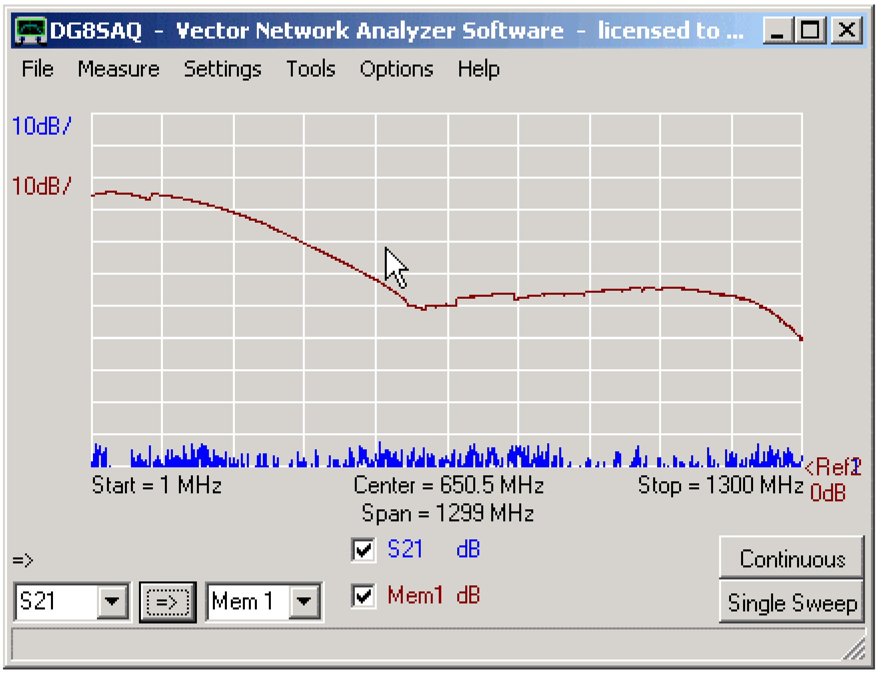
The trace should move down in amplitude. The difference between the previous trace (Open) and the current trace (Load) is the directivity of the VNWA bridge. The above plot proves, that the bridge is operational. If you see no trace change when sweeping the load, you might have missed to select the correct audio channel as reference channel or you might have selected the wrong polarity setting for the S11/S21 switch.

c) Next, connect VNWA TX port with the RX port by means of a coaxial cable. Select S21 and do a sweep:



The result should be similar to the S11 result with open TX port. The trace shows the available signal in transmission.
Save the sweep to Mem1, display Mem1 for comparison and repeat the sweep several times to verify stability and repeatability.

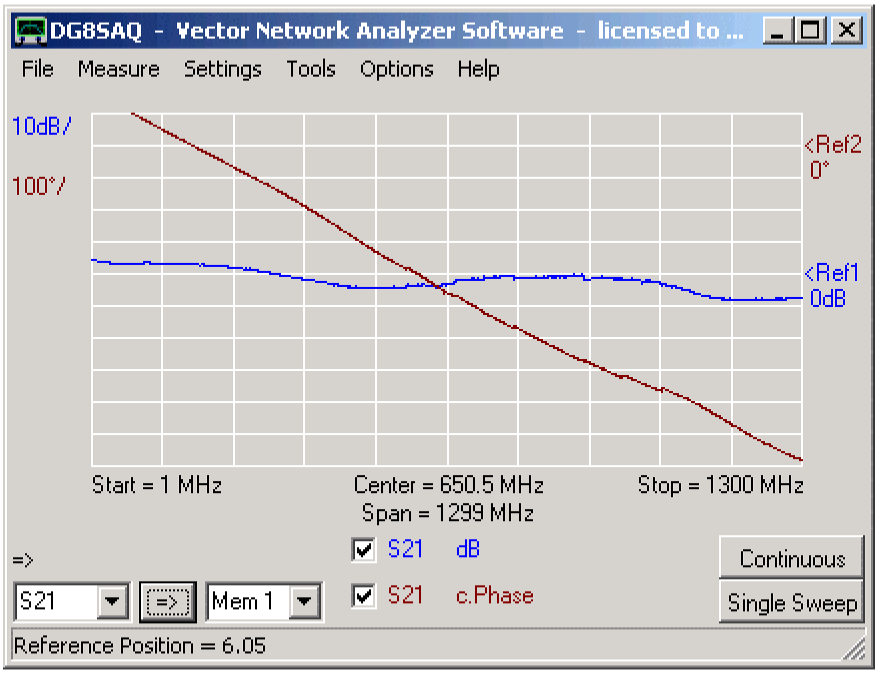
d) Next completely remove the coaxial cable from TX to RX and sweep again:



This test shows the noise floor of the RX. The achievable dynamic range in transmission is the vertical distance between the two traces. Note, that the test was performed at a high sweep rate and thus the noise floor is rather high. This test proves that the RX is operational.

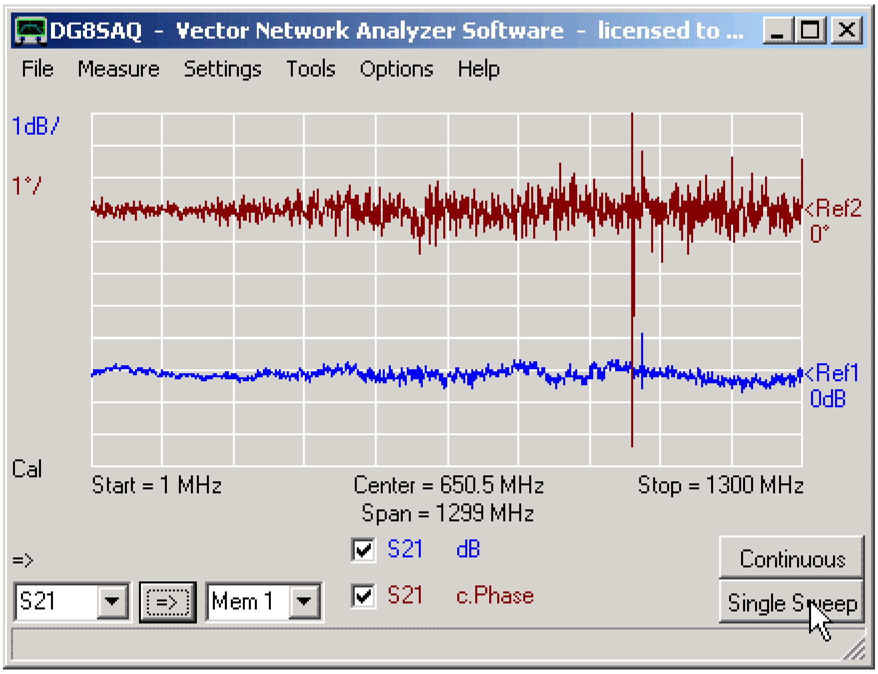
4. Proper operation of calibration
For the following tests you **MUST UN-CHECK "don't normalize to reference channel"** in the setup-misc settings.

a) Connect TX port with RX port and do an S21 sweep. Now the result should look similar to this:



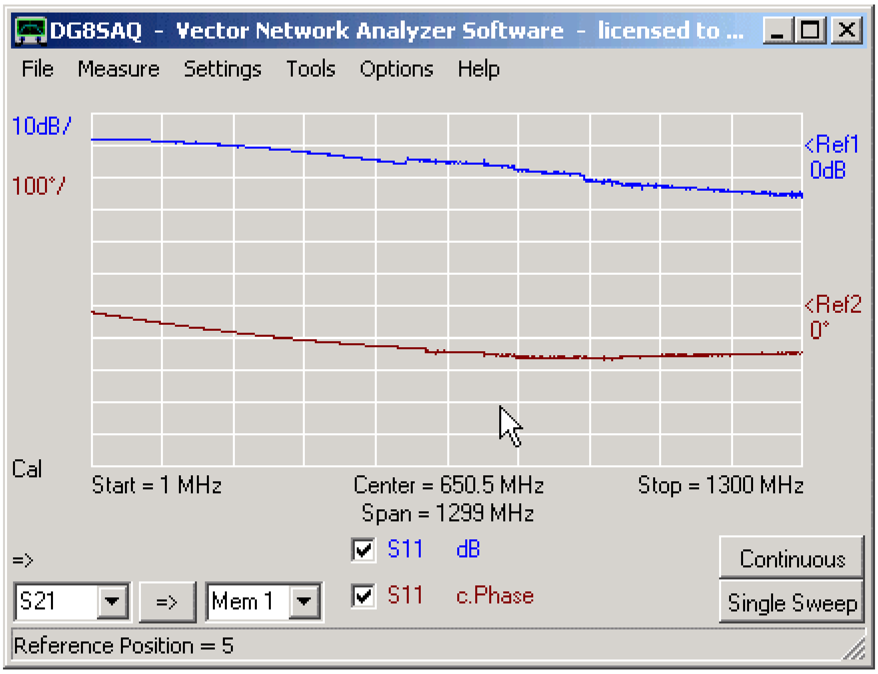
The amplitude should be around 0dB and there should be phase information available.
This proves that the reference signal chain is operational. Repeat the sweep several times to verify that the result is stable and repeatable.

b) Next, do a Thru-calibration (Thru only, no Crosstalk, no Thru Match Calibration!) and repeat the S21 sweep after the Thru-calibration. You should get two straight lines for amplitude and phase then:



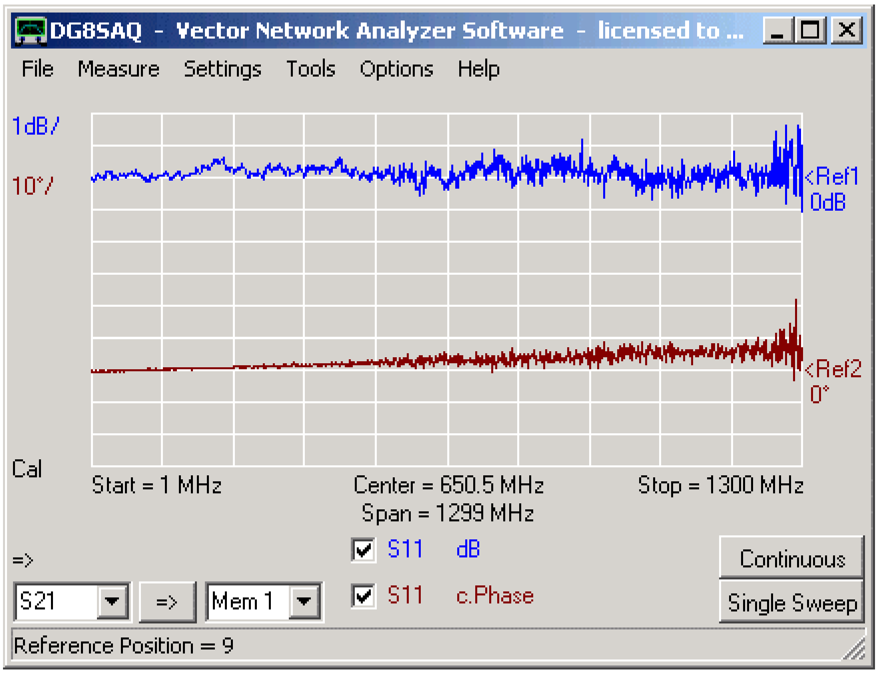
Repeat the sweep several times to verify that the result is stable and repeatable.

c) Next, disconnect the coaxial cable, display S11 and perform a sweep. Note, that you have not done a reflect calibration yet. This is what you should approximately get, an amplitude around 0dB and a fairly constant phase:



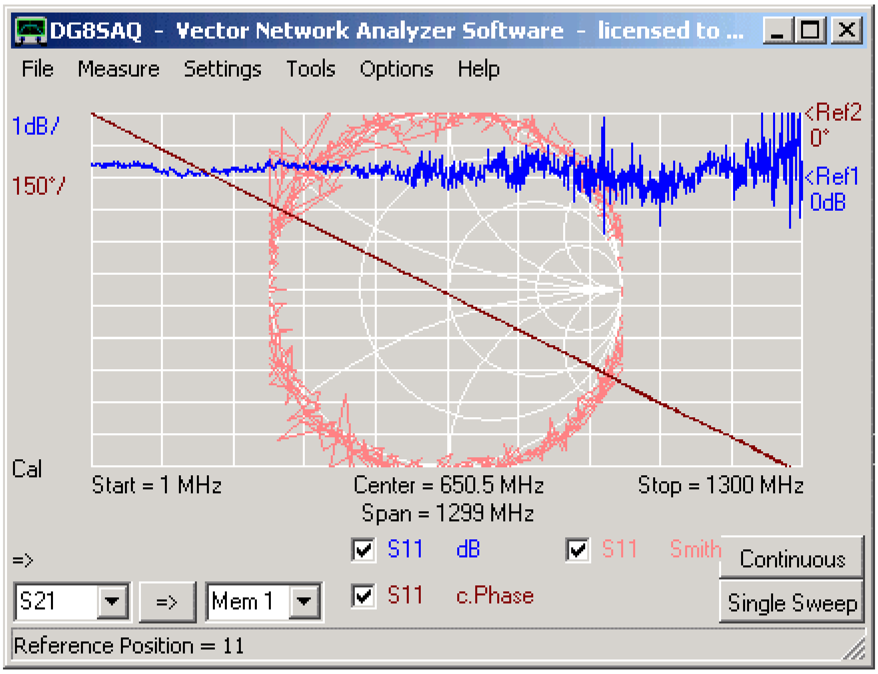
This proves that normalizing also works for the reflect signal. Repeat the sweep several times to verify that the result is stable and repeatable.

d) Next perform an SOL calibration and repeat your sweep of the open TX port. You should get two straight horizontal lines at 0dB and 0°:

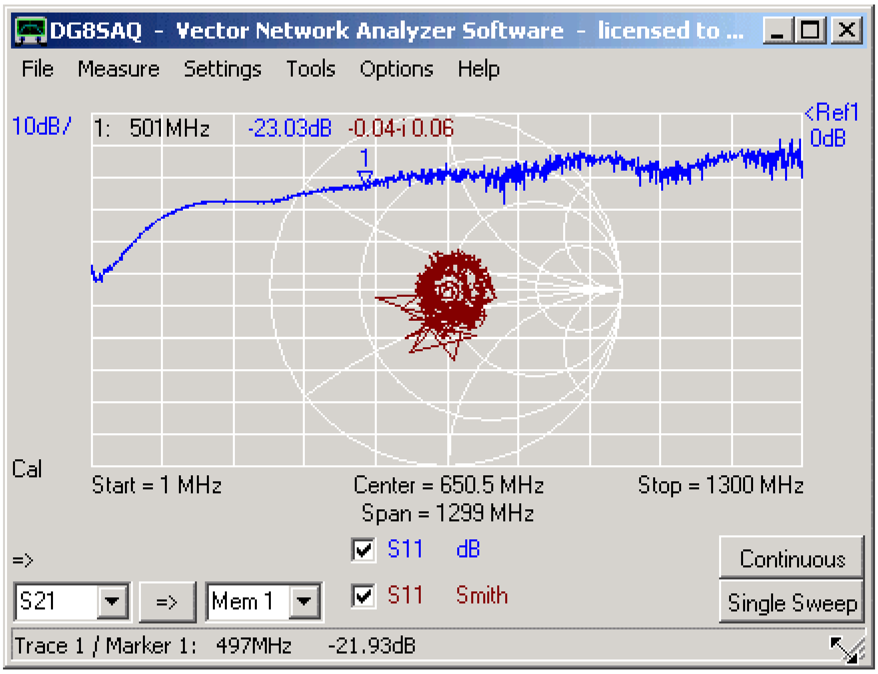


This proves that the reflect calibration is operational. You should also verify, that you can reproduce the reflection coefficients of your other two cal standards (Short, Load).
Repeat the sweeps several times to verify that the results are stable and repeatable.
▸▸▸ **Note:** For the **Load**, the phase information will be arbitrary, as the reflection coefficient of an ideal load is zero, which has arbitrary phase. Or put in math: 0*exp(j*ß) = 0 for any angle ß.
So, all phase information you will get to see for the load is noise. Simply ignore it.

e) Next, connect a short coaxial cable (about 10cm) to the TX port and leave the other cable end open. Sweep and observe, that you should still obtain a magnitude of 0dB but a linear phase increase which leads to a circular trace in the Smith chart:



f) Next connect the TX port with the RX port by means of the short coax cable and perform an S11 sweep. This way, you measure the input reflection coefficient of the RX port:



The RX reflection coefficient should be better than -20dB up to 500MHz.

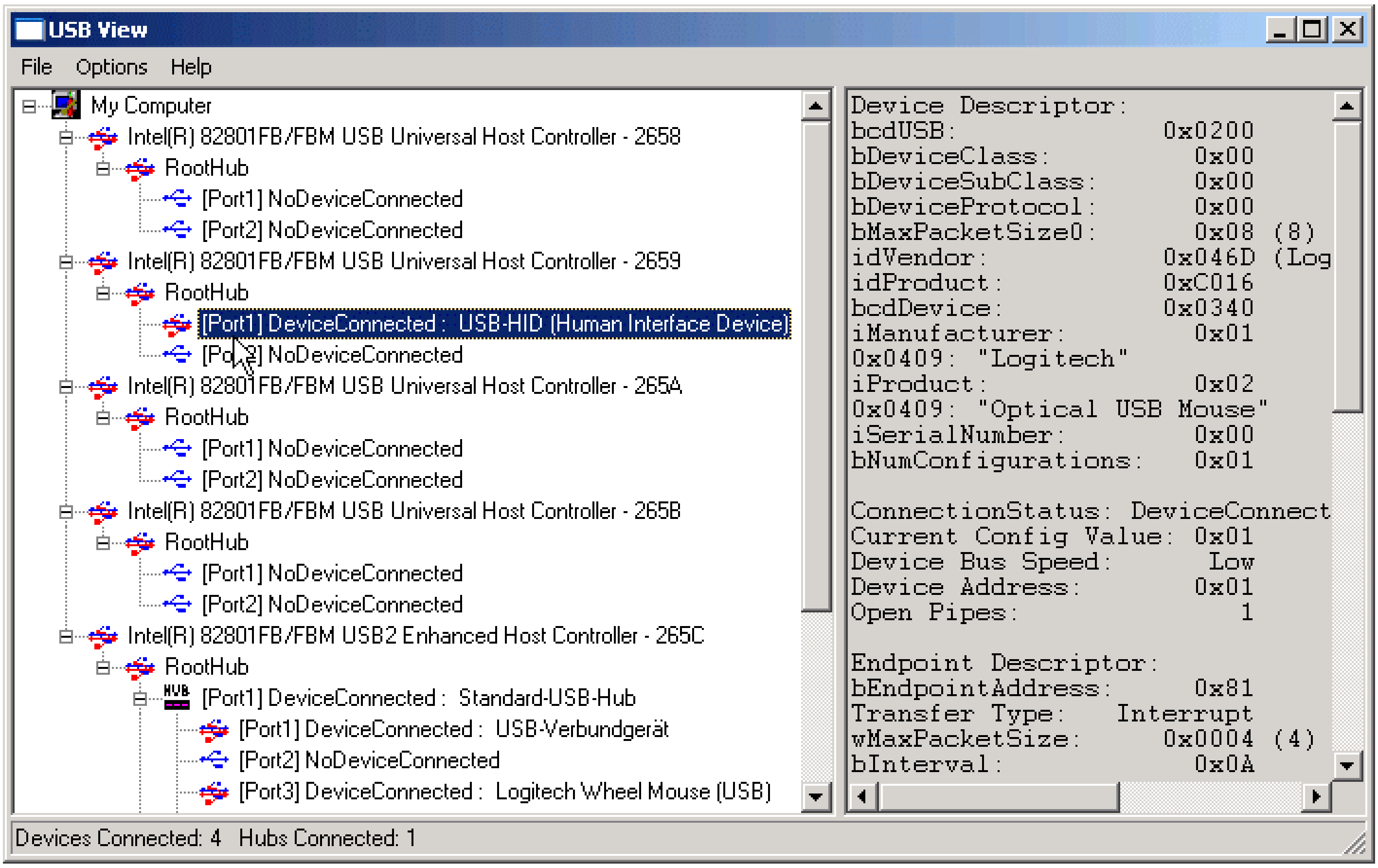
If your tests fail at a certain step, please cite the exact step, at which the test procedure failed.

Using the Microsoft utility **"USB View"** it is possible to detect problems of USB devices. In case the DG8SAQ USB_VNWA device is not properly detected by Windows, do the following diagnosis procedure.

Troubleshooting procedure

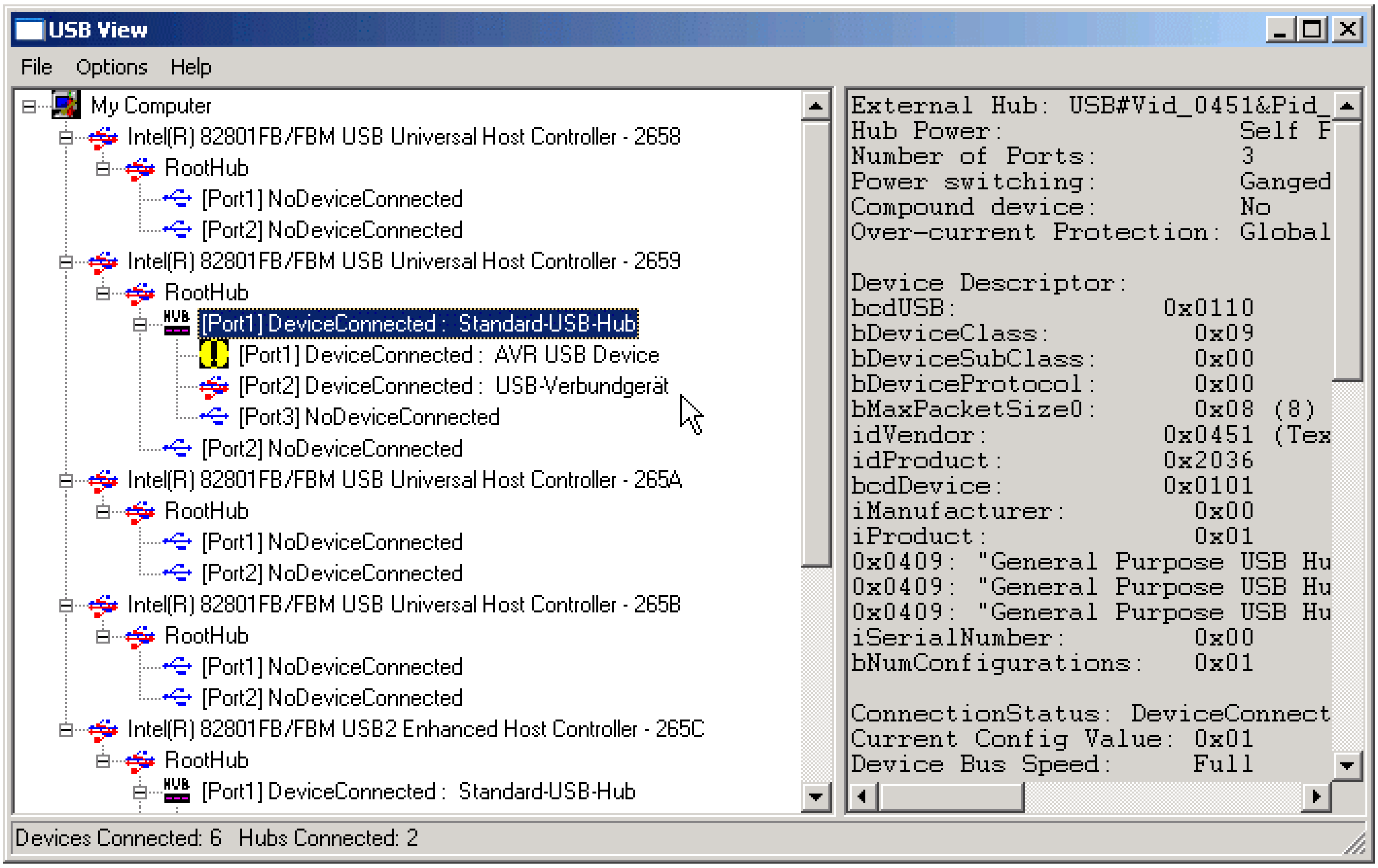
1. Download the Microsoft utility "USB View".

2. Start "USB View" and find the USB outlet in "USB View" where you want to plug in your USB_VNWA interface by e.g. plugging in a known good USB device like a mouse or a memory stick. Make sure you activate the options "Auto Refresh" and "Config Descriptors". In below example a USB_HID mouse is detected. The grey field on the right gives more details about the highlighted USB device. This is proof that your USB PC interface works.



3. Plug in your USB_VNWA interface into the very same USB outlet and observe the change in "USB View". As seen below, the following devices should be detected:
- a standard USB hub
- an AVR USB device (the VNWA controller)
- a USB audio codec

Make sure, the display of "USB View" is refreshed after replugging.

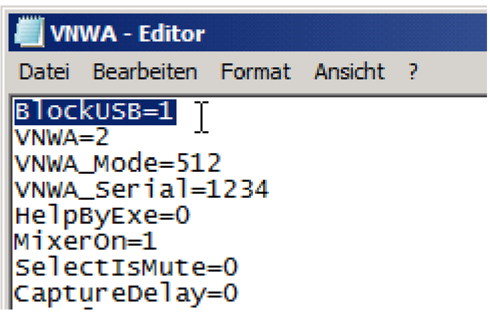


Should at least one of the above devices fail to show operational, then you have likely a hardware problem on your USB_VNWA interface.

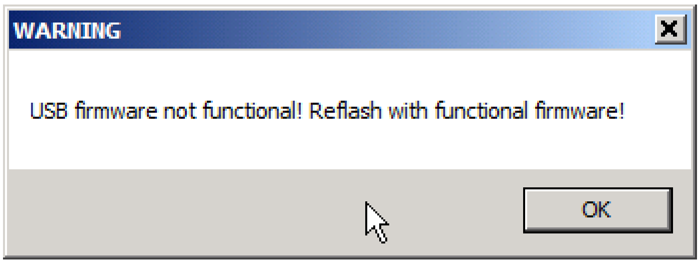
If the above tests perform positive and yet Windows refuses to detect a new device, then you might be confronted with a Windows driver problem. You could try to install the driver manually from the Windows control panel.

Dealing with corrupted Flash

If a power failure or a PC crash is encountered during flashing a new VNWA firmware, the firmware will become corrupted and non-functional. Since the USB core of the firmware is to some extent protected, there is still a chance that the firmware can be restored via USB. If the USB core is still functional, the AVR-device will show up in usbview, as shown above. Since the firmware is corrupted, VNWA will issue multiple errors upon program start. To avoid these error messages, open the file **VNWA.ini** with a text editor (e.g. notepad.exe) and add the line "BlockUSB=1":



Save and close VNWA.ini and restart the VNWA software. A message telling that the firmware is not functional will be shown:



Confirm and VNWA will start. Since all USB functions except reflashing and rescanning the bus are blocked now, VNWA will issue no more error messages. Go straight to "Setup"-"USB Settings" and try to reflash the firmware with the version that failed to flash previously.
If you are lucky, your VNWA will behave normally after closing and restarting the VNWA application then. This procedure has proved successful in one case already!

If your CPU load is 100% during the sweep, the collected data will be corrupt!

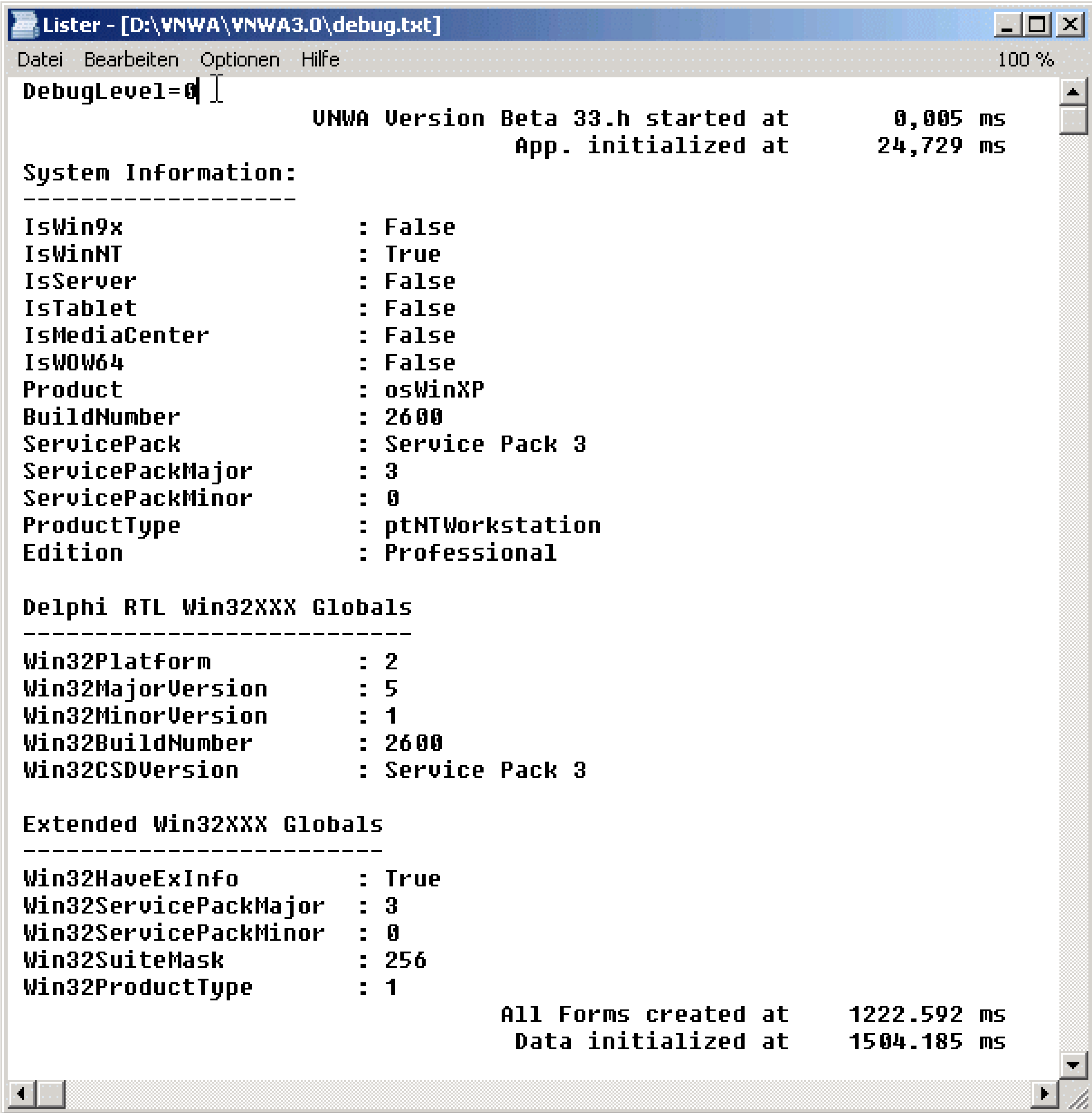
Here are a few things, you can do, if your PC is too slow:

1) Reduce the audio sampling rate
2) Increase the audio buffer size
3) Display as few curves during the measurement as possible
4) Use the smallest number of data points possible
5) Switch off progress bar and progress display
6) Set display background color to "none"

As the VNWA itself does not own any intelligence, all data capturing and analysis as well as the graphics have to be handled by the PC at the same time! The added intelligence in the DG8SAQ USB_VNWA interface relaxes the host PC CPU load considerably.

The VNWA software lets you **generate a debug file** which gives information about your Windows version, your VNWA hardware and its status.
The **debug output is activated** by simply creating an empty textfile with name **debug.txt** in the VNWA program directory:

Next, start VNWA and immediately terminate VNWA again and observe the now modified file debug.txt:

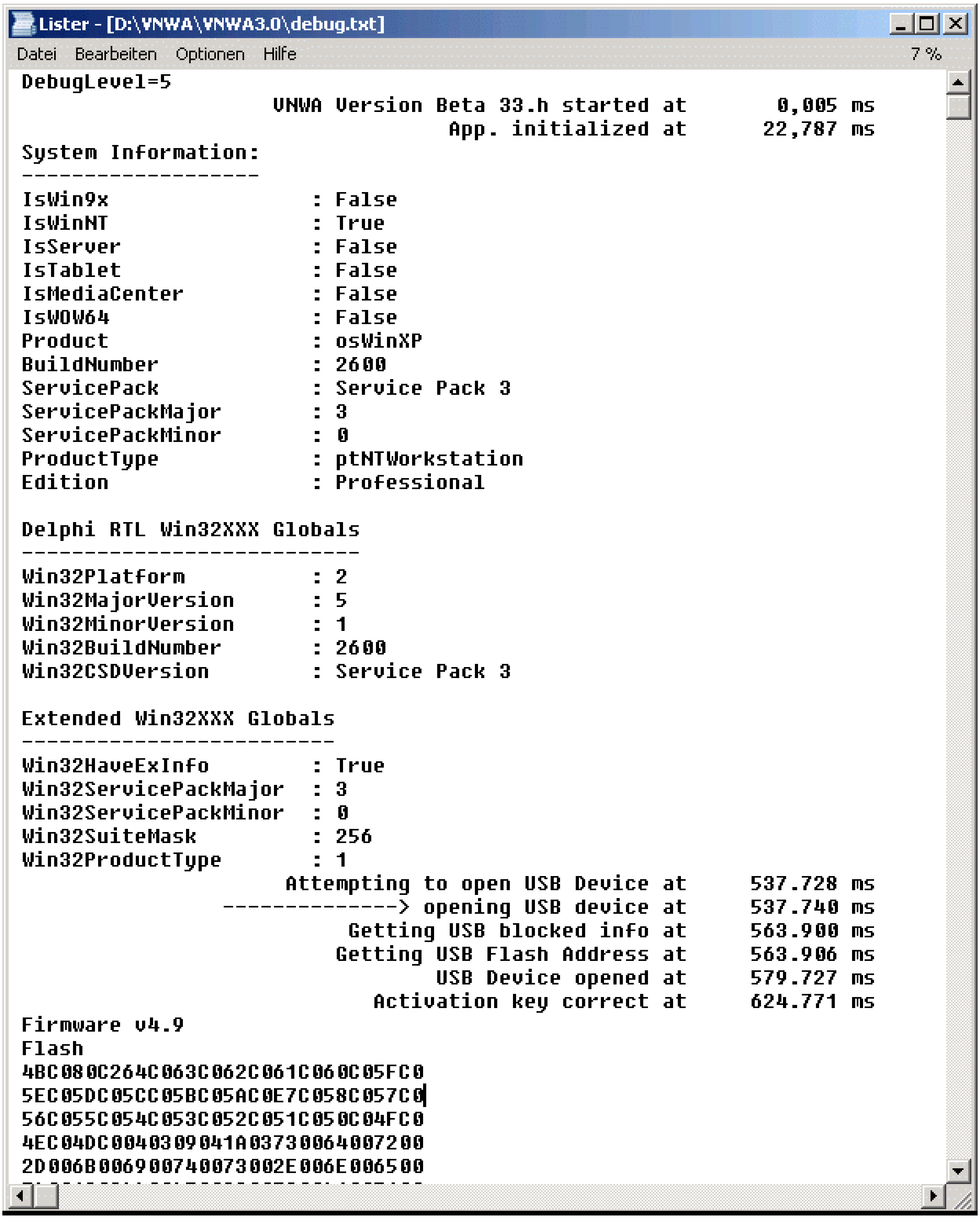



```
DebugLevel=0
                    VNWA Version Beta 33.h started at       0,005 ms
                                 App. initialized at       24,729 ms
System Information:
-------------------
IsWin9x                 : False
IsWinNT                 : True
IsServer                : False
IsTablet                : False
IsMediaCenter           : False
IsWOW64                 : False
Product                 : osWinXP
BuildNumber             : 2600
ServicePack             : Service Pack 3
ServicePackMajor        : 3
ServicePackMinor        : 0
ProductType             : ptNTWorkstation
Edition                 : Professional

Delphi RTL Win32XXX Globals
---------------------------
Win32Platform           : 2
Win32MajorVersion       : 5
Win32MinorVersion       : 1
Win32BuildNumber        : 2600
Win32CSDVersion         : Service Pack 3

Extended Win32XXX Globals
-------------------------
Win32HaveExInfo         : True
Win32ServicePackMajor   : 3
Win32ServicePackMinor   : 0
Win32SuiteMask          : 256
Win32ProductType        : 1
                                 All Forms created at    1222.592 ms
                                  Data initialized at    1504.185 ms
```

For **extended debug information**, edit the file debug.txt with a text editor (e.g. notepad) and change the first line from Debuglevel=1 to **Debuglevel=5**. Close the file and start and stop the VNWA software again. The debug file has grown considerably now. Besides your system and software version info it contains your firmware and a log of all USB communication:



Lister - [D:\VNWA\VNWA3.0\debug.txt]

Datei Bearbeiten Optionen Hilfe 7 %

```
DebugLevel=5
                    VNWA Version Beta 33.h started at       0,005 ms
                                  App. initialized at      22,787 ms
System Information:
-------------------
IsWin9x                : False
IsWinNT                : True
IsServer               : False
IsTablet               : False
IsMediaCenter          : False
IsWOW64                : False
Product                : osWinXP
BuildNumber            : 2600
ServicePack            : Service Pack 3
ServicePackMajor       : 3
ServicePackMinor       : 0
ProductType            : ptNTWorkstation
Edition                : Professional

Delphi RTL Win32XXX Globals
---------------------------
Win32Platform          : 2
Win32MajorVersion      : 5
Win32MinorVersion      : 1
Win32BuildNumber       : 2600
Win32CSDVersion        : Service Pack 3

Extended Win32XXX Globals
-------------------------
Win32HaveExInfo        : True
Win32ServicePackMajor  : 3
Win32ServicePackMinor  : 0
Win32SuiteMask         : 256
Win32ProductType       : 1
                     Attempting to open USB Device at     537.728 ms
                ---------------> opening USB device at     537.740 ms
                          Getting USB blocked info at     563.900 ms
                         Getting USB Flash Address at     563.906 ms
                                 USB Device opened at     579.727 ms
                            Activation key correct at     624.771 ms
Firmware v4.9
Flash
4BC080C264C063C062C061C060C05FC0
5EC05DC05CC05BC05AC0E7C058C057C0
56C055C054C053C052C051C050C04FC0
4EC04DC0040309041A03730064007200
2D006B00690074007300 2E006E006500
```

This extended debug info is very valuable for troubleshooting.

The T-check is a quick test invented by R&S to give an indication whether a calibrated VNA is yielding reasonable results.

See here for more details: **Rohde & Schwarz T-Check** http://www2.rohde-schwarz.com/file/1ez43_0e.pdf