The following chapter explains what a two port device is and how it is being described. Next, actual examples of how two port device measurements are being performed. Finally, the measured data is being analyzed in order to find out what we can do with it.

1. Principles of 2-Port device parameters and their measurement
2. Practical example on performing a 2-Port measurement
3. Interpretation and Usage of 2-Port measurements

Electrical two port devices are devices with two ports or two connectors, where a signal can be sent into one port and be detected at the other port. Typical examples are attenuators, amplifiers and filters.

The description of electrical two port devices follows closely the description of optical devices like a lens. A lens is also a two port device, where a beam of light shines onto one side of the lens. Some of the incident light gets reflected back, some of the light entering the lens will be absorbed and converted to heat, while the rest will be transmitted from the other side:
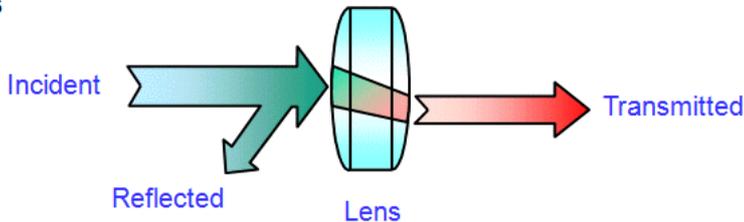
An electrical two port device behaves quite similarly, when inserted between two transmission lines, where the left one is guiding an electrical wave into the device. Some of the electrical power might get reflected at the device input (reflection loss), some will be converted into heat inside the device (dissipation) and the rest will come out of the device at the right hand side.

As a lens can be used to shine light through it it both directions, also an electrical two port device can be operated in reverse direction. Thus, waves may enter the device from both sides (incident waves). Likewise, waves will leave the device from both sides (evanescent waves). Electrical two port devices are being characterized in terms of these incident waves denoted by **a** and evanescent waves denoted by **b**. The two ports are being numbered (1=left, 2=right) and the waves carry subscripts denoting on which side of the device they exist, e.g. a1 is an incident wave (a) coming from the left (1).
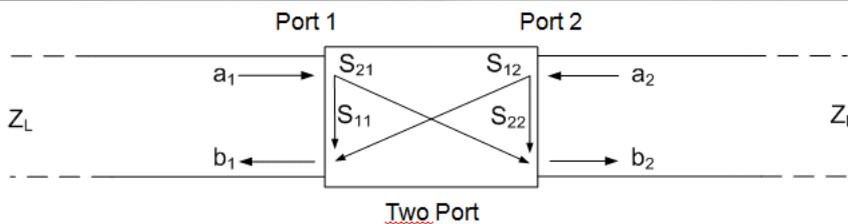
The complex wave amplitudes a1, a2, b1 and b2 are complex numbers that carry both the wave amplitude and the wave phase information.

For linear devices there are linear relationships between these:

$$b_1 = S_{11} \cdot a_1 + S_{12} \cdot a_2$$
$$b_2 = S_{21} \cdot a_1 + S_{22} \cdot a_2$$

The complex numbers S11, S12, S21 and S22 are called scattering parameters or S-parameters. These fully characterize the two port device for linear (=small signal) operation.

Now, we will investigate the meaning of these S-parameters.

Consider a situation where there is a wave a1 being sent into the left side port 1 of our two port device, but nothing is sent into the right side port 2, i.e. a2=0.
In this case the above equations simplify to:

$$b_1 = S_{11} \cdot a_1$$
$$b_2 = S_{21} \cdot a_1$$

These simplified equations allow us to interpret the S-parameters as well as to understand how to measure them.

Let's look at the first equation. Solving it for S11 shows, that S11 is the ratio of reflected wave b1 to incident wave a1 at port 1:
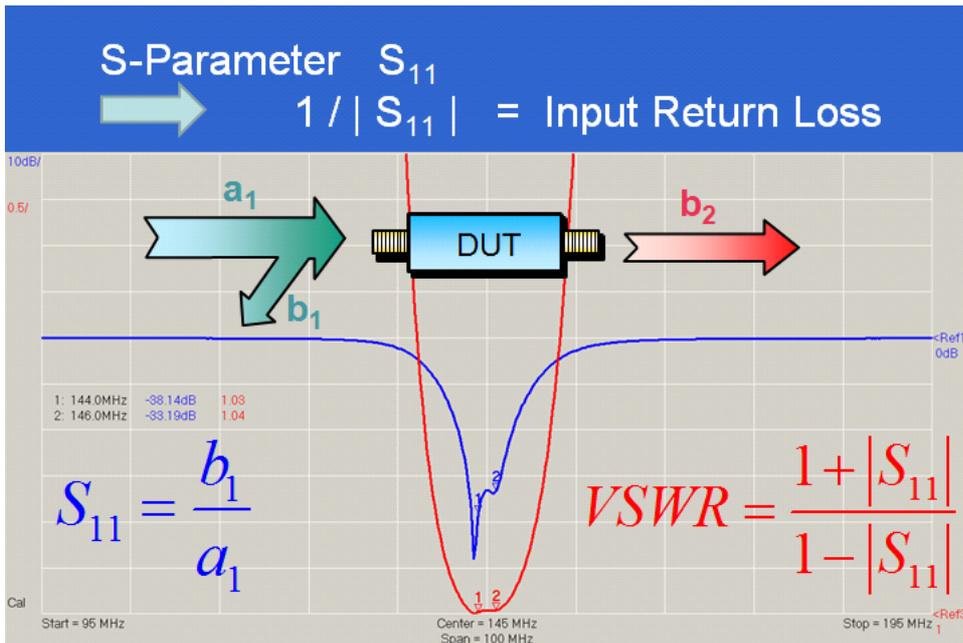
The inverse of the magnitude of S11 is known as input return loss. I.e. a device showing a return loss of 10dB will yield |S11|=-10dB. This means, the reflected wave is 10dB weaker than the incident wave or 1/10th of the incident power is being reflected. But, since S11 also carries the phase information, also the input impedance of our DUT cann be calculated from S11.

Solving the second equation for S21 shows, that S21 is the ratio of the transmitted wave b2 to the incident wave a1:
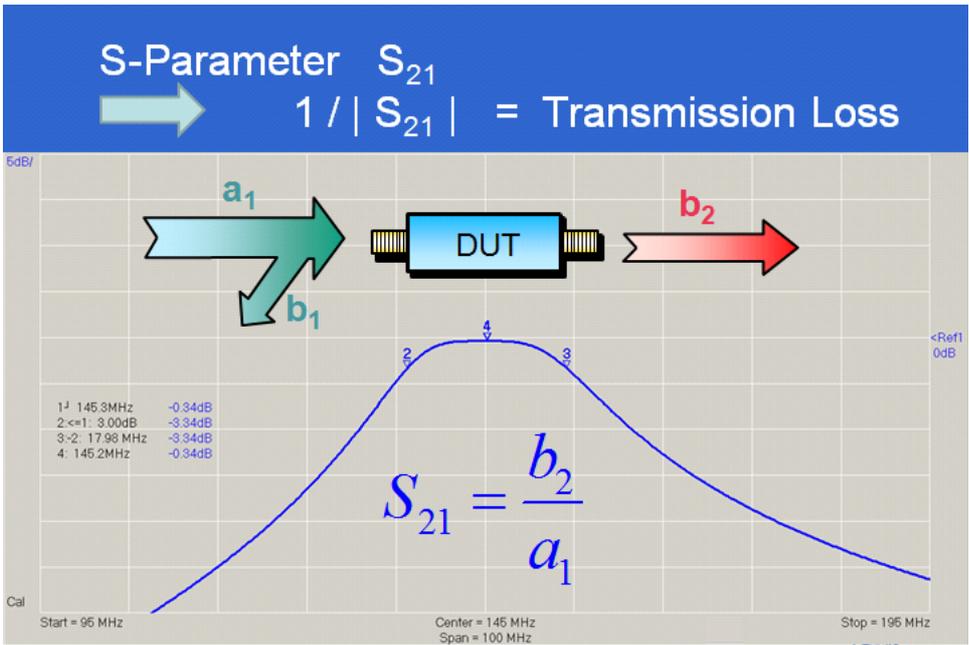
The inverse of |S21| is known as transmission loss. For example, a 10dB / 50 Ohms attenuator will show S21=-10dB.

Now, it is clear that in order to measure S11 and S21 we must feed power into port 1 and detect the incident wave a1, the reflected wave b1 and the transmitted wave b2.
This is exactly what the VNWA is doing:
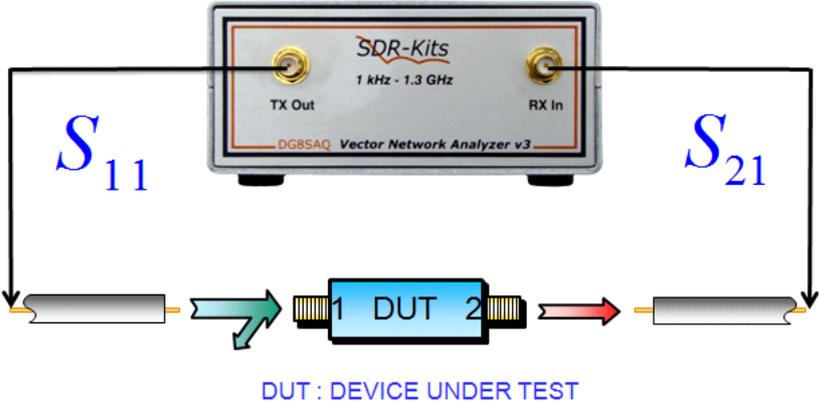
If we want to know S12 and S22, we can measure this in a similar fashion by sending power into port 2. As the VNWA in its standard version is unidirectional, i.e. only the TX port can deliver a signal, we have to interchange ports 1 and 2 of our test object, i.e. turn it by 180 degrees in order to measure S12 and S22:
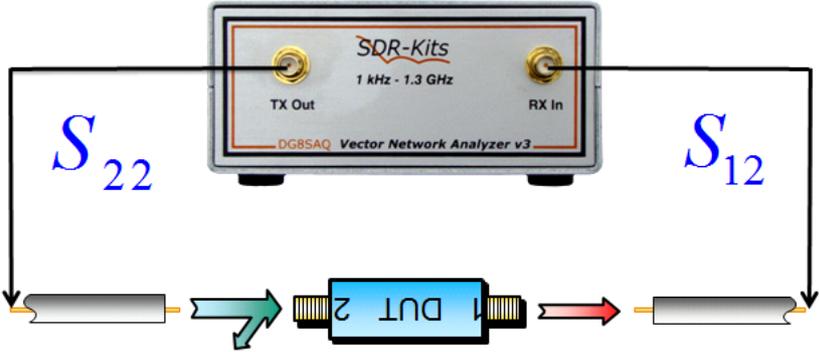
But why should we be interested in how our device performs in reverse direction, when we will only operate it in forward direction in the final application?
There are several answers to this.

Imagine, your test object is a low pass filter to be placed in between a power amplifier and an antenna. Clearly, only the power amplifier will feed power into the low pass filter and all that counts is how much power arrives at the antenna. So, S21 being a measure for the transmission loss is crucial. But, as we do not want power to be reflected back into the power amplifier, we have to make sure that amplifier and filter are impedance matched. So, S11 being a measure for the filter input impedance is crucial. Likewise, we want the antenna to be impedance matched to the filter output. Thus, S22 being a measure for the filter output impedance is crucial, too, even though we won't actively feed power into the filter output. If there is no perfect match between filter output and antenna, which will lead to signal being reflected back into the filter, S12 will become important, too.

Another scenario is a filter, which is to operate in the final application in an impedance environment very different from the 50 Ohms environment of the VNWA like a high impedance crystal filter. The VNWA cal only measure the filter S-parameters in its 50 Ohms environment. Once, the full set of S-parameters is known, it is possible to use these to simulate and optimize the filter performance in any final application with an arbitrary impedance environment. Such an example will be shown in page "Interpretation and Usage of 2-Port Measurements".

The following page "Practical Example on Performing a 2-Port Measurement" will present a step by step example on how the full set of S-parameters is being measured.

The following example demonstrates how the VNWA can be used to measure two port S-parameters of a narrow band 10.7 MHz two pole crystal filter.
We connect the filter using test cables and a test board between the VNWA TX port and RX port:
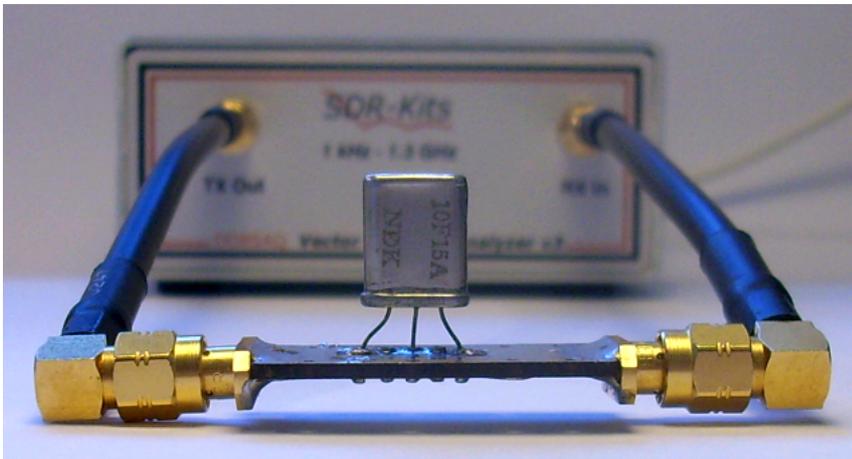
It is recommended that you number your test object ports to make very sure that you always measure in the proper direction:
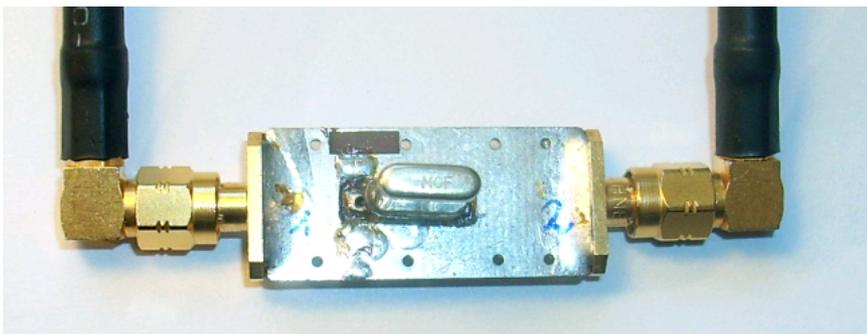
1. Setting up

First, we need to set a frequency range reasonably covering our test object.
To find it, we will do a first transmission measurement without calibration over a relatively large frequency span.
To make sure that we capture all interesting features, we will set a large number of data points (here 8000) and a the fastest sweep time (0.16ms per data point) by pressing the s-key or by selecting Settings-Sweep:
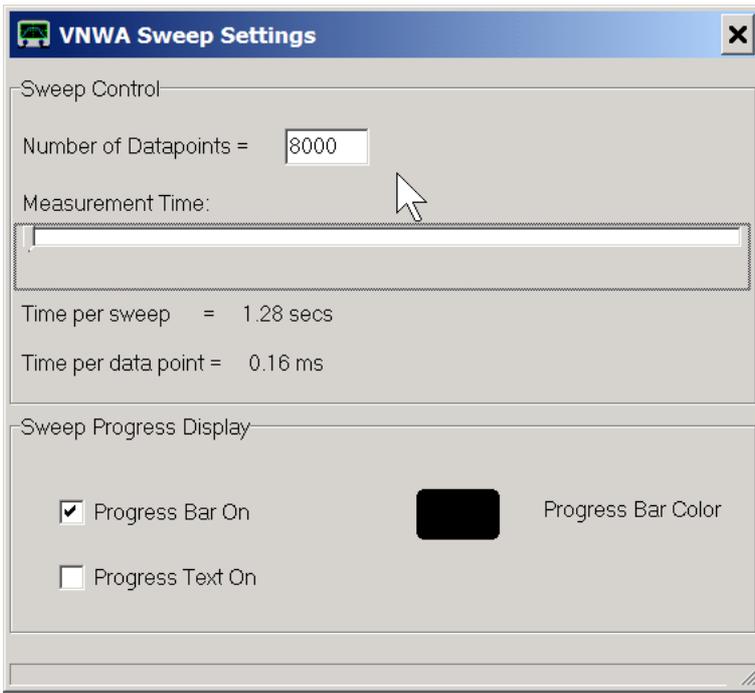
Next, we set the instrument to the desired frequency range e.g. by double-clicking the "Stop" label or by simply pressing the f-key on the keyboard.
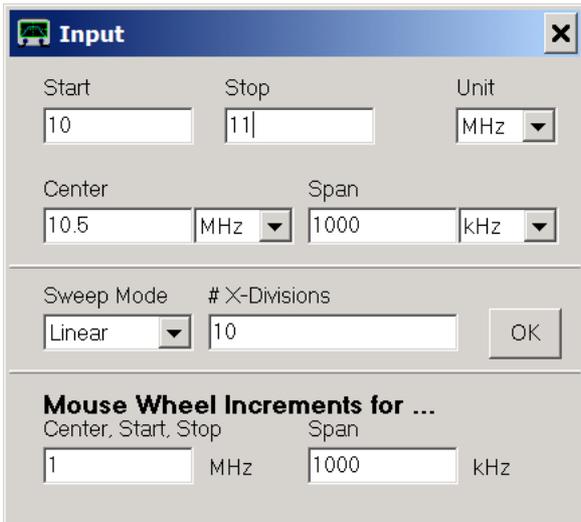
We select S21 (=transmission) for a trace as shown below and do a single sweep by pressing the "Single Sweep" button:
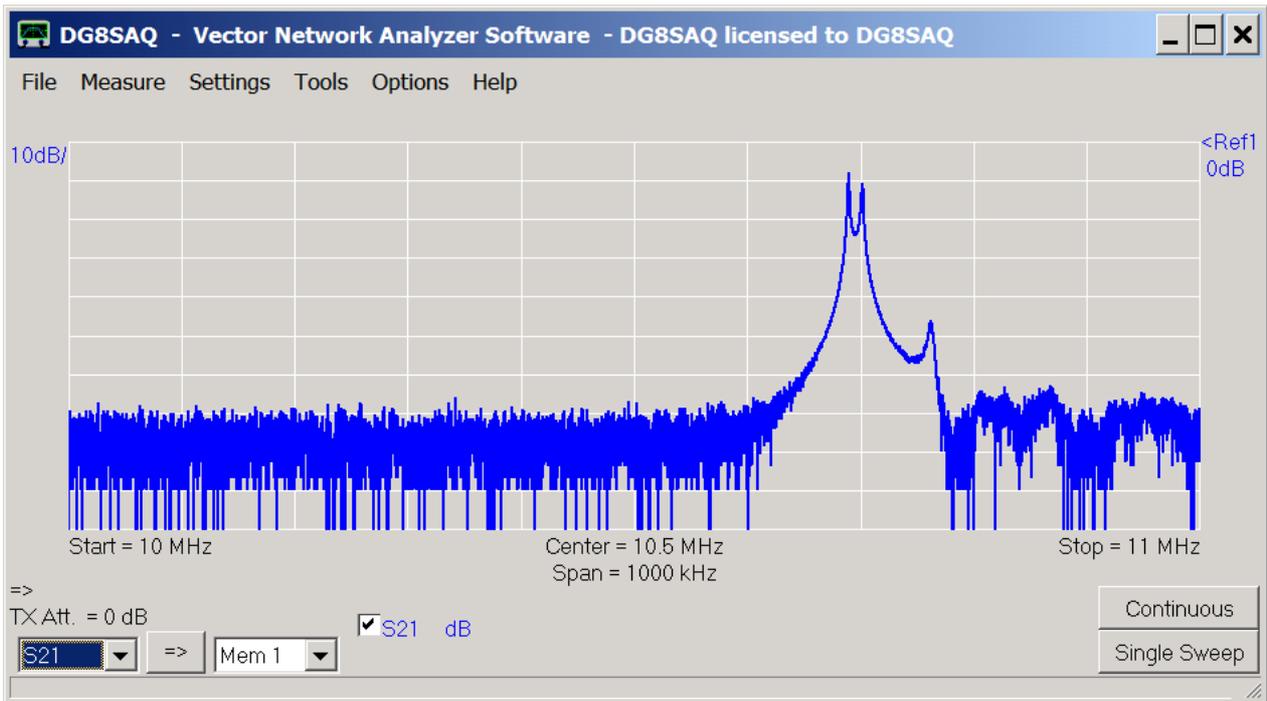
The span (1MHz) is much too high and the center frequency is off. We place a bandwidth marker by right-clicking onto the main grid in order to determine the optimum center frequency and span:
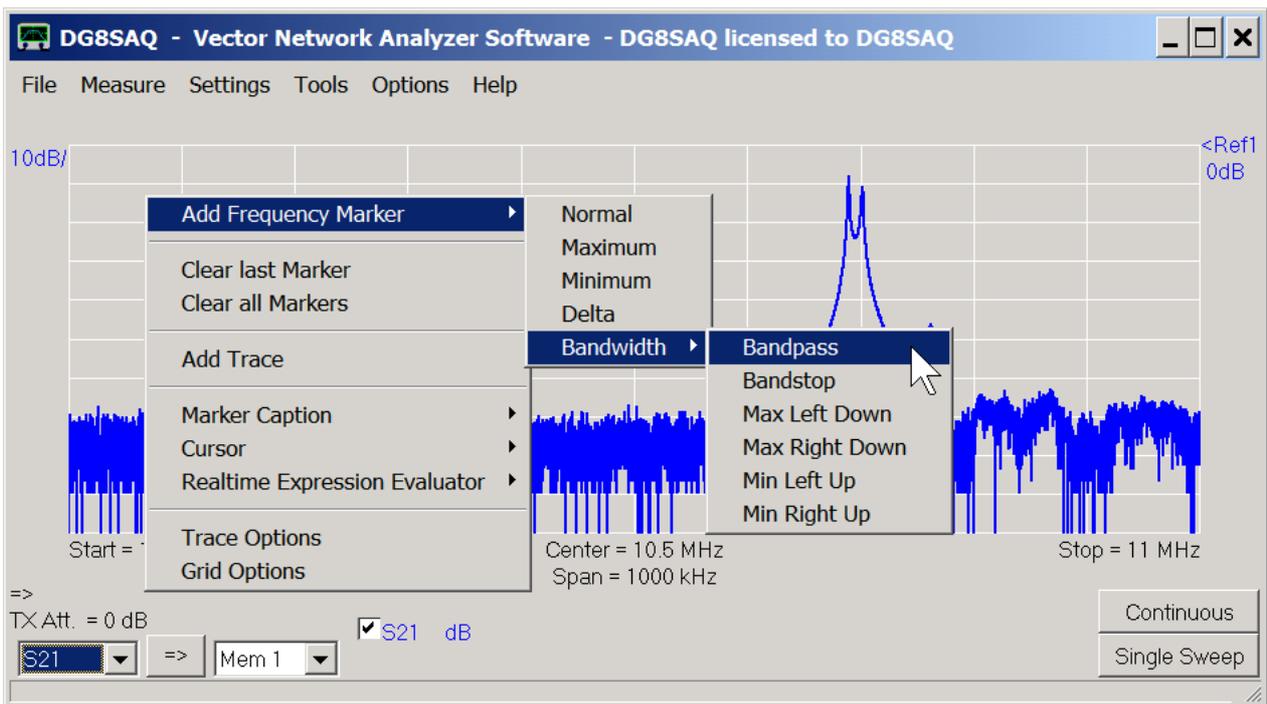
We want a higher bandwidth level, so we right-click on marker4 caption to change it...
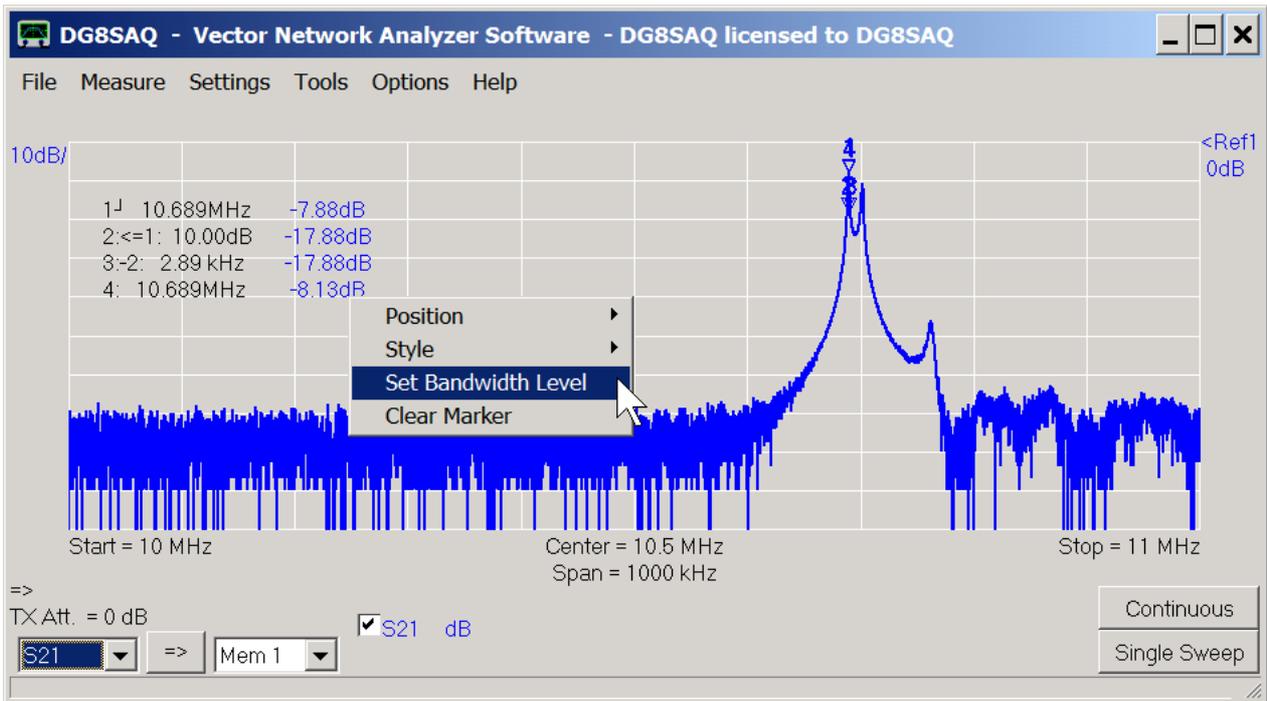
... to 40dB:
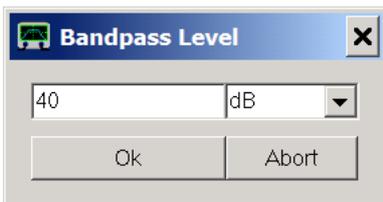
Now, we can read off the desired center frequency and span:
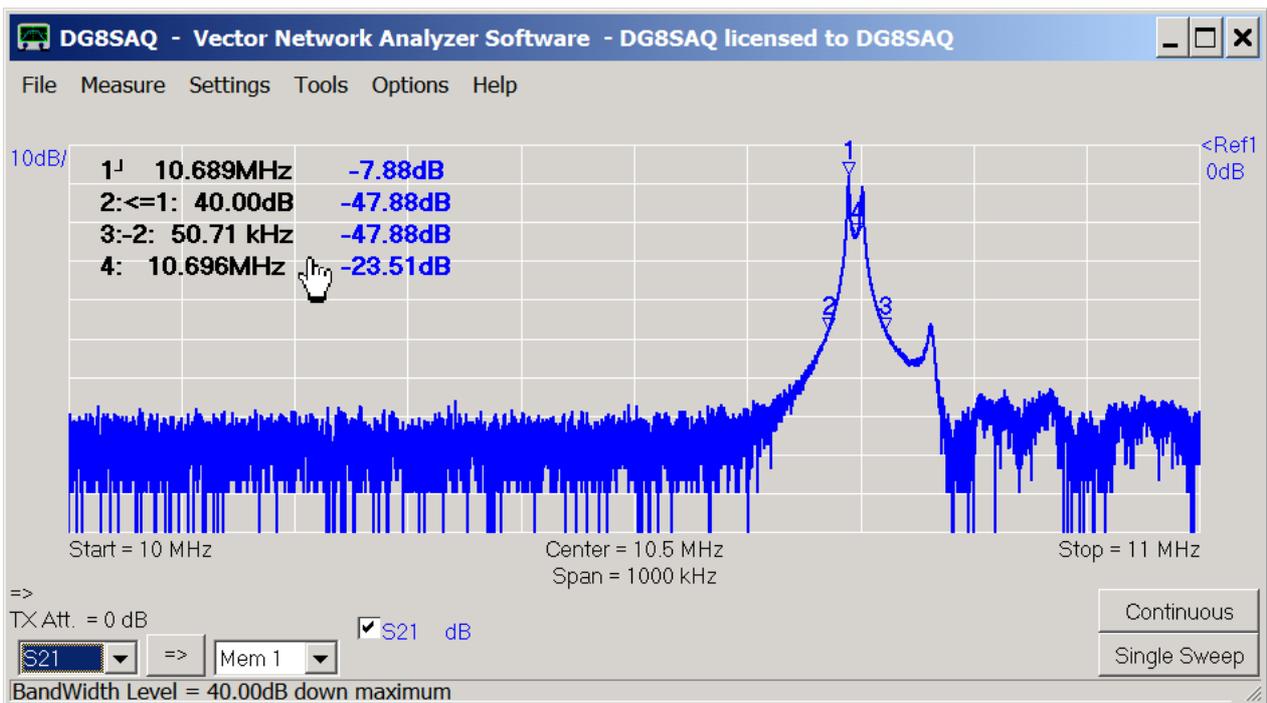
We decide to use a center frequency of 10.7MHz, a span of 80kHz, 500 data points and a total sweep time of 4s, i.e.

8ms per data point:
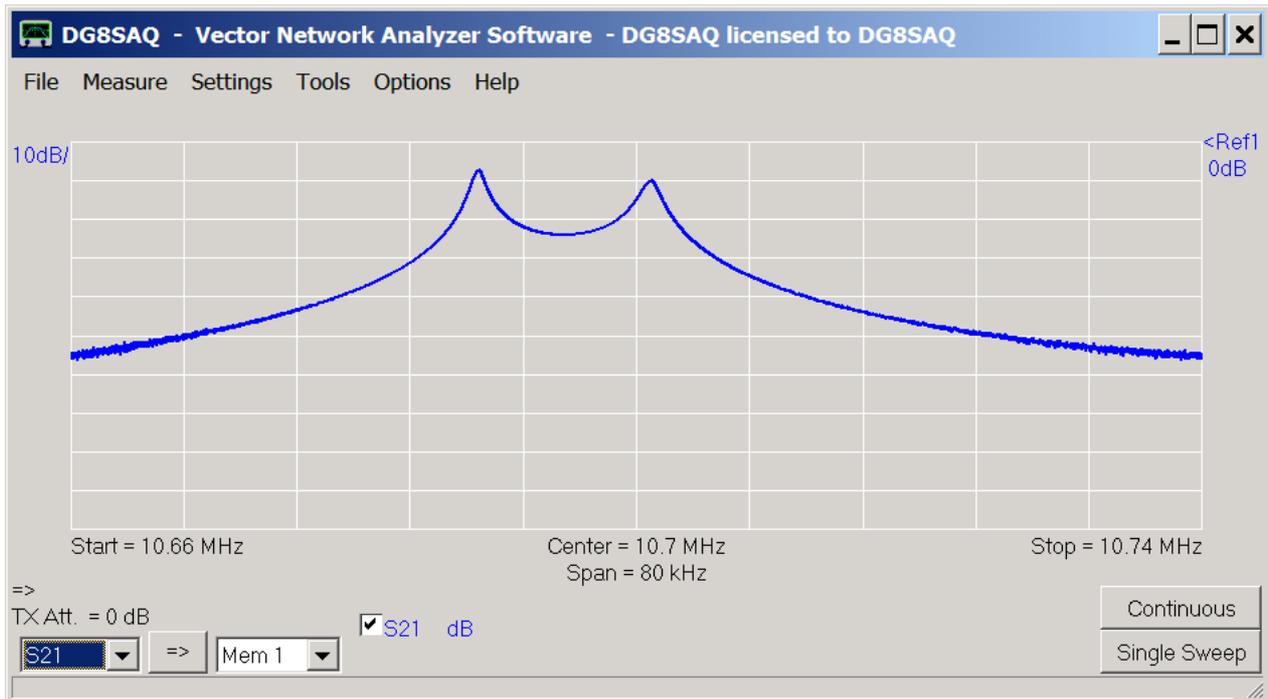
This way, we have a good view of the passband of the filter (between the two maxima) and we also see some of the nearby stopband. Note, that the filter passband is not flat, because the filter is not impedance-matched to the VNWA impedance level of 50 Ohms.

Now, we reduce the number of frequency points to a reasonably low number (500 is quite sufficient) and in turn increase the measurement time such that the whole sweep takes about 4 seconds. Note, that quartz filters have relatively high Q values. Therefore, they need time to settle to a new frequency stimulus. Thus, it is crucial to sweep them slowly. Play with sweeping speed and see how the passband changes. In below view (200 points only), the red trace is sweeped within 30ms, while the blue trace was taken within 4s:
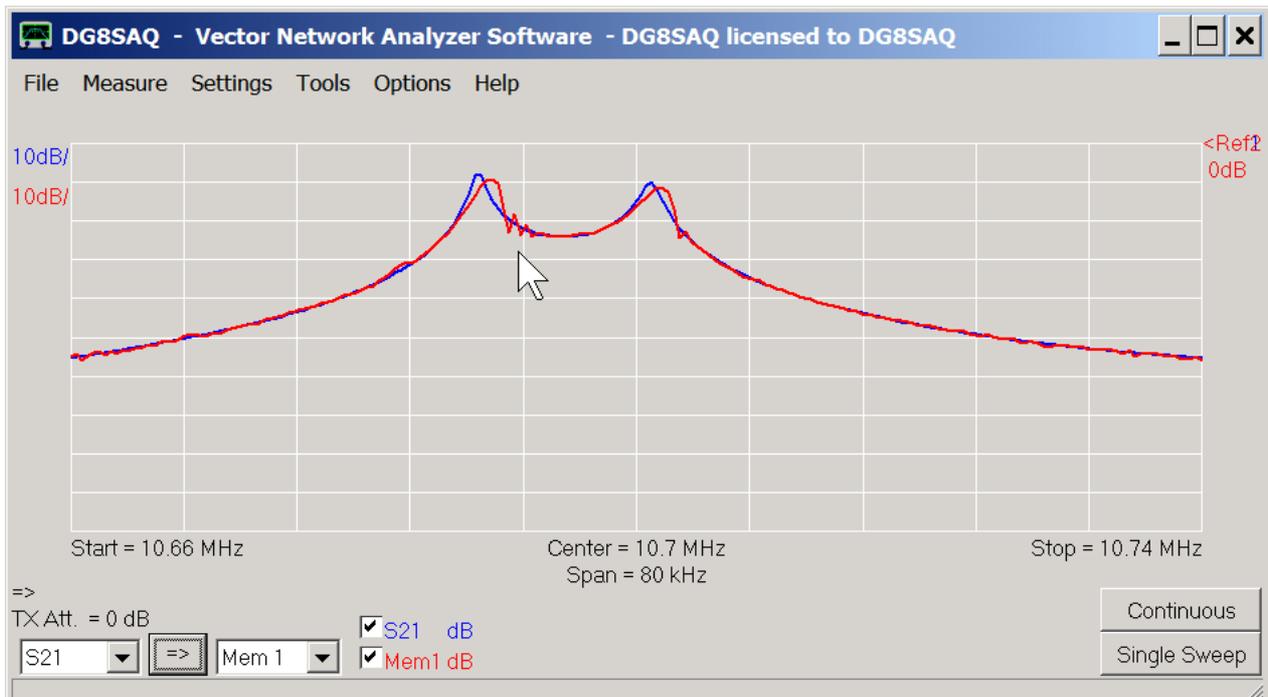
Clearly, the red trace was taken at a too high sweep rate as the maxima are shifted to the right and the filter is ringing after the maxima (marker arrow).

Finally, we activate a second trace for S11 by pressing the + key on the keyboard, we select it to display a Smith chart and we sweep again:
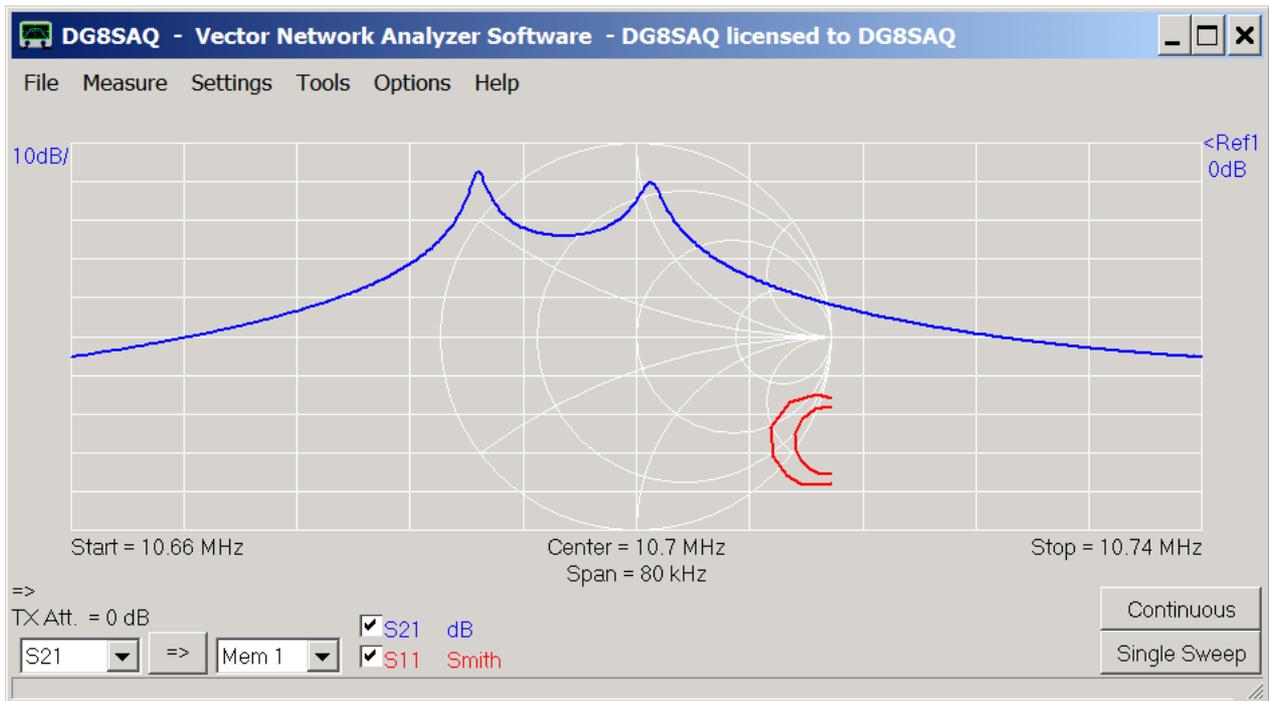
The just measured S11 reflection data is partly outside the Smith chart because our VNWA is not yet calibrated. The data is not valid yet.
Now we are ready to calibrate the VNWA for a two port measurement, which consists of transmission measurements (S21,S12) and a reflection measurements (S11,S22).

2. Calibration

We want to perform a calibration using a female Amphenol calibration kit available through SDRKits:
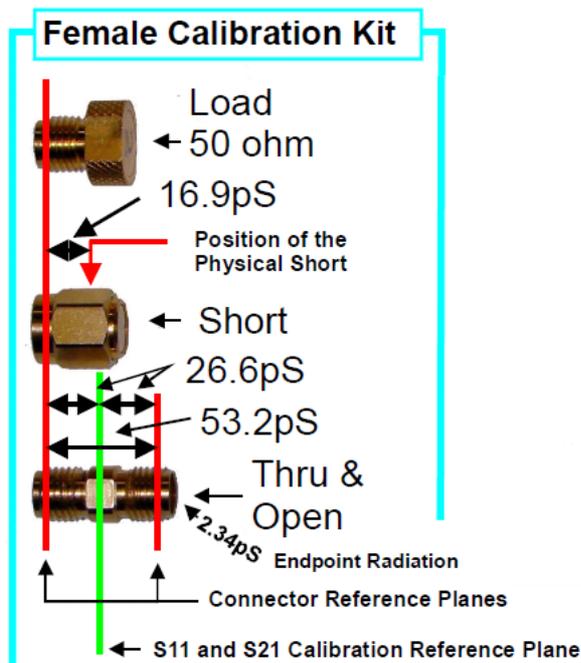
We want to perform a center of thru calibration...
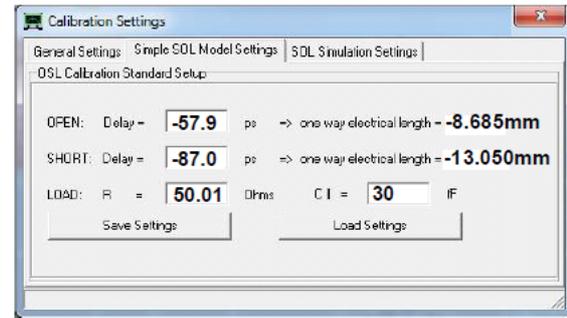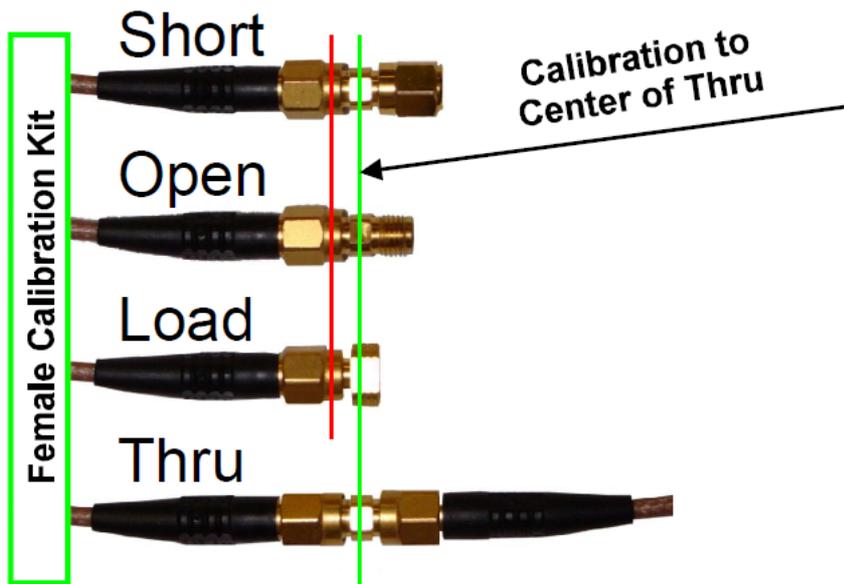
... which places the calibration plane exactly into the middle of the Thru calibration standard.
The above calibration kit data was compiled by Kurt Poulsen and is available from his web page
http://www.hamcom.dk/VNWA/ also for different calibration kit builds and manufacturers.

We open the calibration kit settings window by pressing the k-key on the keyboard, select "simple model custom calibration standards"...
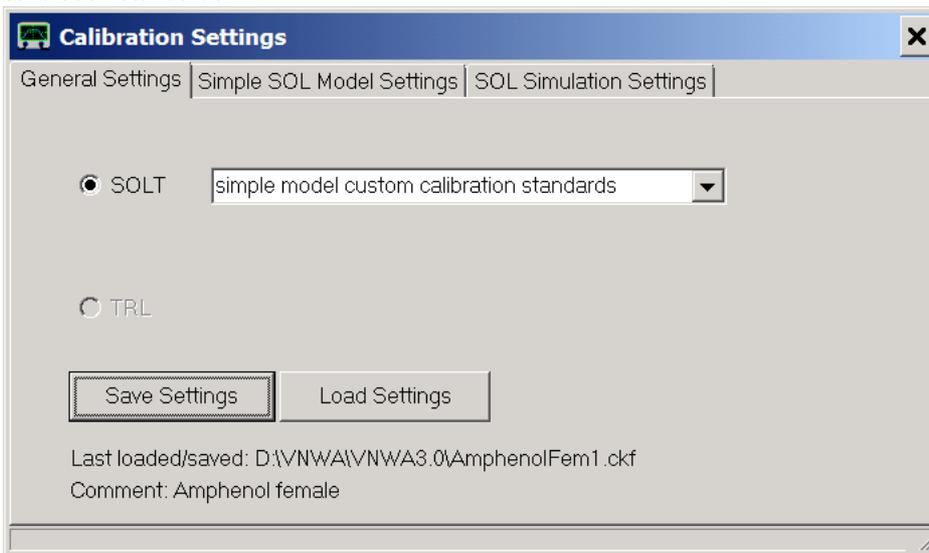
... and copy the calibration kit parameters from Kurt Poulsens document:
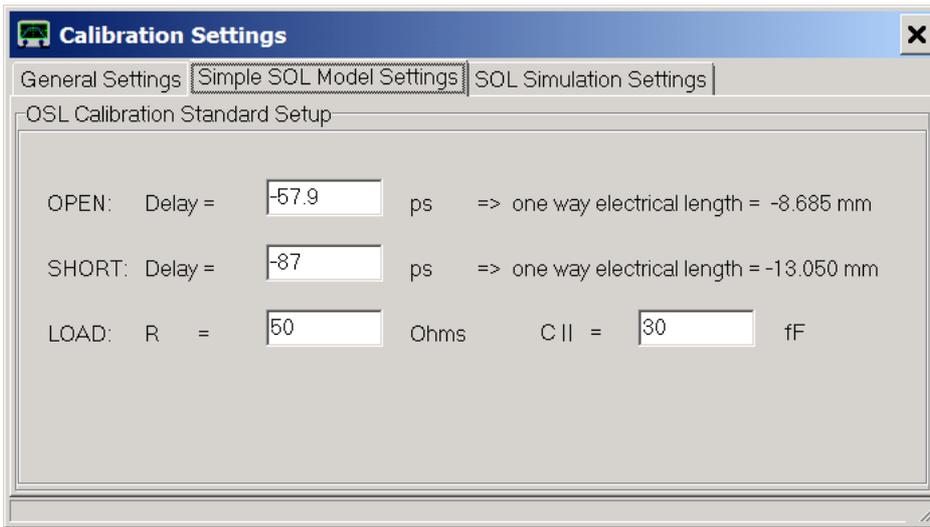
Note: Before performing any calibration measurement, make sure, that the instrument sweep rate is set to the lowest value you intent to use the calibration with, otherwise your later measurement results will be dominated by the noise of your calibration measurements. We have selected a total sweep time of 4s above which is as slow as we want to sweep, so we're ok.

First, we do a reflect calibration of the TX test cable end.
Our test cables remain connected to the VNWA TX and RX ports. We connect our open calibration standard (in this case identical with the female-female through connector) to the far end of the TX port test cable:
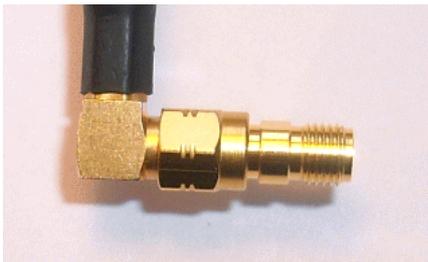
We select "Measure-Calibrate" or simply press the c-key on the keyboard, press the "**Open**" button, confirm that the open standard is connected and observe the instrument measuring the standard.
When the measurement is completed, the red Open light turns green:
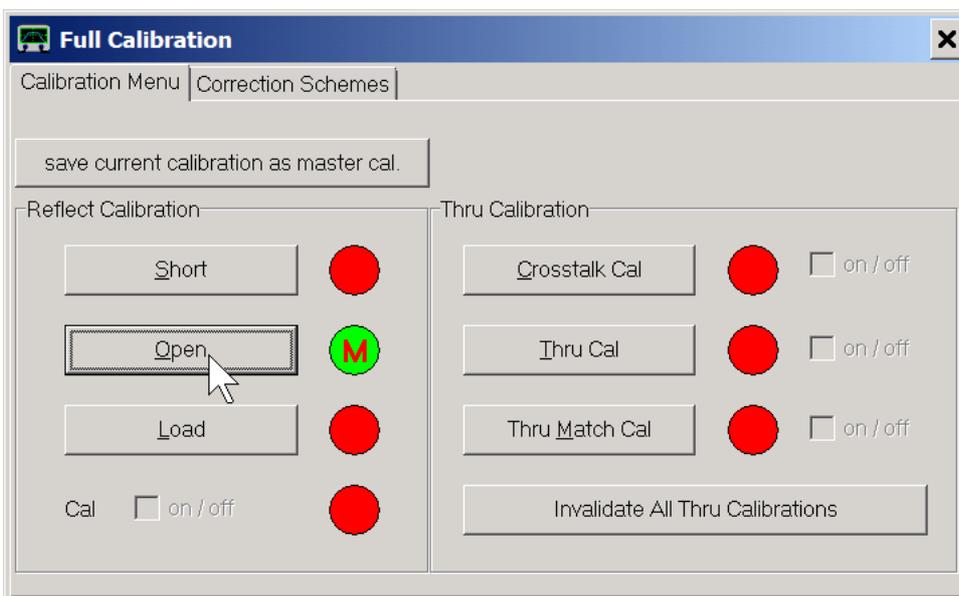
In similar fashion we measure the load calibration standard...
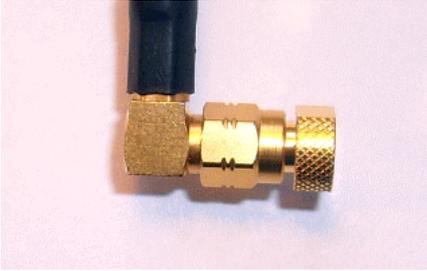
... and the short calibration standard:
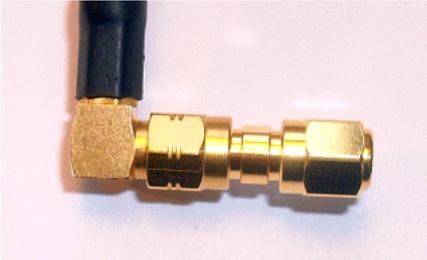
Finally, all reflect calibration lamps should be green:
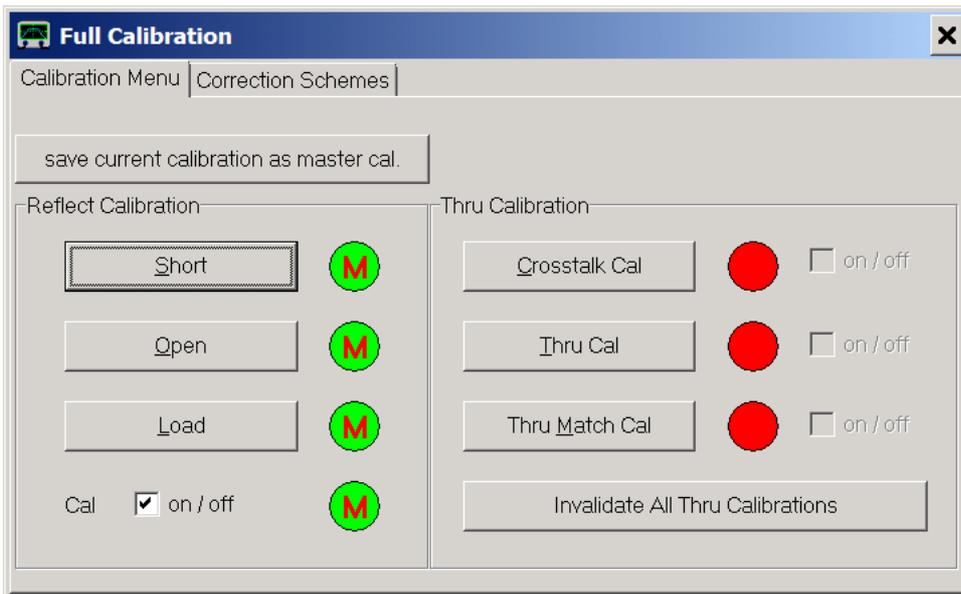
Next, we need to calibrate the VNWA for transmission. We therefore connect the test cable ends of TX port and RX port with the Thru standard...
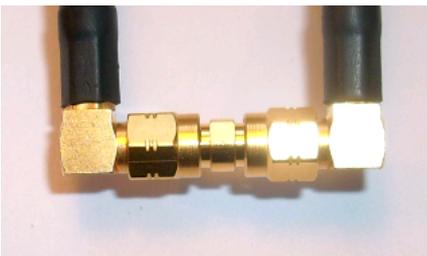
... and press the **"Thru"** button and let the instrument do the calibration sweep.

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.

So, once the thru calibration sweep is done, we press the Thru Match button.

We do omit the crosstalk isolation as this is usually not required. Finally, all lamps except for the crosstalk calibration are green:
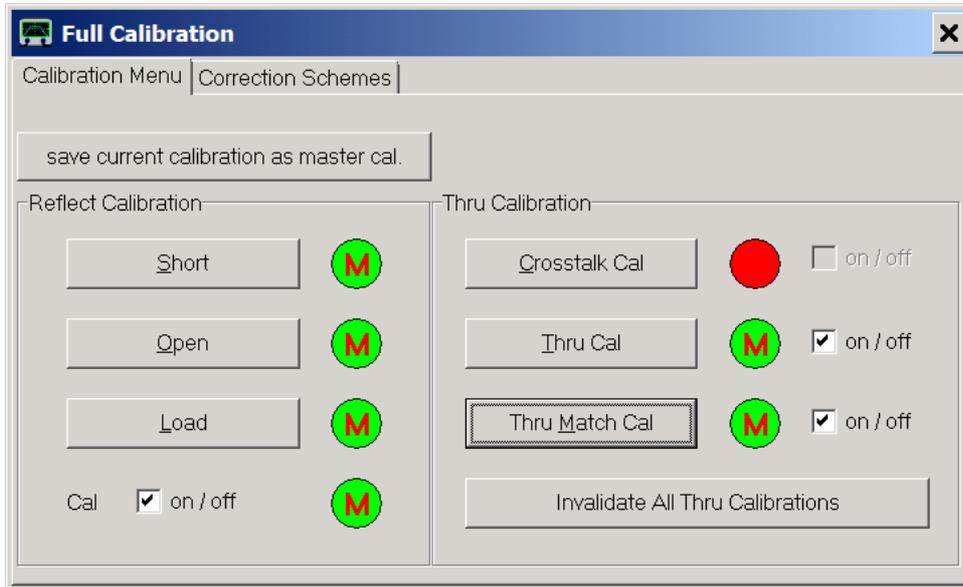
We close the calibration window and double-check the calibration by measuring the "Thru" calibration standard.
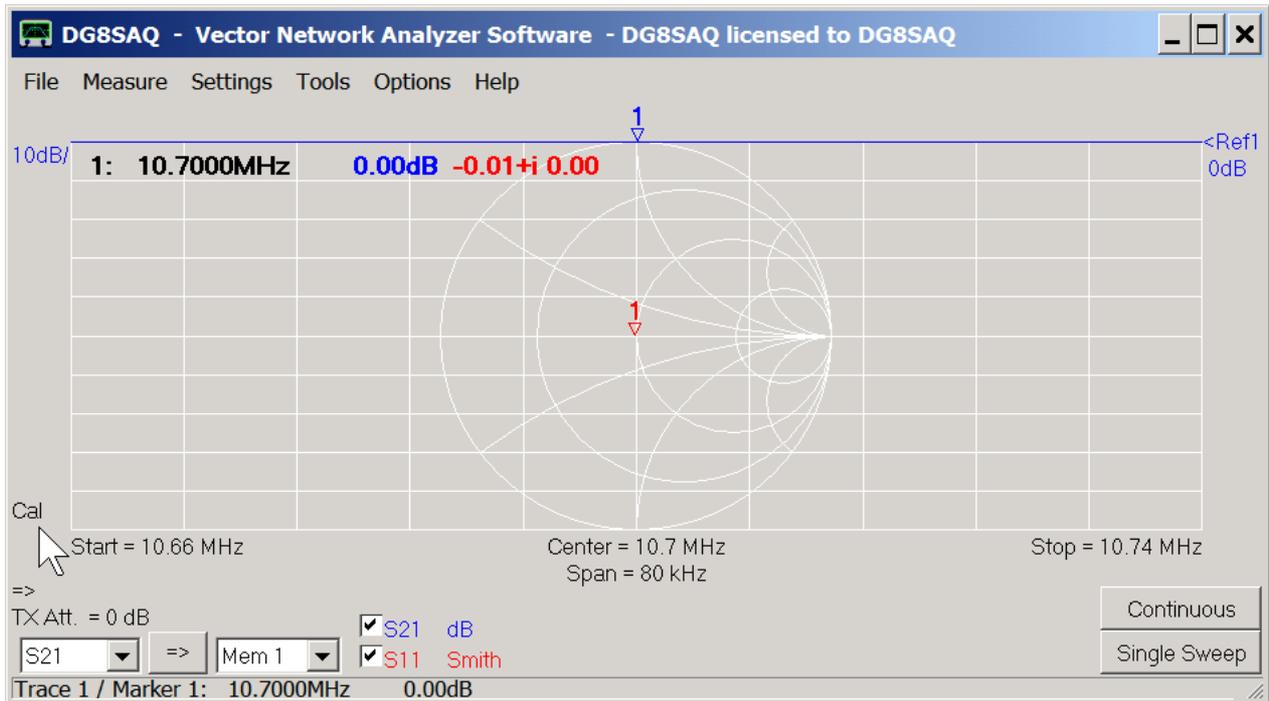
As our Thru standard has by definition 0dB attenuation, we obtain teh flat blue 0dB line. We have also measured the reflection coefficient of the RX port (red trace reduced to a dot, barely visible without marker), which is very close to zero for all frequencies.

Next, we must move the calibration plane from the center of thru position onto our device test board to the wire position of our test object using port extensions. We therefore insert the test board **without** the test object with port 1 connected to the VNWA TX port ...
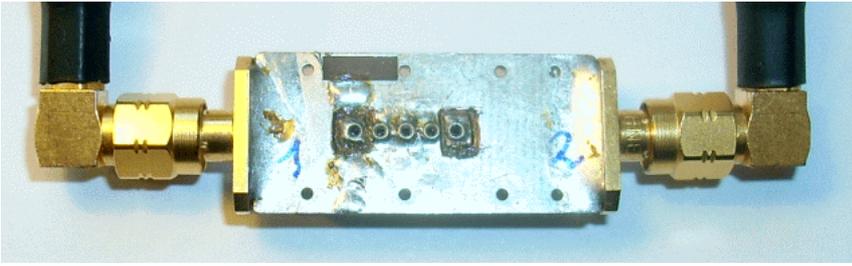
... and do a sweep. We add another trace showing S11 by pressing the + key and select it to show the phase:
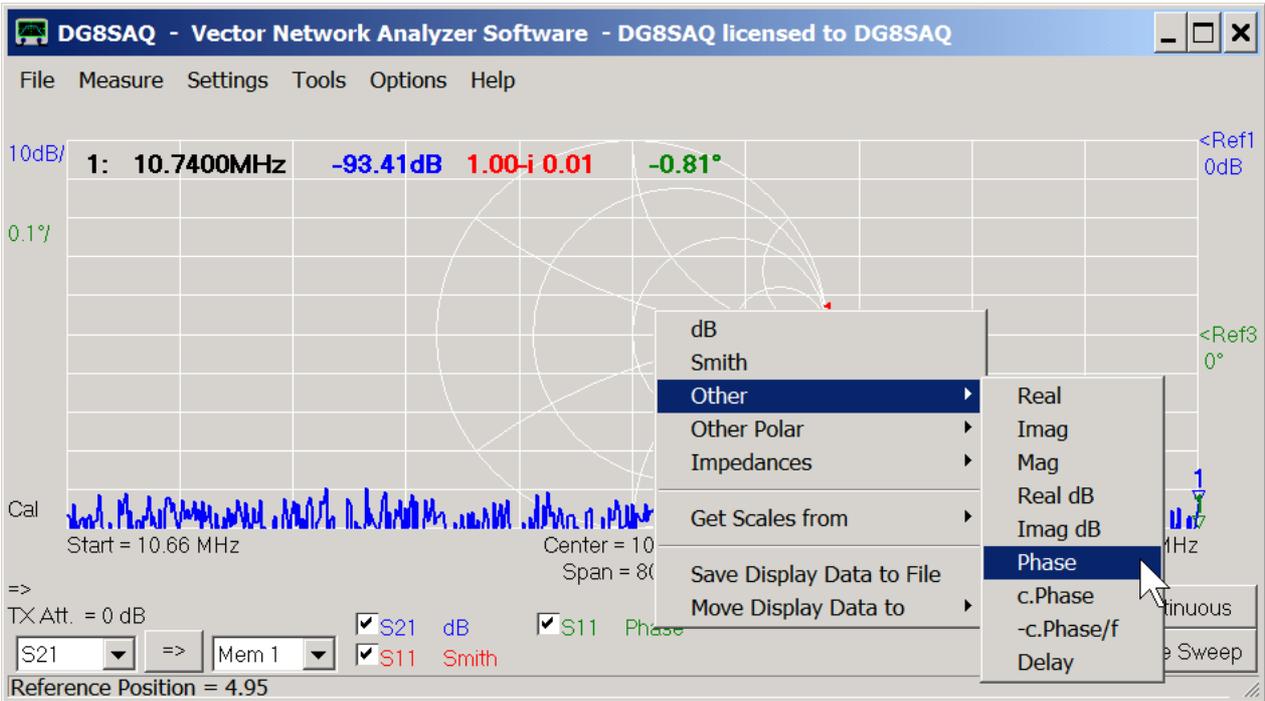
Then we open the **port extensions** menu by pressing the p-key on the keyboard and place the port extensions window next to the main window so both remain visible:
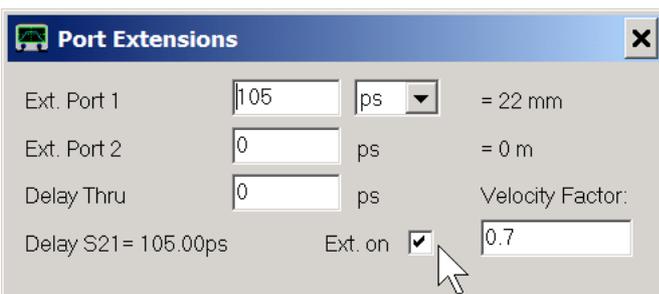
We check the **Ext. on** box (mousepointer) and set focus to the Ext. Port 1 by clicking on it. Next, while observing the green S11 phase trace in the main window, we turn the mouse wheel or edit the field until the the S11 phase is zero:
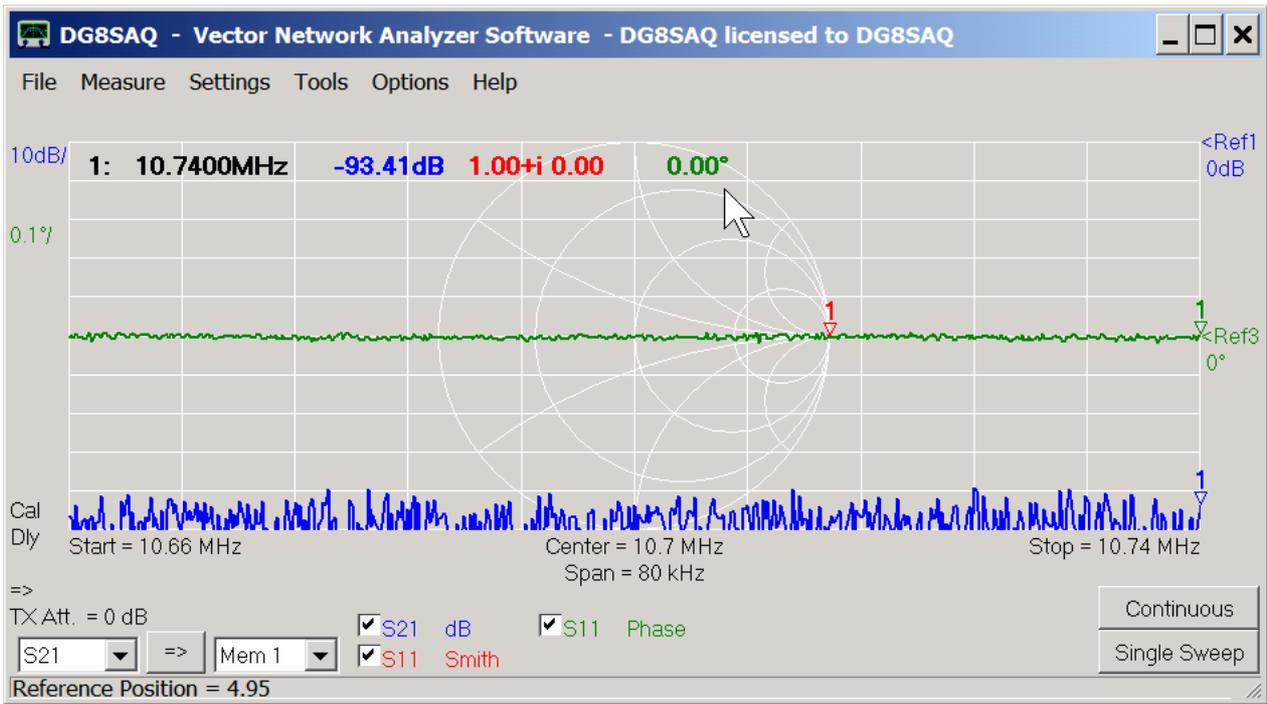
Note, that the reflection coefficient S11 is now exactly 1+i0, which is the open reflection condition, corresponding to a dot at the right edge of the Smith chart.

Next, we reverse the empty test board, such that port 2 is connected to the VNWA TX port:
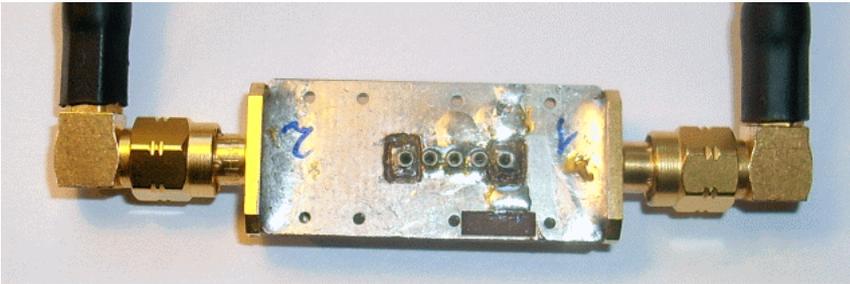
We add a fourth trace S22 also showing the phase and we reverse the measurement direction by clicking onto the direction arrow (hand mouse pointer in below figure):
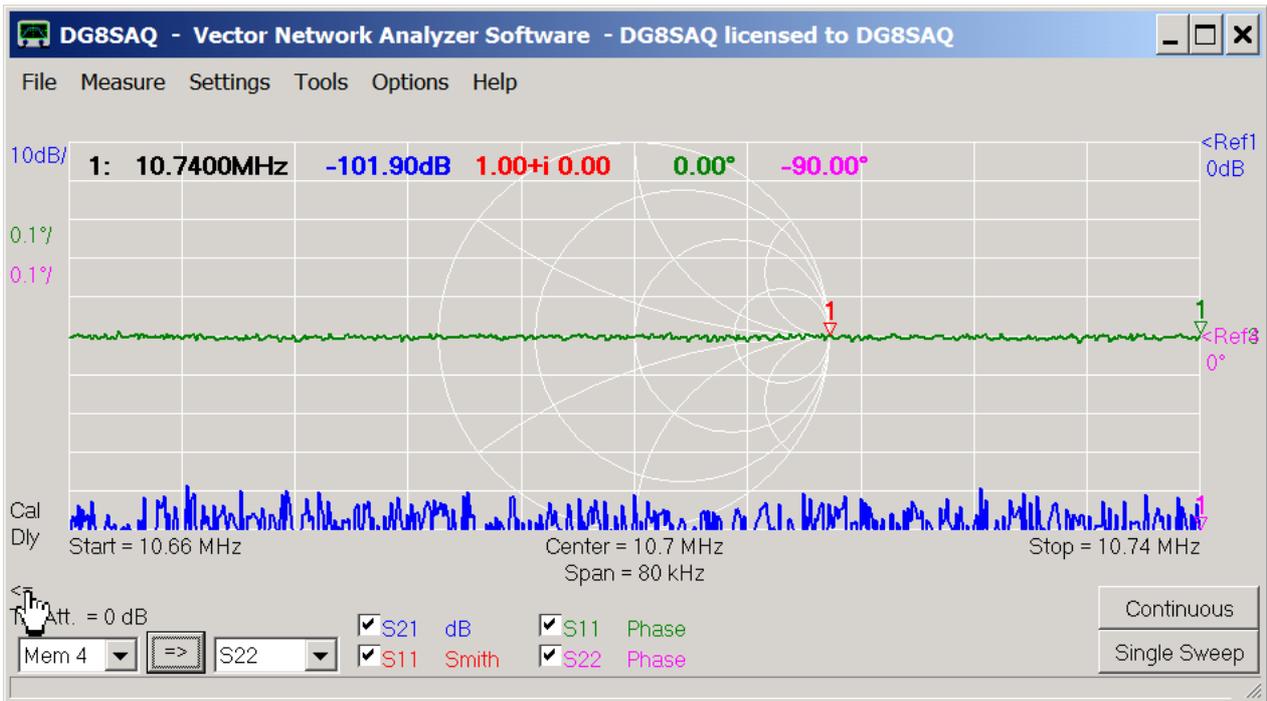
Note, that reversing the VNWA measurement direction arrow does not physically change the measurement direction. The signal still comes out of the TX port. That's why we have physically reversed the test object. Reversing the direction inside the VNWA software causes the reflection data to be collected in the S22 rather than in S11 as before.

Now, do a single sweep and observe data being collected in S22. S21 and S11 data remain untouched by this sweep.

Go to the still open port extensions window (or reopen it by pressing the p-key) and change the **Ext. Port 2** value while observing the pink S22 phase trace ...
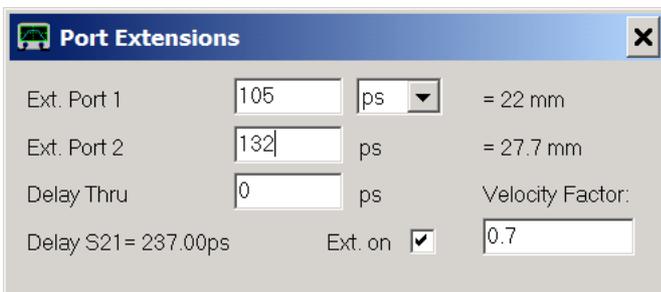
... until the phase is exactly zero:
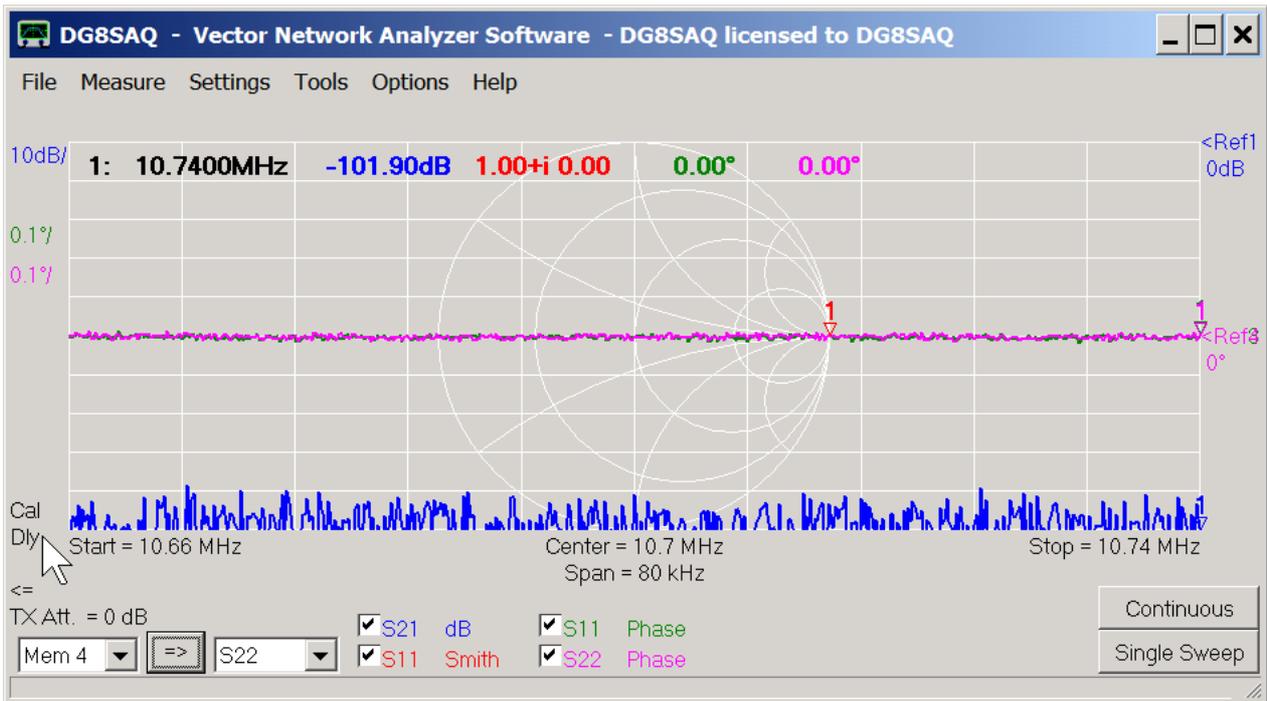
Note the labels Cal and Dly in above figure at the mouse arrow indicating that a calibration is active and port extensions (=port delays) are activated.

Note, that the **Delay Thru** setting in the port extensions window remains zero for the Amphenol calibration kit. Looking at our port extensions, we observe, that the path from the test board connector to our test object on port 2 side is somewhat longer than on port 1 side, which is confirmed by visual inspection of the test board.

Now, the VNWA is fully calibrated and ready for measuring two port s-parameters and we can close the port extensions window.

3. Measurement

Next, we put our test object (=crystal filter) onto the test board and connect the test board to the VNWA such that port 1 is connected to the VNWA TX port and port 2 is connected to the RX port, ...
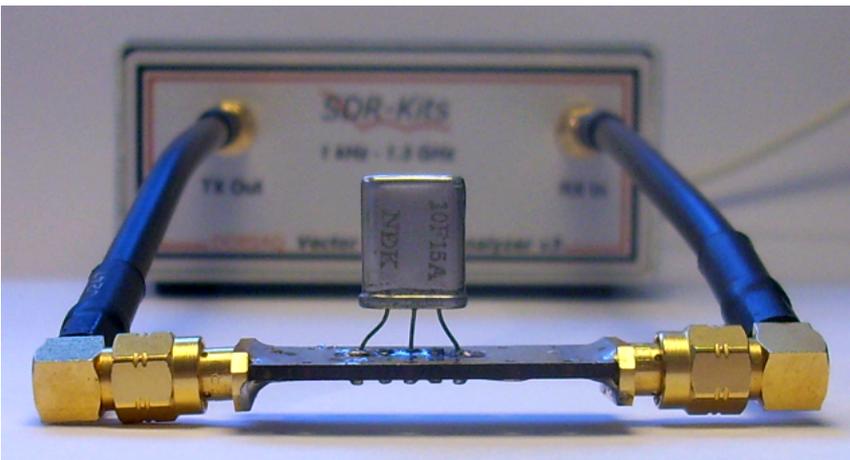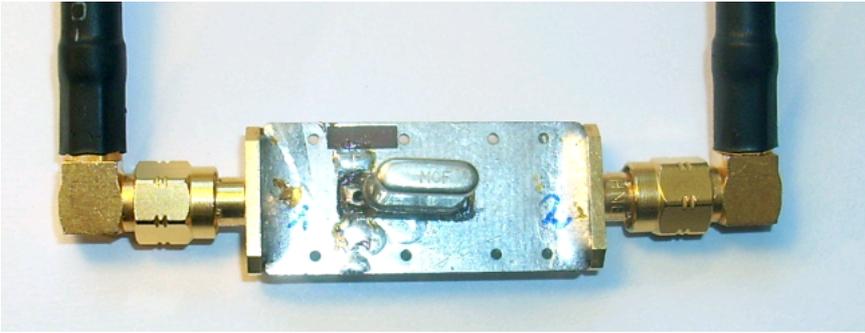
... switch the software to measure forward direction by clicking the arrow do a single sweep:
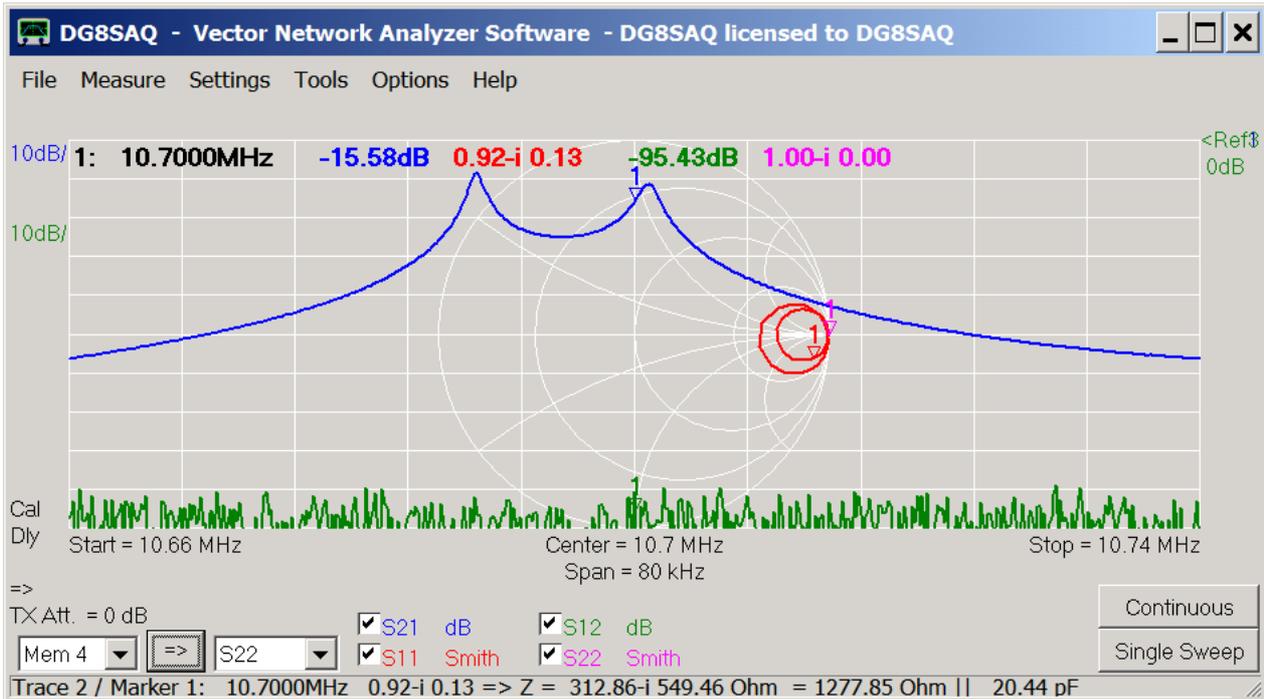
Now, we have already measured S21 and S11. Note, that the S11 data now lies completely inside the Smith chart as must be for a passive device. When clicking onto the red Smith chart marker 1, we get additional information on the S11 data at the marker position in the bottom status line. We do not only see the measured reflection coefficient (0.92-i*10.13), but also what this means in terms of impedance in the series equivalent circuit model (Z=312.86 Ohms-i*549.46 Ohms) as well as in the parallel equivalent circuit model (Z=1277.85Ohms || 20.44pF). So, the input impedance of our filter is quite high.

We could now collect reverse data by reversing the test object and manually reversing the measurement direction. But there is a more convenient way to collect 2 port data, which is the 2 port measurement assistant. We launch a 2 port measurement into memory by pressing the F2-key or via the main menu:
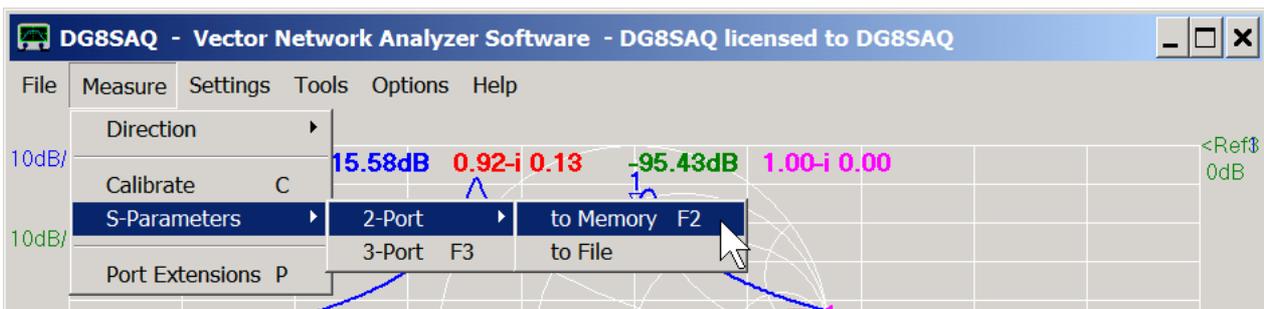
The assistant instructs us that the signal flow is Terminal1 => Terminal2, which means that the TX port should be connected to port 1 and the RX port to port 2.
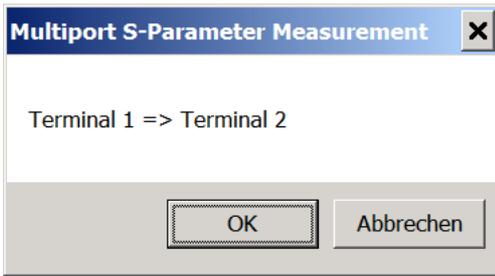
After inspection, that this is indeed the case, we press **OK** and observe how S11 and S21 are being remeasured:
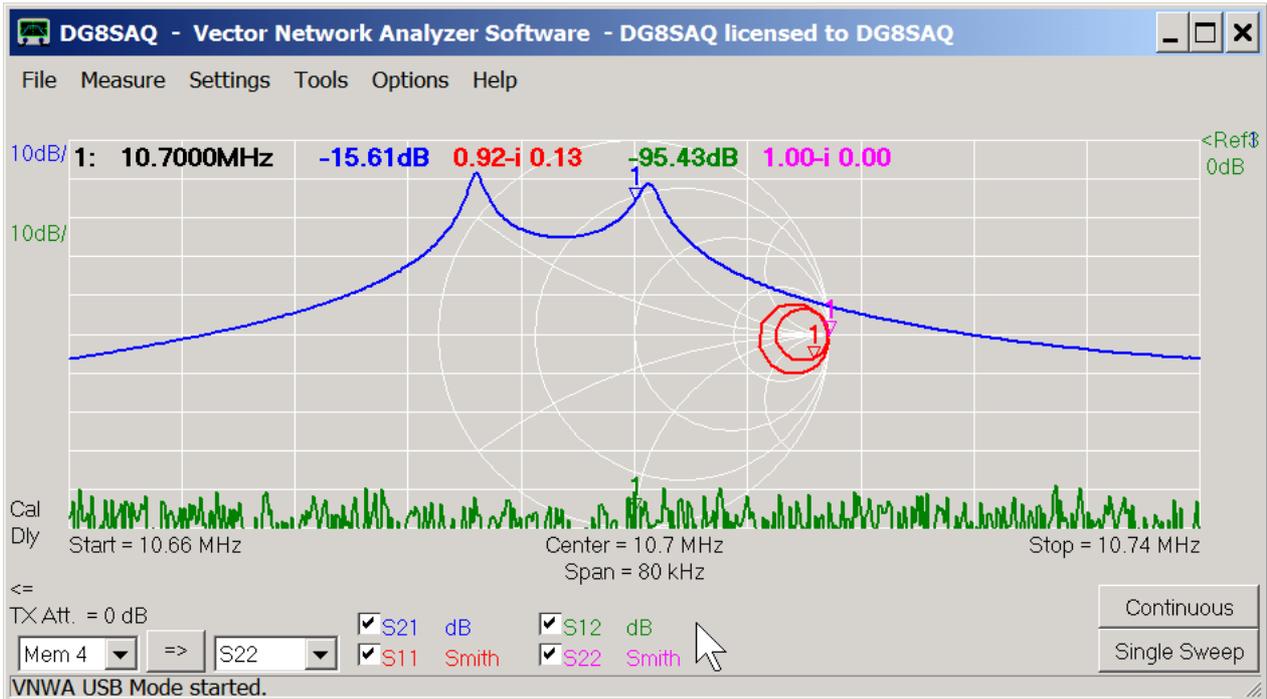
Note, that meanwhile we have chosen to display all four S-parameters S11, S21, S12 and S22.

When the forward sweeps are done, the assistant pops up again indicating that next the reverse direction is to be measured:
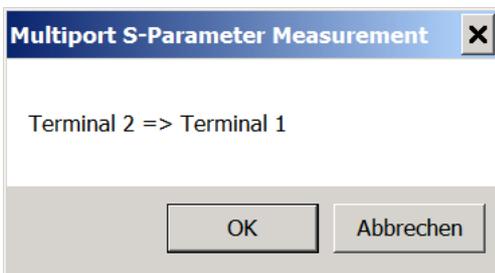
This means, that we have to reverse our test board, i.e. connect port 2 to the VNWA TX port and port 1 to the VNWA RX port:
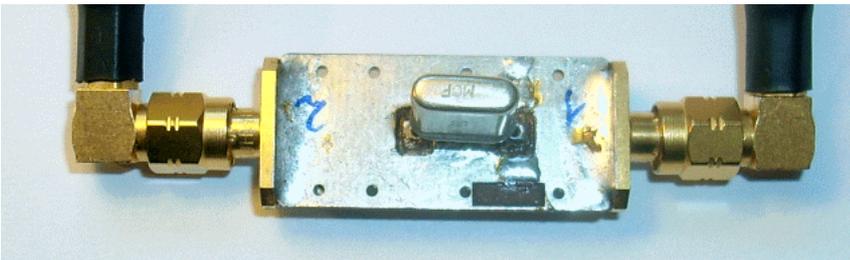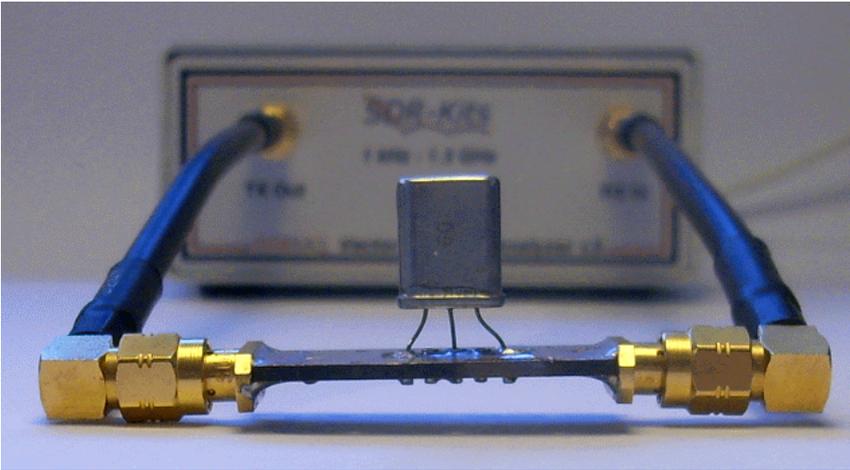
Once we are done with reconnecting the test board, we press **OK** on the multiport assistant and observe S12 and S22 being measured:
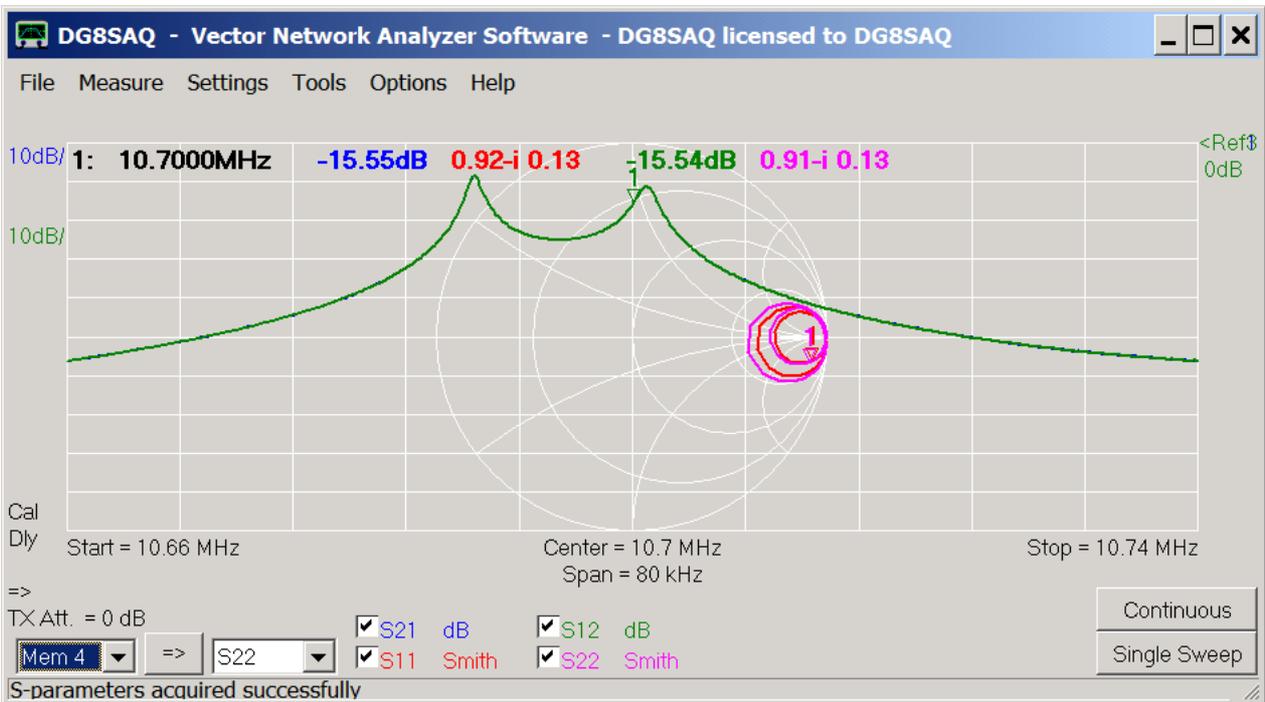
Note, that the forward Sparameters (S21, S11) might slightly change during the reverse measurement as the software uses the additional information of the reverse measurement to correct the forward measurement results in order to obtain the highest possible accuracy.

Note, that S12 and S21 are identical as must be the case for reciprocal devices like passive filters. S11 and S22 are quite similar indicating that the filter input and output are designed in similar fashion.

Finally, we save the full set of measured S-parameters into a Touchstone s2p-file for later use:
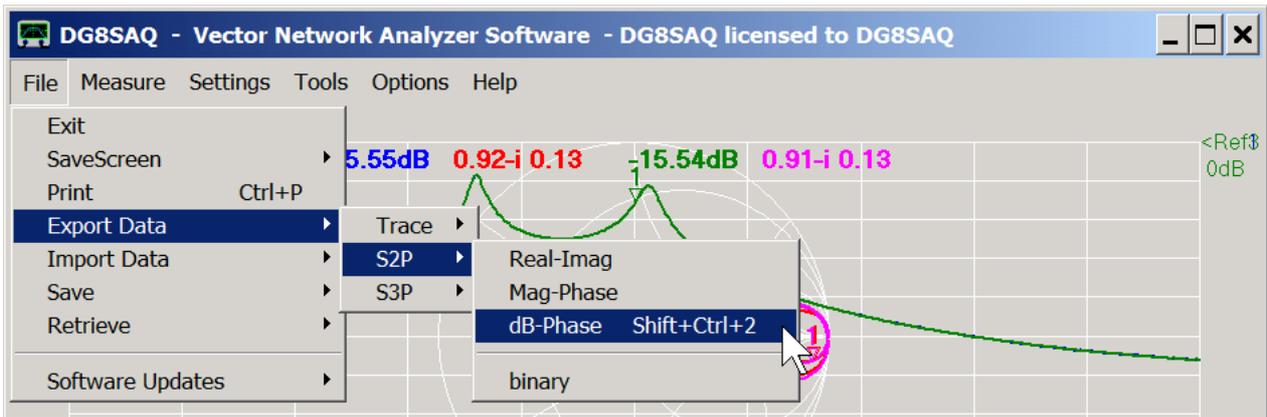
We may add a comment to the s2p-file...
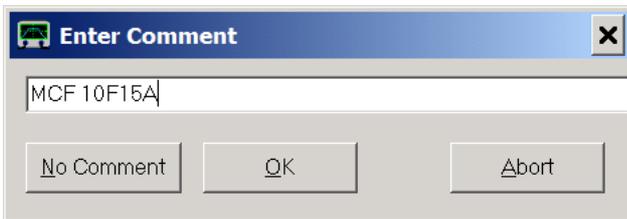
... and save it:
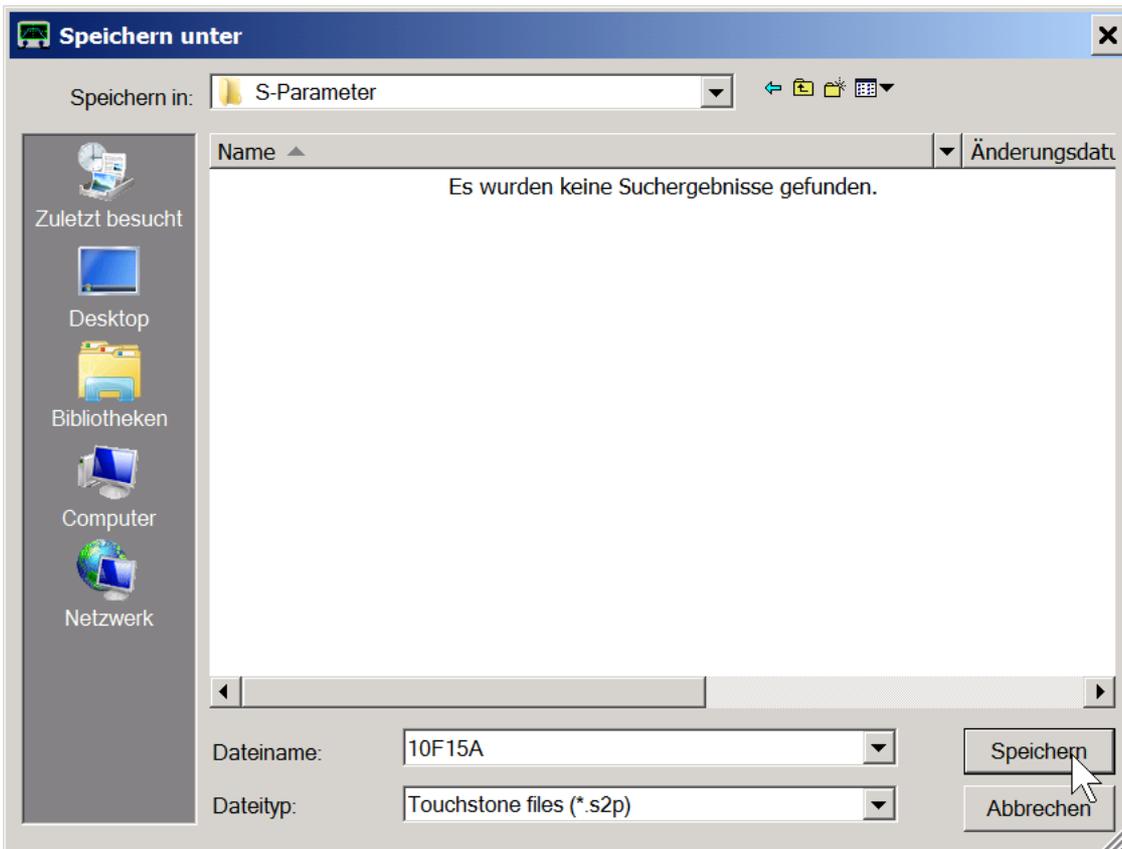
Now, that we have obtained the full set of S-parameters, what can we do with them?
This will be demonstrated on the following page "Interpretation and Usage of 2-Port Measurements".

In the previous page "Practical Example on Performing a 2-Port Measurement" we have demonstrated how to measure the two port S-parameters of a two pole crystal filter:
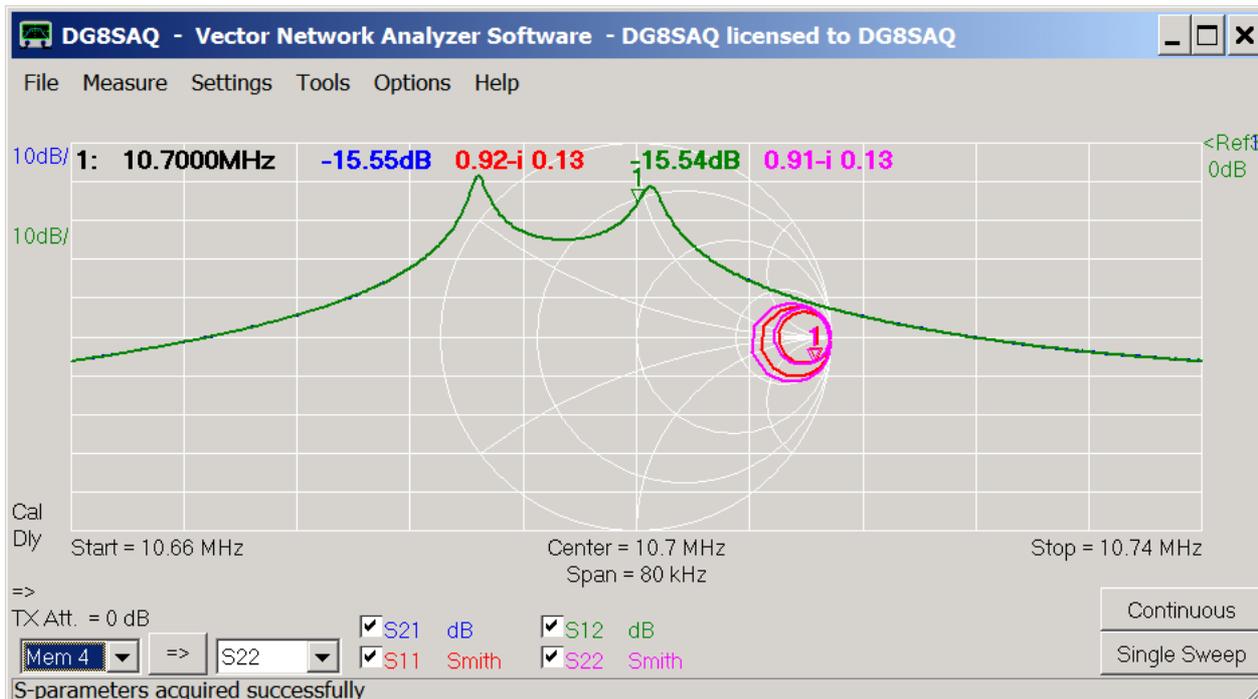
Now, what can we learn from these and how can we make use of them? We can do some simple analysis right inside the VNWA software. For more sofisticated simulations we need to make use of network simulation software, which will be described at the end of this page.

Analyzing S-Parameters inside the VNWA Software

Note, that both S11 and S22 data lies at the open circuit side in the Smith chart, i.e. our filter appears to have a relatively high input and output impedance compared to 50 Ohms. We have measured the transmission in the 50 Ohms impedance environment of the VNWA, though. Now, it could be interesting to find out, how the same filter would behave in a high impedance environment, i.e. in an environment where both the filter input and output are power matched. We actually can do this analysis inside the VNWA software using the Matching Tool, which we activate by pressing the m-key on the keyboard:
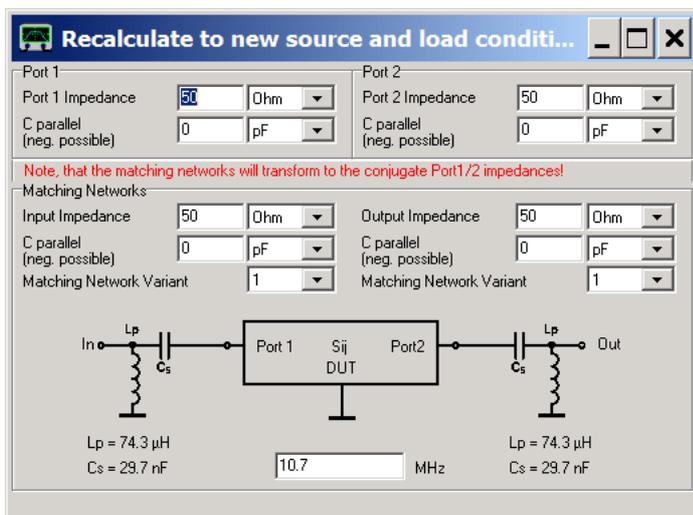
We can now modify port 1 and port 2 impedances by using the keyboard or the mouse wheel (right-click onto the edit fields to change the mouse wheel increments) and observe in me main window how the S-parameters change when transformed to different terminating conditions, e.g. changing the terminating impedances on both sides to 1550 Ohms

...
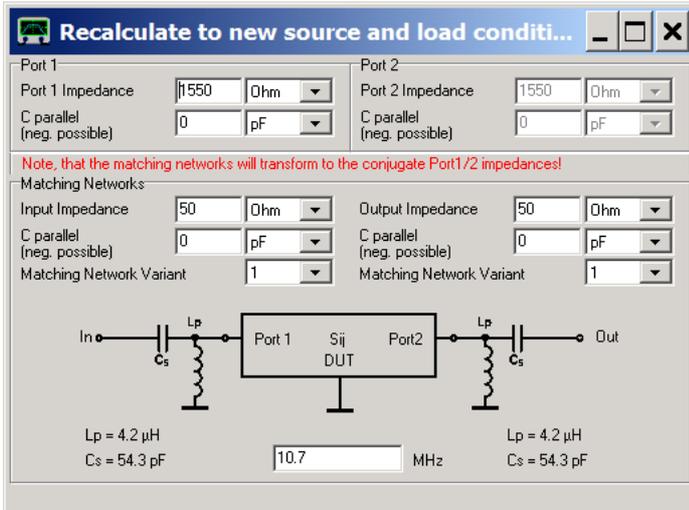
... will yield a reasonable power match condition and a flat passband:
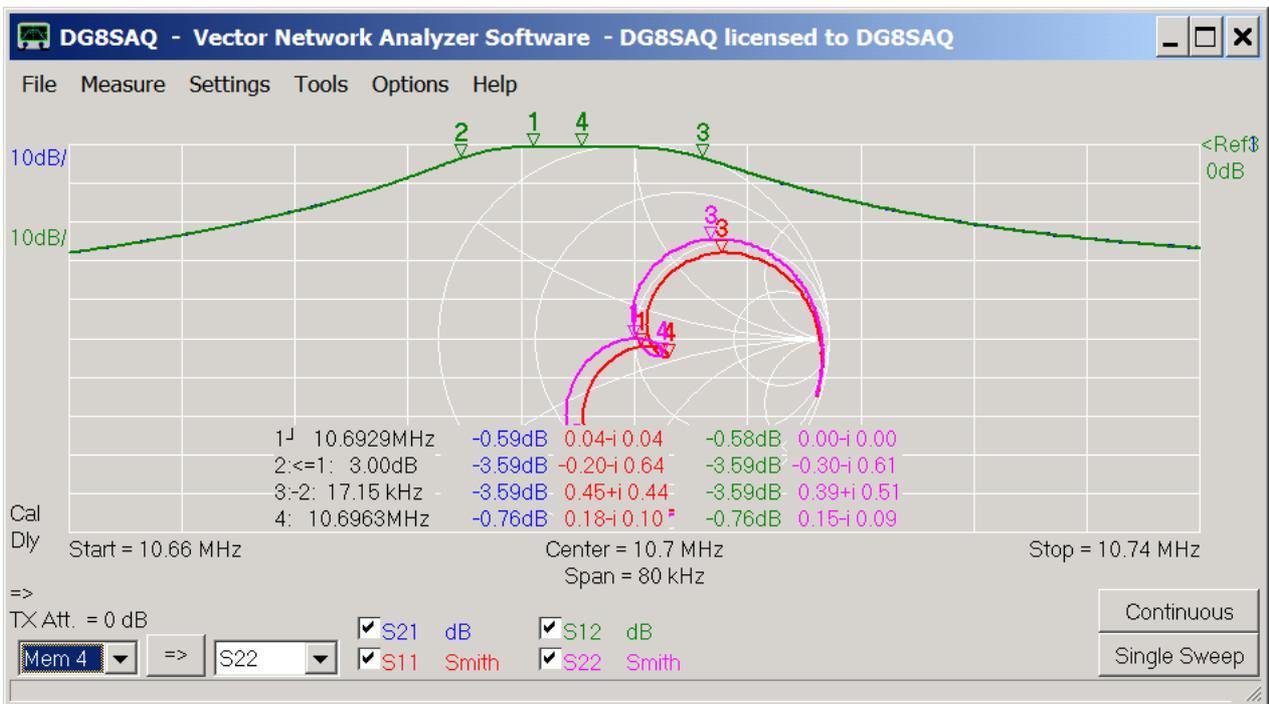
We can easily implement this impedance condition in an experiment by connecting 1500 Ohms resistors in series with the VNWA RX and TX ports. The RX and TX port impedances of 50 Ohms will add to these 1500 Ohms to yield 1550 Ohms impedance at the filter:
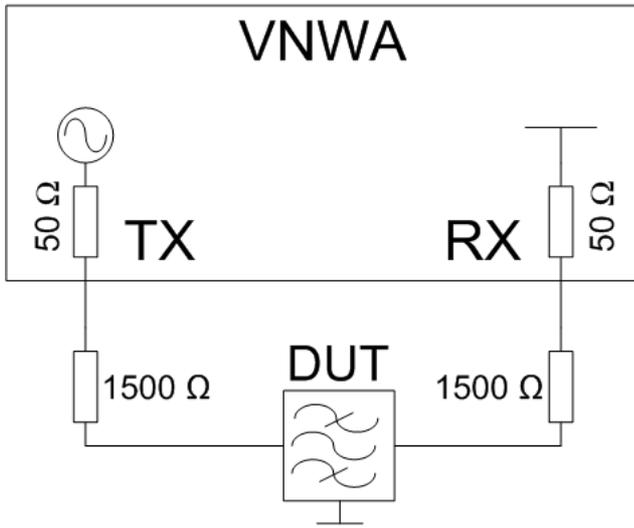
We therefore modify the bottom of our test board ...
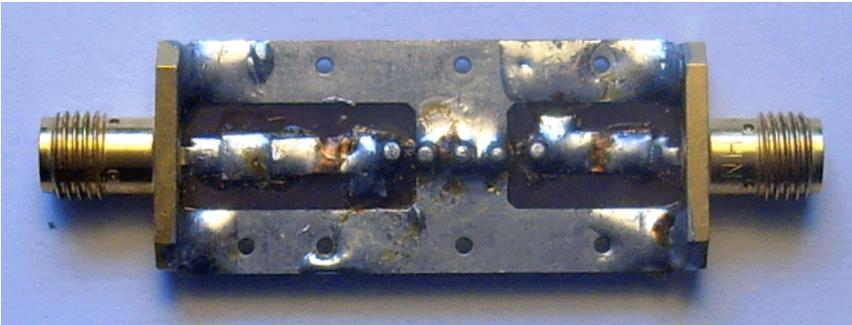
... to include two 1500 Ohms resistors:
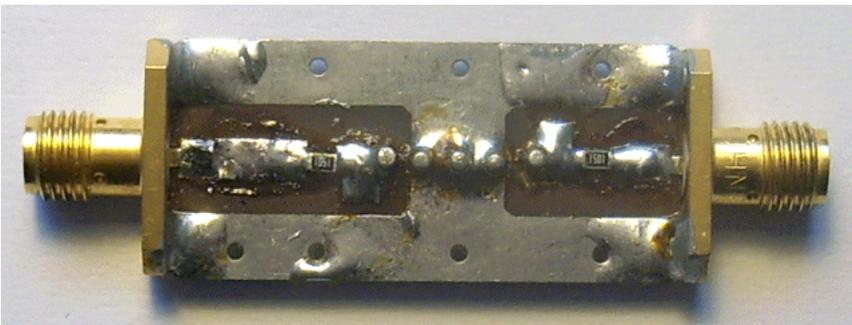
We connect the modified test board with the crystal filter to the VNWA and do a single sweep in forward direction using the original calibration and settings from the S-parameter measurement:
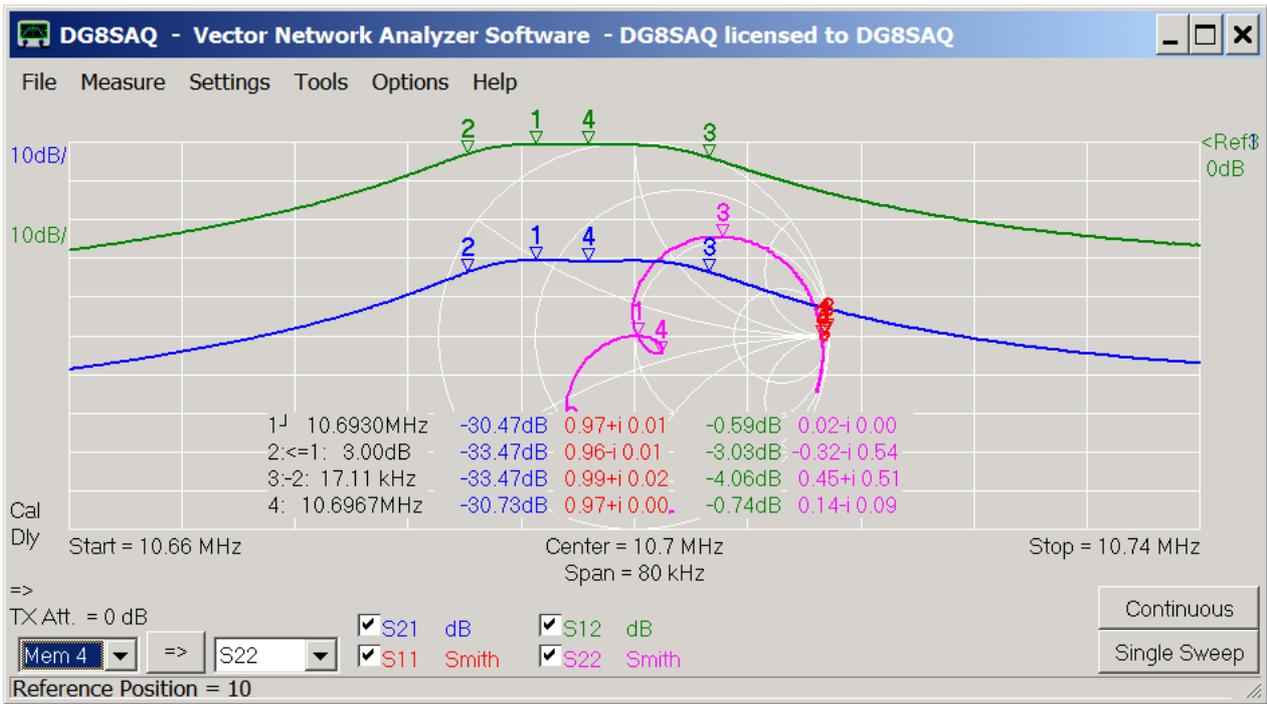
Indeed, the filter passband is flat now, but the insertion loss has increased by about 30dB in comparison to our S12 trace which has survived from our software matching experiment!
We get a better comparison when we adjust the scales and shift the reference level of the blue S21 trace to -30dB:
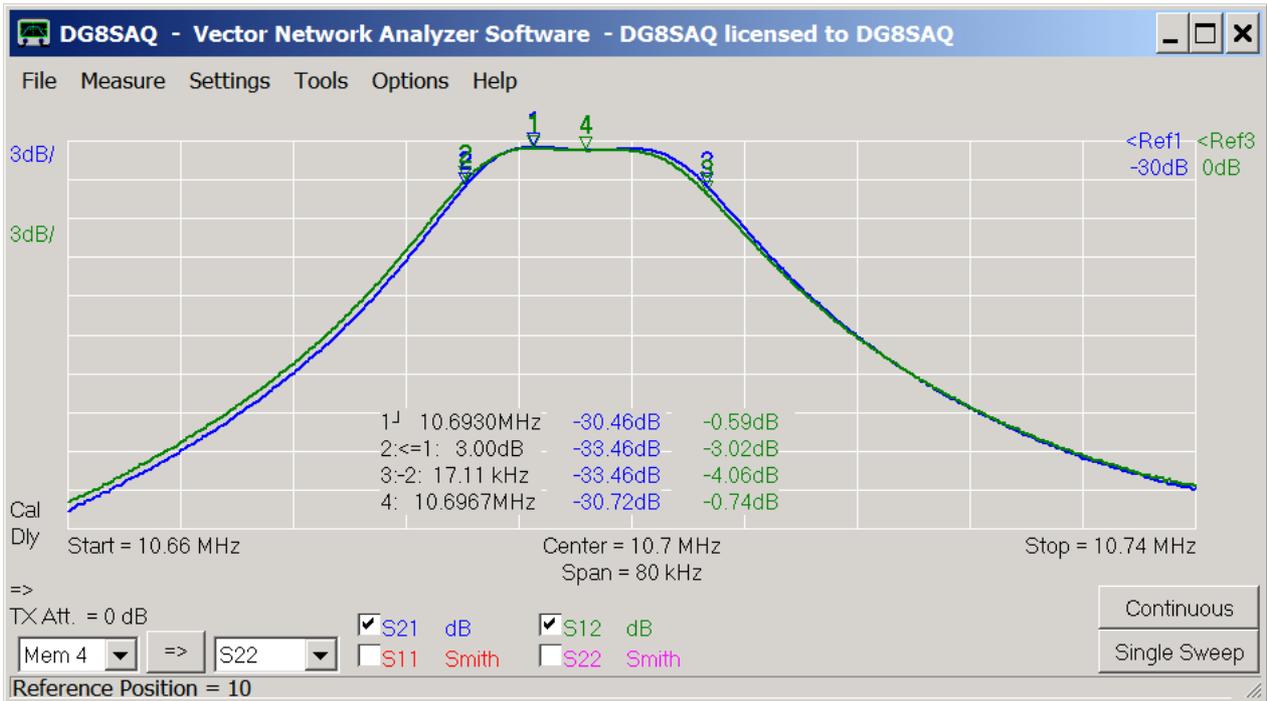
So, except for the high loss, our measurement looks like anticipated.
The 30dB extra loss is no surprise, though, as we have effectively introduced 2x1500 Ohms=3000 Ohms into the signal path.
Next, we want to quickly find out, how much extra loss 3000 Ohms in the signal path will yield. We add another trace and make it a Custom trace:
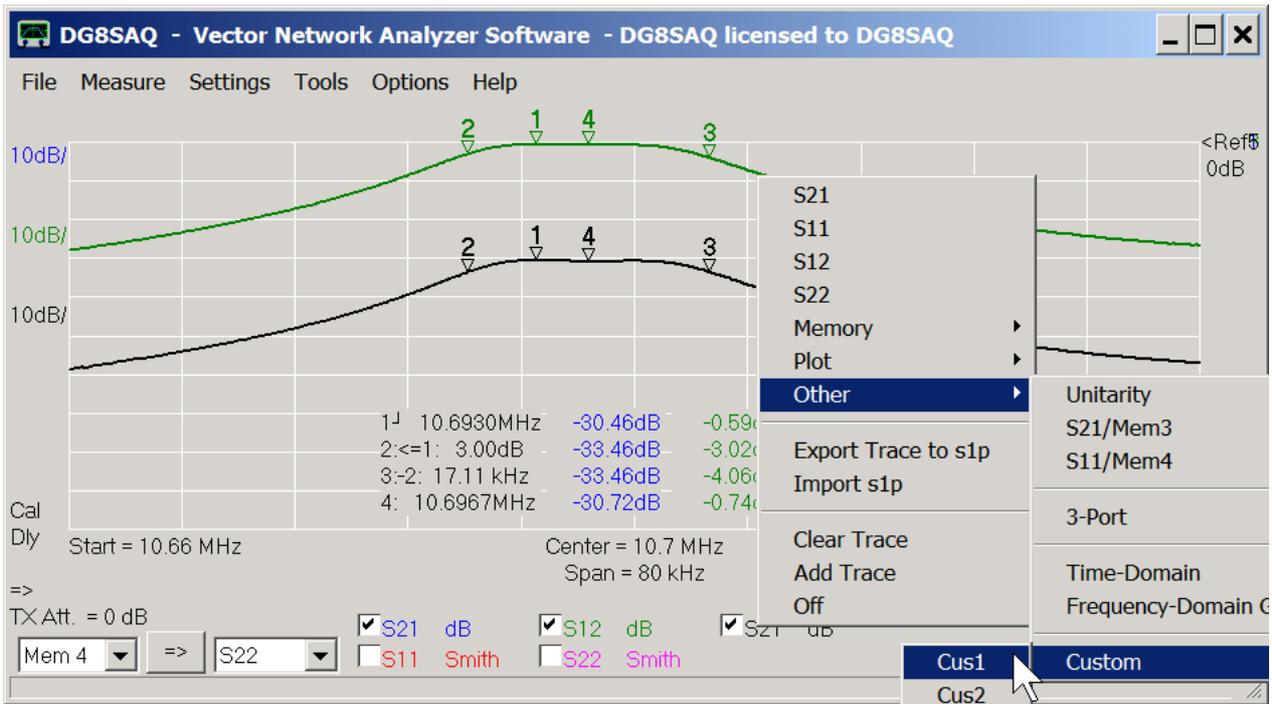
The custom trace editor will open:

We edit the expression for the function to be plotted as shown above. The number 3000 denotes our resistor of 3000 Ohms which we want to simulate. The function z2s() converts this resistance into a reflection coefficient. The outermost function s2t() converts the reflection data (which could also be measured as S11 with our 3000 Ohms connected from the TX port to ground) into transmission data (which could be measured as S21 with our 3000 Ohms connected between TX port and RX port). We also assign a descriptive caption, which will later show on the main window. Closing the custom trace editor, the new trace will be computed and plotted:

DG8SAQ - Vector Network Analyzer Software - DG8SAQ licensed to DG8SAQ

File   Measure   Settings   Tools   Options   Help

	1	10.6930MHz	-30.46dB	-0.59dB	-29.83dB
	2:<=1:	3.00dB	-33.46dB	-3.02dB	-29.83dB
	3:-2:	17.11 kHz	-33.46dB	-4.06dB	-29.83dB
	4:	10.6967MHz	-30.72dB	-0.74dB	-29.83dB

Start = 10.66 MHz   Center = 10.7 MHz   Stop = 10.74 MHz
Span = 80 kHz

TX Att. = 0 dB

Trace5: s2t(z2s(3000))
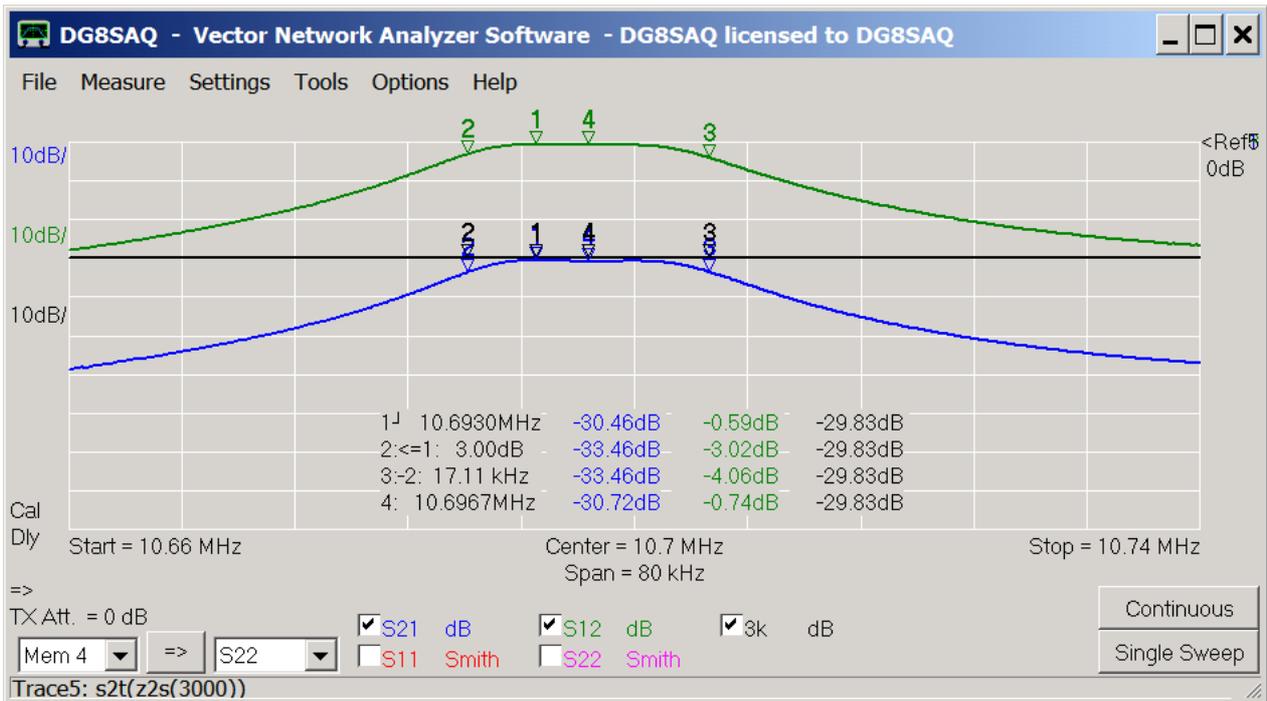
For clarity, we have reverted to the original scales 10dB/ and a unique reference level of 0dB on top. We can read off the black 3k simulation trace, that 3000 Ohms will introduce an extra loss of 29.83dB, which looks about right compared to the blue S21 trace.

Now, can we correct the green S12 trace by the just computed attenuation to directly compare it to the measured blue S21 trace? Yes, we can. We open the custom trace editor again by double clicking onto the black 3k label and modify the experssion such that we multiply S12 with the previous expression:

Enter Expression 1 for trace 5:

Expression:
s12*s2t(z2s(3000))

Aliases:

S21 =   S11 =   S12 =   S22 =   Caption: 3k

Mem1 =   Mem2 =   Mem3 =   Mem4 =

ok   Save   Load

Indeed, the simulated black transfer characteristic and the measured blue S21 trace are in good agreement:
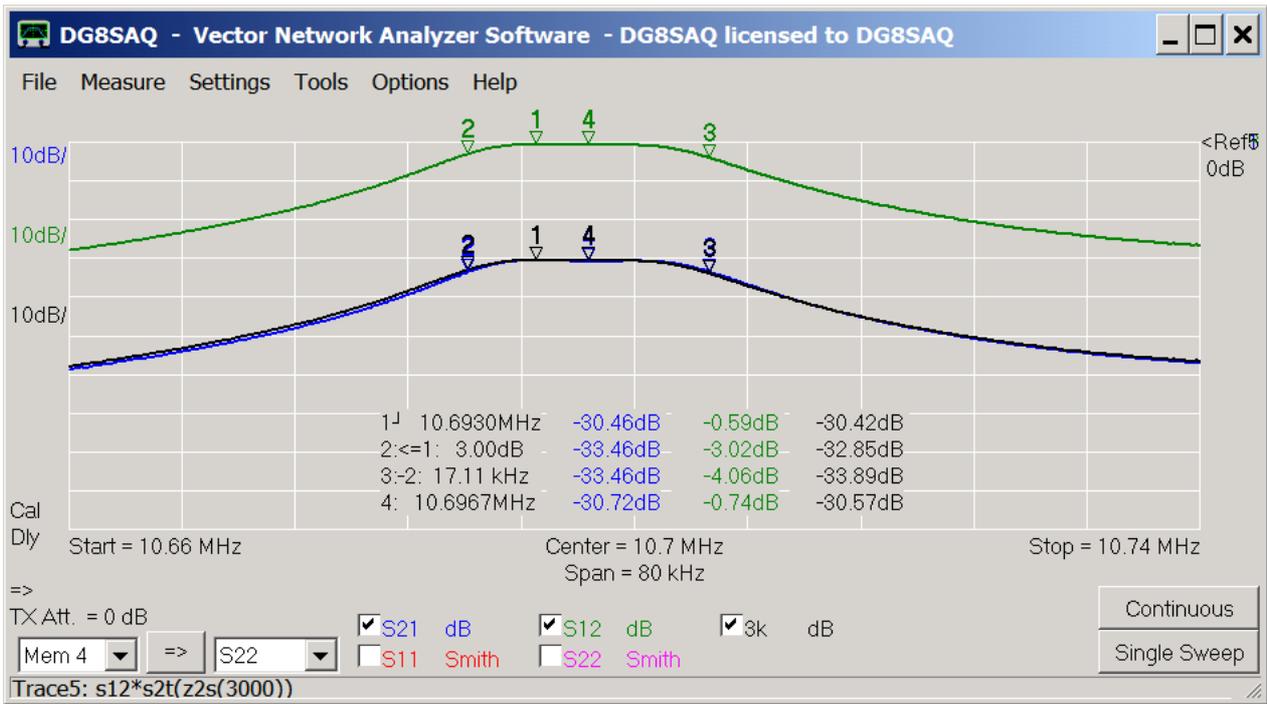
We have arrived at our black simulation curve in a handwaving way. Next, we want to verify it by doing a real network simulation

Playing with S-Parameters using the Network Simulator QUCS

QUCS is a powerful yet free network analysis software available for download at http://sourceforge.net/projects/qucs/. In the following, we will set up a QUCS project using our crystal filter S-parameters which we have just measured. First, we want to simulate how the filter behaves in between the two 1500 Ohms resistors as measured above.

After launching QUCS, we create a new project by clicking onto **New** in the **Projects** tab and by assigning a new project name:
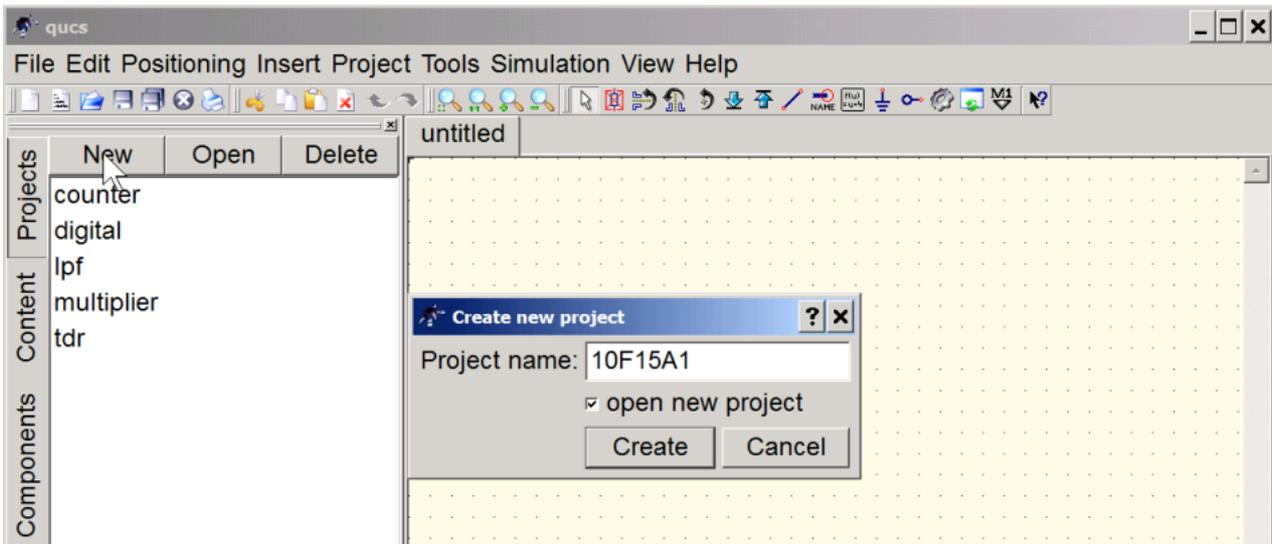
An untitled schematic opens:
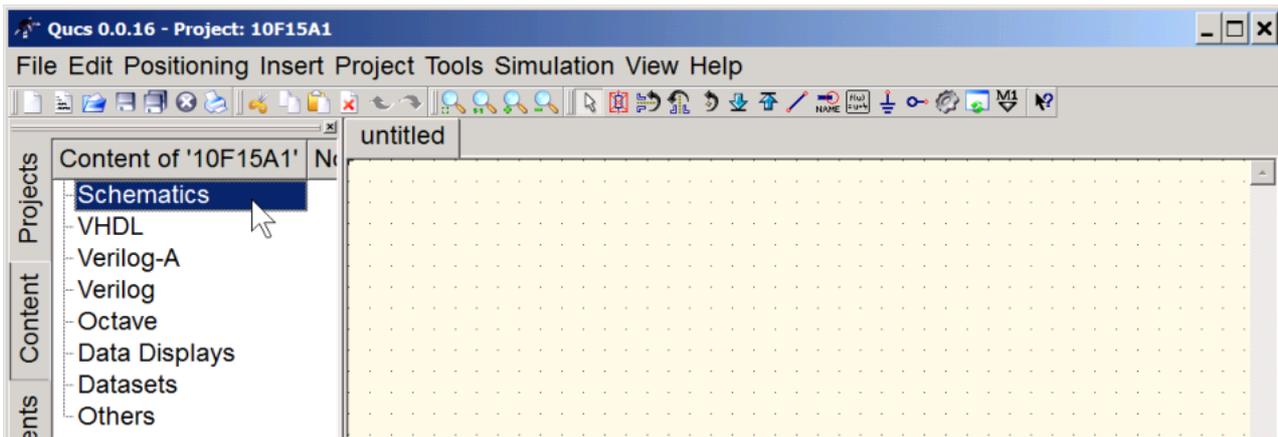
We save it ...
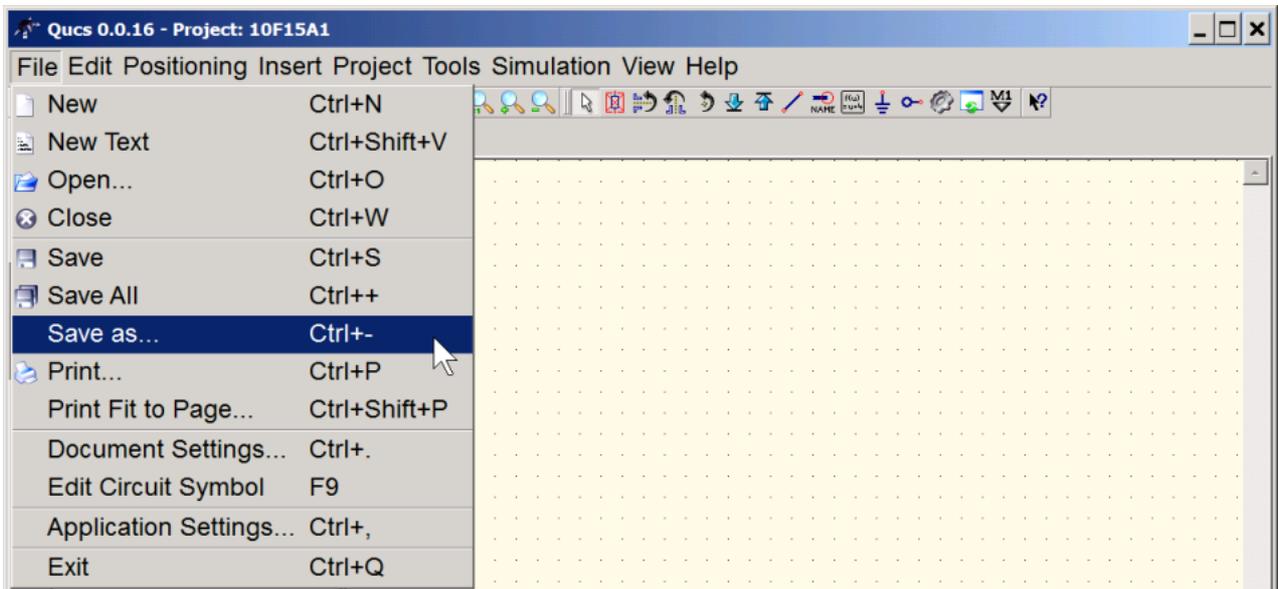
... and thus assign a name to it:
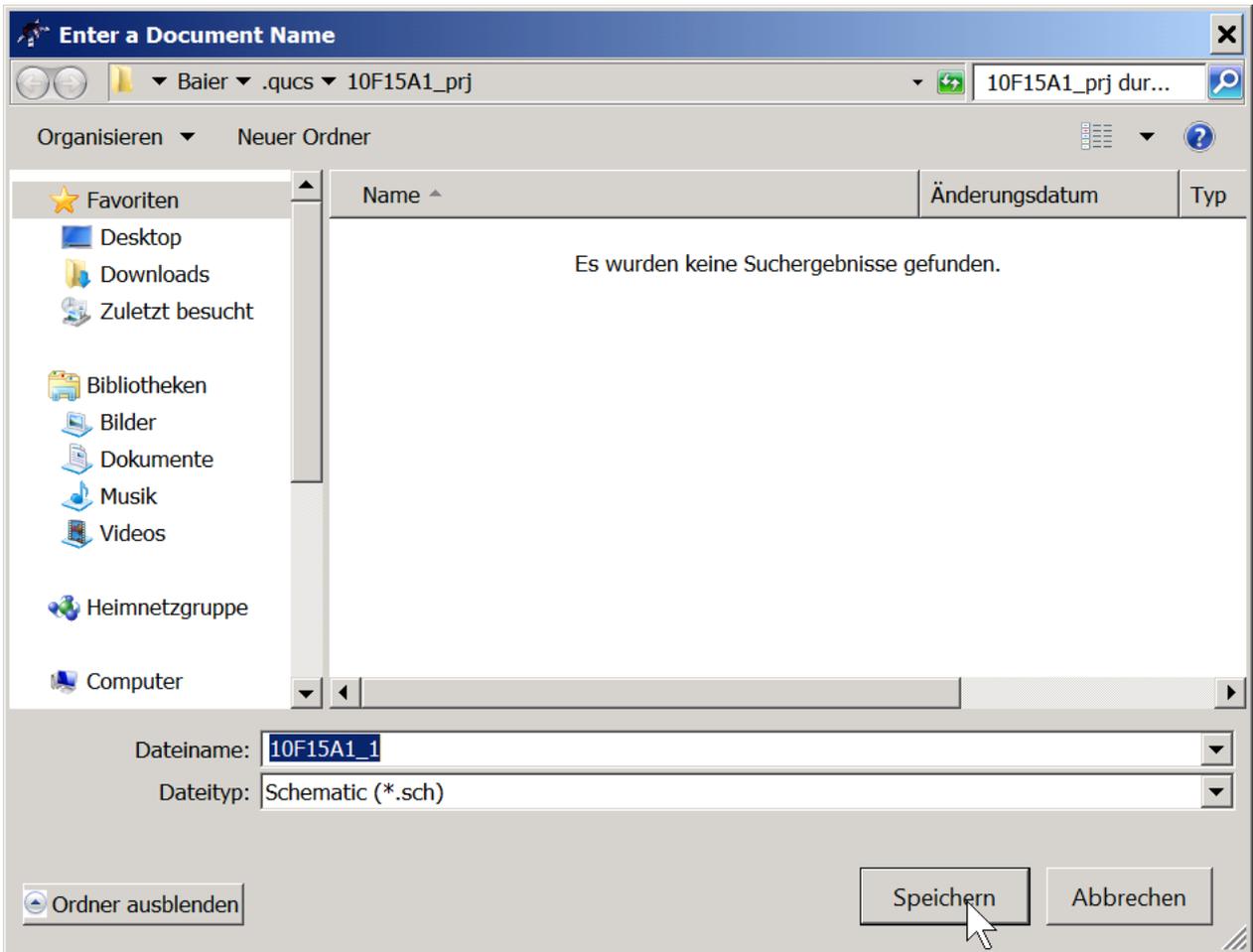
We change to the **Components** tab, ...
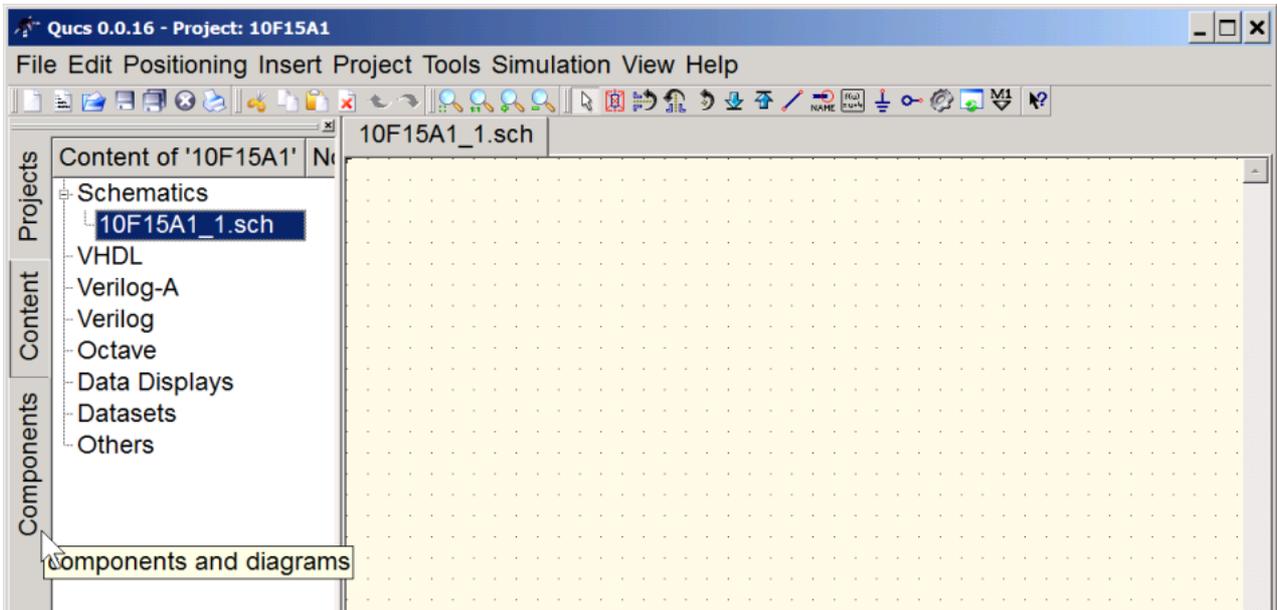
...select **lumped components** dropdown menu and add two resistors and two ground symbols:
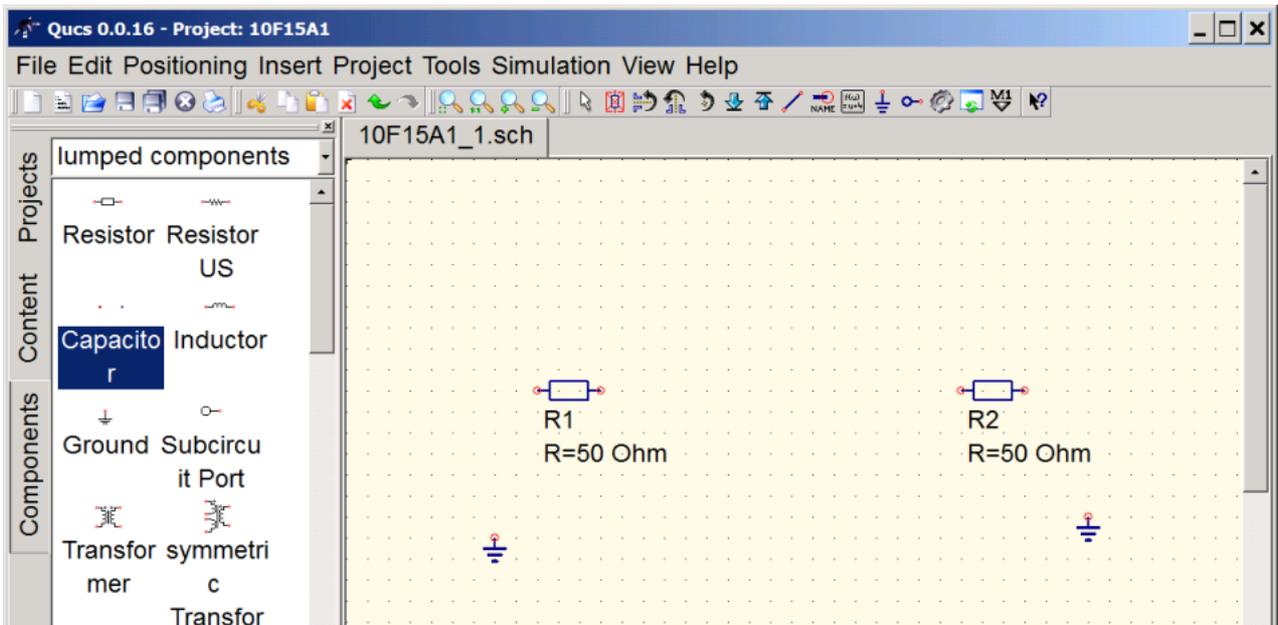
Next, we select the **sources** dropdown menu and add two power sources.
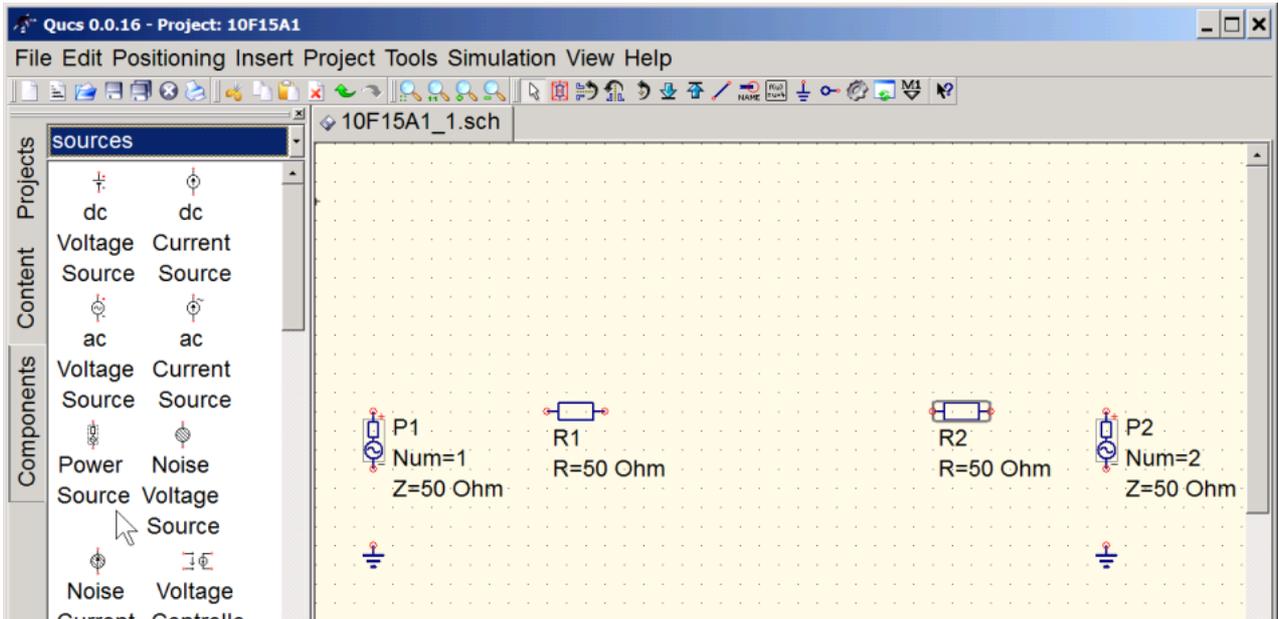
Then, we switch to **file components** and add a **2-port S parameter file** component (and from the lumped components another ground component which we have previously forgotten) :
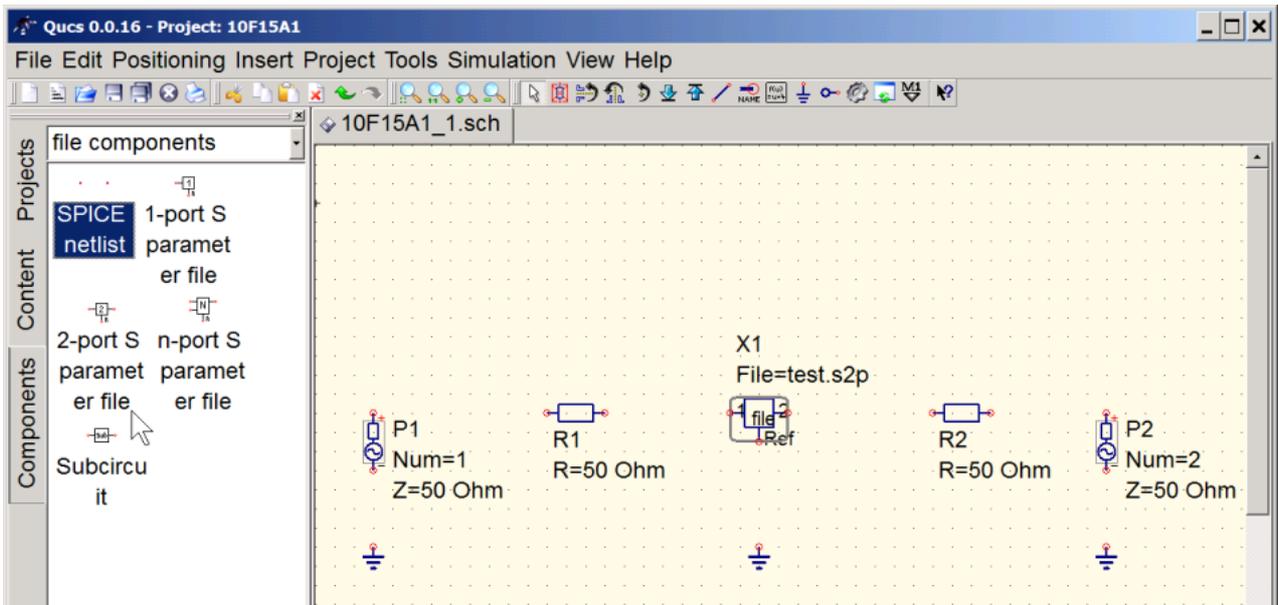
We need to change our resistors to 1500 Ohms, therefore we right-click the resistors and select **Edit Properties**:
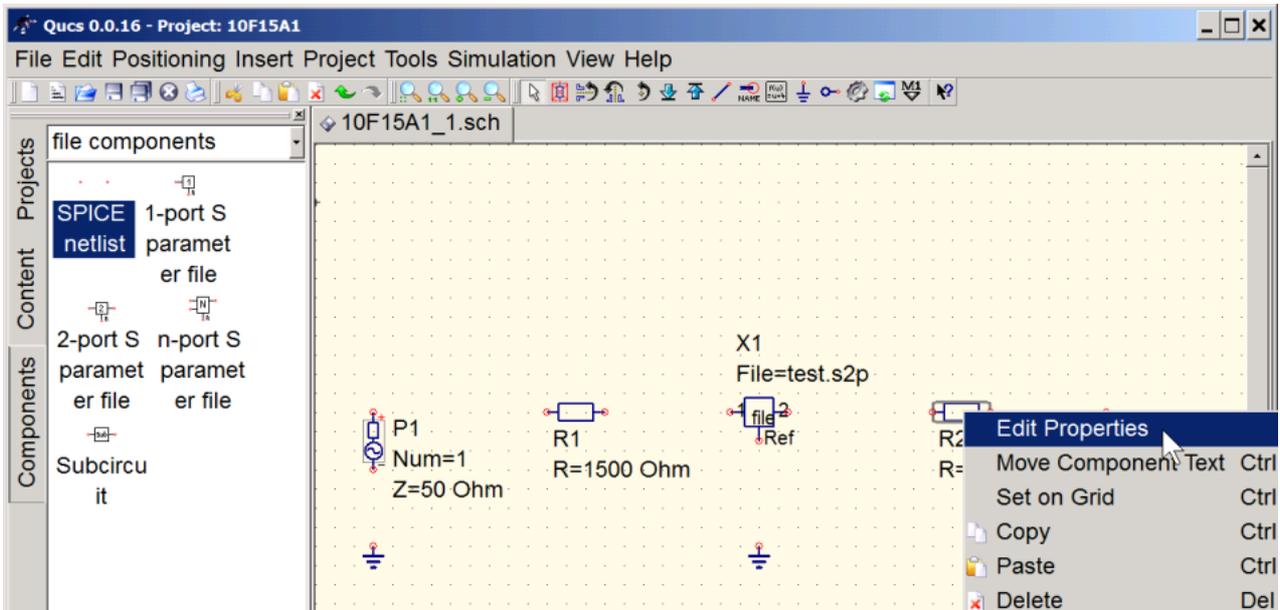
We highlight the R-property and change the value to 1500 Ohm
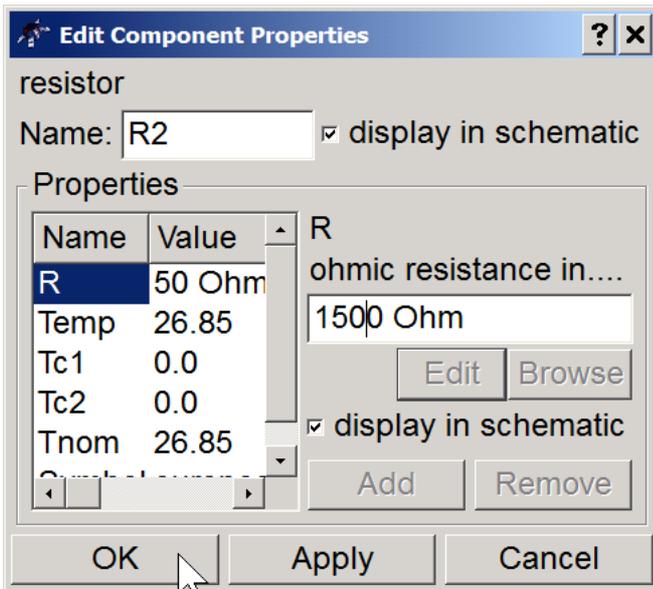
In similar fashion, we edit the property of the **2-port S parameter file** component by right-clicking on it and selecting **Edit Properties**:
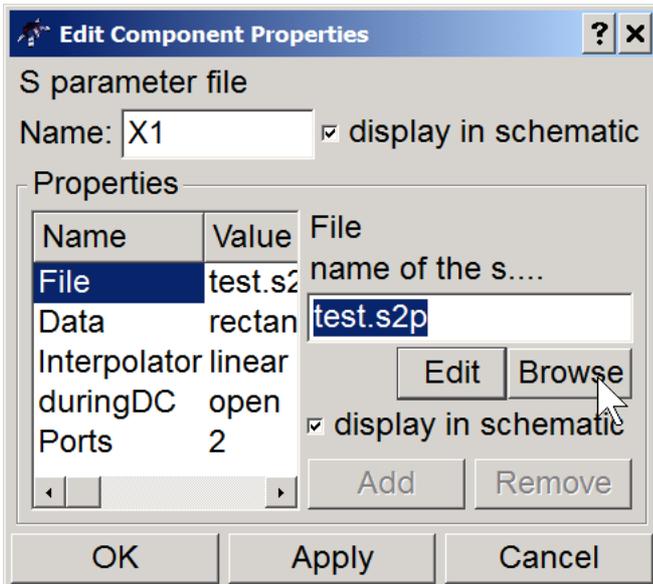
Here, we browse for our measured S-parameter file which we have saved from the VNWA software in the previous section.

Next, we wire the components:
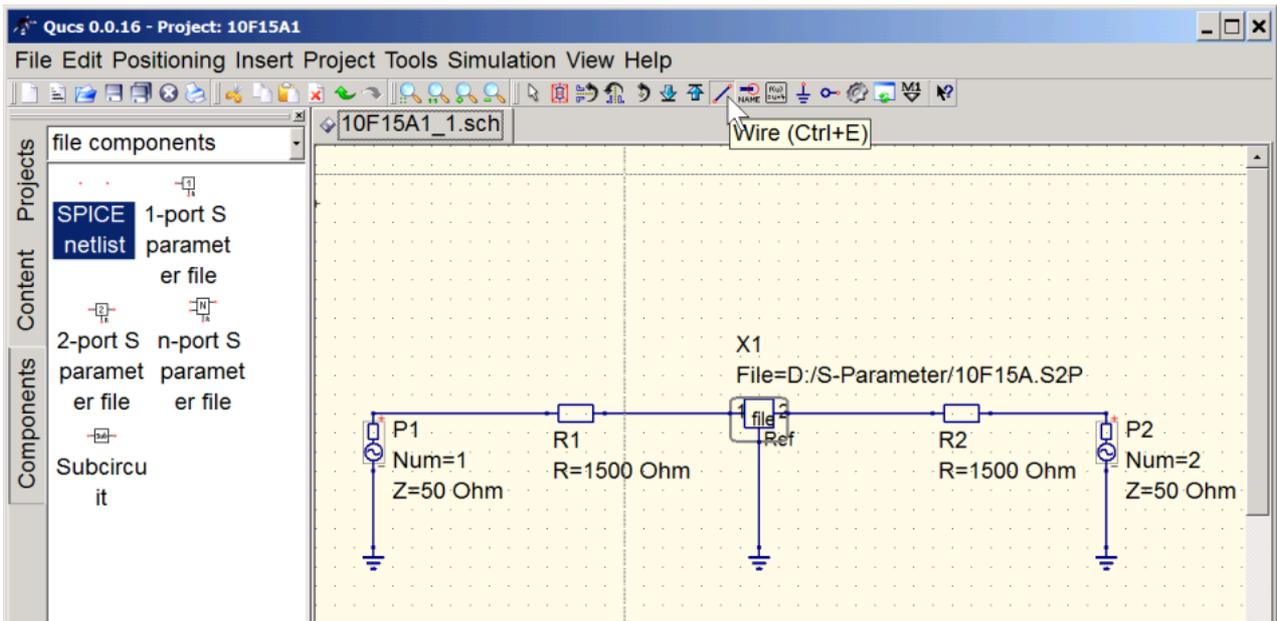
The sematic is done. It looks like the hardware in our previous measurement.

Now, we have to tell the simulator what kind of simulation we want. We change to the **simulations** components and add an **S parameter simulation**:
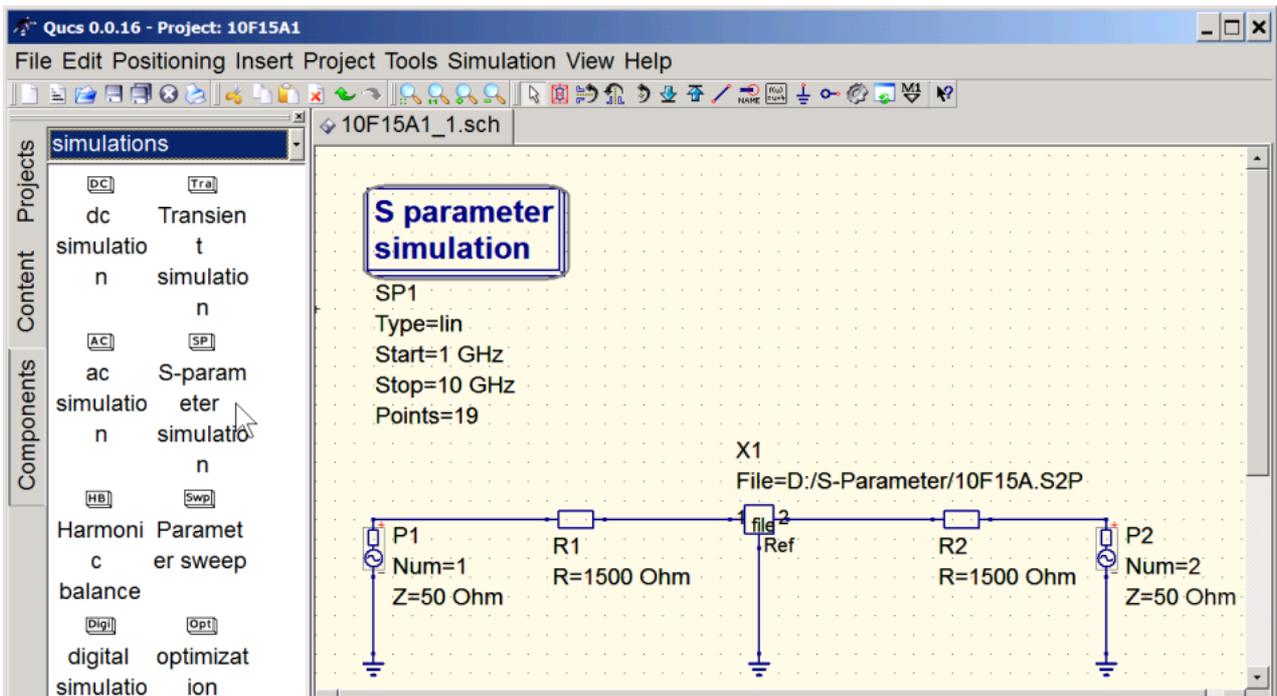
We edit the S parameter properties by right-clicking them and entering the desired frequency range and number of points:
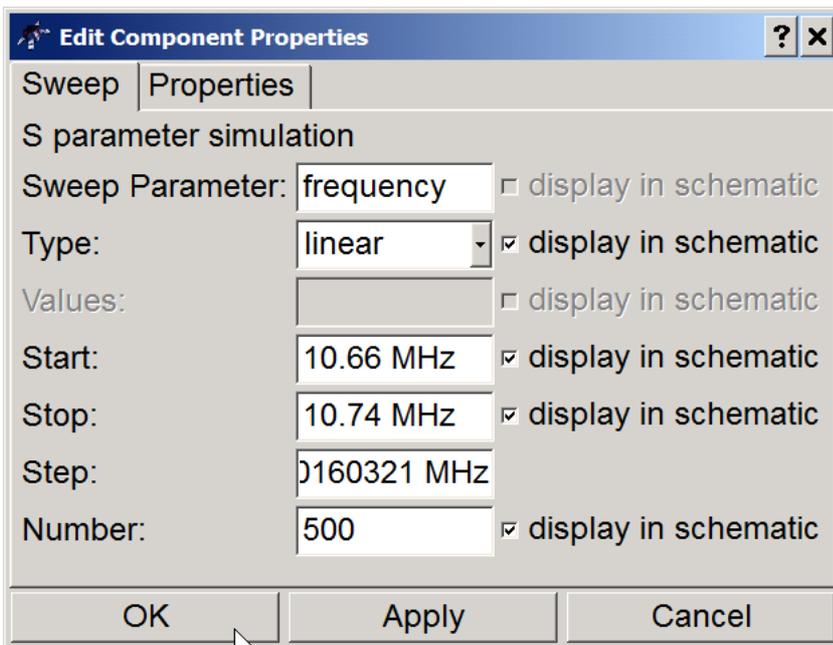
We use the same frequency grid as in our S-parameter file. Now, we can launch the simulation by pressing the **Simulate button**:
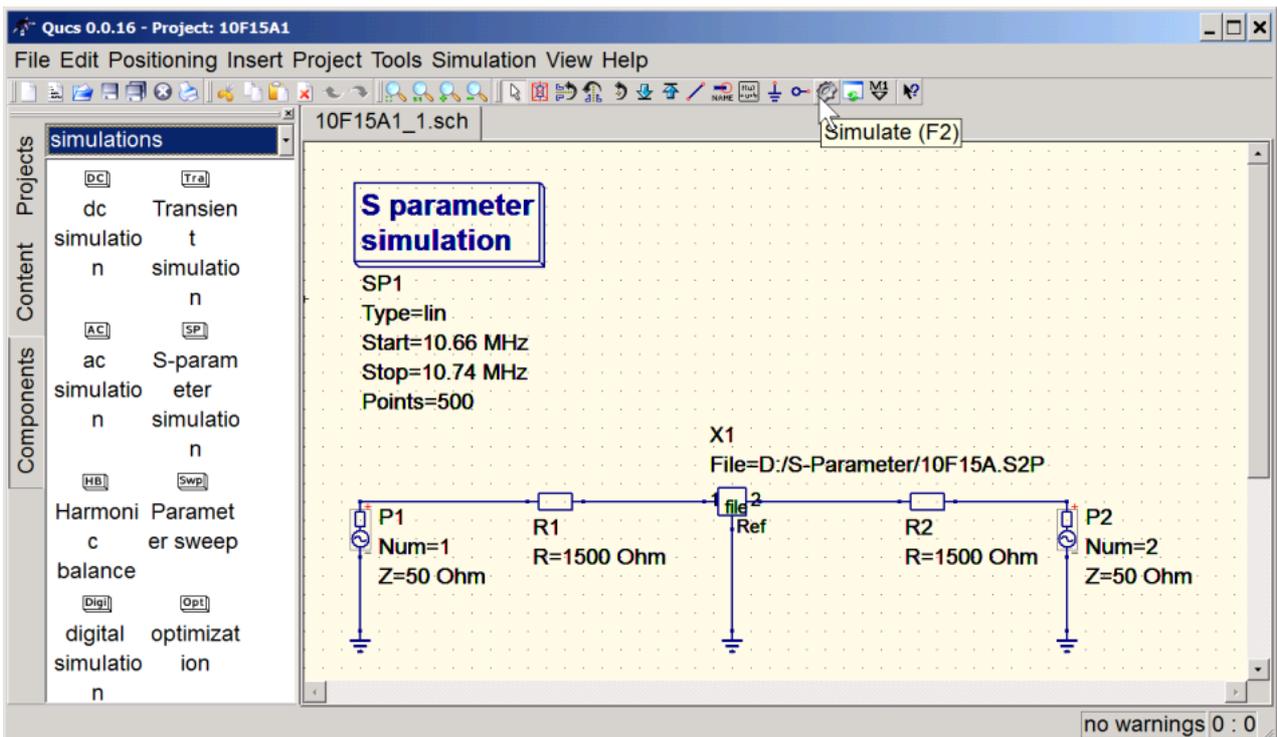
The plot window opens, but it is empty. We still need to add a diagram component. We select a **Carthesian** plot...
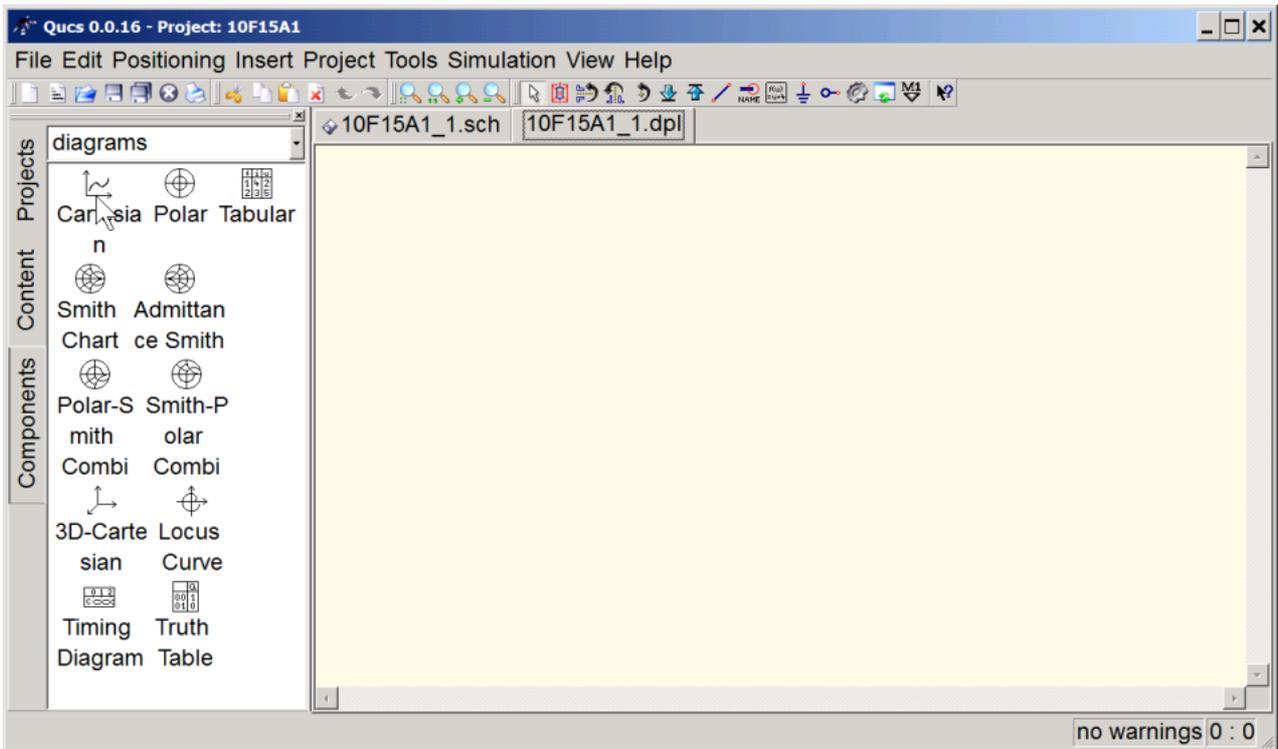
... and add it to the document. The diagram properties pop open. We add an S21 trace by double-clicking onto the highlighted **S[2,1]** property ...
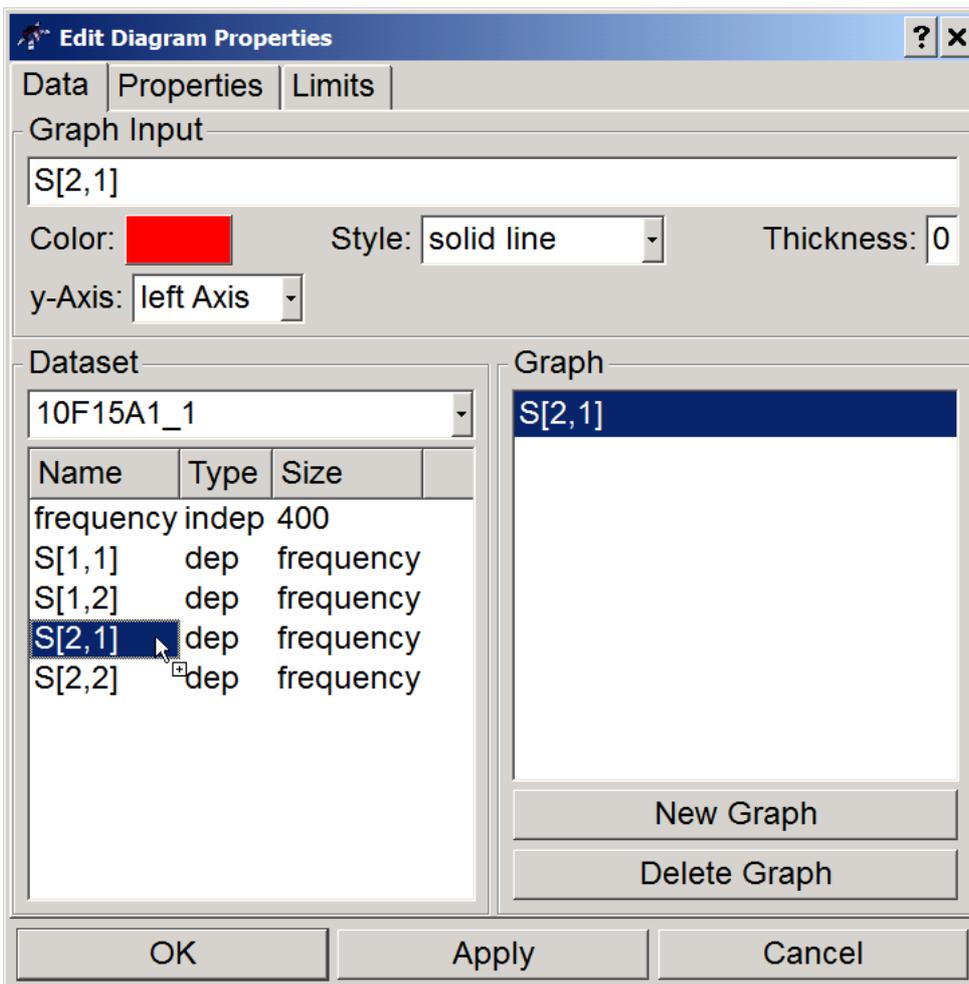
... and specify that we want a logarithmic left axis:
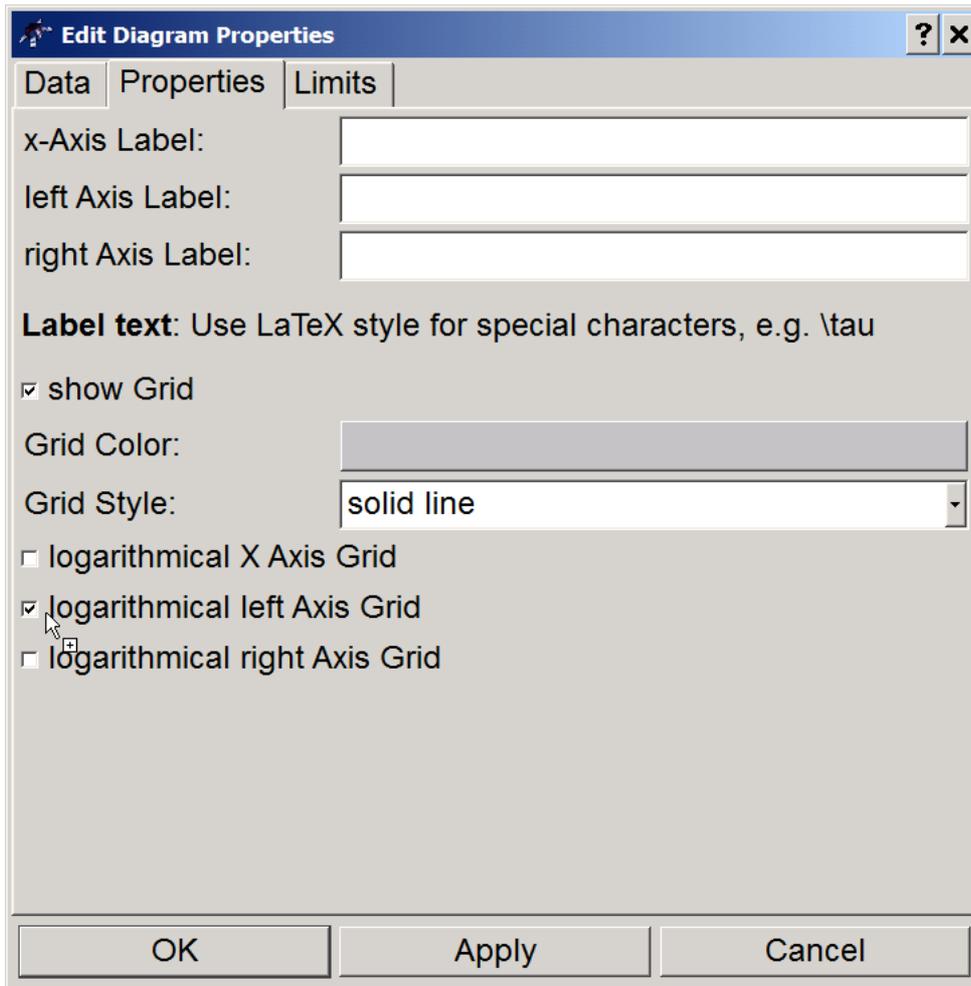
So, finally, we see our simulation result:
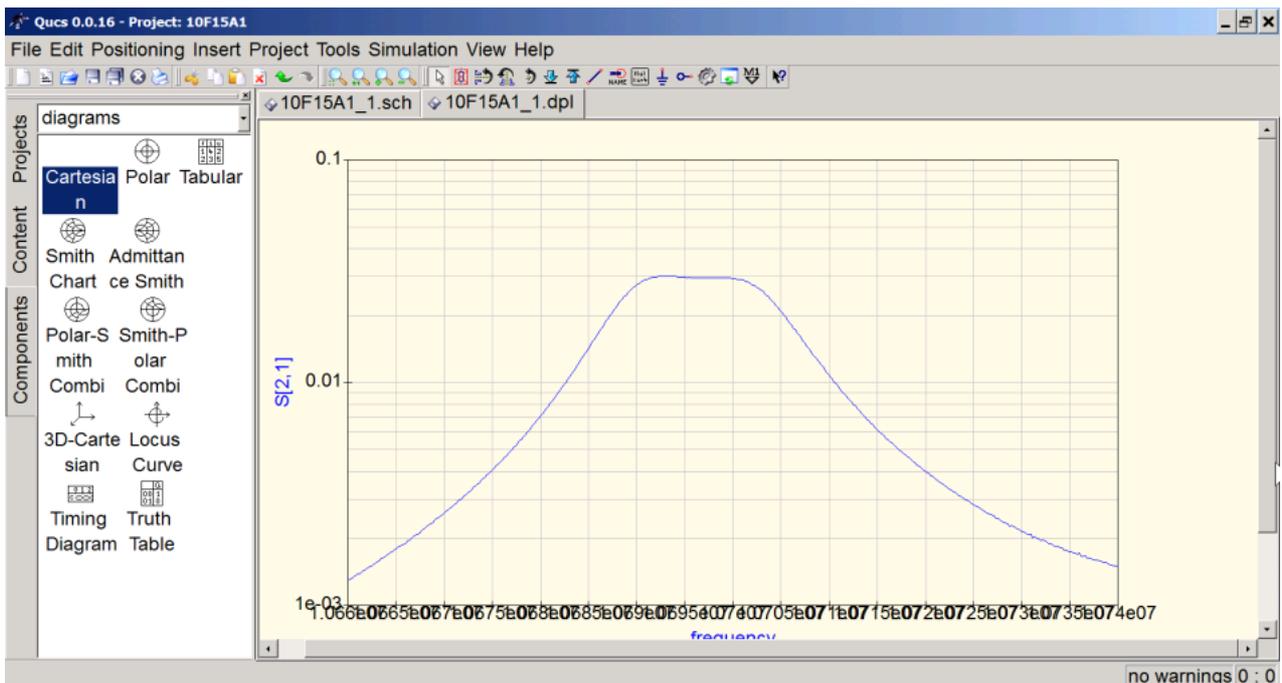
The plot is not very nice, and we want to compare this simulation with our previous measurement. Therefore, we are

going to export the simulation data back into the VNWA software.
To do so, we right-click the bulu simulation curve and select **Export to CSV**:
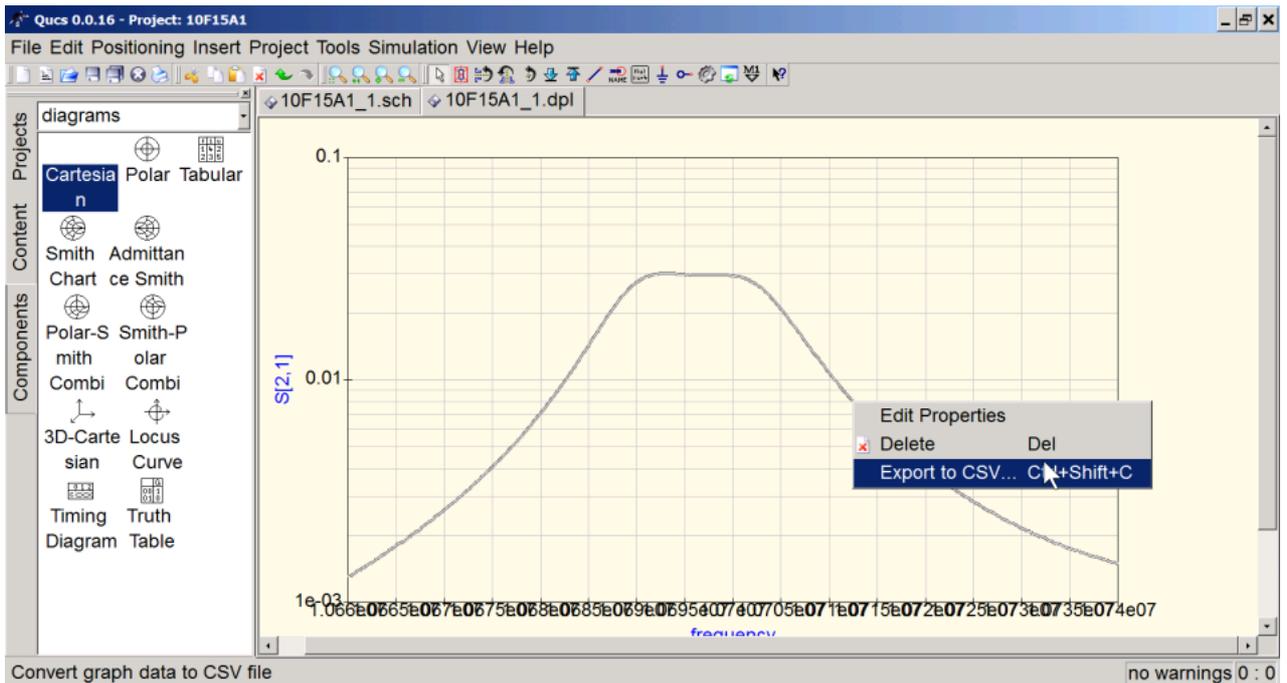
We save the simulation result as csv-file on disk:
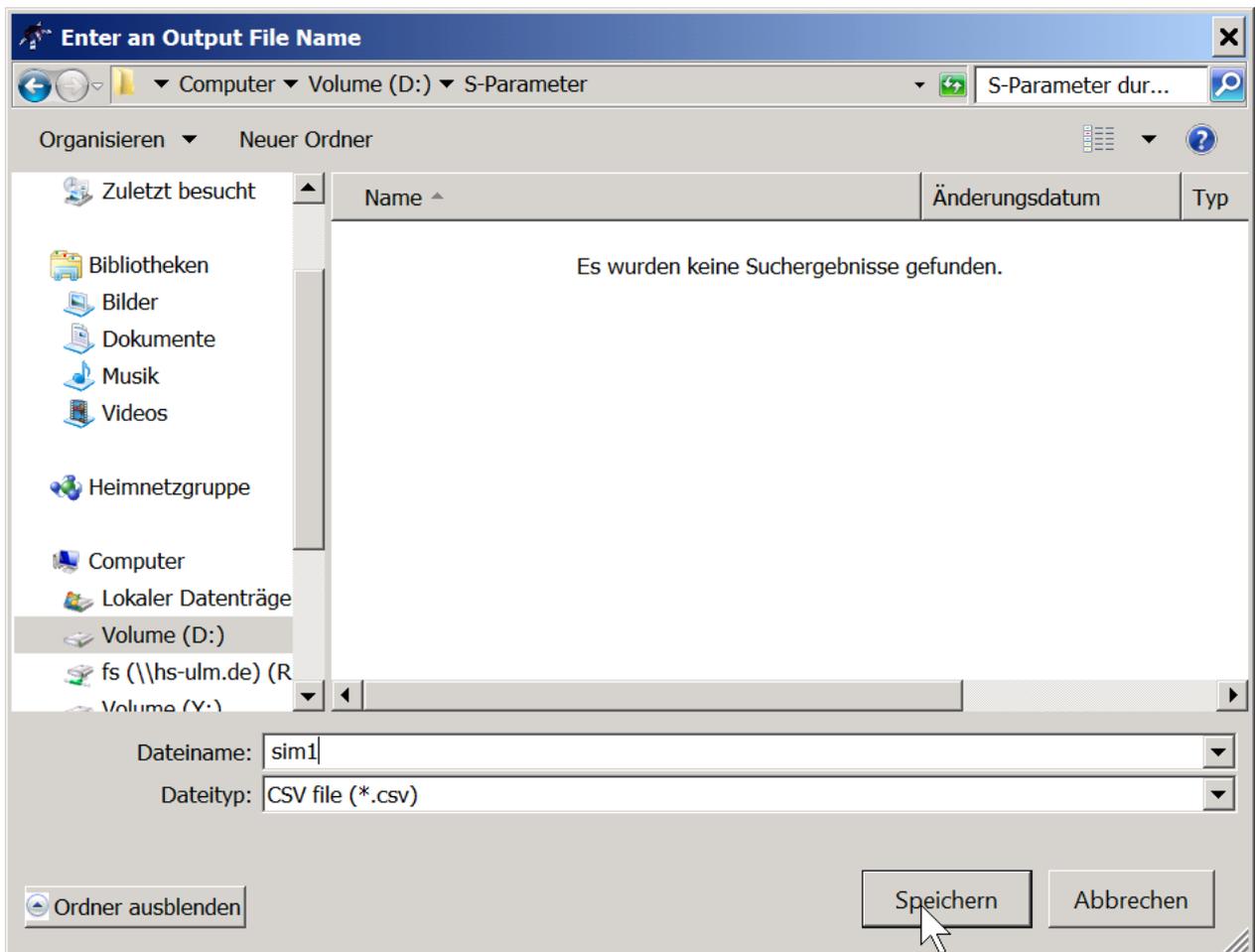
And we switch over to the VNWA software, which still has our old measurement and our handwaving simulation result on

screen. There, we import the just saved QUCS simulation result into Memory 1:
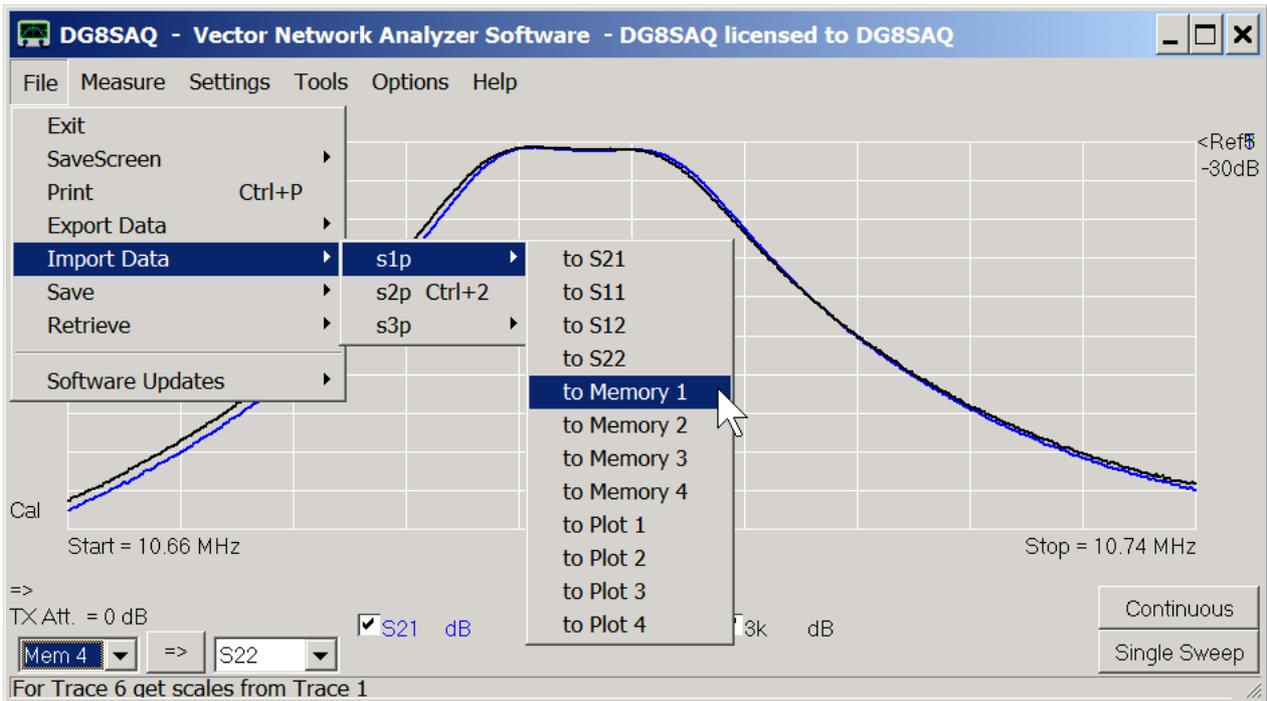
We select file type QUCS, ... (*.csv) and select our simulation file from disk:
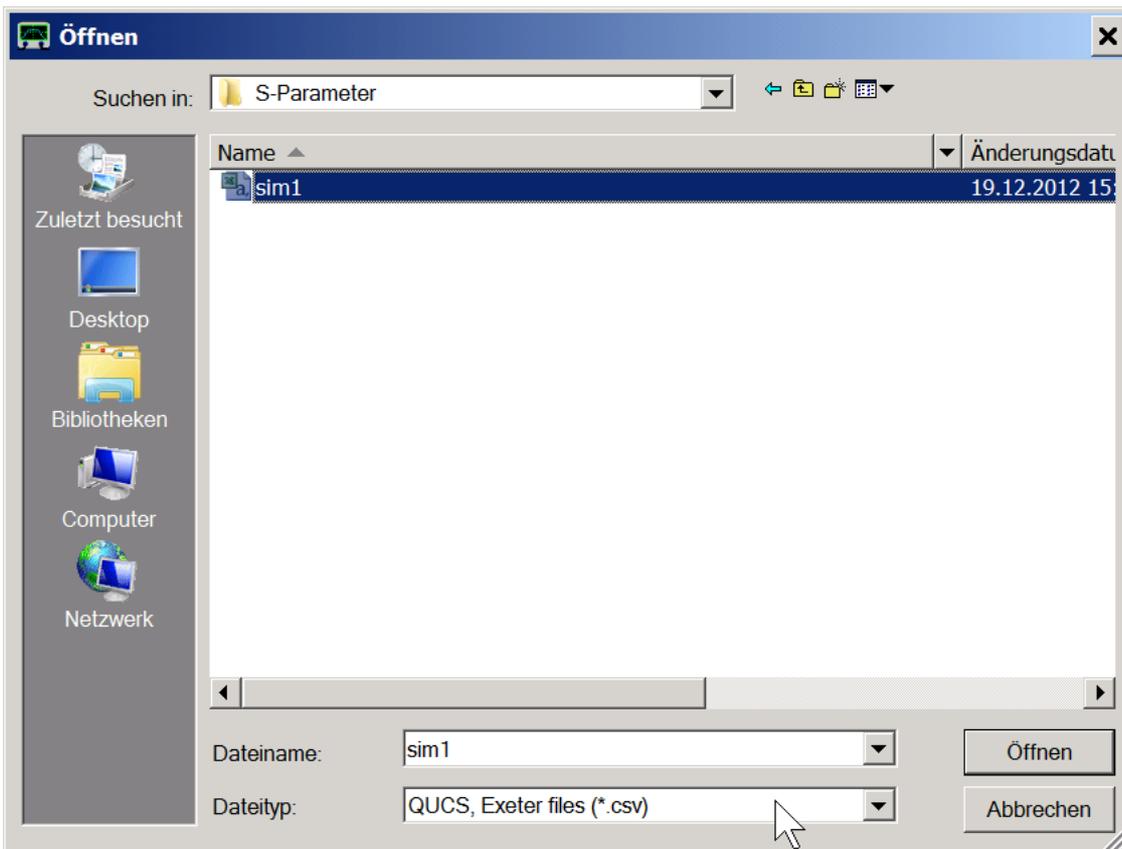
We open another trace to display our QUCS simulation in Mem1:
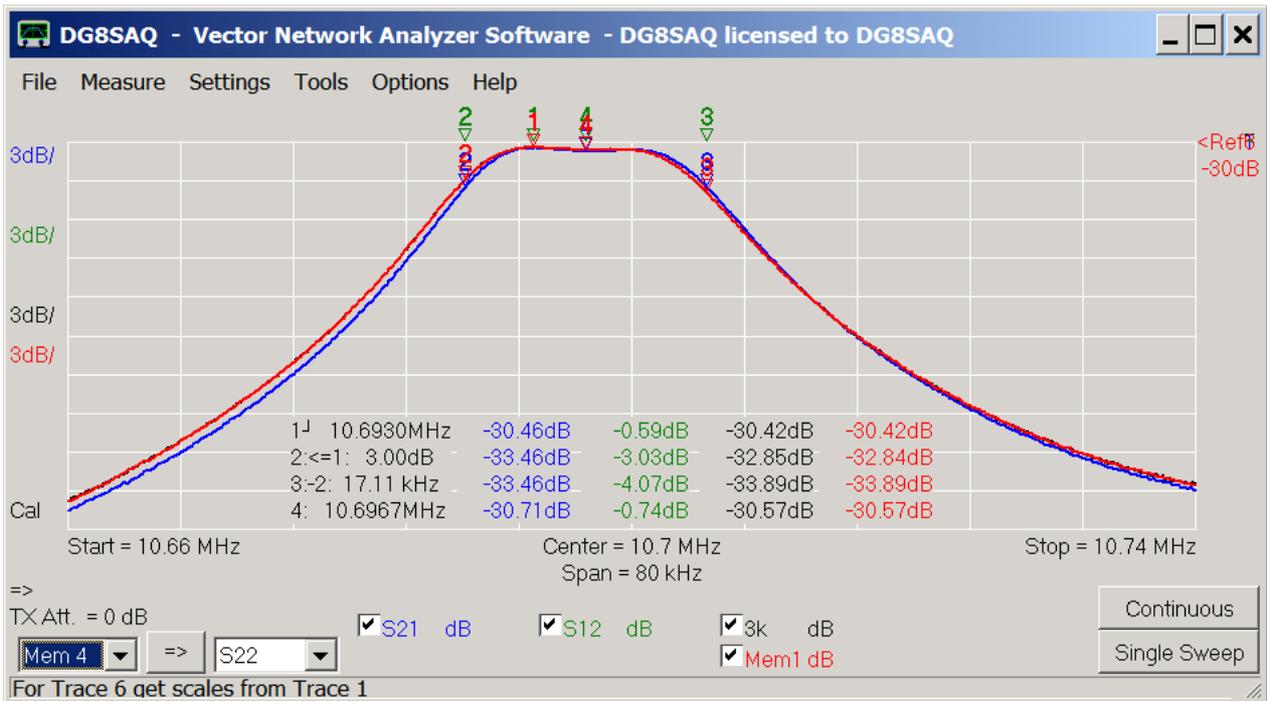
Note, that our handwaving black simulation agrees with the red QUCS simulation to almost the last digit. Therefore, the black trace lies hidden behind the red trace.

What else can we simulate with our S-parameters?

We can simulate what our filter would look like in a 1550 Ohms environment like we have done with the VNWA matching tool. To do so, we remove the resistors and change the sources' impedances to 1550 Ohms:
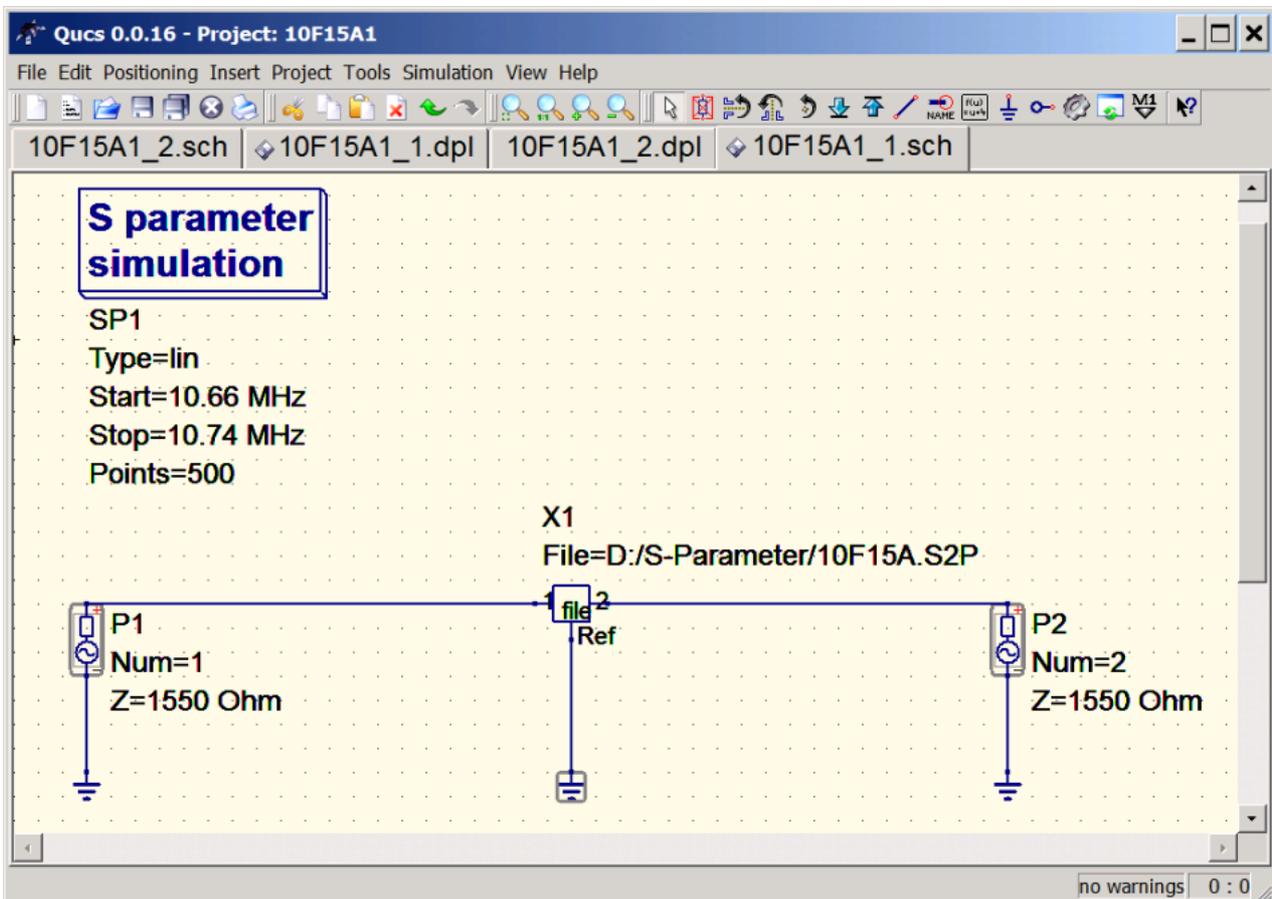
Not surprising, the result is identical with the VNWA result:
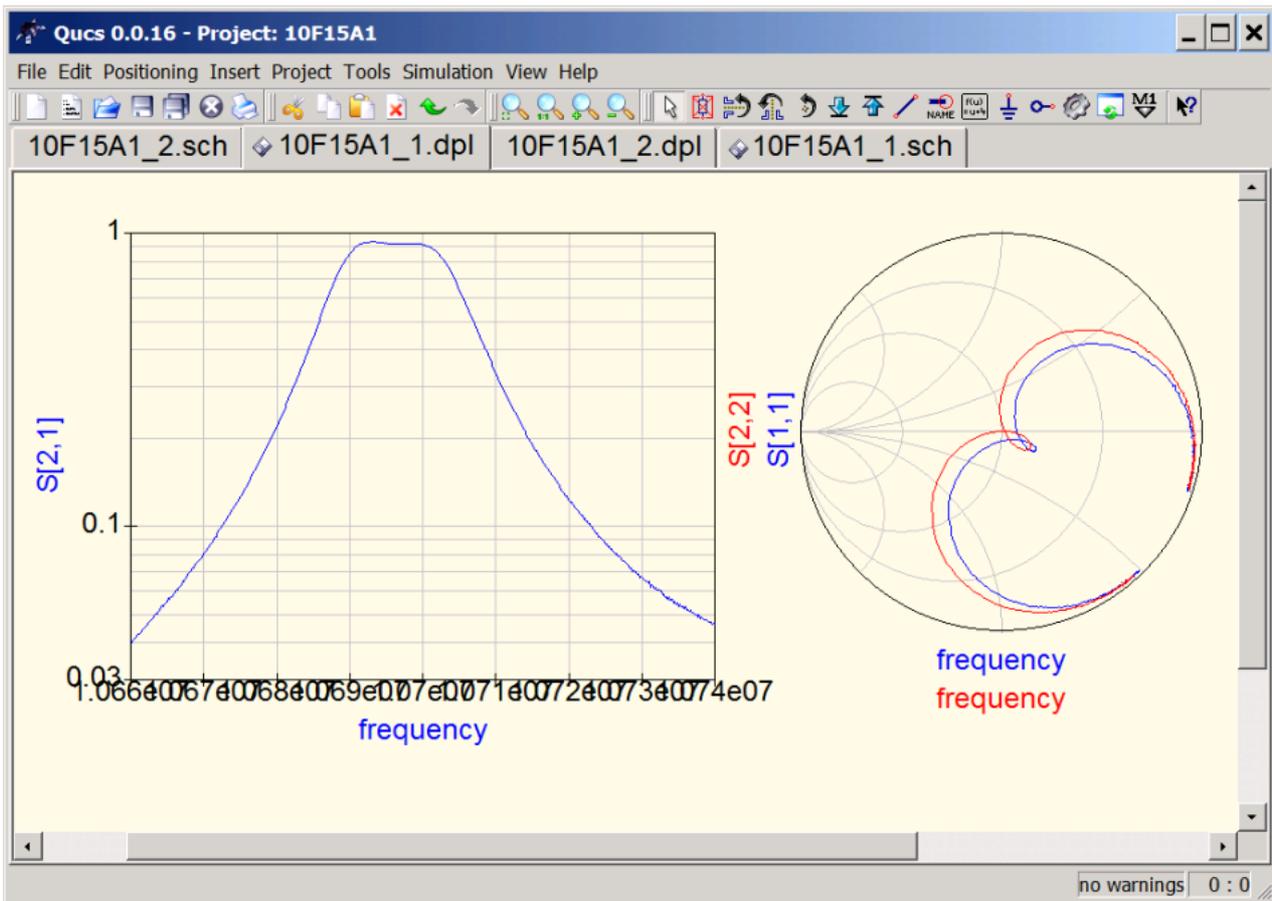
We have added a Smith chart to display the simulated S11 and S22.

Note, that the stopband attenuation of our two pole filter is not very high. We might improve it by cascading two of these filters. We can easily simulate the expected performance by adding another S parameter file component which actually points to the same file as the first one, since we want to cascade two identical filters:
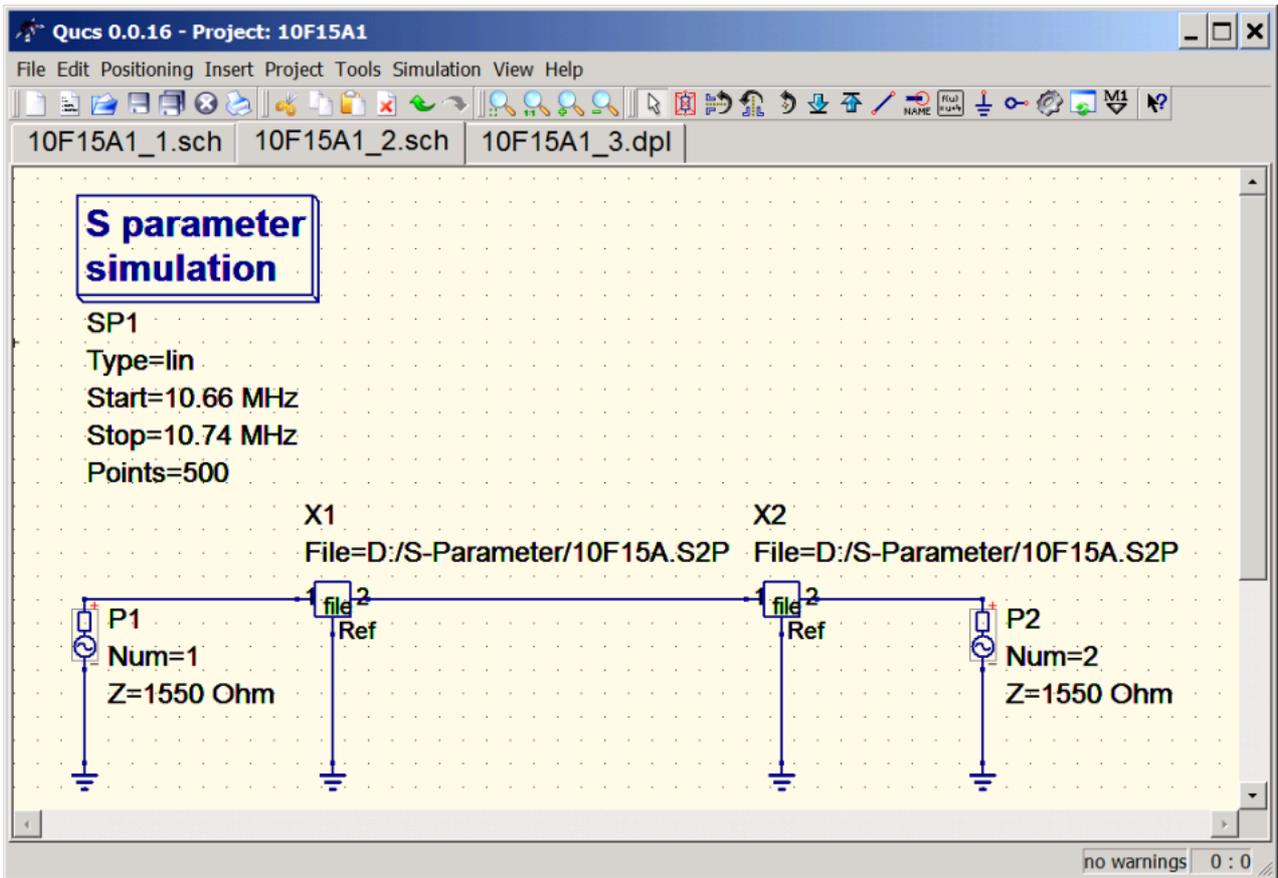
The simulated S21 data shows an ugly hump at the high frequency skirt which is due to internal mismatch between the two pole filters:
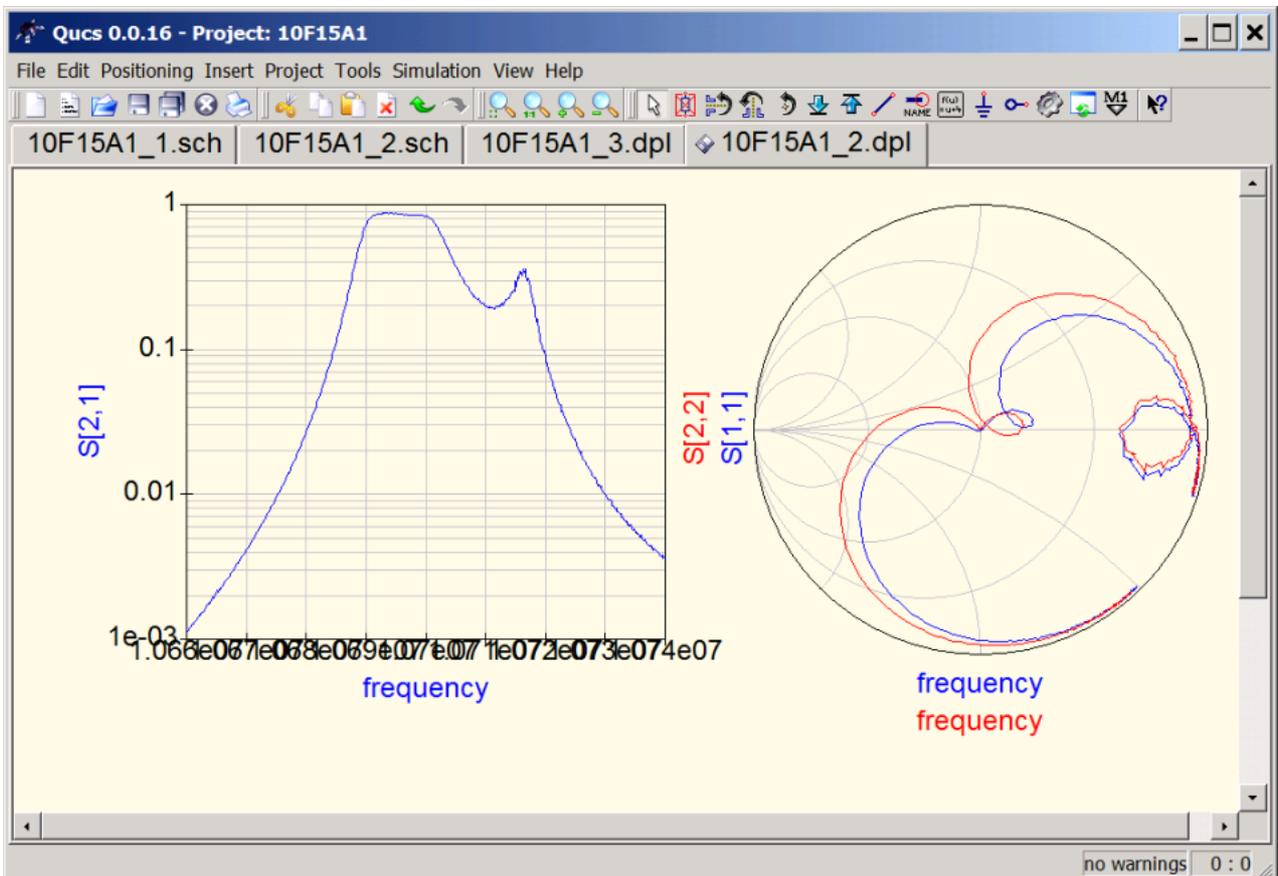
We add a capacitor to the internal filter connection to improve the match:
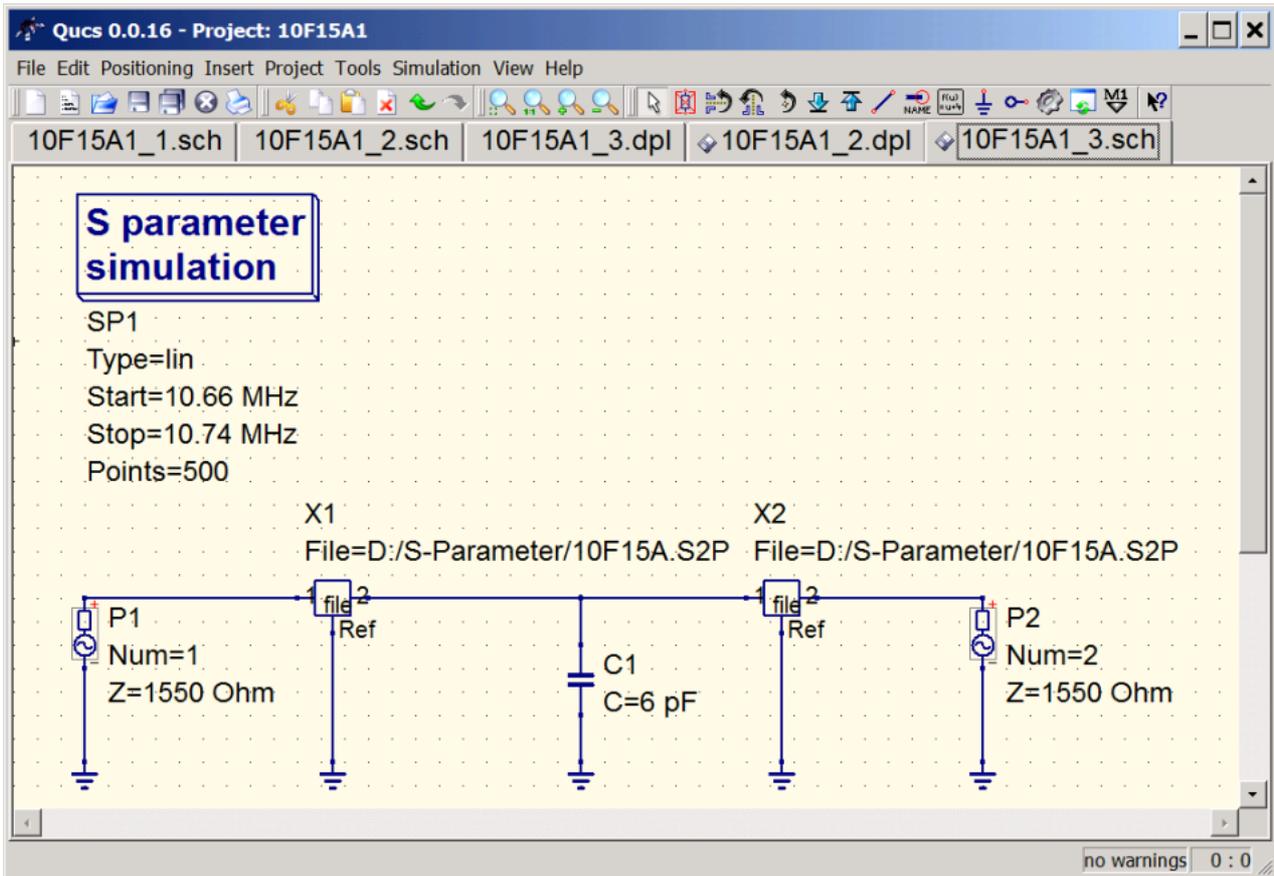
Now, we have a decent passband and a considerably larger stopband attenuation.
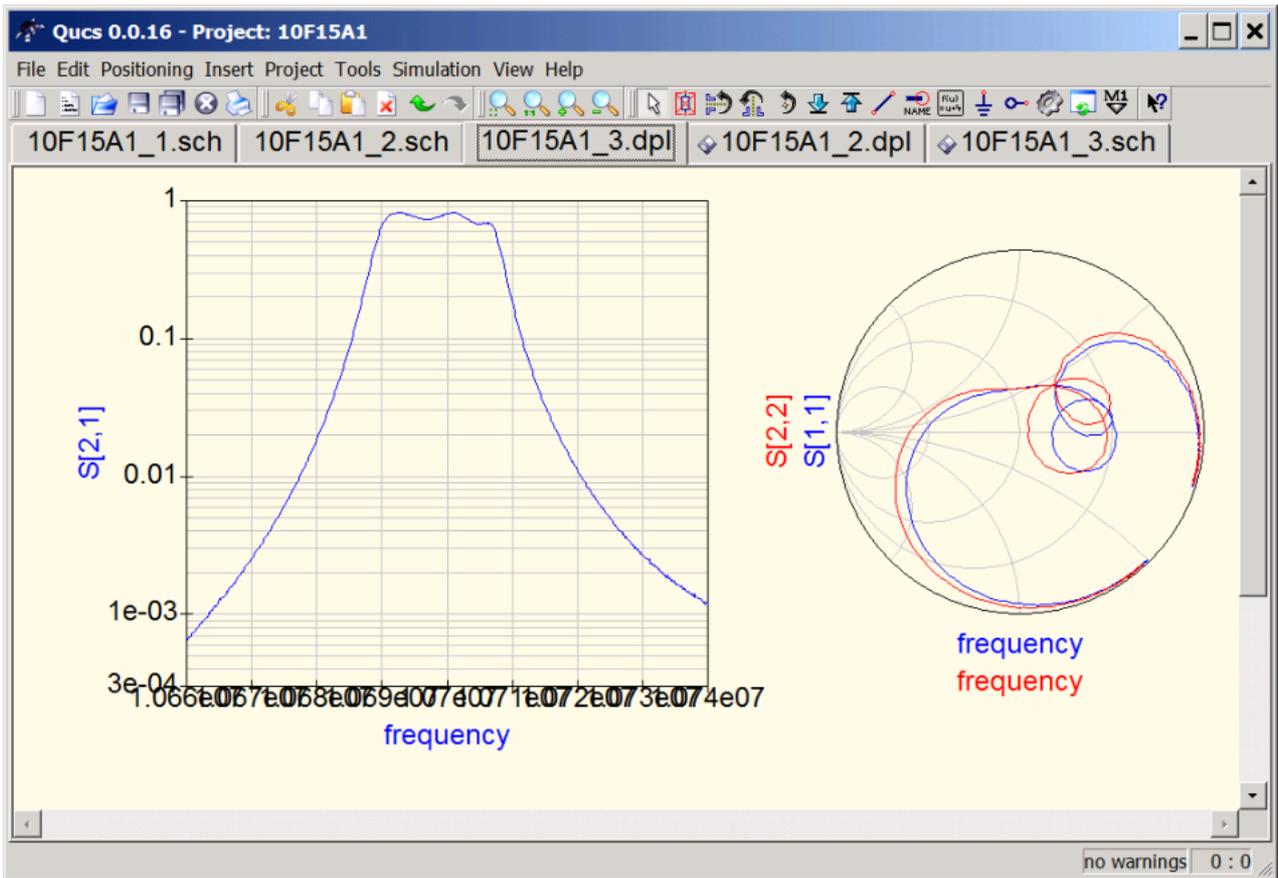
We could easily continue to optimize internal and external matching and thus design a perfect four pole crystal filter. Obviously, a simulator like QUCS together with the VNWA for acqiring S parameter data is a powerful pair for RF design.