# QucsStudio Tutorial

# Part 1:

## Simulations in the Time Domain and in the Frequency Domain


## Version 1.6

**Author:**
**Gunthard Kraus, DG8GB**,
Guest Professor at the Duale Hochschule Baden-Wuerttemberg (Friedrichshafen / Germany), assisted by the Program's Author Michael Margraf, DD6UM

Email: mail@gunthard-kraus.de
Homepage: www.gunthard-kraus.de

**March 10th, 2016**

# Content

**Page**

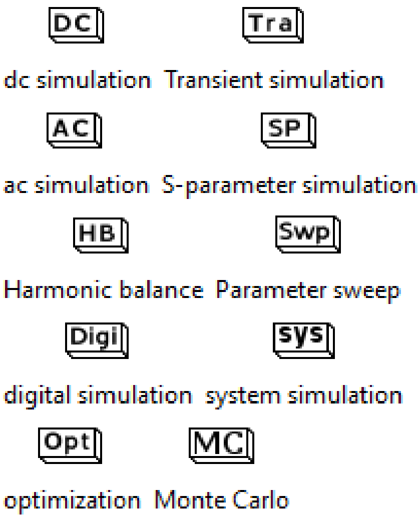
# 1. Introduction
We start with a confusing fact:

## „Qucs“ and „Qucs-Studio“ are existing!

Let's at first have a short look at „Qucs“.
„**Qucs**“ means „**Quite Universal Circuit Simulator**“ and is free of charge. There are no restrictions and you can either use the **English or the German Language**.

**(...Switching is possible in the menu „Files / Application Settings / Language).**

**Time Domain and Frequency Domain Simulations are possible.** And the Microwave Specialist will find his beloved Smith Diagram, the Noise Simulation and the Stability Check. But the most important feature is „**Harmonic Balance**“ – a simulation in the frequency domain in a circuit which also contains nonlinear devices. So qucs is the only software on the market which offers this option without any charge.




**So have a look at this illustration with the simulation possibilities to get hungry...**

When examining the program you'll at once recognize the usage philosophy: it comes from the university campus and offers a lot of tools for any problem (..but not to save time in comparison to other software).

**But its famous operating range is the RF- and Microwave Technique including communications by light. For these applications you'll always find a solution.**

So please like the software and do not complain about the necessary efforts to get familiar with it.....

But some day happened what also happens when parents die: the children are not able to find a common sense for the future. So chief designer Michael Margraf (DD6UM) decided to leave the author team and to walk his own way. He wrote the software completely new, continued to develop new features and named it „**QucsStudio**“.
So there are sometimes some „incompatibilities“ between qucs and qucsstudio. But it seems now that only qucsstudio is updated and improved and bug fixed.

## 2. Download and Installation of QucsStudio

# QucsStudio Homepage

deutsch

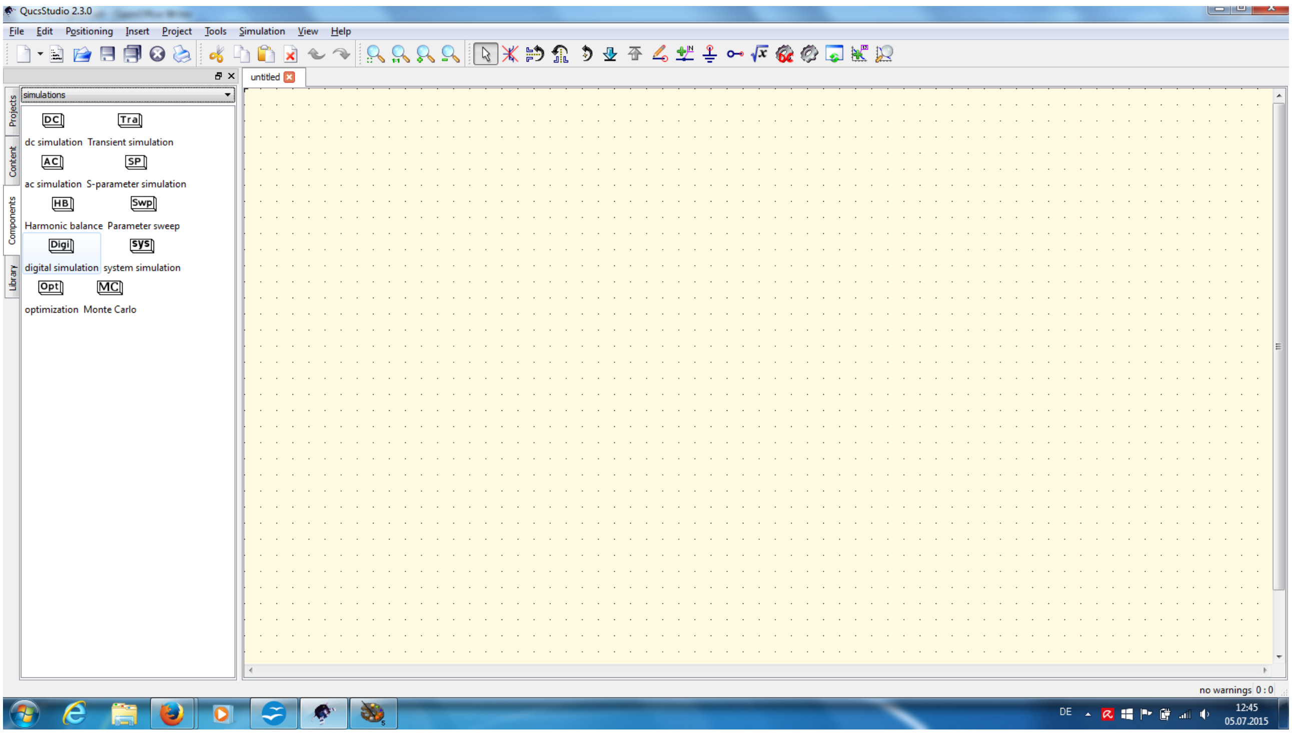
news - about - screenshots & videos - download - examples - FAQ - li

*Latest News*

| | | |
|---|---|---|
| 23th June | 2015 | **version 1.2 of the german tutorial by Gunthard Kraus available** |
| 24th April | 2015 | **version 1.1 of the german tutorial by Gunthard Kraus available** |
| 10th April | 2015 | **QucsStudio Version 2.3.0 released and examples updated** |
| 25th February | 2015 | **first version of the german tutorial by Gunthard Kraus available** |


Use Google, enter „qucsstudio" and download the software from Michael's homepage. At this moment it is version „2.3.0" and you get it in zipped form. But pay attention: the used zipping software is „7z".....

The installation runs without any problems and when you see the left icon......please start the program to get this screen:




**Remarks:**
Have a look at the other offers in the homepage (examples. Screen shots etc.). They are sometimes very helpful if you fight against problems.

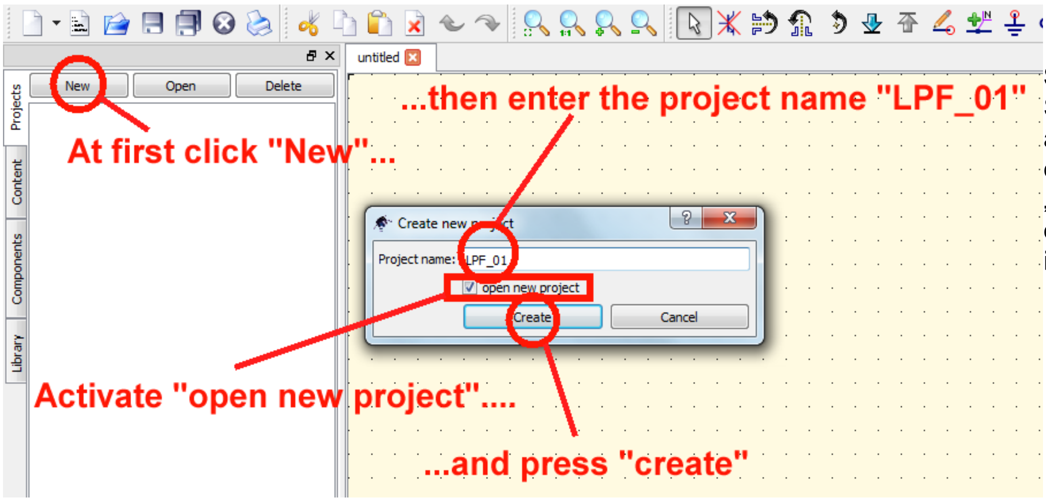
**The program does not produce any Windows registy entries**. So it can be marked, shifted, deleted...without problems.

# 3. First Project: RC Low Pass Filter

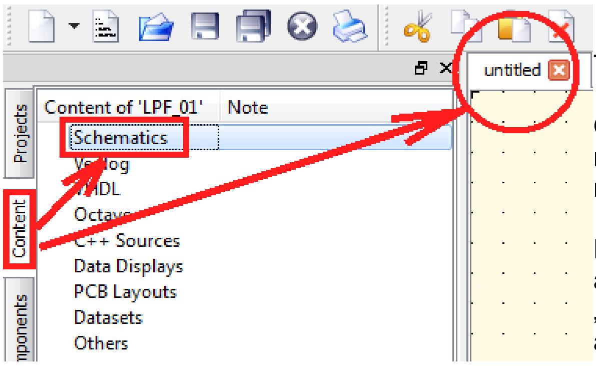
## 3.1. Simulation in the Time Domain: the Transfer Function

### 3.1.1. Linear and logarithmic Presentation of Simulation Results

We start with a simple circuit = an RC LPF and simulate the transfer function versus frequency for a Sine Wave signal at the input.




**Step 1:**
Start the software and install the desired project „LPF_01“ in the order as given in this illustration.

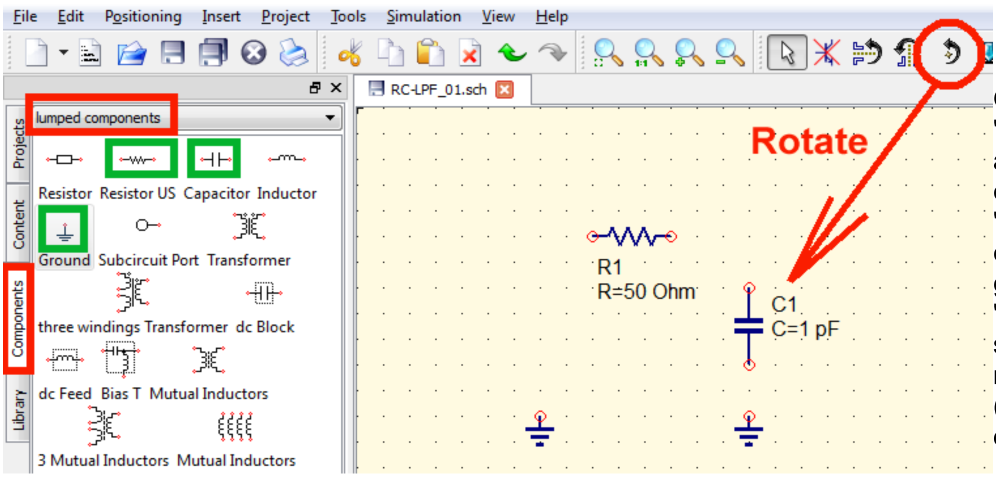



This should now be your screen.

Qucsstudio has opened the project folder and you have to name **and to save your schematic** of the LPF (….at this moment still named „untitled“).

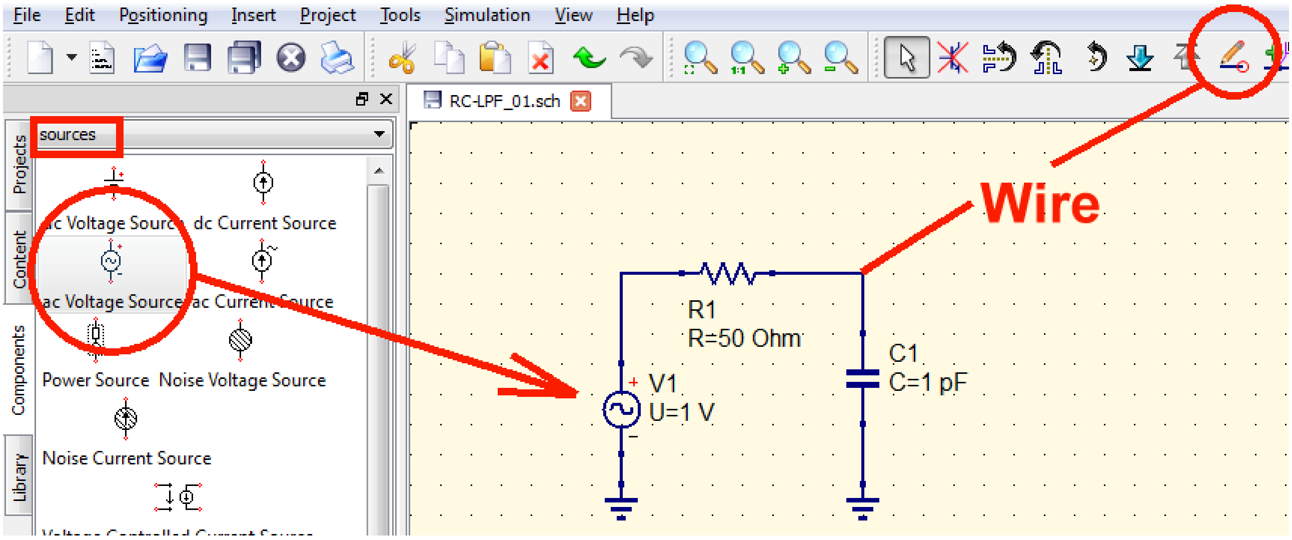
Please give it the name „**RC-LPF_01**“ and save it as usual and familiar under Windows (= open „File“ and „save as“. The necessary path will be set automatically).

**Step 2:**

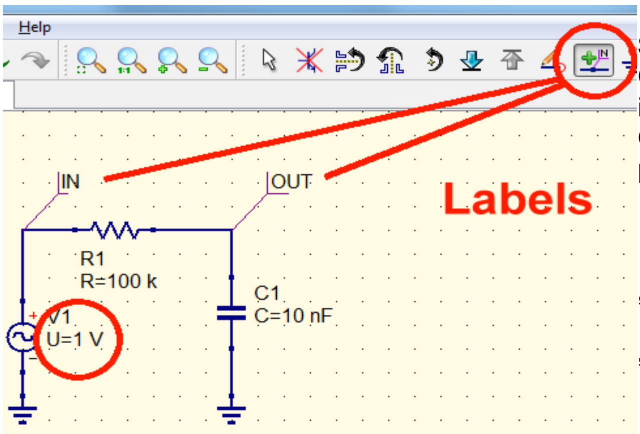



Open the **"Components"** menu and **"lumped components"**. Place a **"resistor US"**, a **capacitor** and **two ground** symbols by "drag and drop" on your screen. Do nor forget to rotate the capacitor.
(= mark the part and enter <Control> + <r>)

**Step 3:**
Change to the „**sources**“ menu, pick an AC Voltage Source an place it in your schematic. Activate „Wire“ and create all necessary connections of your parts :




**Step 4:**
We set the correct resistor and capacitor value.
At first right click on the symbol of the **resistor**. In the opened property menu of this part press „**edit properties**“ and enter a value of **100 k**. Close by „Apply“ and „OK“.
In the same manner right click on the symbol of the **capacitor**, edit the properties and enter the value of **10 nF**, followed by „Apply“ and „OK“.

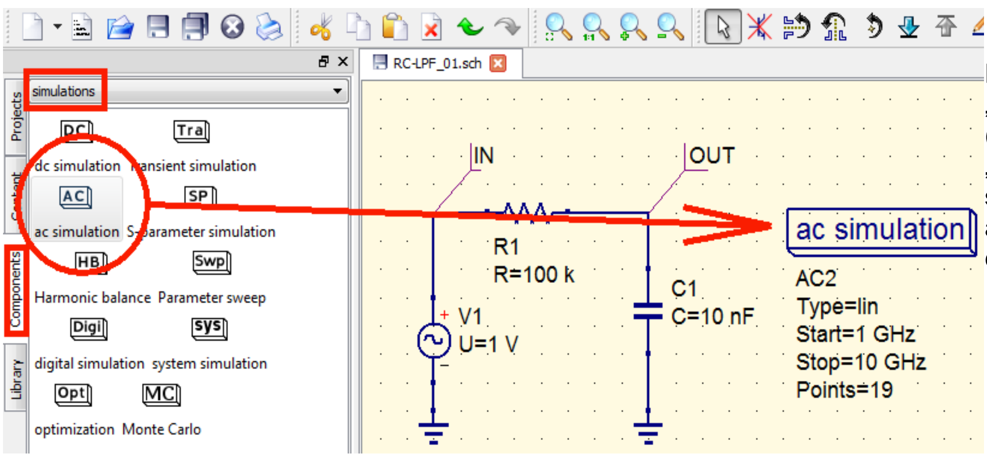



**Step 5:**
Check whether the amplitude of the voltage amplitude in the voltage source is already set to „**U=1V**“. Otherwise right click on the symbol to open the property menu and to modify the entry.

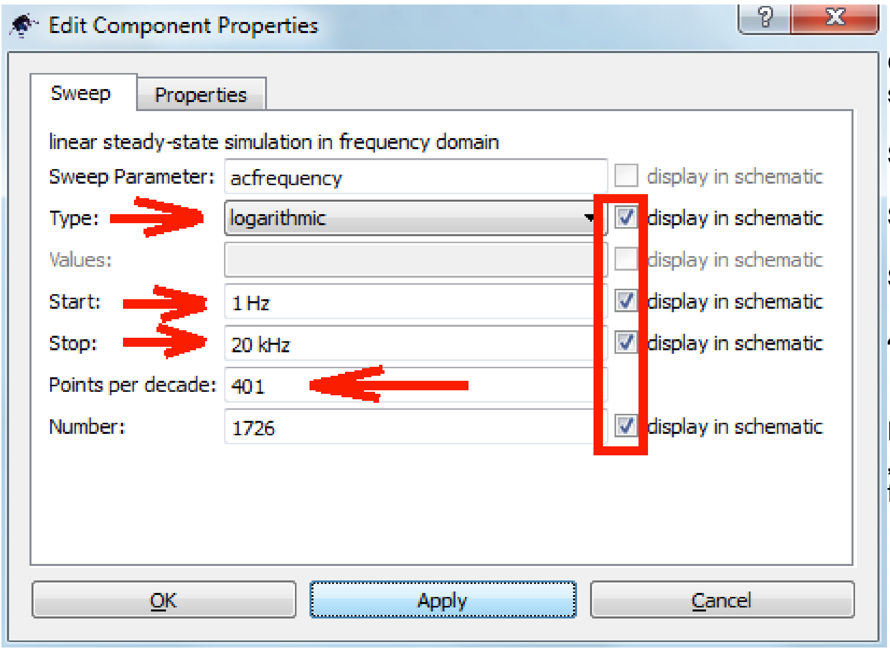
Place **two labels** in the circuit:

**„IN“ for the input**

**„OUT“ for the output.**




Pick „**ac simulation**“ (which can be found in „**components / simulations**“) and place it beside your circuit on the screen.




Open the property menu of „ac simulation“ and enter:

**Sweep type = logarithmic**

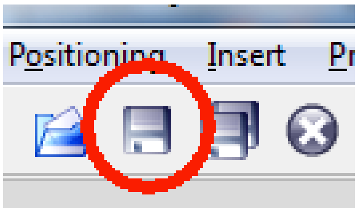
**Start = 1Hz**

**Stop = 20 kHz**

**401 points per decade**

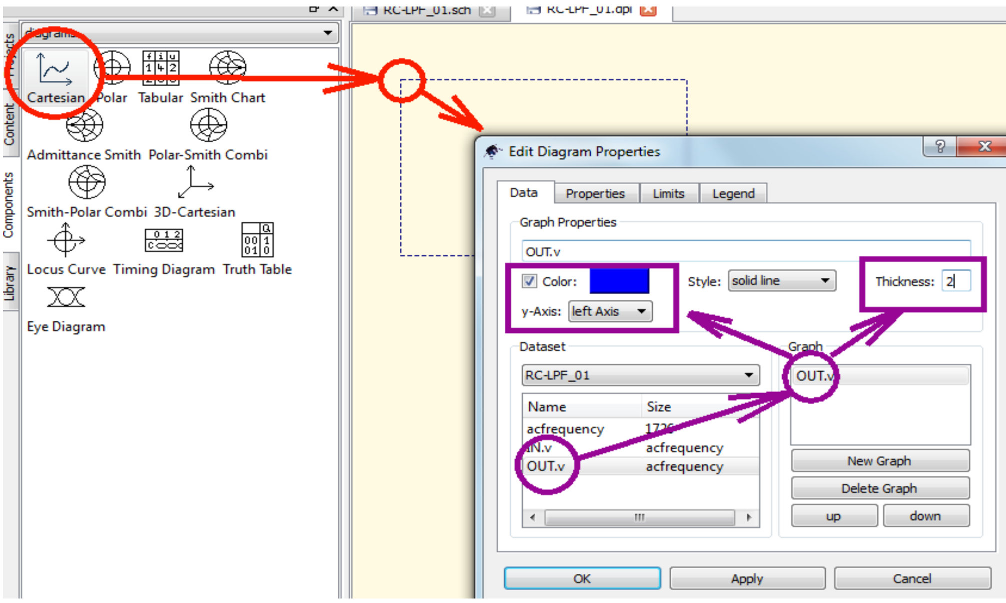
Do not forget to activate **„display in schematic“** for all items.

Now save your work



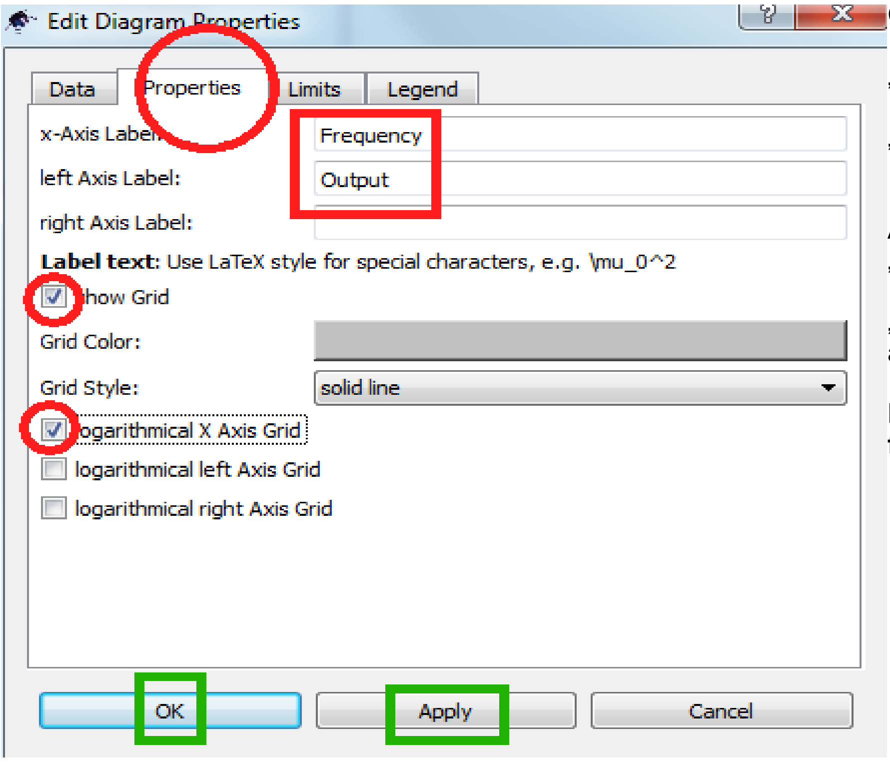

....and start the simulation

**Step 6:**




The screen switches automatically to the „Display mode“ when the simulation has finished.
Click on „**Cartesian**“ to open such a diagram. This is placed on the screen by a left mouse click and opens automatically its property menu.
Then click on „**OUT.v**“ in the data set to add it to the Graph list.

Now you can modify the color (template: blue) and the **thickness** of the displayed curve. Set **thickness to „2“**.




Change to „**Properties**" and enter:

„**Frequency**" as x-Axis Label and

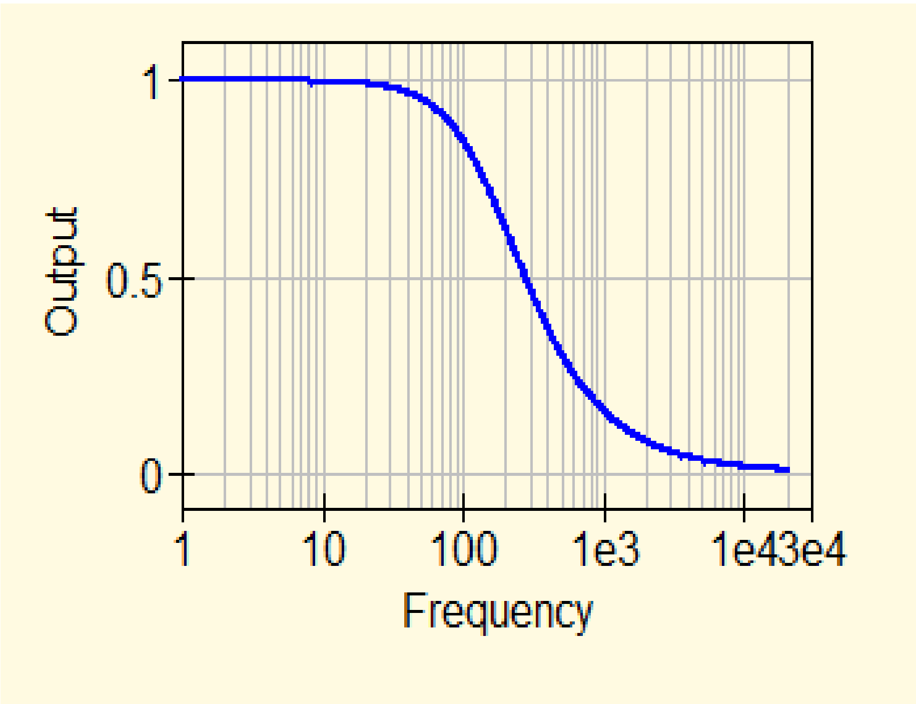
„**Output"** as left Axis Label.

Activate
„**show Grid**" and

„**logarithmic X Axis Grid**"
as grid style.

If all this is done, press „**Apply",
followed by OK.**

That is what we wanted to see.



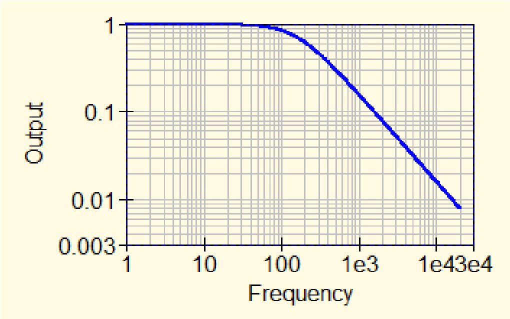

And because the input voltage has always an amplitude of

**U =1**

this is automatically the

**magnitude of the Transfer Function**

**Task 1:**
**Repeat the procedure with a new Cartesian diagram, but set both axes to „logarithmic".**
**Result:**

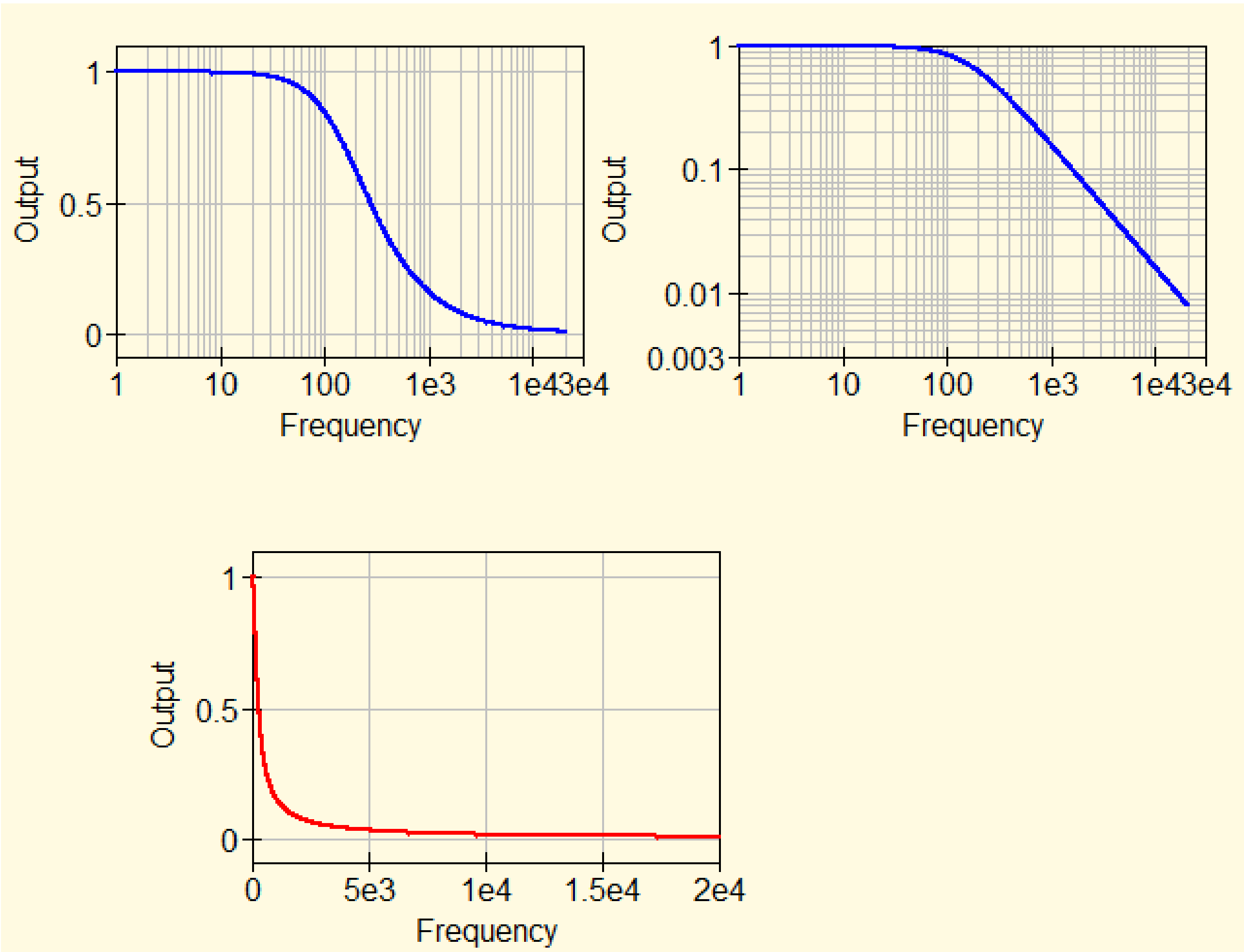



**Task 2:**
**Repeat the procedure with another new Cartesian diagram, but set:**

**a) both axes to „linear" (= delete the activation for „logarithmic")**

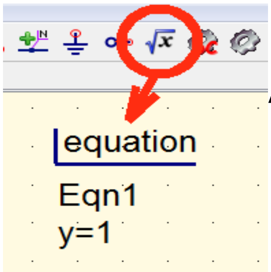
**b) Choose a red color for this curve** (= delete the color activation and afterwards switch it on again. Then click on the color palette and choose the new color = red).

**c) Arrange these three diagrams on the screen as follows:**



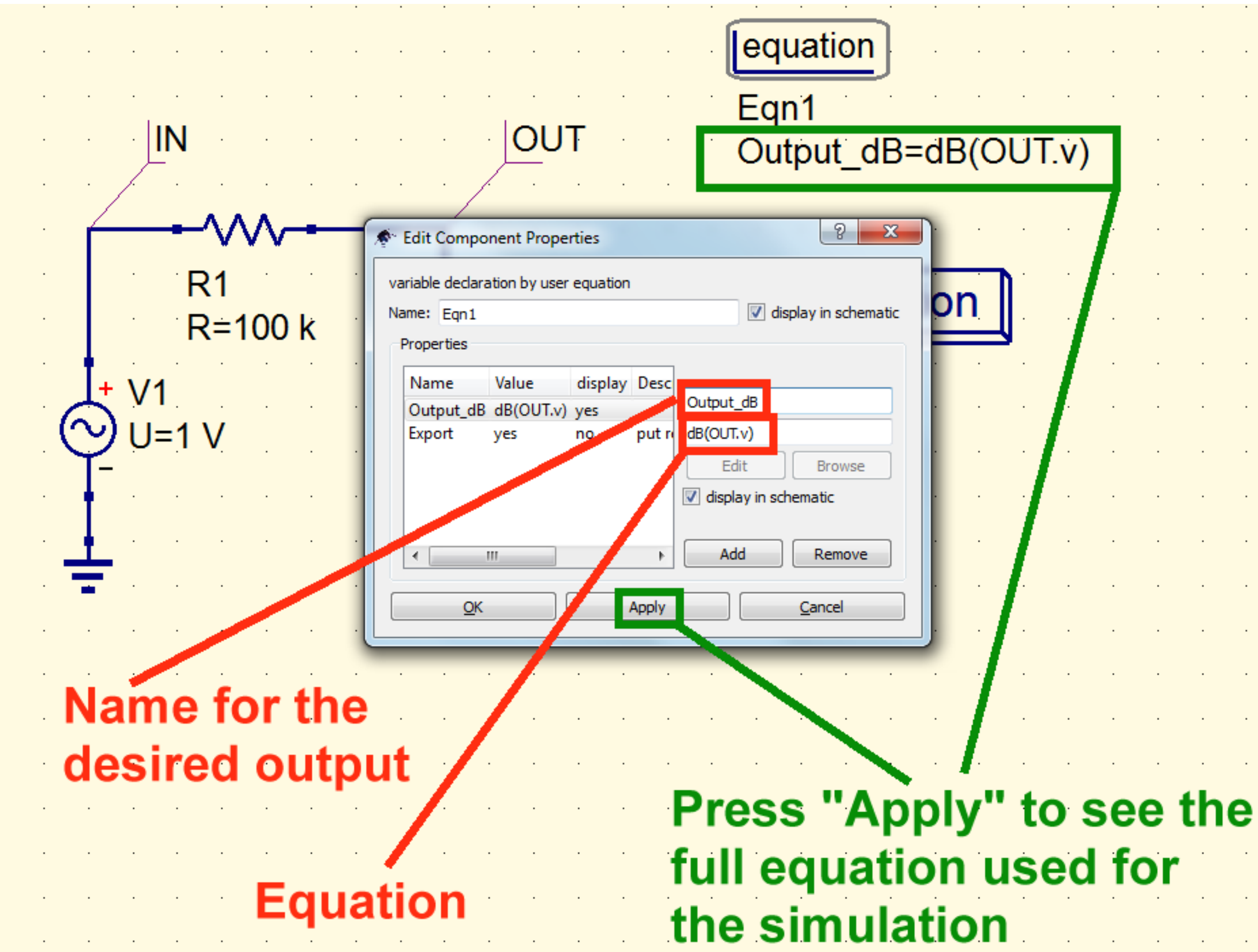
### 3.1.2. Result Presentation in „dB“

There is no button for this purpose – this job must be done by an **equation**!




At first press the equation button and place the equation on the screen.

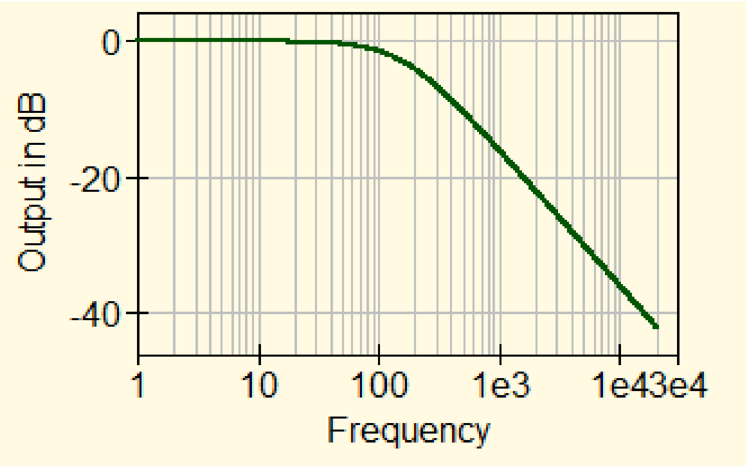
Then right click on "equation" to get the following property menu and to fill in the necessary entries.
Close by pressing "Apply" and OK.




## Now simulate again and use a new Cartesian diagram for the result!

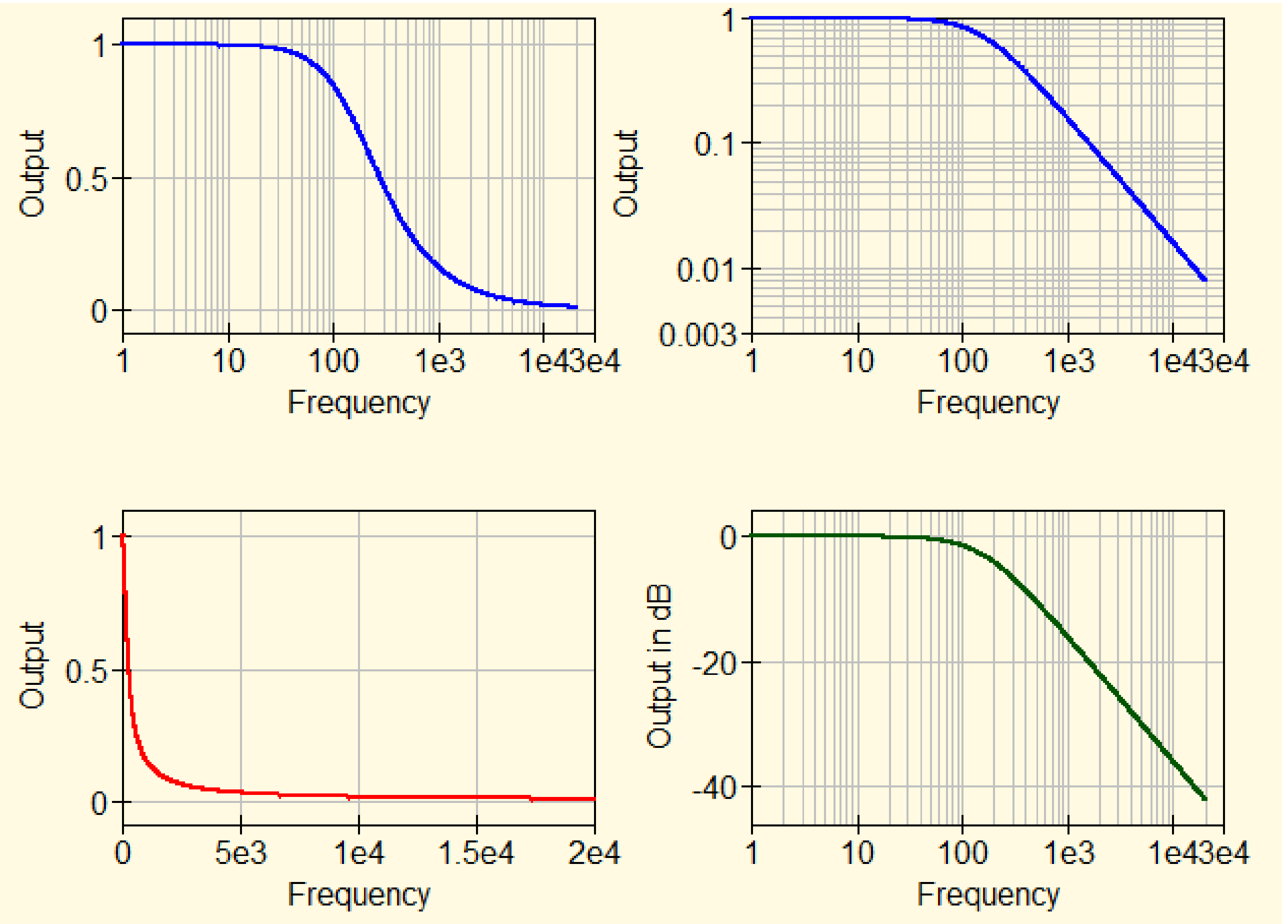
Open the property menu and enter:
a) in the folder "**Properties**" use **"Frequency" as x-Axis Label** and **"Output in dB" as y-axis label.**

b) Also choose **"logarithmic x-axis grid"**




This is the success.

**Task:**
**Show the four simulation results of the project together on this page.**



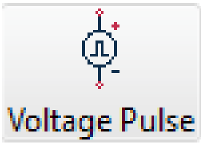
## 3.2. Simulations in the Time Domain

### 3.2.1. Impulse Response of the RC Low Pass Filter

Very important, because this is the way to get the Transfer Function by a simulation in the Time Domain!
A „Dirac“ Impulse is fed to the input of the system. But the pulse length must be extremely short (absolute border = near to ZERO) and the pulse amplitude = nearly infinite. But the pulse area is „1“.
This is impossible to realize, but the following idea does nearly the same job:

**Choose a pulse length which is „the systems time constant divided by 100 to 1000“. Increase the pulse amplitude bye same factor to hold the pulse area at its old area value of „1".**
**In this case you get the same results as when using a „true Dirac impulse“.**

So we look again at our well known Low Pass Filter.
Open a new schematic in our folder „LPF_01“, name it „LPF_TD“ and save it. Now you can either drawn the well known circuit once more or insert a copy of the circuit used in the last simulations (schematic „LPF_01“).




Replace the voltage source (used for the AC sweep) by a **Voltage Pulse** (which can be found in „**components / sources**“).

Open the Property Menu and enter:

**Minimum Voltage U1 = 0 V**

**Maximum Voltage U2 = 1 MEG**

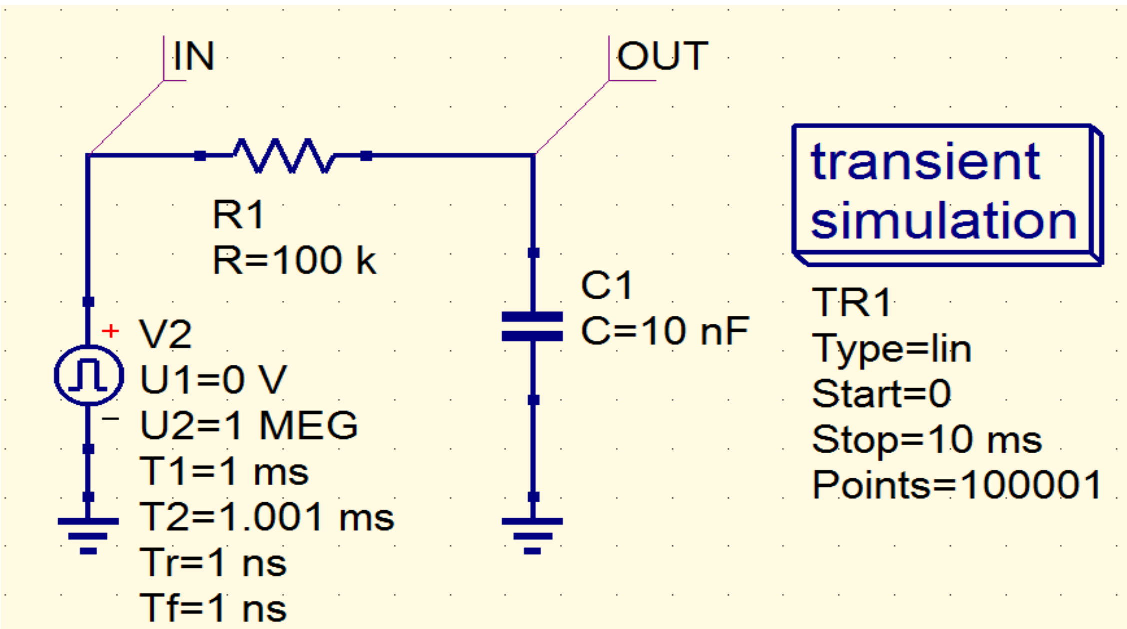
**Start point of the pulse = T1 = 1 ms**

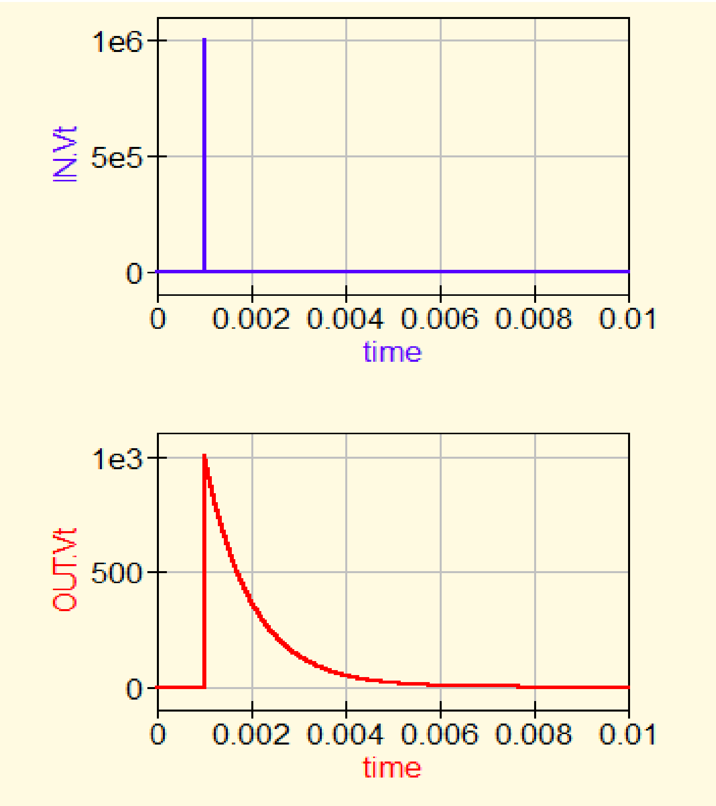
**End point of the pulse = T2 = 1.001 ms (...gives a pulse length of 1 Microsecond)**

**Rise time tr = fall time tf = 1 ns**

Make all entries visible.

At last we need a „**Transient simulation“** (from „**Components / Simulations**“) for **t = 10 ms and with 100 001 points (which gives a time step of 100 Nanoseconds)**

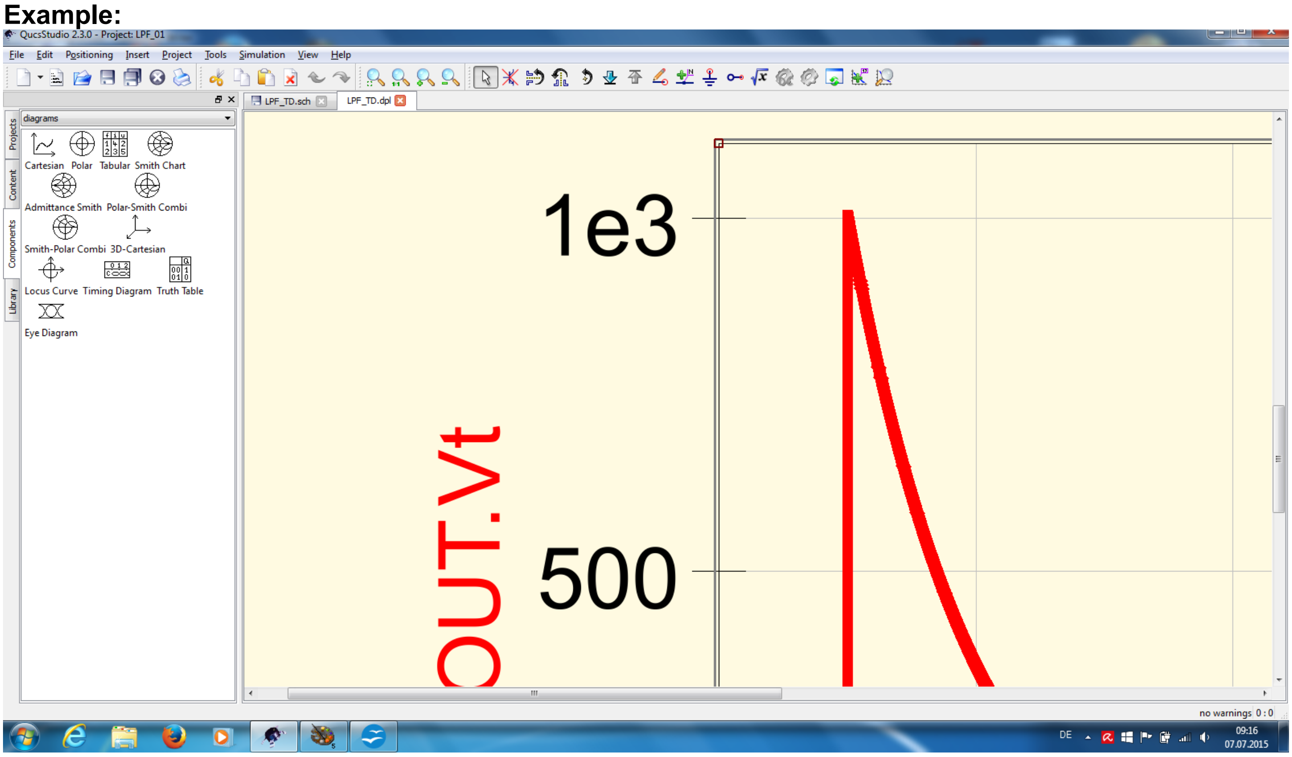





Now simulate and show „IN.Vt“ and „OUT.Vt“ in two different cartesian diagrams and different colours.

**Remark:**
If you are interested in details of a curve, simply use the **Zoom function.**

**Example:**



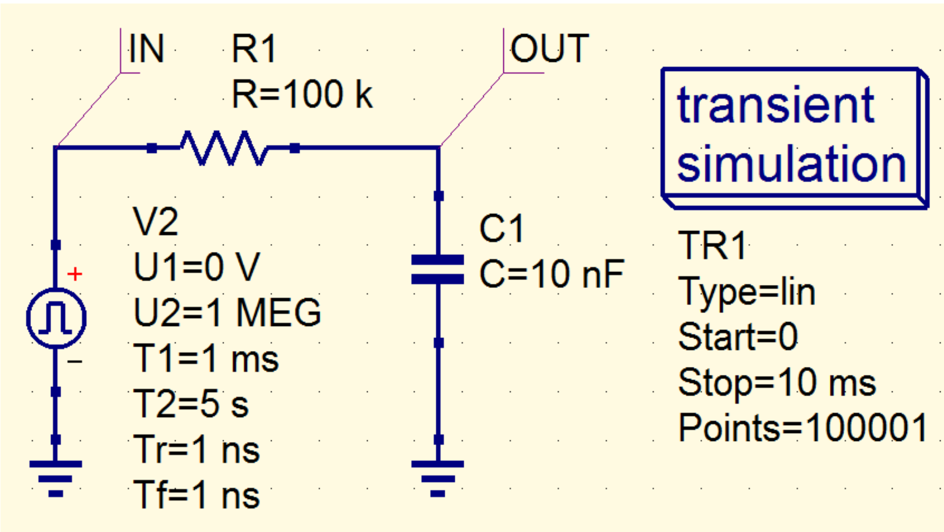

**For a more detailed analysis you can also walk on this way:**
**Edit the diagram properties, open the „limits“ menu and use the possibilities „manual“ for the x-axis and / or the left axis.**

### 3.2.2. Step Response of the LPF

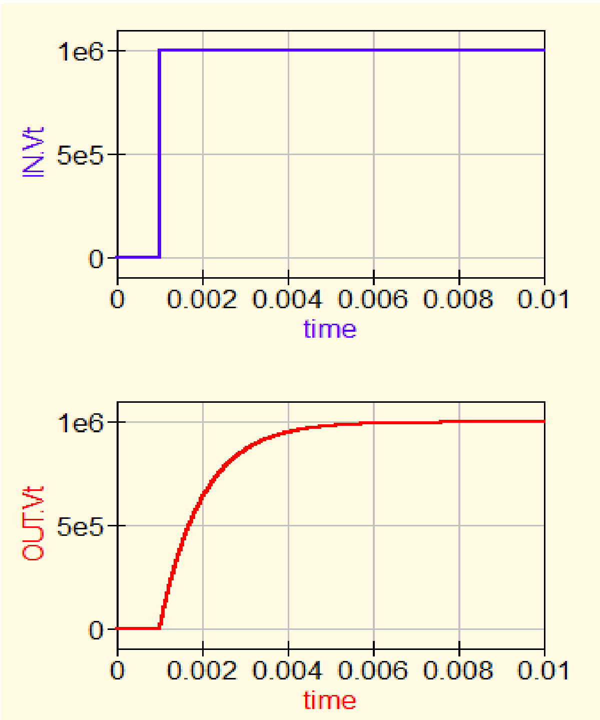
You have two possibilities. This is the easy way.

#### 3.2.2.1. Simulation using a Pulse Source

This is the same circuit as in the last chapter and it is only necessary to modify the properties of the pulse source for this purpose:




**Simply change T2 to 5 seconds to get a long pulse length .....**




**...and regard only the first 10 Milliseconds!**

## 3.2.2.2. Simulation using a „PWL“ (= piece wise linear) Signal

This is an option of every simulation software: a voltage curve is represented by a series of samples. Every sample must be defined by the famous couple „amplitude at a defined time“. Alll the samples are afterwards connected together by the program to the desired curve.

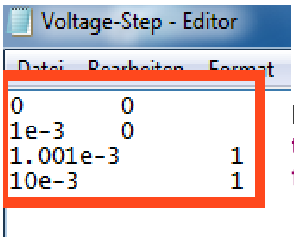
For the simulation in qucsstudio you need:

a) the „**File Based Voltage Source**“ from „**Components / Sources**“.

b) A „**Voltage-Step.csv-file**“ which contains the series of samples („csv“ means „comma separated values“). You have to write this file with a text editor and to save it in your project folder (here: LPF-01).

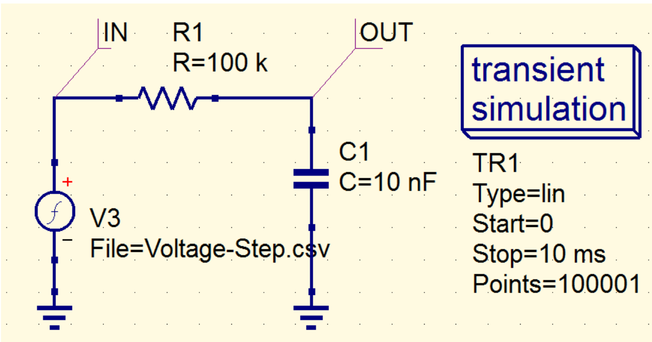
In our example we start at t = 0 with U = 0V. At t = 1ms the amplitude has still a value of 0 V, but already 1 Microsecond later  the amplitude has switched to 1 V. And this value of 1 V is constant to 10 ms.B

**Attention:**
**Every couple of values (= at first the time, then the amplitude) must be written in an own line. And this line must be terminated by a „carriage return“.**

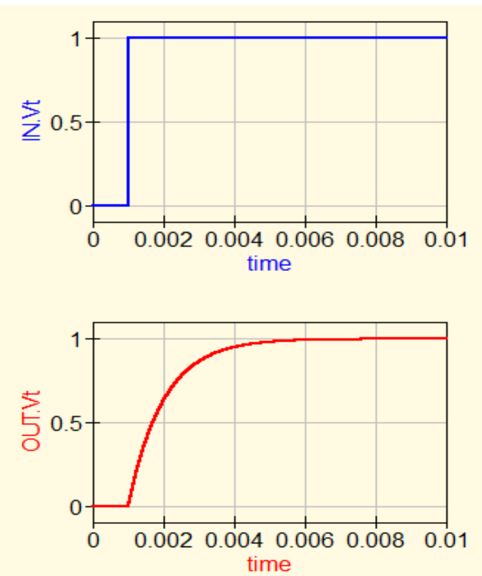



So the Voltage-Step.csv file must look like:

Please write this now with your text editor and **do not forget to switch your editor to „All Files / ANSI“ before saving as „Voltage_Step.csv“ in the LPF project folder**. Otherwise qucsstudio can't read it....




This is the circuit after **entering „Voltage_Step.csv“ in the property list** of the File based Voltage Source.




...and the simulation result is as expected.

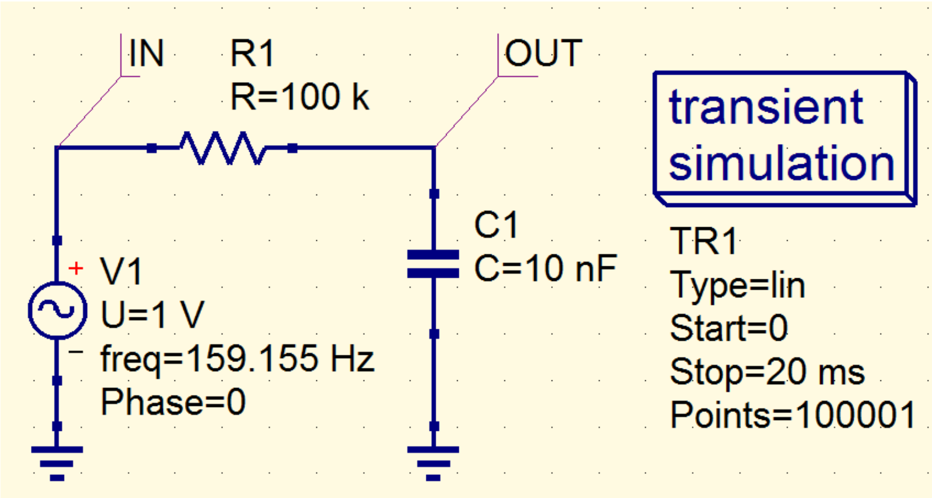
## 3.2.3. Periodic Signals at the Input

### 3.2.3.1. Sine Wave

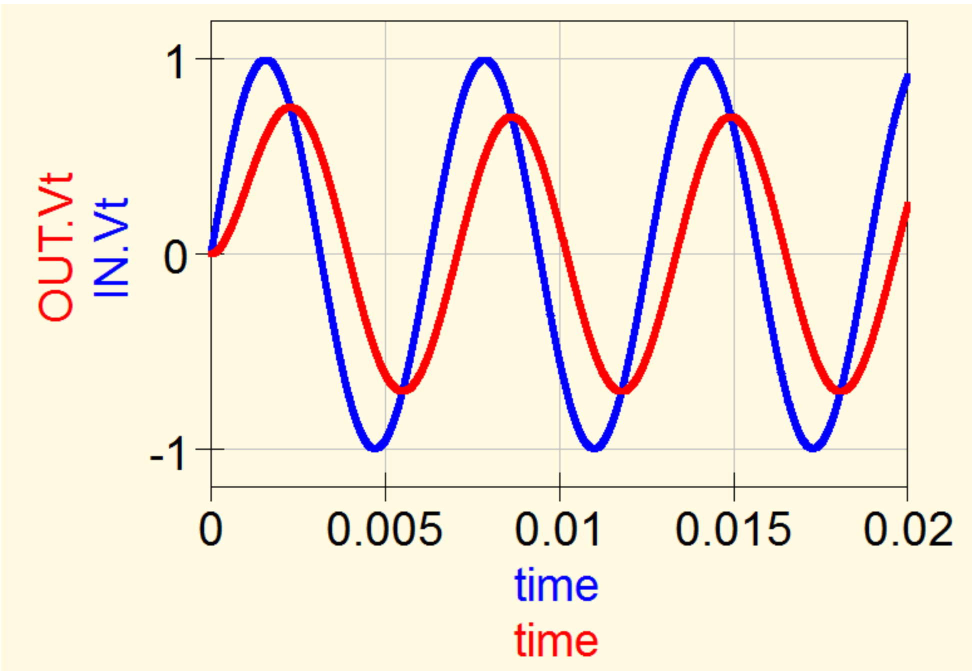
We examine the circuit exactly at its corner frequency which can be calculated as follows:

**fc = 1 / 2πRC = 1 / 2 * π * 100k * 10nF = 159.155 Hz**

The circuit is fed by an „ac Voltage Source“ (from „components / sources“). The Sine voltage amplitude is 1 V at this frequency of 159.155 Hz. Do not forget to make these entries visible.

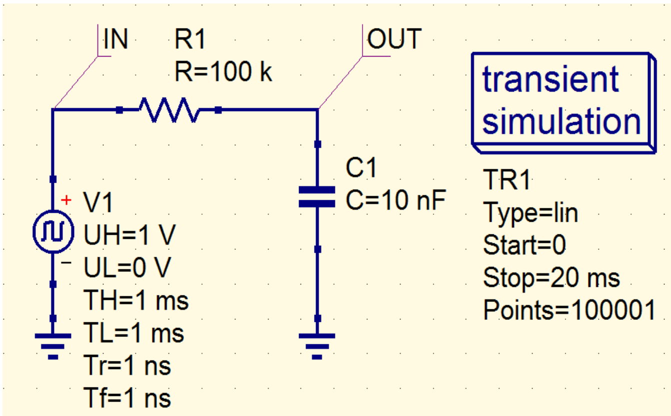



Please use the circuit of the last example, replace the data based voltage source bay the ac voltage source and set **the simulation time to 20 ms (with 100 001 points to simulate).**




The output shows an attenuation of -3dB and a phase shift of -45 degrees at the corner frequency.

### 3.2.3.2. Pulse Signal



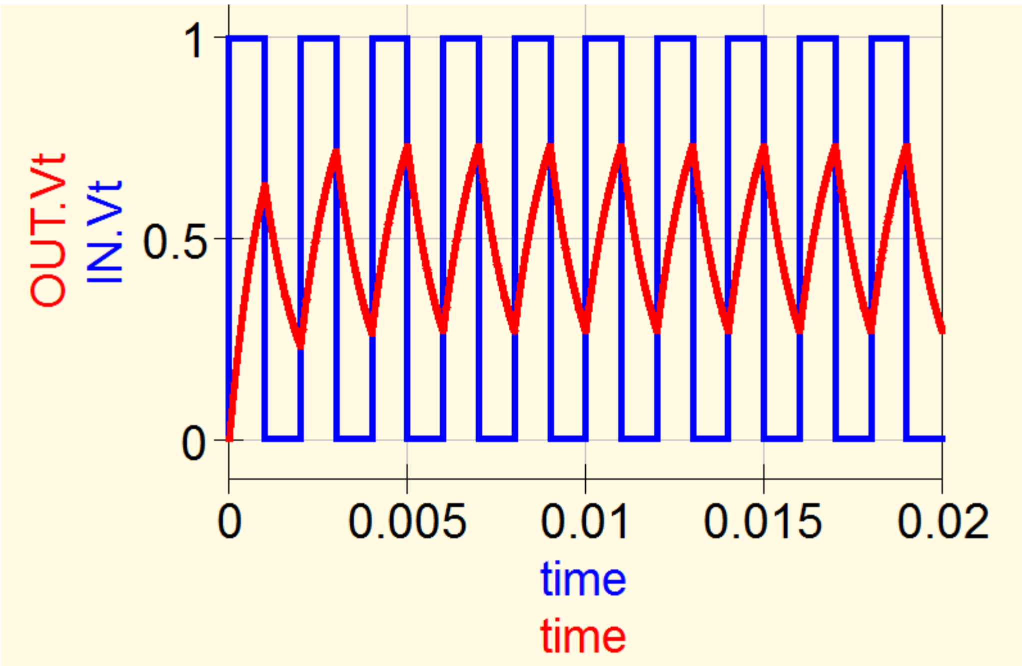

Use a periodic and symmetric pulse voltage with a frequency of f = 1 kHz, a minimum value of 0 V and a maximum value 0f +1 V.

TH and TL are 1 millisecond.

Simulate for 0....20 ms with 100001 points.

The „**Rectangular Voltage**“ source can be found under „components / sources“.

Please make all properties of the voltage visible (...you know: right click on the symbol, then „edit properties“)



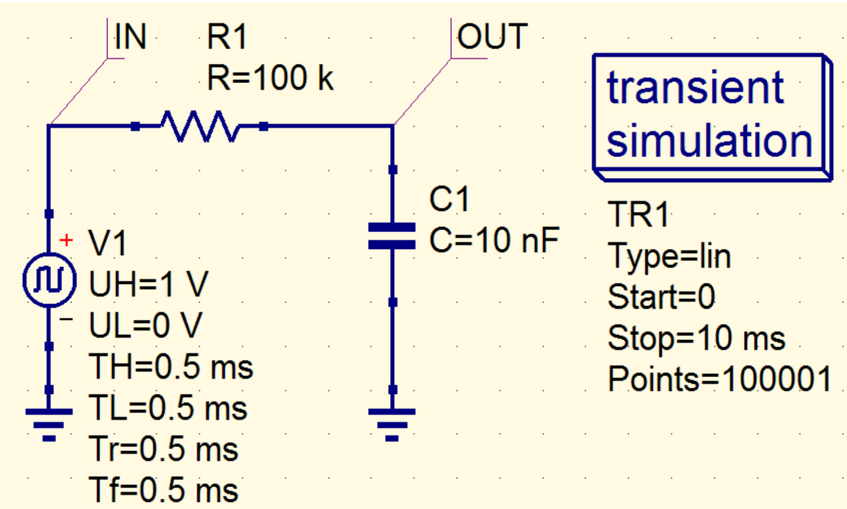

A fine result....

### 3.2.3.3. Triangle Voltage

A very simple task, because we use again the complete circuit of the last example including the rectangular voltage with f = 1 kHz.

a) With a long rise and a long fall time we get the desired triangle, but

b) the times TH (= High level) and TL (= Low level) must be set to Zero.

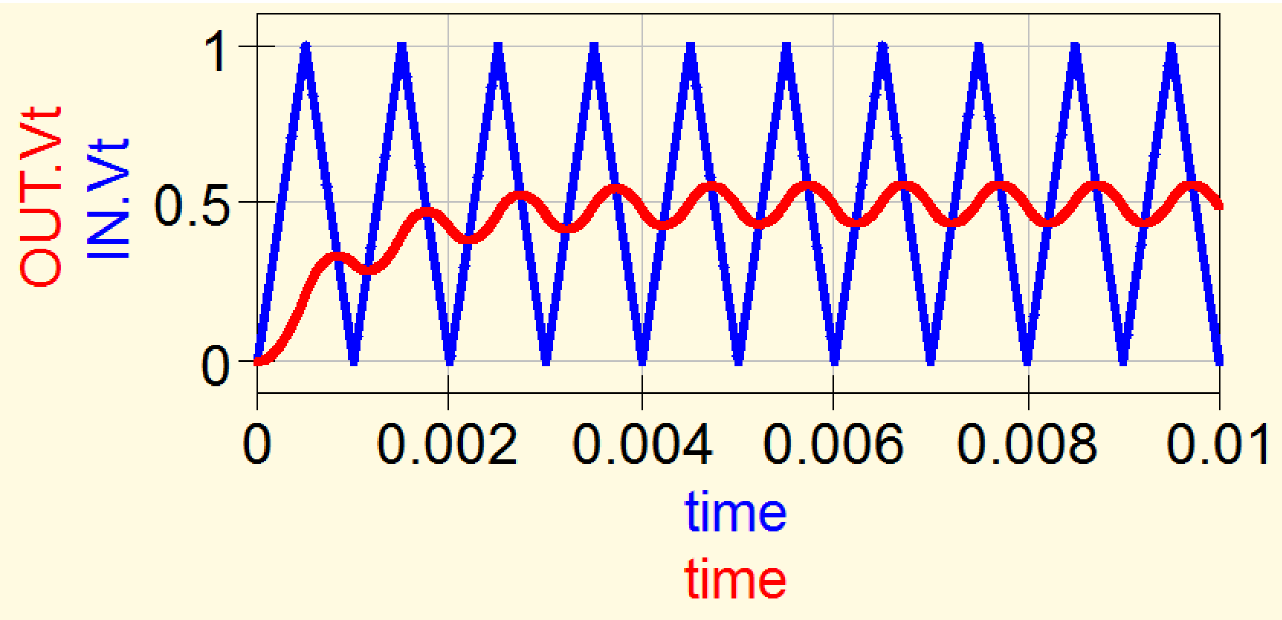



If you know that qucsstudio always defines the rise time as a part of the „High Level Time" and the fall time als a part of the „Low Level Time", then the problem is solved in a minimum of time:

**make Tr = TH = Tf = TL = 0.5ms and the job is done!**

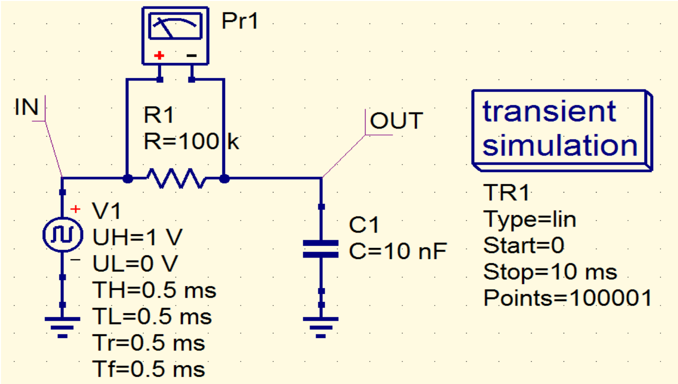
Now simulate for 0....10 ms.

Result:

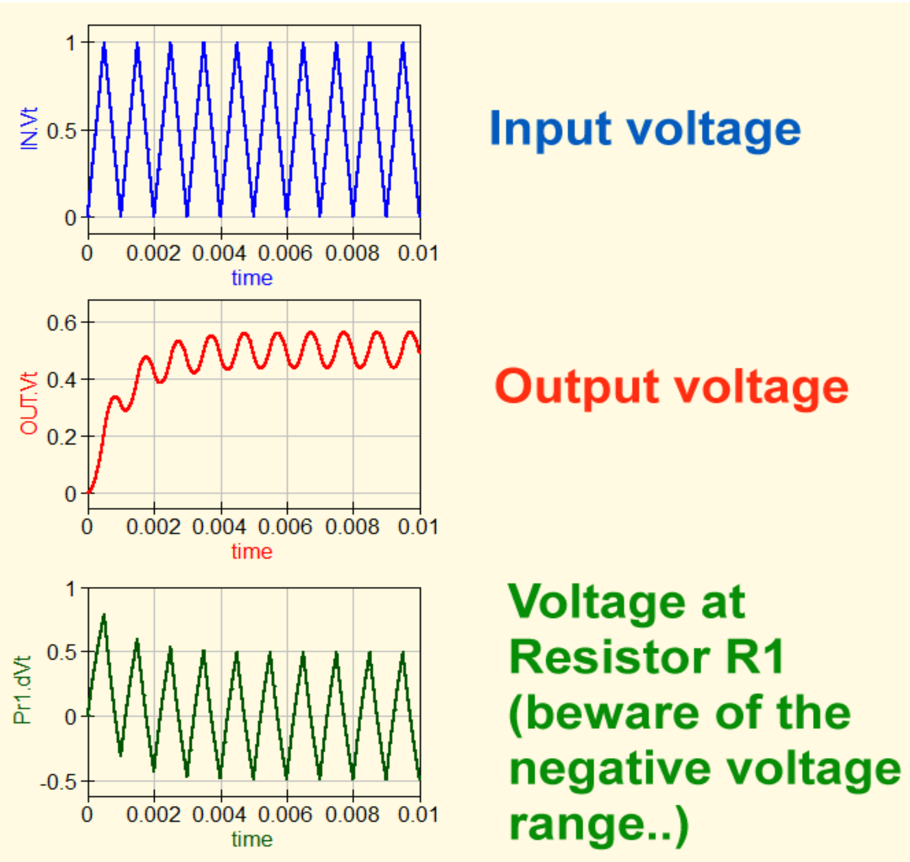



(To get this „CinemaScope" presentation:
mark the diagram by a left mouse click. Then pull with the mouse on the upper or lower right corner).

### 3.2.3.4. Differential Measurements




If you want to see the voltage at the resistor R1, use a „Voltage Probe“ (= components / devices and connect it in parallel to R1.

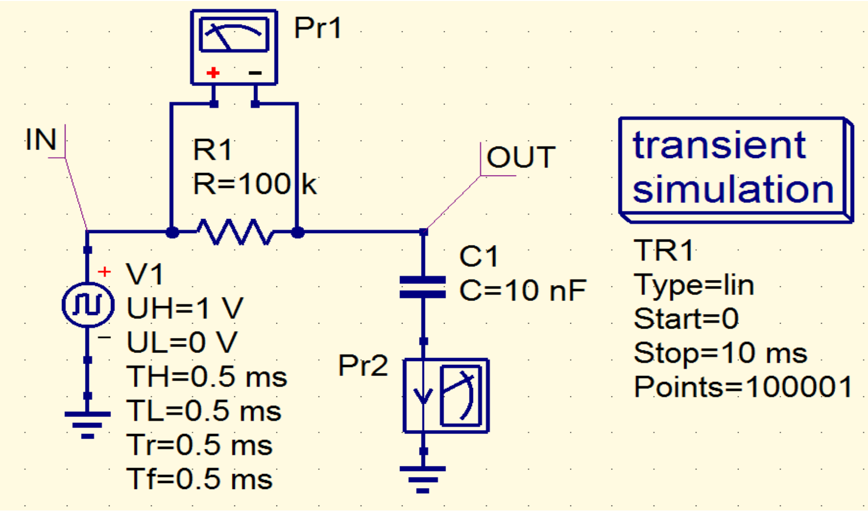



Use 3 different diagrams to present the input voltage IN.Vt, the output voltage OUT.Vt and the voltage across the resistor R1 (Pr1.Vt).

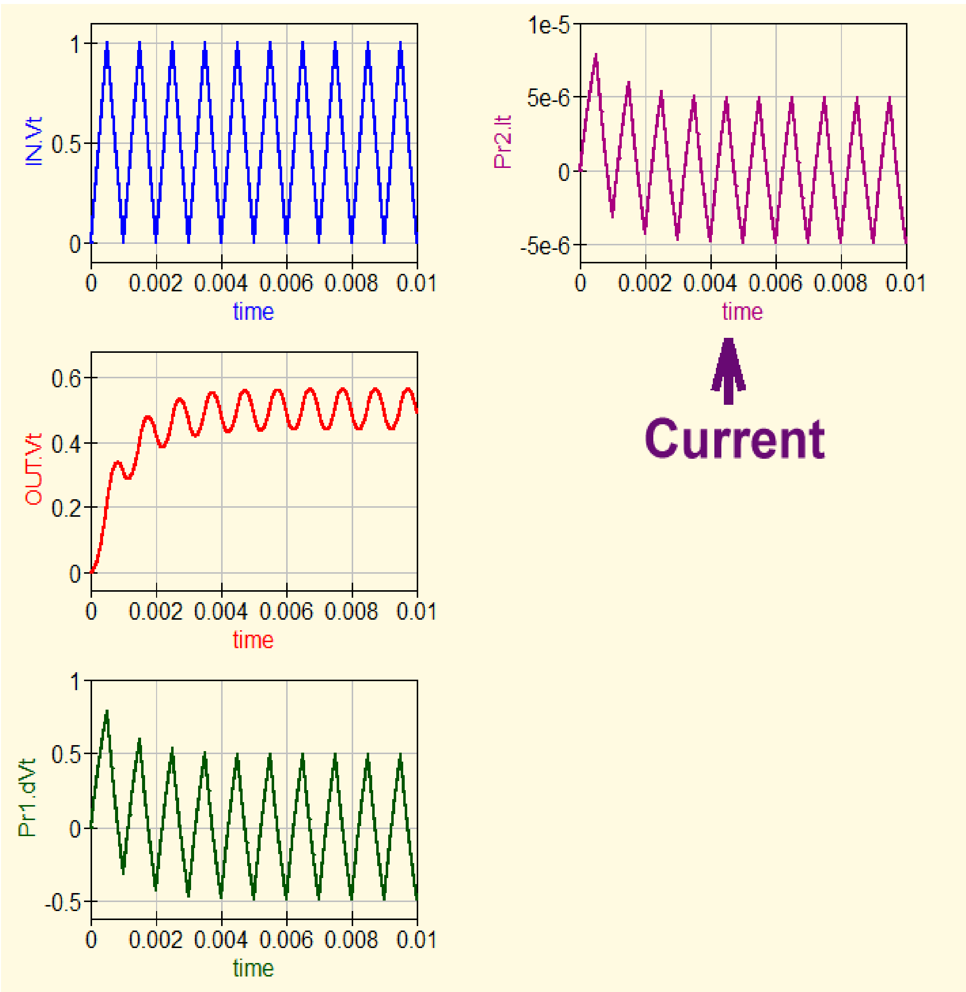
Very fine, isn't it?

But pay attention: the voltage at R1 shows also negative values....L

### 3.2.3.5. Current Measurements




You find a „Current Probe“ in „components / devices“ for this purpose. So insert this measuring set Pr2 to your circuit and simulate the last example using a copy.




Use 3 different diagrams to present

the input voltage IN.Vt,

the output voltage OUT.Vt,

the voltage across the resistor R1 (Pr1.Vt)

and the current in the circuit-

Attention:
The current and the voltage across R1 show also negative values...

## 3.3. FFT (= Fast Fourier Transformation) = the Bridge between the Time Domain and the Frequency Domain

### 3.3.1. Fundamentals

Very often we are not only interested in the properties of a signal given as a „curve in the Time Domain". It is also important to know the „spectral content" if a voltage curve differs from the „ideal sine wave". In this case we find Harmonics or we see a mixture of different frequencies or we get sum and difference of input frequencies.

**The FFT calculates this spectral content using a signal given in the Time Domain!**

***But for a FFT success you have to pay a lot of attention to the necessary preparations.***
==============================================================================

**The simulation time determines the start frequency of the displayed frequency spectrum, the „width" of every calculated frequency lines and the minimum frequency step on the x-axis.**

## Minimum frequency step = 1 / Simulation Time

This **ratio of the simulation time and the period of the regarded signal should be integer.** Violation of this law can produce noise and / or non existing additional spectral lines.(= Leakage).
=============================================================================
The „**Maximum Time Step**" in the Time Domain needs also attention:

a) Use as **much samples as possible** to get a „soft, smoothed curve".

b) The „**sample frequency = 1 / time step" muss not be smaller than twice the highest existing frequency in the input signal to satisfy the „Shannon Law" and to avoid „Aliasing Effects"** (which can cause a totally useless result).

c) You need always a **minimum lot of samples for a convenient FFT**. This „package" determines (together with the used sample frequency) the **highest shown frequency value on the x-axis in the output spectrum AND the dynamic range of the vertical axis.**

============================================================================

## The number of used samples must always be a multiple of „2".

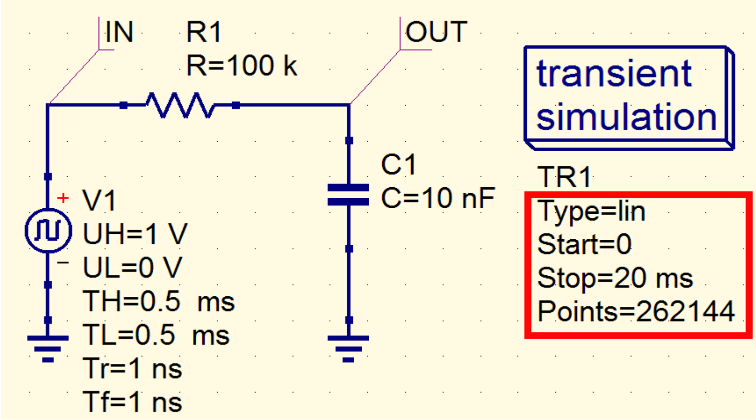
Otherwise you do not work with the **Fast** Fourier Transform!

But do never forget:
Use more samples and a smaller time step (if possible) to get higher amplitude and frequency resolution. But you pay in this case with a longer calculation time and an increased result file size....
============================================================================

## 3.3.2. An Example

### 3.3.2.1: Preparation and FFT of the Rectangular Input Signal

Let us have a look at the last example = the RC LPF fed by a rectangular voltage (= pulse signal). The pulse frequency is 1 kHz, UH = +1 V, UL = 0 V.



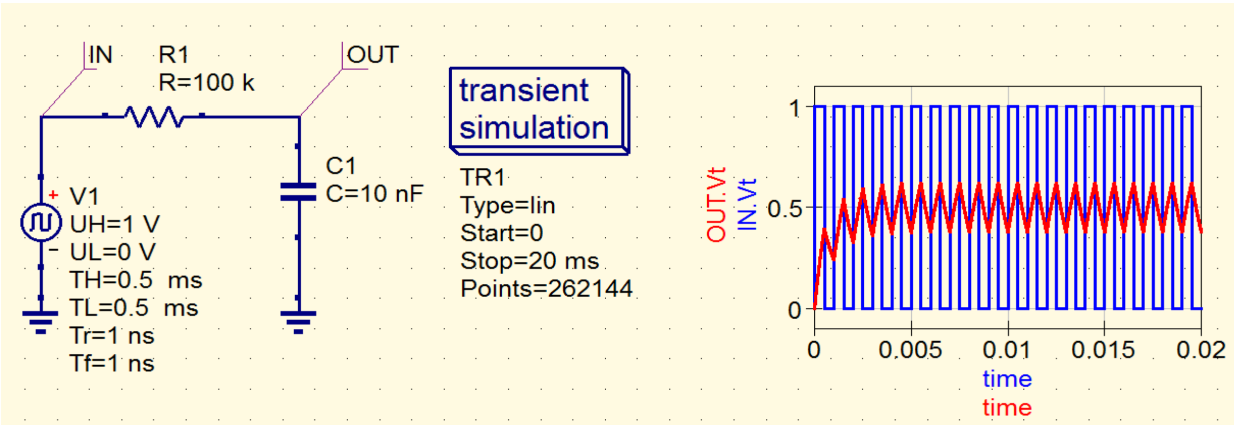

**The most important decision is the number of samples, which must be a multiple of 2.**

To get a very high resolution we use

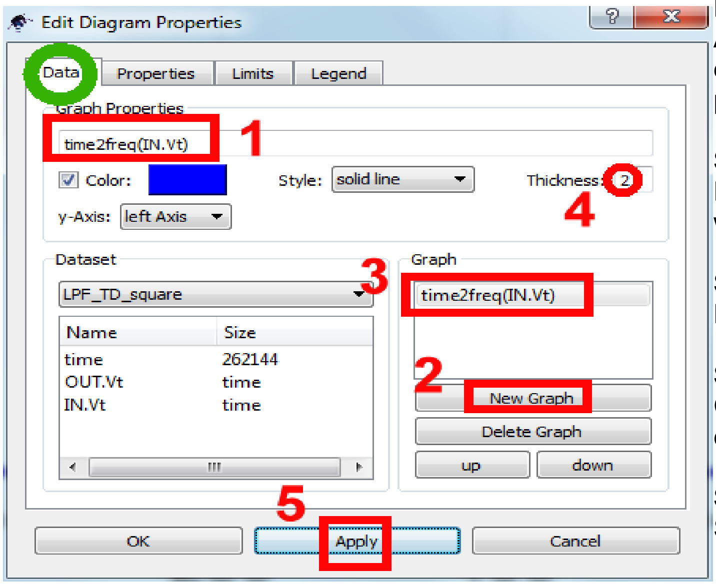
**262 144 time steps (= points)**

and a **simulation time of 20 Milliseconds.**

The stop at 20 ms sets the minimum frequency step in the output spectrum to **1 / 20 ms = 50 Hz**




After the simulation present IN and Out in the same Cartesian diagram. (If you want to see circuit diagram and simulation result on the same screen: mark the diagram, copy and paste it in the schematic beside the circuit).




Now back to the result screen:
At first we need an additional new Cartesian diagram. Then follow these steps for a correct preparation of the entries in the „data“ card:

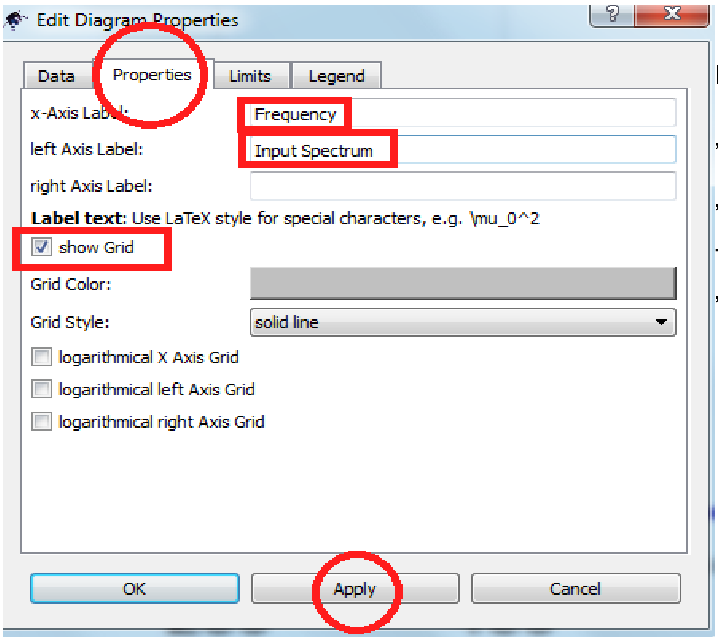
**Step 1:**
Enter „**time2freq(IN.Vt)**“ in the Graph Properties window for a FFT

**Step 2:**
Press „**New Graph**“

**Step 3:**
Check whether this command was accepted correctly.

**Step 4:**
Set line thickness to „2“

**Step 5:**
Press „Apply“

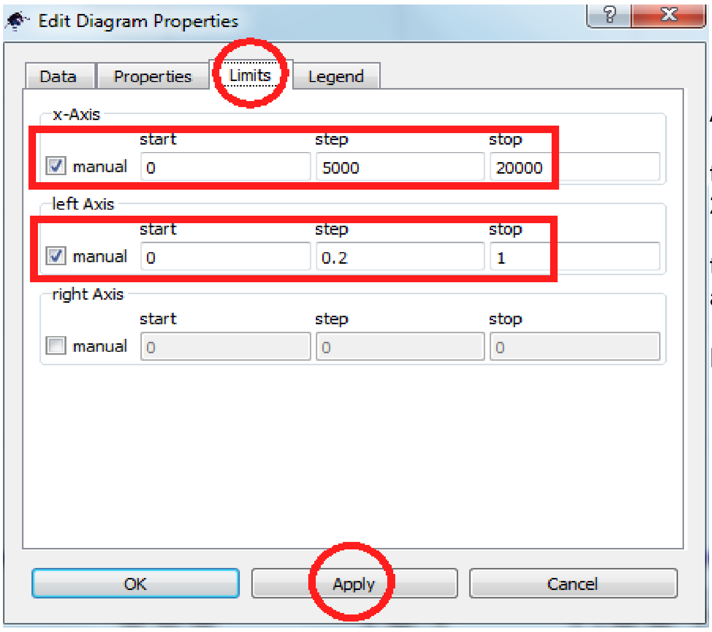



Now change to the „Properties“ card and enter:

„**Frequency**“ as x-axis label and

„**Input Spectrum**“ as left axis label.

Test that „**show grid**“ is marked, then press „**Apply**“

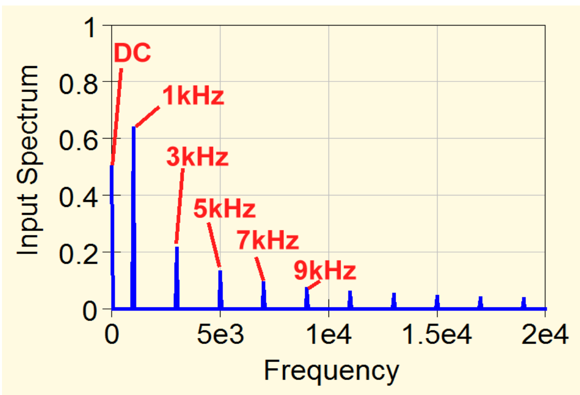



At last go to the „Limits“ card and set:

the-axis to show frequencies between 0 and 20000 Hz with a step width of 5000 Hz

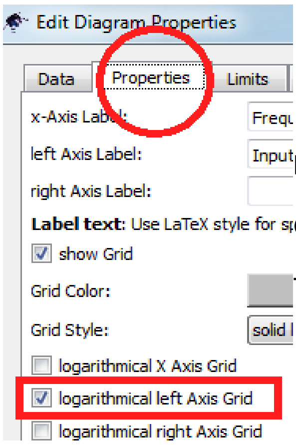
the left axis to show values between 0 and 1 with a step of 0.2

Press „Apply“




This diagram is correct for a strictly symmetrical rectangular signal.

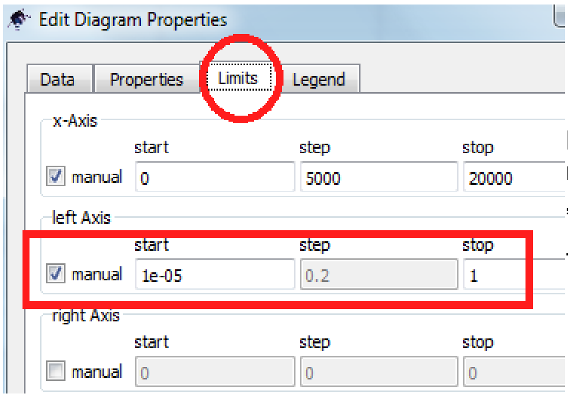
Very nice....




Now we repeat the result presentation with a **left logarithmic axis**:

Go to the „**Properties**" card and mark „**logarithmic left Axis Grid**".

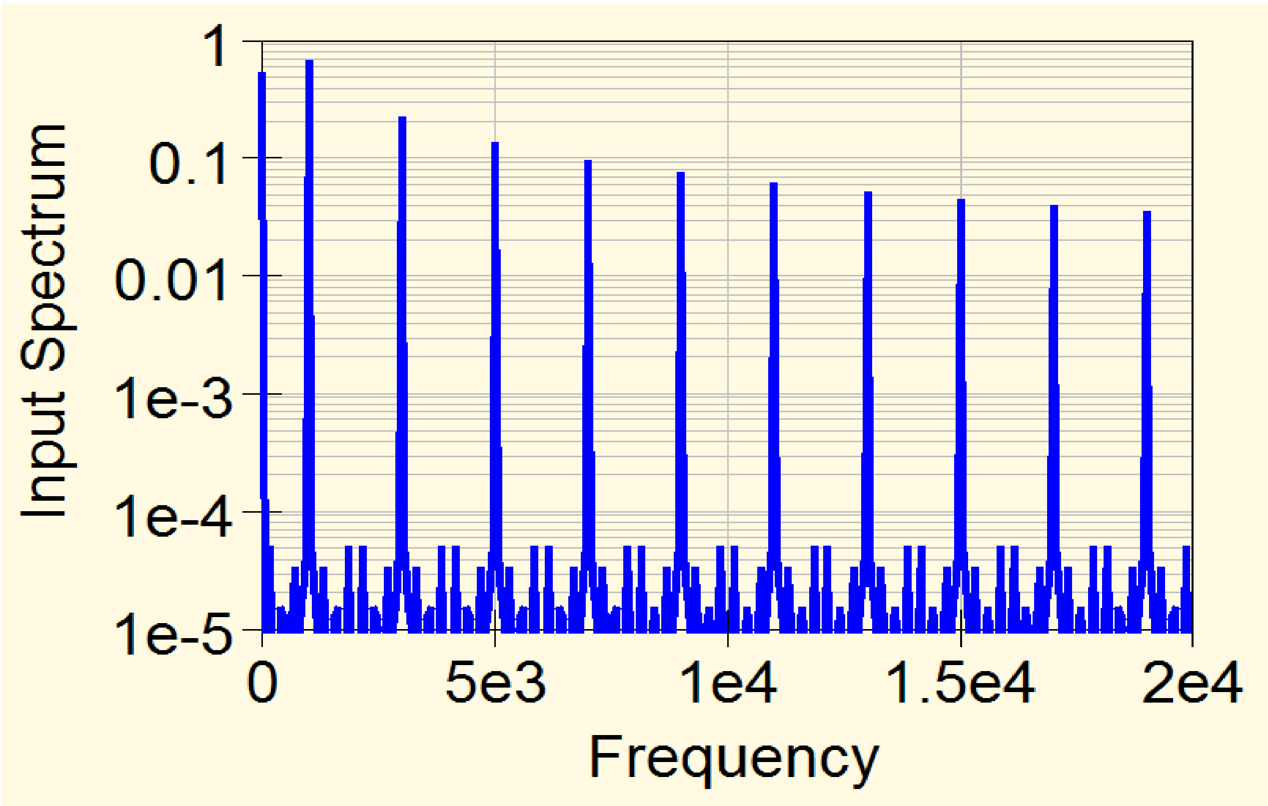
Then press „Apply"




But do not forget to enter the **minimum value (1e-5)** and the **maximum value (= 1)** of the left logarithmic axis on the **„Limits" card.**

Then press „Apply".

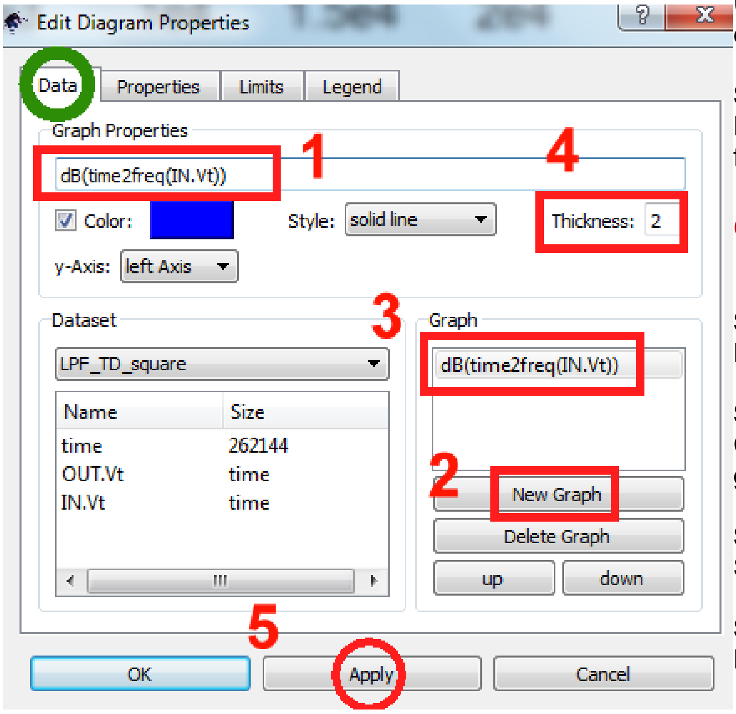
A nice result:



## 3.3.2.2. „dB“ Result Presentation

**Task:**
**Replace the logarithmic scale of the left axis by a „dB“ scale!**

Solution:




Use a new Cartesian diagram, go to the „Data“ card and enter the (well known) settings:

**Step 1:**
Enter under „Graph Properties“the following line for the conversion to dB:

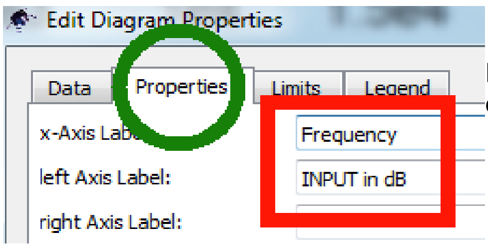
**dB(time2freq(IN.Vt))**

**Step 2:**
Press „New Graph“

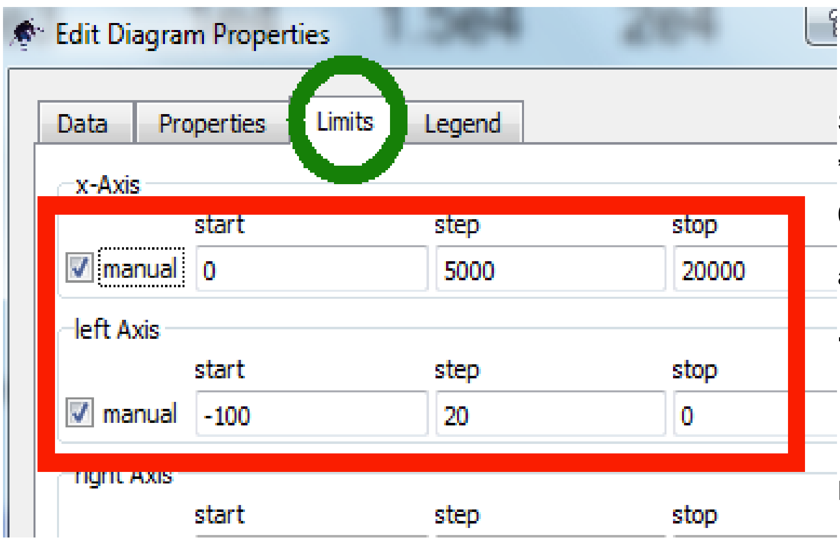
**Step 3:**
Check that the formula was accepted for the graph

**Step 4:**
Set Line thickness to „2“

**Step 5:**
Press „Apply“




Enter „Frequency“ as x-axis label and „INPUT in dB“ as laft axis label on the „Properties“ card.



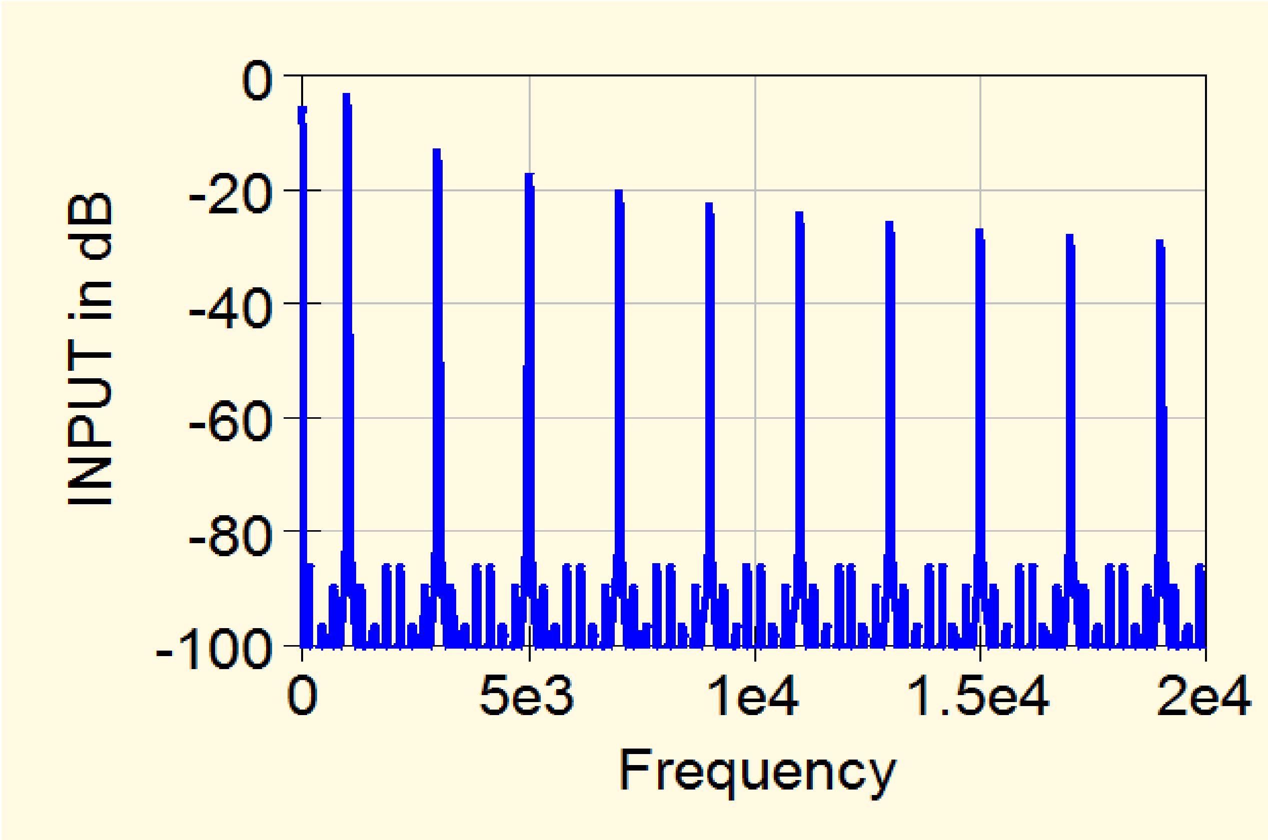

Set the **frequency range** on the „**Limits**“ card to

**0...20000 (with step = 5000)**

and the **dB range of the left axis to**

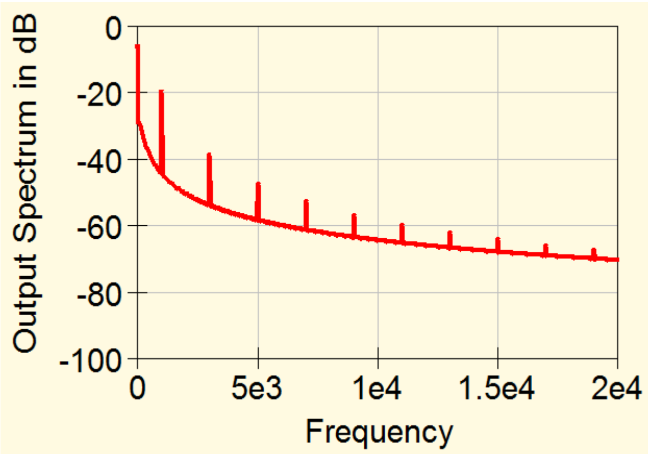
**-100.....0 with a step of 20 dB**

Press „Apply“


0
-20
-40
-60
-80
-100
INPUT in dB
0
5e3
1e4
1.5e4
2e4
Frequency

## 3.3.2.3. FFT of the Output Signal

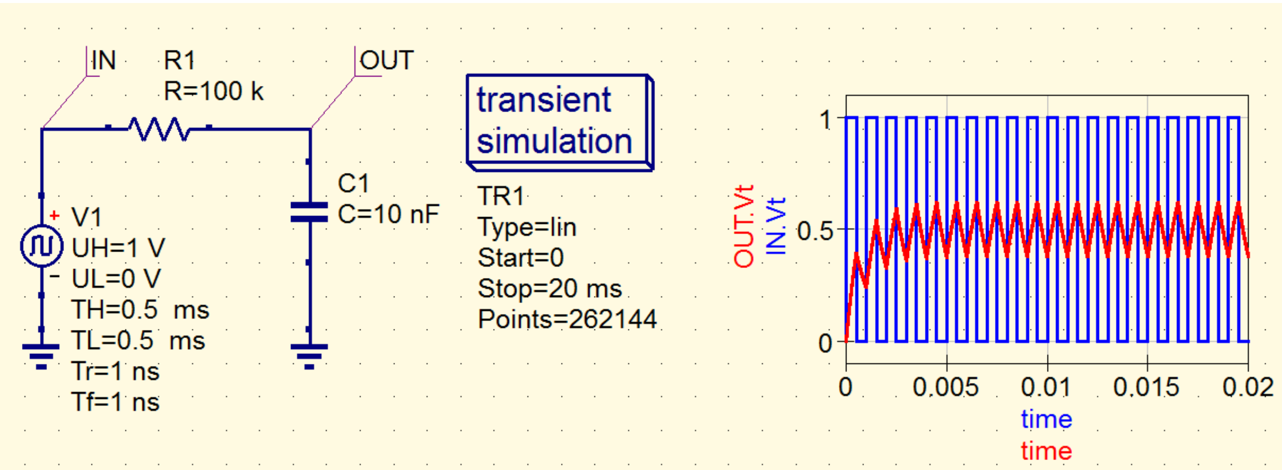
The task „**show the spectrum of the Output signal in a diagram using a linear frequency axis from 0 to 20000 Hz and a vertical dB scale from -100 to 0 dB**" should now not cause any difficulties or problems. Pick a new Cartesian diagram, enter **„dB(time2freq(OUT.Vt))"** for the Graph calculation and set the conditions for the two axis as proposed.




In comparison to the Input spectrum we find a reduced dynamic range with a big noise floor -- decreasing with increasing frequency.

Let use have a look at the reasons for this phenomena.

## 3.3.2.4. We need a „Window" function for a better result



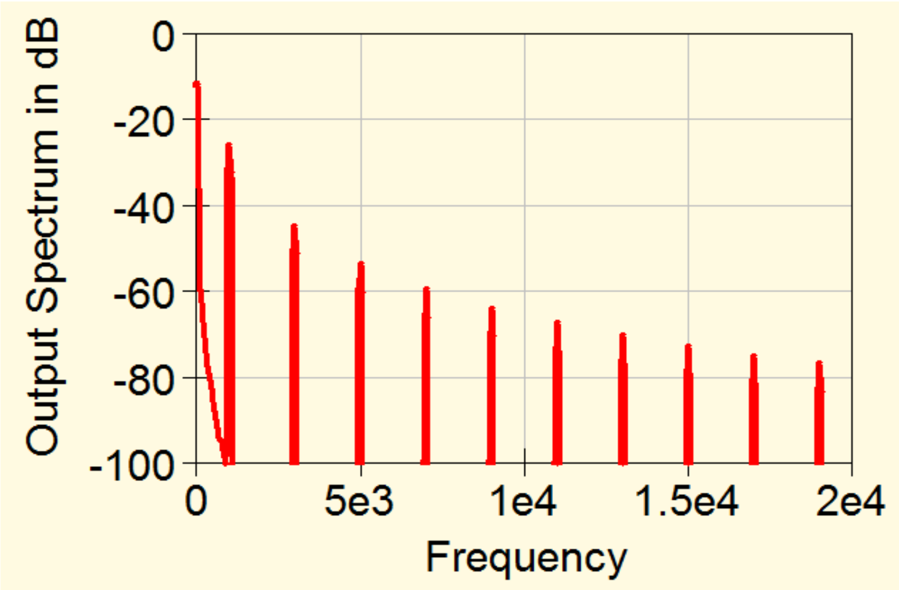

Look at the used circuit and the diagram which shows the Input and the Output signal. In the time range form 0....5 ms the capacitor is charged and this is a „non repetitive event". Such an event produces a spectrum which fills the complete frequency range with energy and this is added to the line spectrum of the periodic Output signal.

Standard help is to multiply the signal to be transformed by a Window Function which reduces the influence of start and stop (which are both „abrupt transitions" and therefore produce such problems). Such a Window function starts and stops usually with the value „0" and reduces the influence of the transitions.

Qucsstudio offers 5 functions ….and I use either number 3 (= Hann) or number 5 (Blackmann) which give good results (..but prefer Blackmann...). The application is easy: simply write for a Hann Windowing

**dB(time2freq(OUT.Vt,[3])) or dB(time2freq(OUT.Vt,3))**

into the „Graph Properties". But **NEVER** forget the comma in front of number „3"!



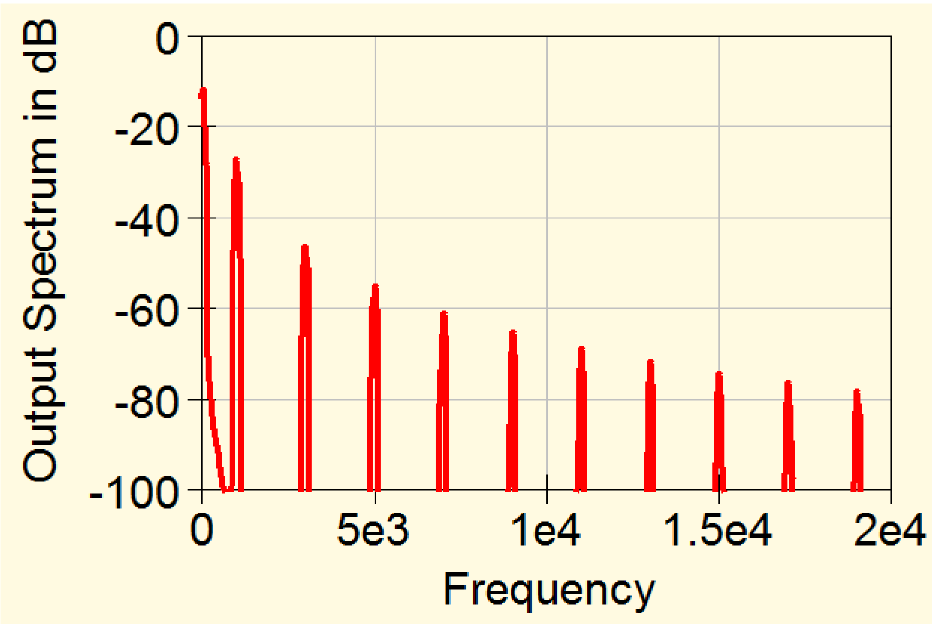

The result is improved and now we have a dynamic range down to -100 dB!

**Task:**
Repeat the FFT but use a Blackmann Window (index = 5).

**Solution:**

The formula for the Graph Properties is **dB(time2freq(OUT.Vt,5))**

and gives this result:




The width of every line has increased, because the Blackmann Window Function has more samples (at the start and at the end) strongly reduced in their amplitudes.

But this **equates to a „reduced simulation time"** and so the

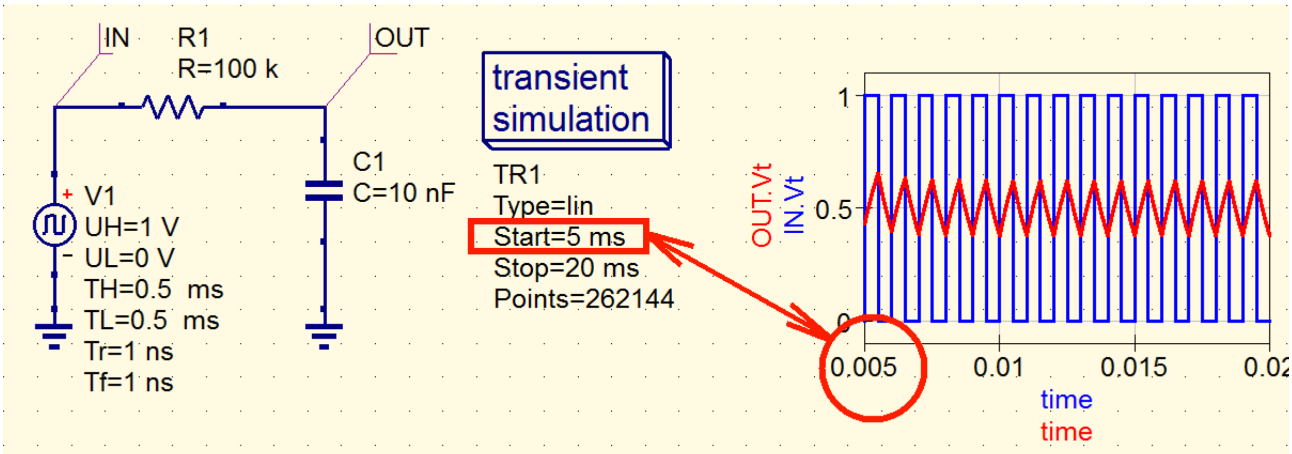
**minimum frequency step = minimum line width = 1 / simulation time**

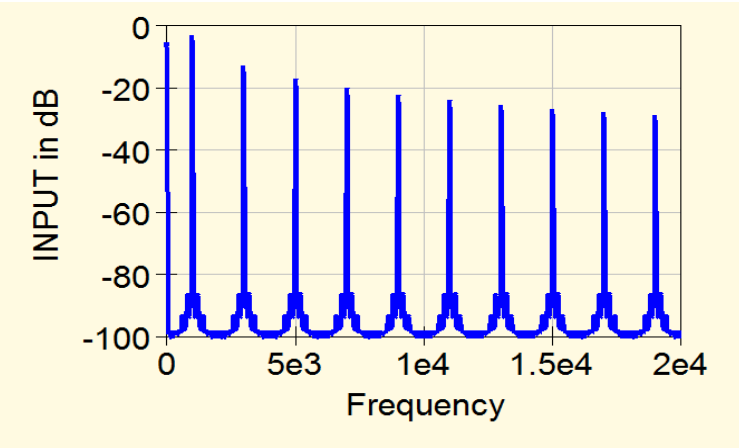
must increase....

**The real advantage of Windowing is the increased dynamic range and the reduced influence of the starting process (= time for loading the capacitor after switching on).**

### 3.3.2.5. Masking the Start

A fine and simple method.
Examine the curve in the Time Domain and try to find out the „**beginning of the steady state**“. This is at **t = 5 ms** in our example. Now set the start point of the Transient Simulation **to the end of the Start Up process (t = 5ms)** and simulate:




Go to the Cartesian FFT diagram of the last example but **DO NOT USE ANY WINDOWING!**

= **enter the formula**

**„dB(time2freq(OUT.Vt))“**

We see now all the lines with a dynamic range down to -90 dB. And also the DC value can be found again.

**But do not forget: the simulation time has been reduced by the windowing process and so the line width must increase....**

# 4. Second Project: Signals and Harmonics = a little FFT Exercise

## 4.1. Fundamentals

We start with a question:

**What means „Ideal Sine Wave"?**

The correct answer is not so simple because you have to distinguish:

**a) The wave form of this signal must be a perfect sine. Otherwise (= caused by periodic distortions) you get a line spectrum with „Harmonics" = additional new lines with new frequencies.**

**b) You must not see the start point and the end of the curve – it must be a Sine Wave from eternity to eternity. Otherwise you get an additional „energy floor" caused by transition processes and „switching on or off".**

## 4.2. Spectrum of a Single Voltage Pulse

Let us examine a voltage pulse with the following properties:

**Umin = 0V / Umax = 1000 V / Rise Time and Fall Time = 1 Microsecond / Pulse Length = 1 Millisecond.**

We add a delay time of 2 Milliseconds after the start and choose a simulation time of 50 Milliseconds in the Time Domain for the output voltage. 262144 Samples will give a good resolution in the frequency domain.

For the curve generation we use the „file based voltage source" (= PWL = piece wise linear voltage source) and feed this source with the following collection of pairs:

| | |
|---|---|
| **at t = Null** | **U = 0** |
| **at t = 2 Milliseconds** | **U = 0** |
| **at t = 2,001 Milliseconds** | **U = 1000 V** |
| **at t = 3,001 Milliseconds** | **U = 1000 V** |
| **at t = 3,002 Milliseconds** | **U = 0** |
| **at t = 50 Milliseconds** | **U = 0** |

Please start your text editor and write this file, but

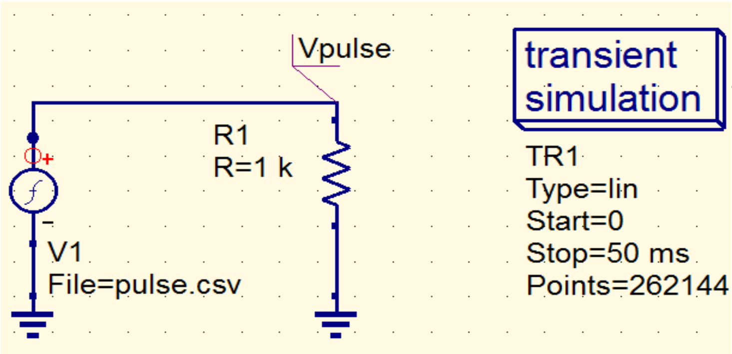
a) use an **own line for every couple**

b) terminate every line with a „**carriage return**"

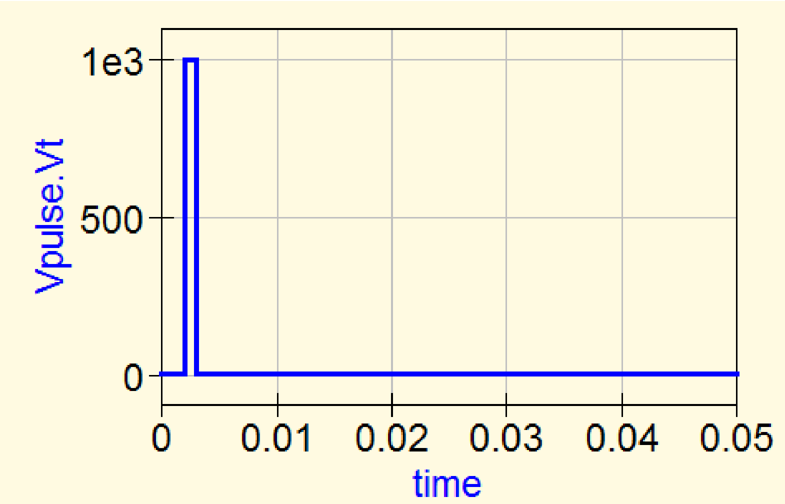
b) use the file name „pulse.csv" and save it in your actual project folder. But do not forget to set the editor properties to „**All files / ANSI**"....

So the content of „pulse.csv" must look like:

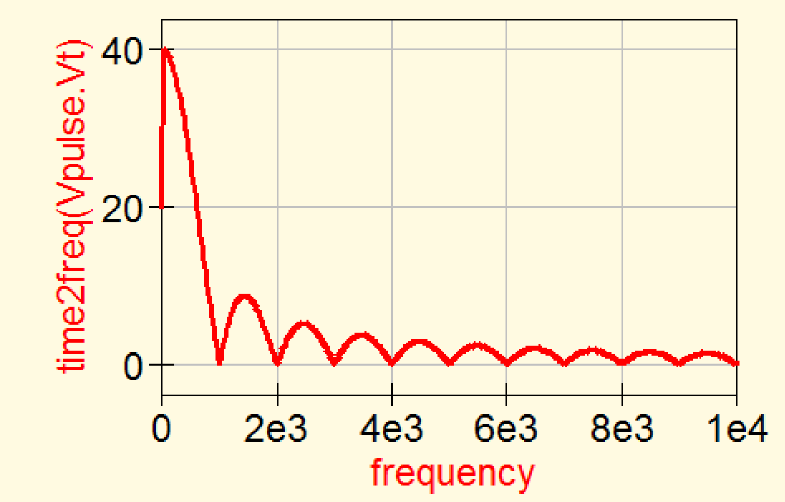
```
0         0
2e-3      0
2.001e-3  1e3
3.001e-3  1e3
3.002e-3  0
50e-3     0
```




This is the used circuit diagram.




Please show at first the Vpulse curve in the Time Domain, using a Cartesian diagram.




With a new diagram and the equation

time2freq(Vpulse.Vt)

you get the spectral content of this signal.

Please use the **manual scaling for the x-axis for 0.....10000Hz using a spacing of 2000 Hz**

Now you can admire the **sinx/x curve for the envelope** of the curve.

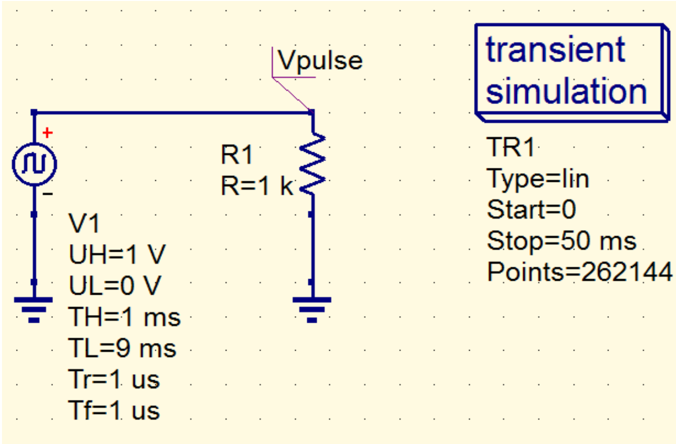
And do not forget:
**every Hertz of bandwidth is completely filled with energy** – so the curve shows the "**spectral energy density"**

You find the "**Zero Values" at every multiple of**

**f = 1 / Pulse Length**

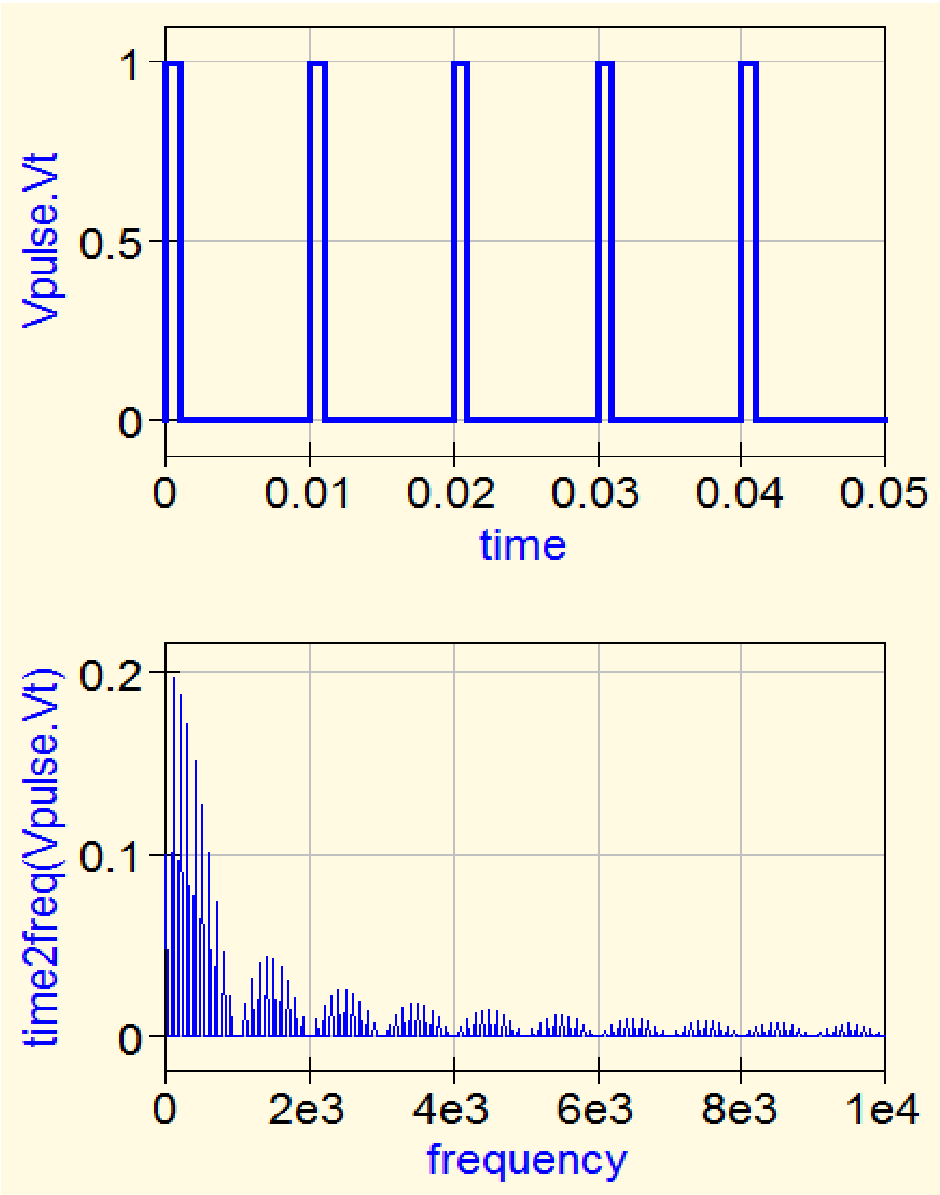
Please check this in the diagram...

## 4.3. Spectrum of a Periodic Pulse Signal




We replace the file based voltage source by a **Pulse Voltage Source and the same pulse width of TH = 1 ms.**
But now the pulse is periodically repeated 100 times per second.

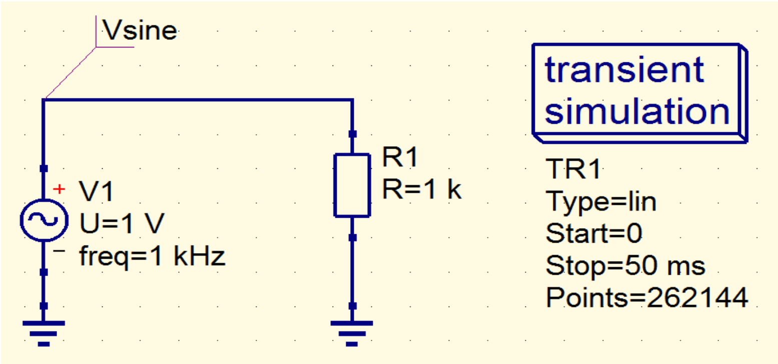
Simulation time is 50ms, thus we get a resolution of 1 / 50 ms = 20 ms in the frequency domain.




The envelope of the curve in the Frequency Domain equates the simulation result of a single pulse (= **Zeros at every multiple of f = 1 / Pulse length = 1 kHz)**

But now we have a periodic input signal and thus we find a **collection of lines (= Harmonics = multiples of the input frequency f = 100 Hz below the envelope.**

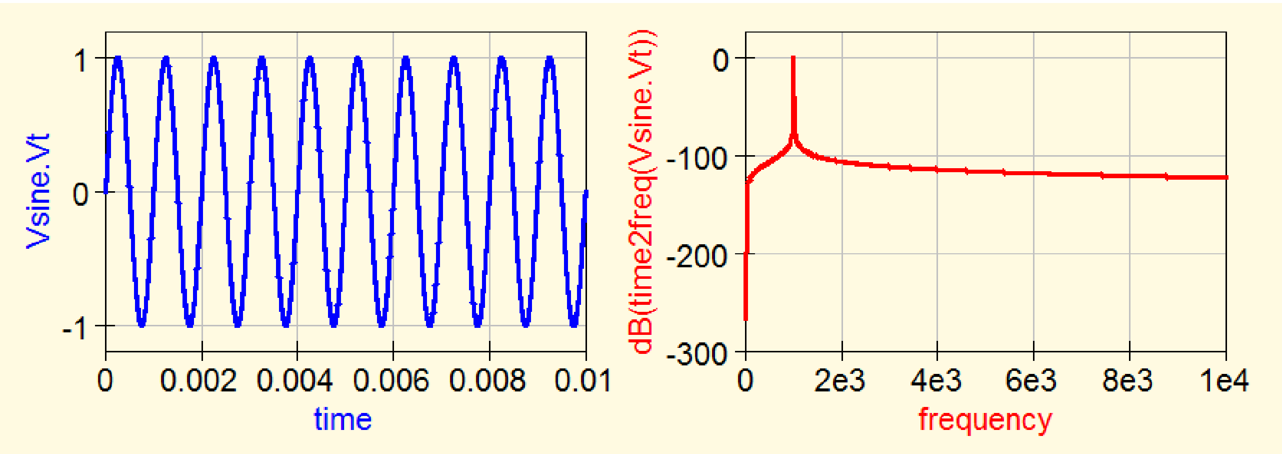
## 4.4. An ideal Sine Wave




In the introduction to this chapter we exclaimed:

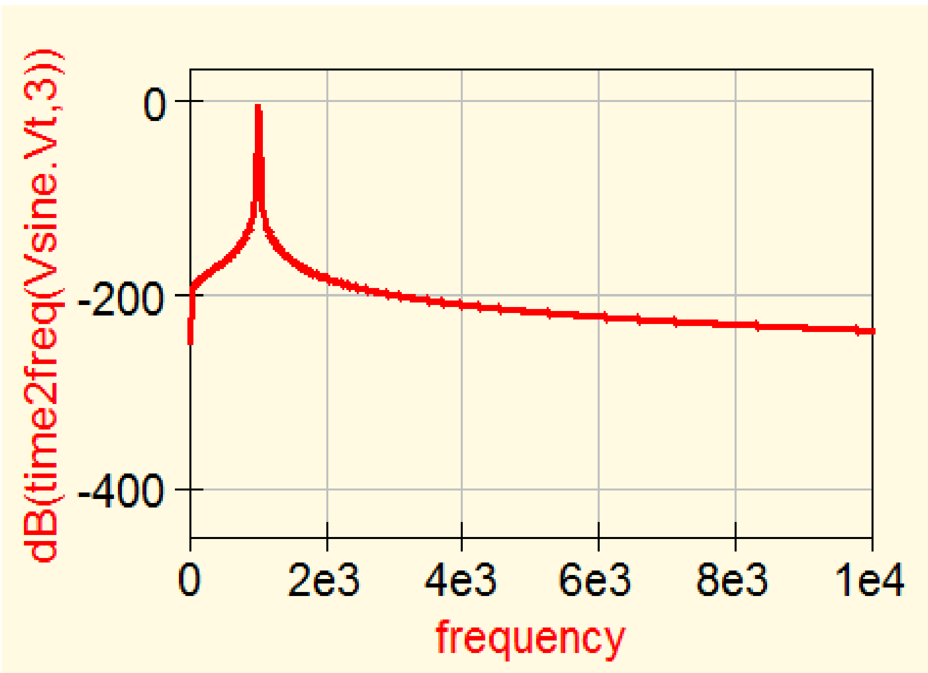
**„Only a sine wave signal with no end consists only of one spectral line“**

Let use proof this by testing a Sine Voltage with f = 1 kHz and an amplitude of U = 1V. (stop time = 50 ms, 262144 samples, result presentation in dB):



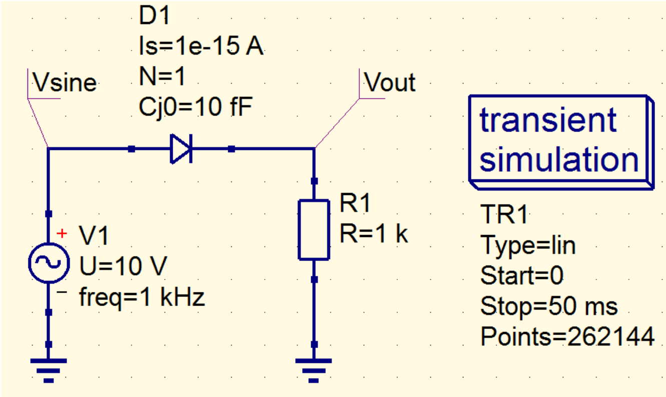

A dynamic range of only 100 dB for an ideal sine wave is a little disappointing – caused by the well known influence of the „start transition“.
So please repeat the FFT using a Hann Window (index = 3) by the formula

## dB(time2freq(Vpulse.Vt,3))



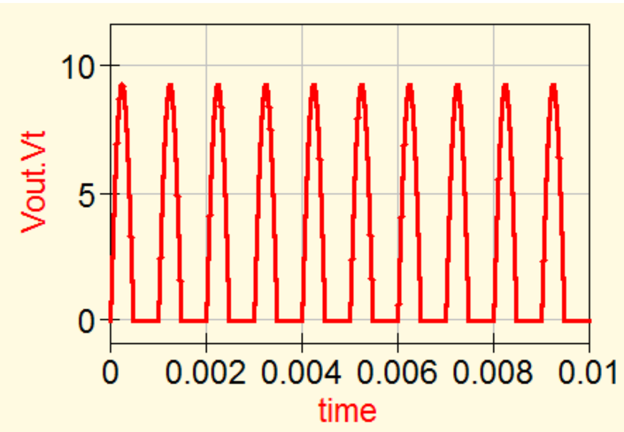

The dynamic range has now increased to 200 dB by this action – this should be a perfect sine wave.

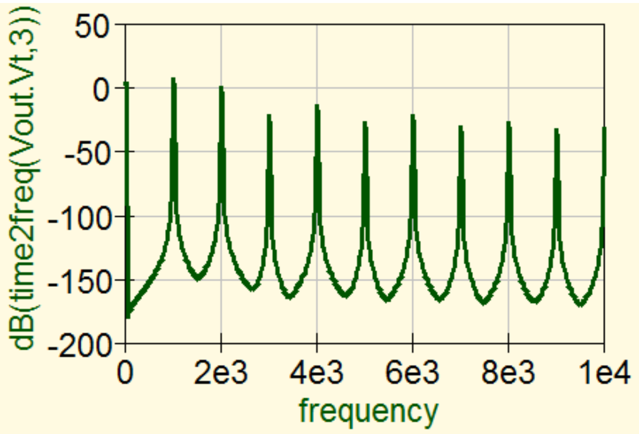
## 4.5. An asymmetrically distorted Sine Wave



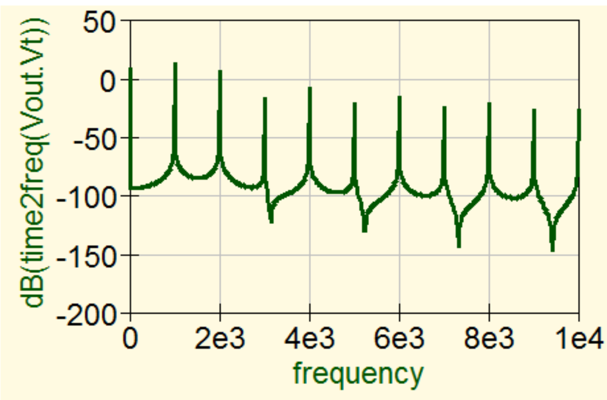

We insert a diode between the source and the load resistor (the diode can be found in „components / **nonlinear components**“).
So the negative part of the sine wave is cut off.

The peak value of the voltage is 10 V, the frequency is f = 1 kHz.
Simulation time is 50 ms with 262144 samples.




The left illustration (with tmax = 10 ms) shows the rectified output voltage Vout. The peak value is 10 V reduced by diode's „ON“ voltage of 0.7 V.
The right illustration is the FFT of Vout using a **Hann window (index = 3)** for the frequency range from 0.....10 kHz. This gives a dynamic range of ca. 150 dB and shows – as theory demands for a unsymmetrically distorted periodic signal! – **all the additional even and odd harmonics** (= **2kHz + 3kHz + 4kHz +5kHz.....**)




But pay attention to a nice little trap which causes errors in the above FFT result diagram:

Using the Hann windowing has successfully increased the dynamic range. But if you compare the **above diagram with windowing** to the **left diagram without windowing,** there is an important difference:

**windowing has reduced the total energy content and thus the absolute amplitudes are reduced and not correct in comparison to the left diagram!**

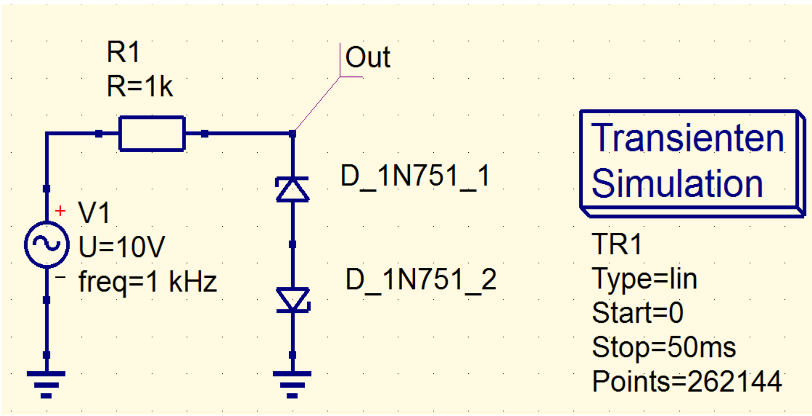
A control calculation shows this:
In the left diagram the **DC value is +10 dBV**. This equates to a voltage of ca. 3.1 V.
The theory says that the DC value of such a rectified signal is

### Udc = Umax / π

Please calculate **(10V – 0.7V) / π** …....and compare this result to the DC value in both diagrams.....

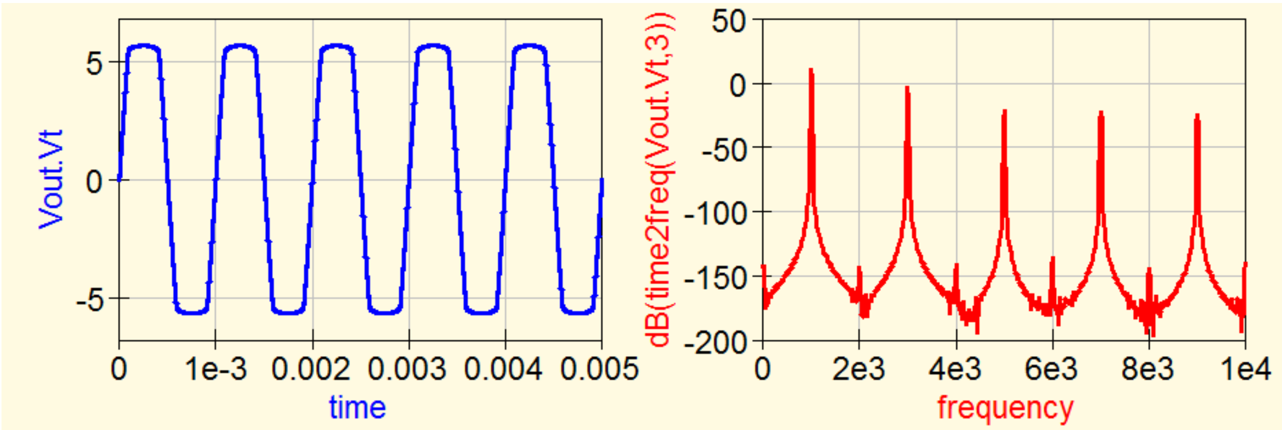
(correct result is 3 V)

## 4.6. A symmetrically distorted Sine Wave

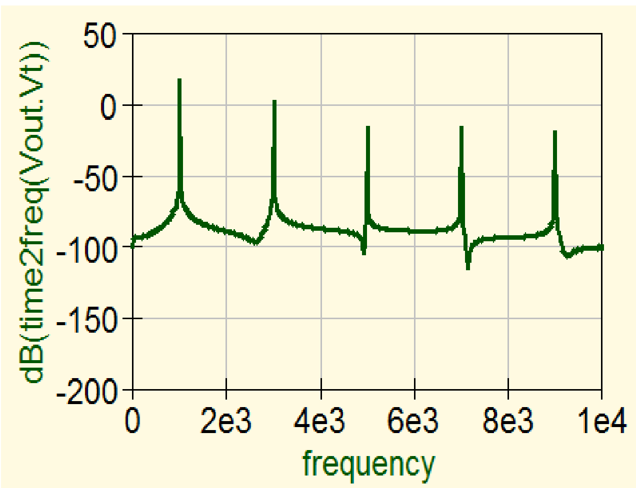



A simple exercise:
Use **two Zener diodes in series** in the manner as shown in this circuit diagram and use the type „1N751" with a breakdown voltage of 5.1V.
This part can be found in „**libraries / Z-diodes**".

Apply again a sine wave with a peak value of 10 V and a frequency of 1 kHz. Simulate for 50 ms with 262144 samples.




The left illustration (with **tmax = 5 ms**) shows the **„symmetrically clipped" output voltage with a peak value of (5.1V + 0.7V) = 5.8V.**
The right illustration shows again the FFT of Vout using a **Hann window (index = 3)** for the frequency range from 0.....10 kHz. This gives a dynamic range of ca. 150 dB and contents – exactly as theory demands for a symmetrically distorted periodic signal! – only the odd harmonics (= **3kHz + 5kHz + 7kHz +9kHz +.....**)




**And once more the same warning:**
Pay attention to the nice little trap which causes errors in the above FFT result diagram:

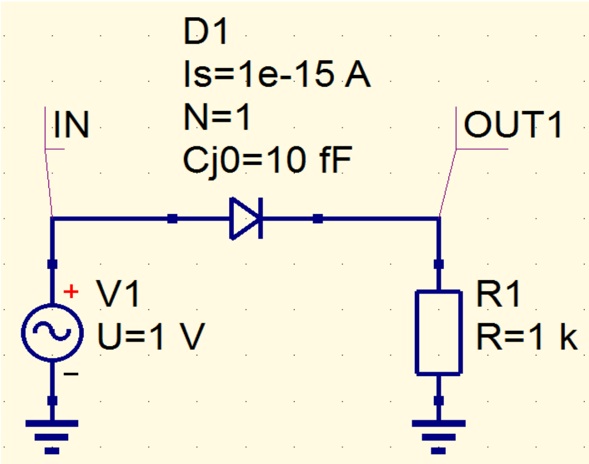
Using the (Hann) windowing procedure has successfully increased the dynamic range. But if you compare the **above diagram with windowing** to the **left diagram without windowing,** there is an important difference:

**windowing has reduced the total energy content of the signal and thus the absolute amplitude values of the lines are reduced and not correct in comparison to the left diagram!**

Hope you'll remember that in the future...

# 5. Third Project: Application of Diodes

## 5.1. One Pulse Rectifier with an ideal Diode




This is a very simple affair.

Use an AC Voltage Source (peak value = 5 V, frequency = 100 kHz)

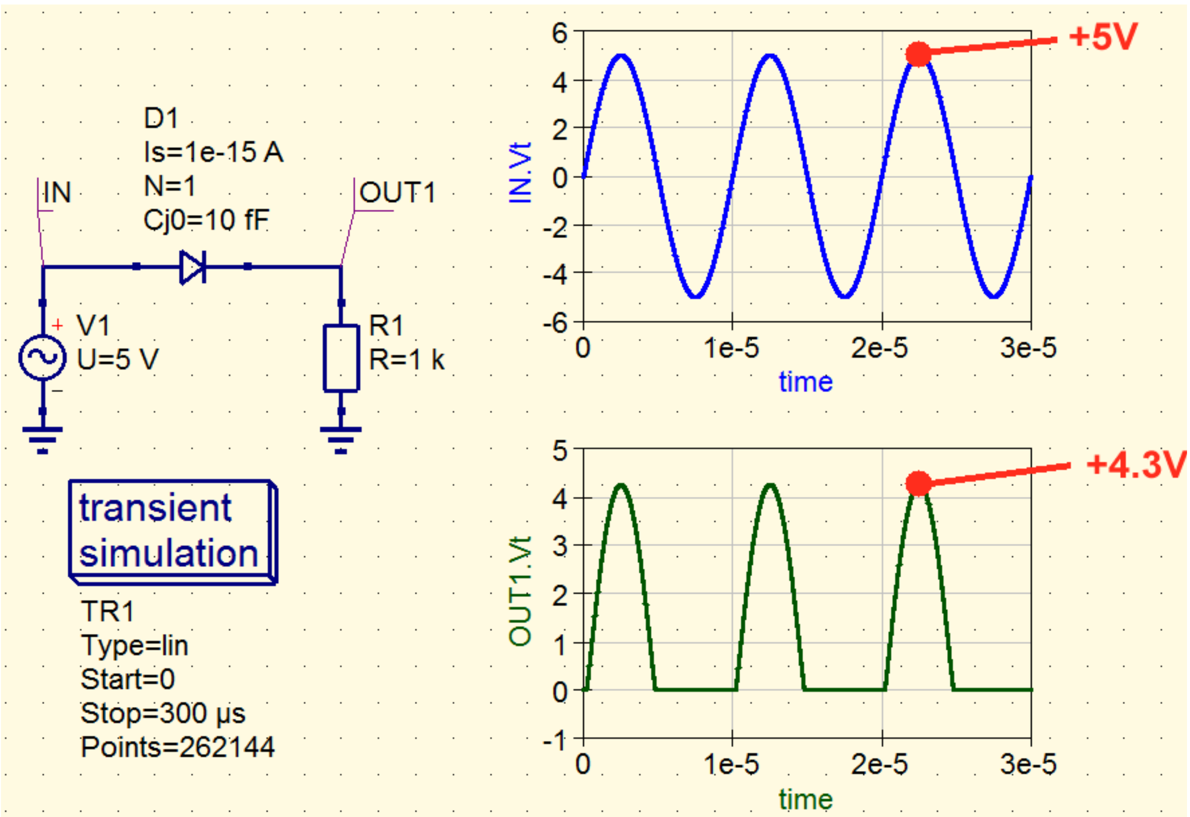
an ideal diode (can be found in „components / nonlinear components)

a load resistor with **R = 1kΩ**.

Labels „IN“ and „OUT1“ for the input and the output side of the circuit.

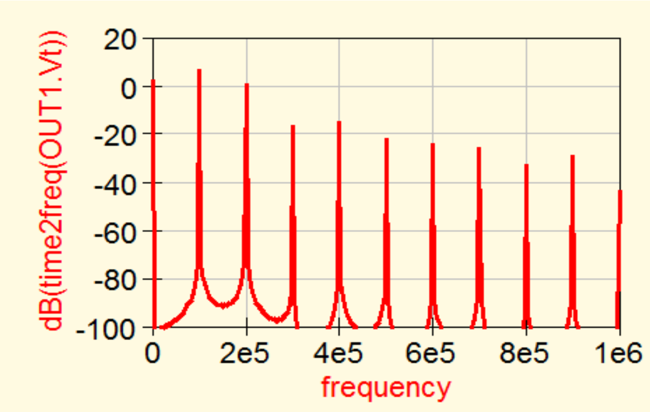
**(Remember:**
**To change a property of a part needs a right mouse click on the symbol to open the property menu)**.

The transient simulation time is **300 Microseconds with 262 144 samples (..use always a multiple of 2 for a correct FFT...). Please present the two voltages beside the circuit diagram.** Show the time range from 0...30 µs in the diagrams.



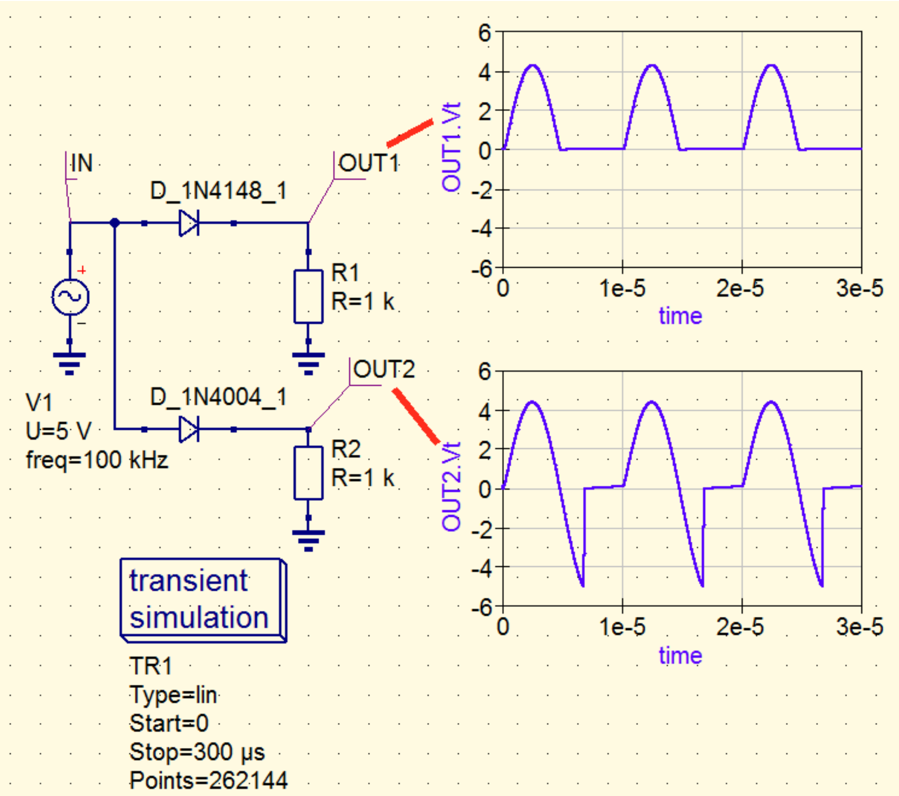

The peak output voltage must be (5V – 0.7V) = 4.3V due to the ON voltage drop of 0.7V at the diode.

**Task:**
**Show the spectrum of the output voltage by an FFT for frequencies from 0.....1 MHz.**




**The input for the FFT is an unsymmetrical distorted periodic voltage and thus we find ALL harmonics (= even + odd multiples of the fundamental input frequency with f = 100 kHz)**

## 5.2. Using Diodes coming from the qucsstudio Component Library




We use two well known parts:

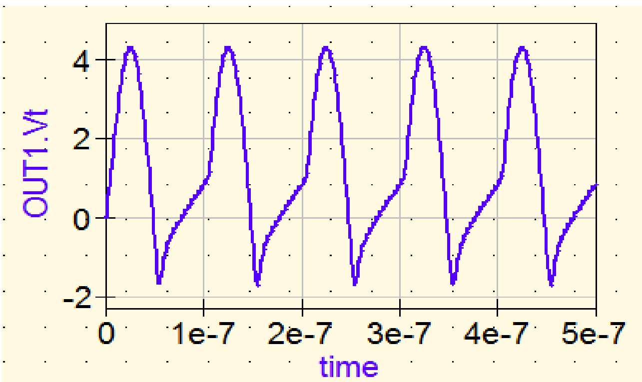
a) the famous universal purpose diode „1N4148“

b) the often used line rectifier 1N4004

They can be found in the „library“ menu at the left hand side of your screen.
Use the „drag'n drop“ in the left lower screen corner to place them in the schematic.

Now simulate for 300µs with 262144 samples.
Open two Cartesian schematics for OUT1 and OUT2. Set the properties and make a copy to place them beside the schematic.

At the curve for the 1N4004 we see the effect of the „**Reverse Recovery Time**“. The diode is not able to stop the diode current at once when the diode voltage has reversed. So this type of diode cannot be used at high frequencies.




The 1N4148 is better but at a frequency of f = 10 MHz we get the same effect.

**A radical correction is only possible by the usage of a super fast Schottky diode with low capacitance!**

In the next chapter we look for the SPICE File of the well known Schottky diode BAT17 and learn how to use it in qucsstudio.

## 5.3. Very important: usage of SPICE Models from the Internet

**Sorry, but this is a fact:**

**Qucsstudio cannot handle SPICE models when coming from a library or from the Internet. These models must at first be converted to a own qucs syntax!**

It is sometimes hard to get everything running properly. So the correct way is now presented and exactly described for the BAT17 Schottky diode.

**Step 1:**
Enter „BAT17 spice model“ in Google and open the **NXP homepage** in the result list. There you find a file named „**BAT17.zip**“. Download it and zip it out. The result is a new file named „**BAT17.prm**“.

**Step 2:**
Open this file with your text editor **at save it at once – but now named „BAT17.cir“. – in the „.qucs“ project folder for your transistor amplifier.**
**Attention**
**Set your text editor to „ANSI“ and „All Files“ before saving. Otherwise “*.txt“ is added and qucsstudio will not be able to read this file.**

This must be the content:

```
*
*******************************************
*
*BAT17
*
*NXP Semiconductors
*
*Schottky barrier diode
*
*
*IFSM =      @ tp = s
*VF   = 600mV @ IF = 10mA
**
*Package pinning does not match Spice model pinning.
*Package: SOT23
*
*Package Pin 1: Anode
*Package Pin 2: not connected
*Package Pin 3: Cathode
*
*
*Simulator:
*
*******************************************
*#
.SUBCKT BAT17 1 2
*
* The resistor R1 does not reflect
* a physical device.  Instead it
* improves modeling in the reverse
* mode of operation.
*
```

```
R1 1 2 6E+07
D1 1 2 BAT17
*
.MODEL BAT17 D
+ IS = 1.419E-09
+ N = 1.022
+ BV = 6
+ IBV = 2.45E-06
+ RS = 5.112
+ CJO = 7.867E-13
+ VJ = 0.1043
+ M = 0.1439
+ FC = 0.5
+ TT = 0
+ EG = 0.69
+ XTI = 2
.ENDS
```

**I recommend you to delete all comments (= lines beginning with a star) in the file and to save only the following corrugated part in your actual project folder (= .qucs) as „BAT17.cir". Avoids errors and an angry user...**
**Then check that the following rules are not violated:**

**First succesfull version:**

```
.SUBCKT BAT17 1 2
R1 1 2 6E+07
D1 1 2 BAT17

.MODEL BAT17 D
+ IS = 1.419E-09
+ N = 1.022
+ BV = 6
+ IBV = 2.45E-06
+ RS = 5.112
+ CJO = 7.867E-13
+ VJ = 0.1043
+ M = 0.1439
+ FC = 0.5
+ TT = 0
+ EG = 0.69
+ XTI = 2
.ENDS
```

**Terminate all these lines with "Enter" (= eol = end of line = carriage return)**

**If the model parameters are listed in separated lines, start every value with "+ " and termintate the line again with "end of line"**

==========================================================================

**Second successful version:**
In the Google result list you find a Siemens – Infineon library which contains also a BAT17 model

```
*******************************************************************
*SRC=BAT17s;BAT17s;Diodes;Schottky;Siemens
.MODEL BAT17s D(IS=4n RS=3.3 N=1.03 XTI=2 EG=.65
+ CJO=415f M=.156 VJ=.115 FC=.5 BV=12 IBV=5U TT=25p)
* Chip parameters; for higher frequencies package must be added
*******
```

Delete all lines starting with a star. Then change the file to the following manner and **save it as „BAT17_Infineon.cir"**

```
.MODEL BAT17s D
+ IS=4n RS=3.3 N=1.03 XTI=2 EG=.65 CJO=415f M=.156 VJ=.115 FC=.5 BV=12 IBV=5U TT=25p
```

**Attention when handling the „.model" part of the file:**

**a) The first line „.MODEL BAT17s D" must be terminated with „eol= end of line"**

**b) The second line with the model parameters starts with „+ „.This line must content all parameters and also be terminated with „eol".**

**Third successful version.**
This version uses the same Siemens Infineon file. But now it is written as follows:

```
.MODEL BAT17s D IS=4n RS=3.3 N=1.03 XTI=2 EG=.65 CJO=415f M=.156 VJ=.115 FC=.5 BV=12 IBV=5U TT=25p
```

**Write the complete content of the model file into only one line and terminate this single line with „eol = end of line“. Save the file as „BAT17_infineon_02.cir“.**

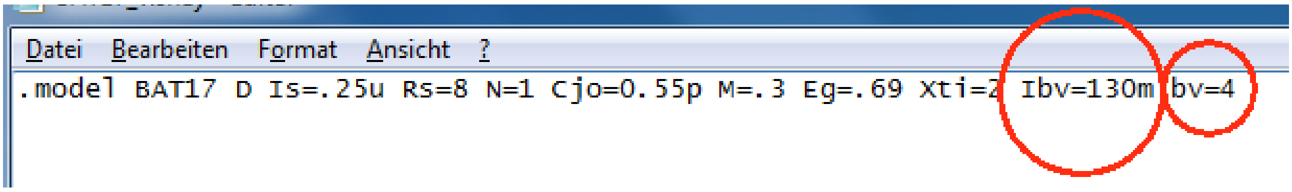
=====================================================================

**Fourth successful version:**
In the Internet you can also find the following BAT17 model, published by Vishay:

**.model BAT17 D(Is=.25u Rs=8 N=1 Cjo=0.55p M=.3 Eg=.69 Xti=2 Iave=130m Vpk=4 mfg=Vishay type=Schottky)**

This must be modified as follows to get it running in the simulation.

**First running solution:**




**Use only one line and terminate it with „eol = end of line = Enter“**. Delete the last two entries (= „mfg=Vishay“ and „type=schottky). **Modify the last two model parameters:**

**„Iave=130m“ must now be „Ibv=130m“**

**„Vpk=4“ must now be „bv=4“** (….“bv“ means: breakdown voltage...)
...
At last save it as „**BAT17_Vishay_01.cir**“

**Second running solution:**

**.model BAT17 D**
**+ Is=.25u Rs=8 N=1 Cjo=0.55p M=.3 Eg=.69 Xti=2 Ibv=130m bv=4**

**Modifications:**
a) entries for type and manufacturer deleted

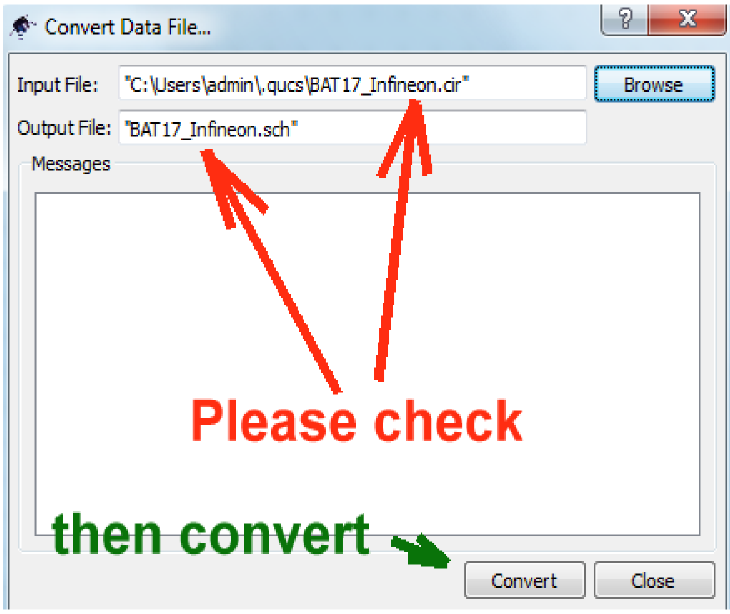
b) „Iave=130m“ changed to **„Ibv=30m**“, „Vpk=4“ changed to **„bv=4**“

c) **„.model BAT17 D“ in an own line, terminated by eol = end of line = Enter**

d) **All model parameters written in only one line**. The line starts with „+ „ and is terminated by „eol = end of line = Enter“

At last save it as „**BAT17_Vishay_02.cir**“ in your „.qucs“ project folder.
=====================================================================

**Step 4:**
Let us test the model file „BAT17_Infineon.cir".




Open the „**Project**" menu, go to „**Import Data**". Browse for „**BAT17_Infineon.cir**" in the list and open it.
This should now be your screen.

Check the path for the input and the output file, then convert.

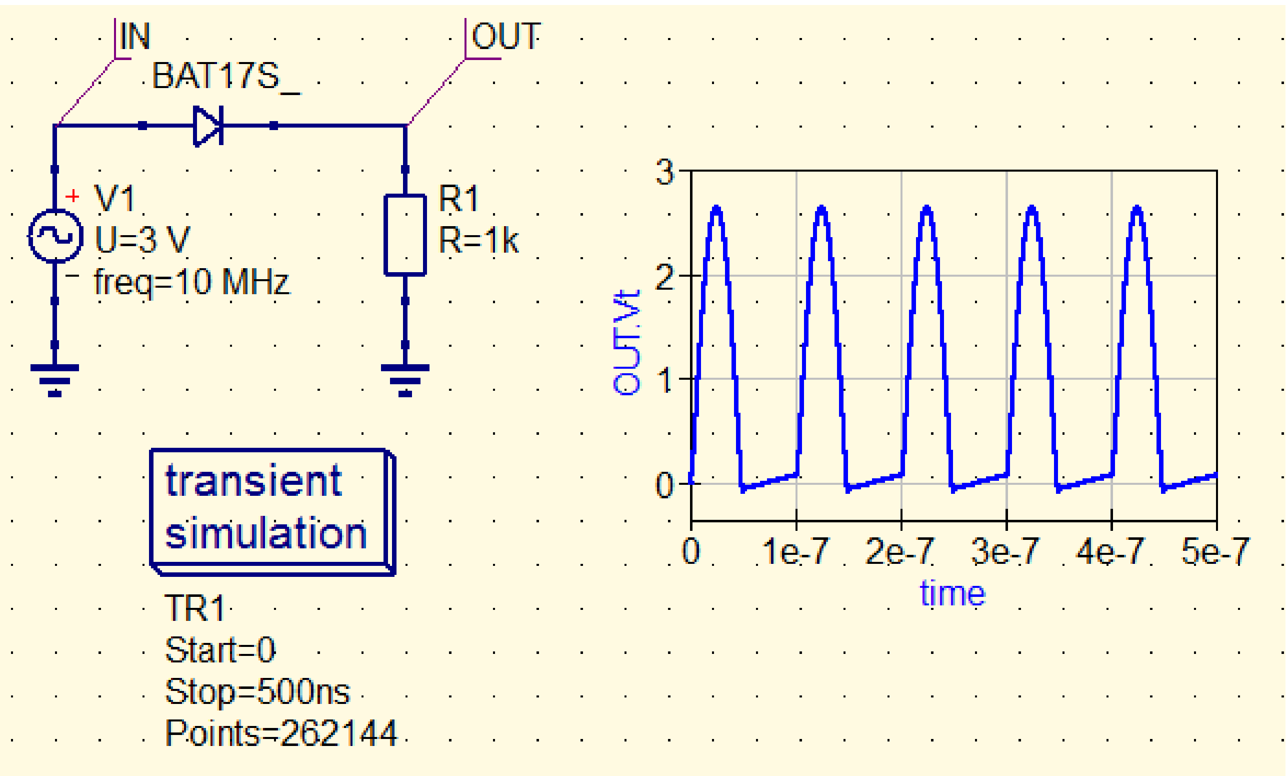
A new schematic named „**BAT17_Infineon.sch**" will be created and you find the message

**Successfully converted file 'BAT17_Infineon.sch'.**

on your screen.

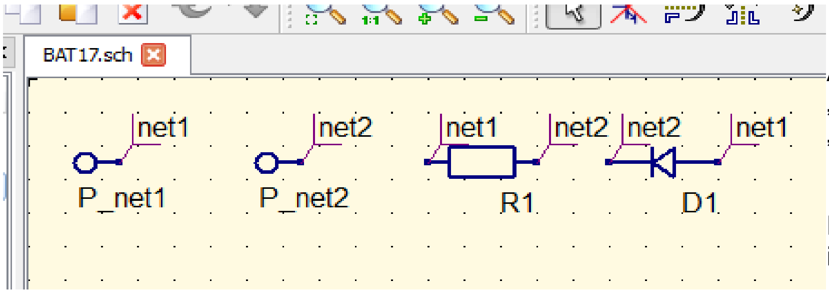
Now open this new schematic, draw the circuit and simulate the output voltage.

**Attention:**
**The peak value of the input voltage must be reduced to 3 V due to the breakdown voltage of 4 V for the BAT17**




You see why only Schottky diodes can do the rectifier job well at high frequencies....

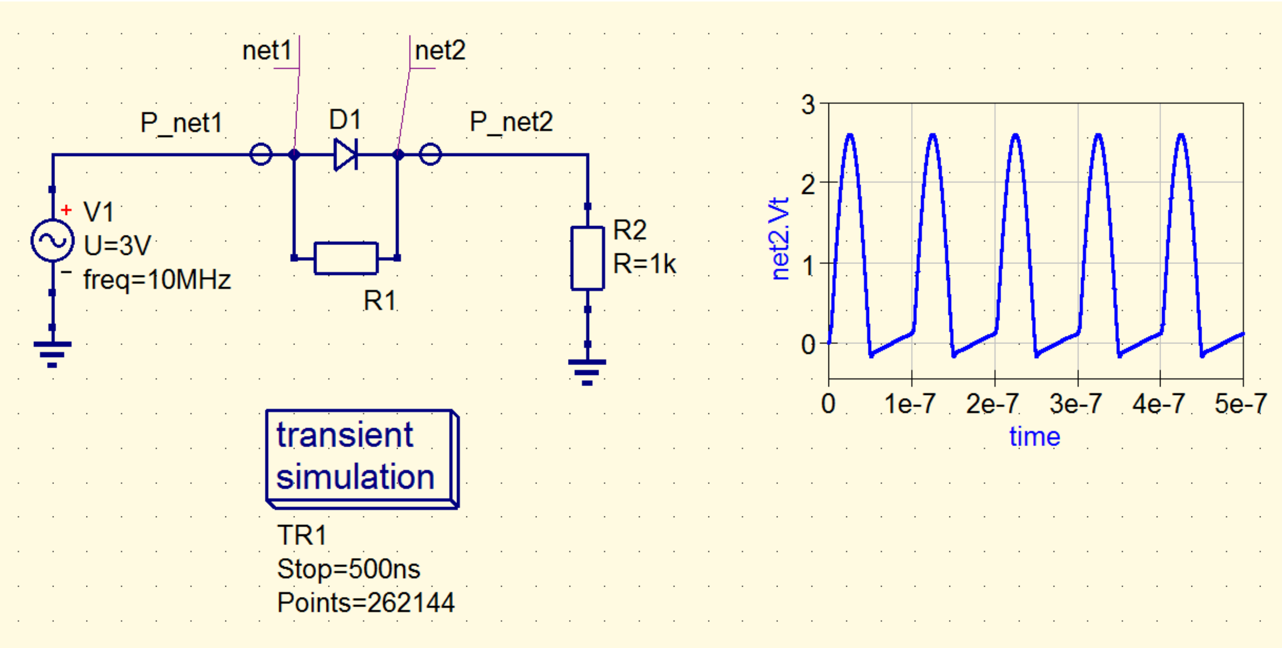
It is also interesting to use the converted **NXP-File** because there is a resistor added to improve the simulation precision.




After the conversion of the file „**BAT17.cir**“ you get this schematic „**BAT17.sch**“:

Don't worry – this schematic is a little irritating but not complicated.

This is the final schematic for the simulation.



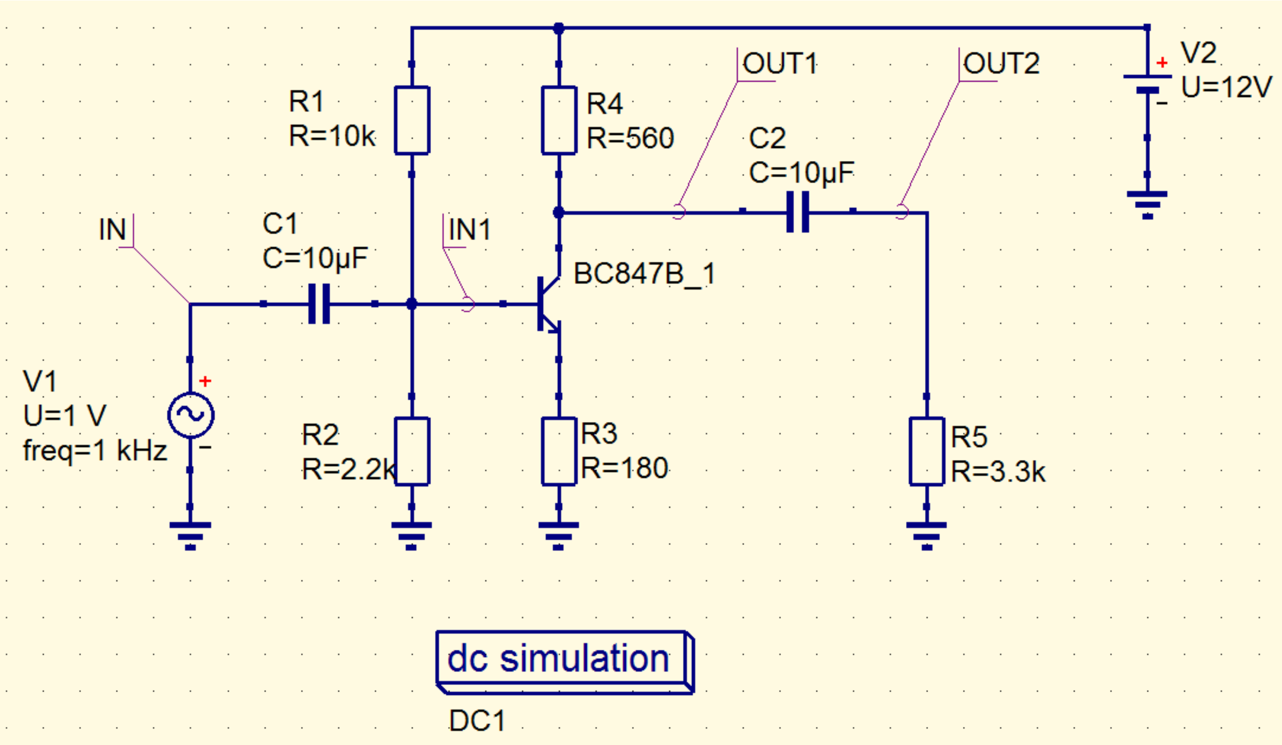

The result is the same as before.

**Task:**

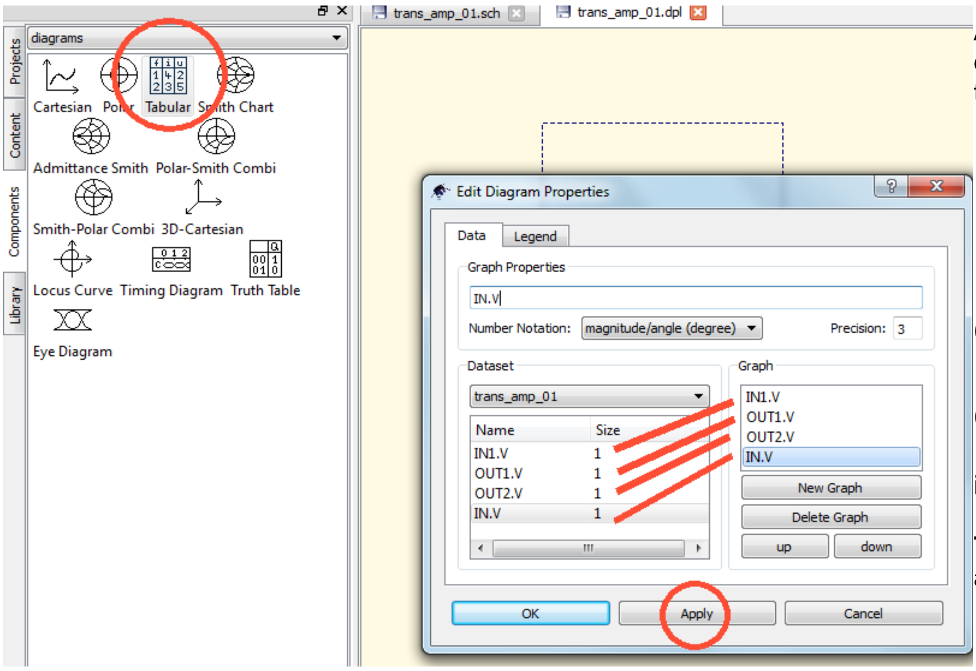
**Test all the other converted model files of the chapter for correct function.**

# 6. One Stage Transistor Amplifier

## 6.1. DC Simulation to check the correct Biasing




Draw this schematic and add „**DC Simulation**“ (from „Components / Simulations“). Set the four **Labels** (**IN / IN1 / OUT1 / OUT**) and save the schematic file as „**trans_amp_01.sch**“.




After the simulation drag'n drop a „**tabular**“and enter the four voltages

**IN.Vt**

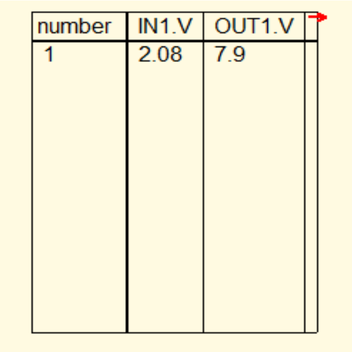
**IN1.Vt**

**OUT1.Vt**

**OUT2.Vt**

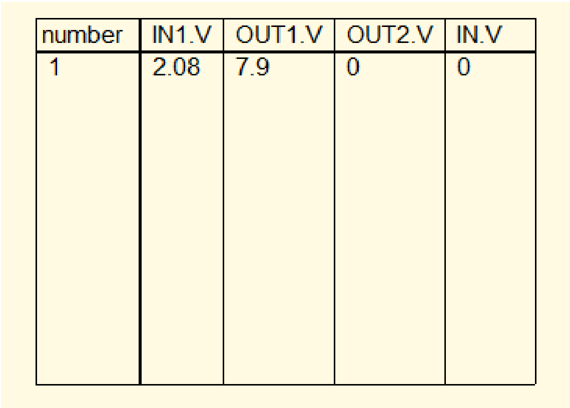
into the Graph list.

**Then click on „Apply“ and OK.**



| number | IN1.V | OUT1.V |
|---|---|---|
| 1 | 2.08 | 7.9 |

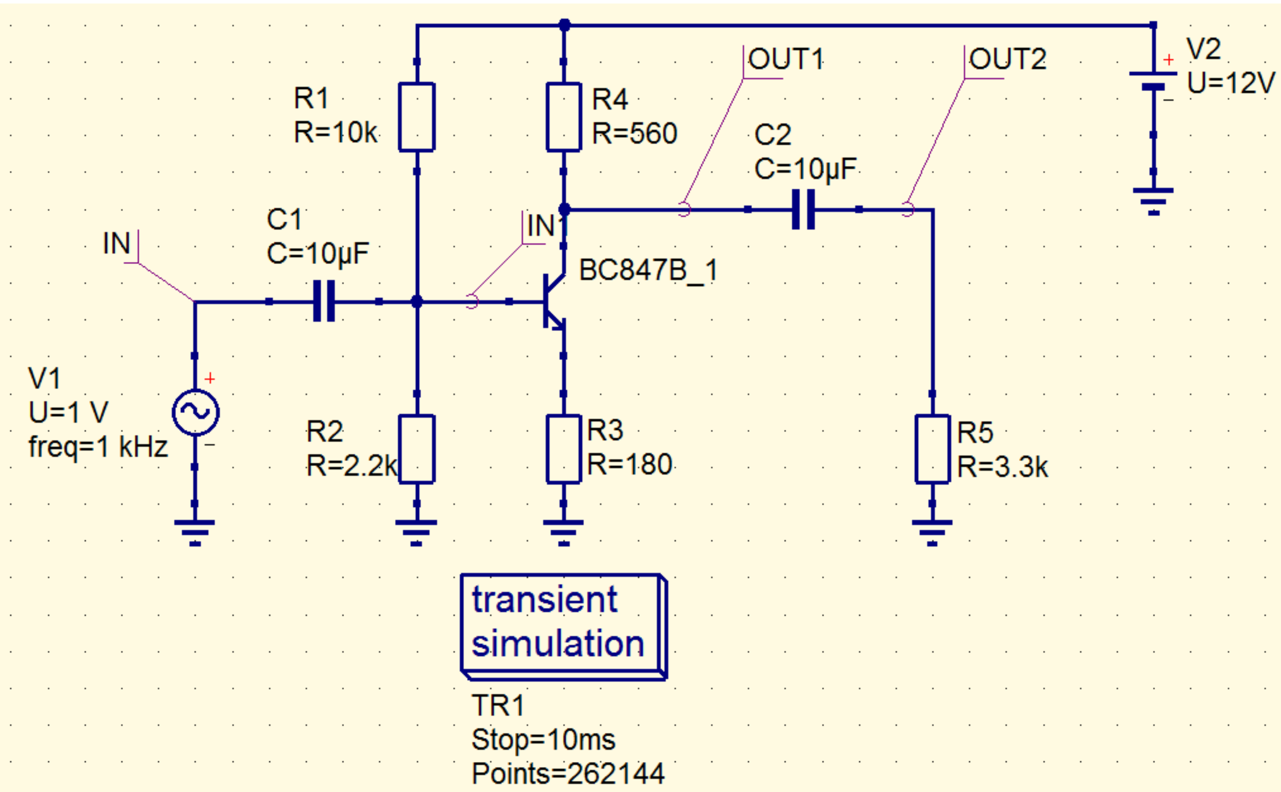
The result is not convenient, because we see only the values of IN1.V and OUT.V.



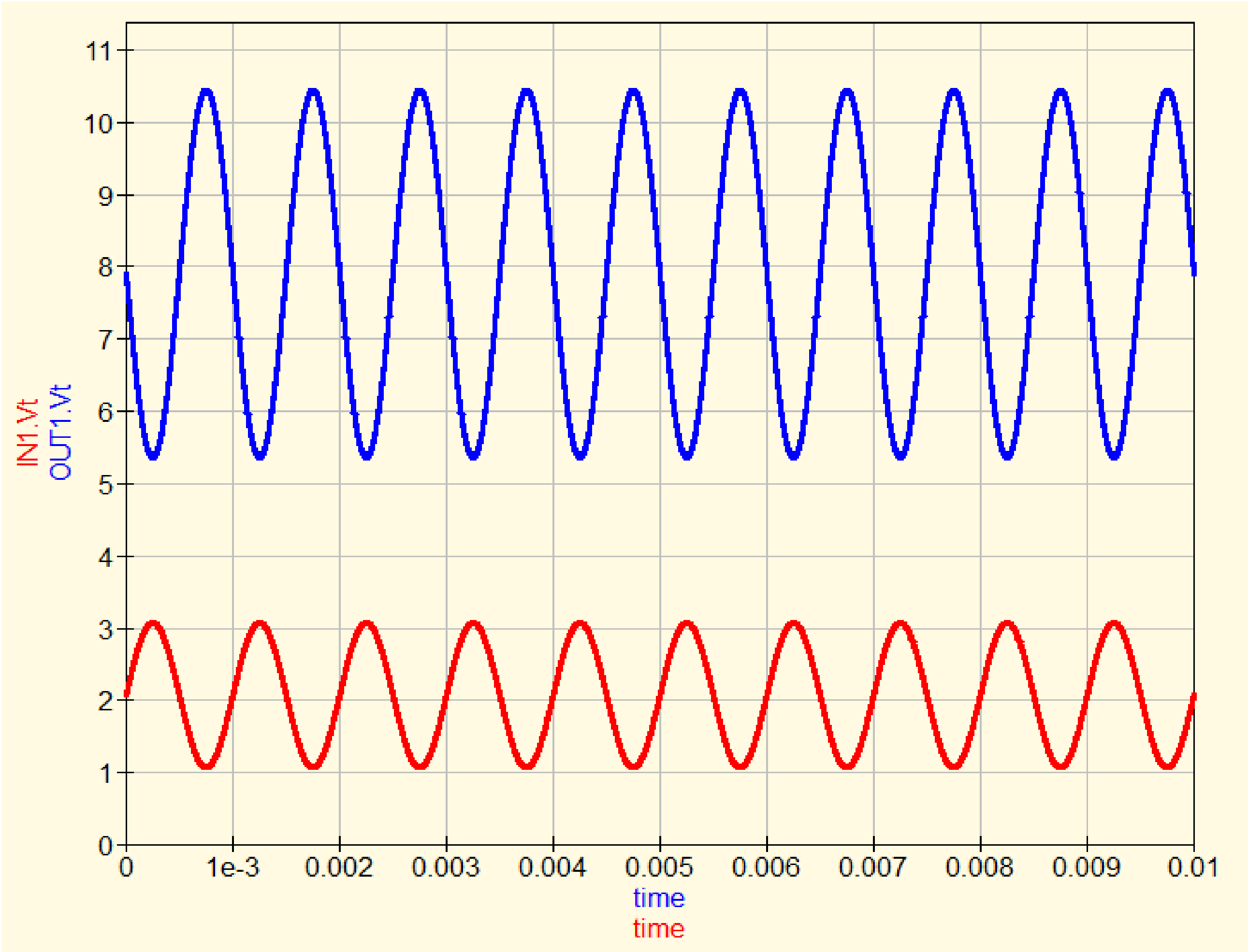
| number | IN1.V | OUT1.V | OUT2.V | IN.V |
|---|---|---|---|---|
| 1 | 2.08 | 7.9 | 0 | 0 |

But the secret and the trick is the **little red arrow in the upper right corner** of the tabular. At first mark the complete tabular. Then roll your mouse cursor to the arrow, press the left mouse button and drag to the right: you will see the rest of the voltage values:

## 6.2. Transient Simulation: Sine Signal at the Input




**Use f = 1kHz and a peak value of 1 V for the Input Sine Voltage. Simulate for 10 ms with 262144 points.**




There is no surprise when regarding the voltages

**IN1.Vt**

and

**OUT1.Vt**

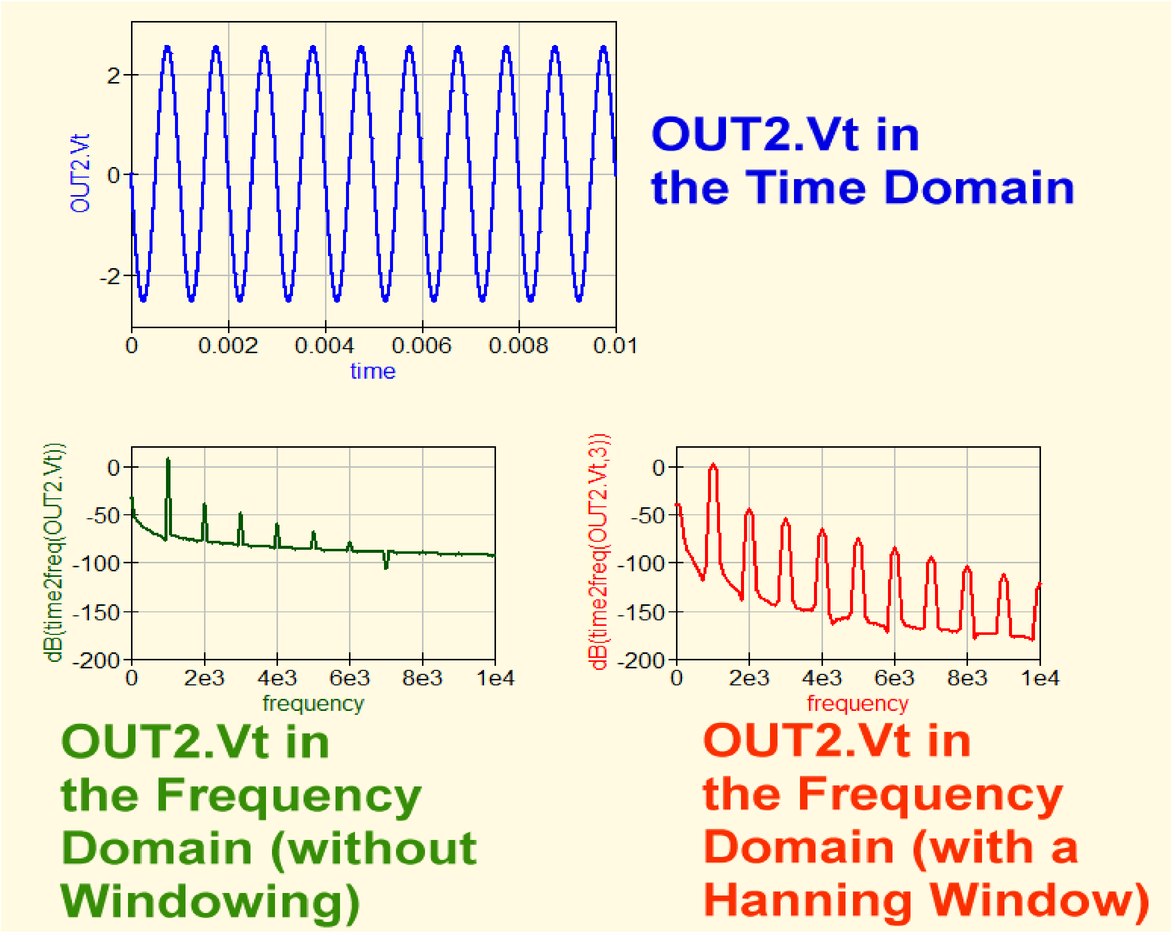
There exists (as theory demands) a phase shift of 180 degrees between the input and the output signal. The operating point is correct = no distortions.

The voltage gain can be calculated by the formula:

**Gain = (R4 || R5) / R3 = (0.56k || 3.3k) / 180) = 2.66**

Please check this result.

Now let us have a sharp look at the **output load resistor R5 and its voltage drop OUT2.Vt using Time Domain and Frequency Domain presentation:**




The first diagram shows **OUT2.Vt** in the Time Domain = that is what you would see on your oscilloscope screen.

Please compare now the two Frequency Domain diagrams:

The **left diagram is generated with the formula:**

**„dB(time2freqOUT2.Vt))"**

and shows a low dynamic range due to the „switch ON transition" at the start. This transition adds an additional „energy floor".
But this diagram has also two advantages: the **amplitude calibration of all lines is absolutely correct** (...calibration of the vertical axis is „dBV" = dBVolt" = 1 Volt as reference). And the **simulated lines of the spectrum are very small**.

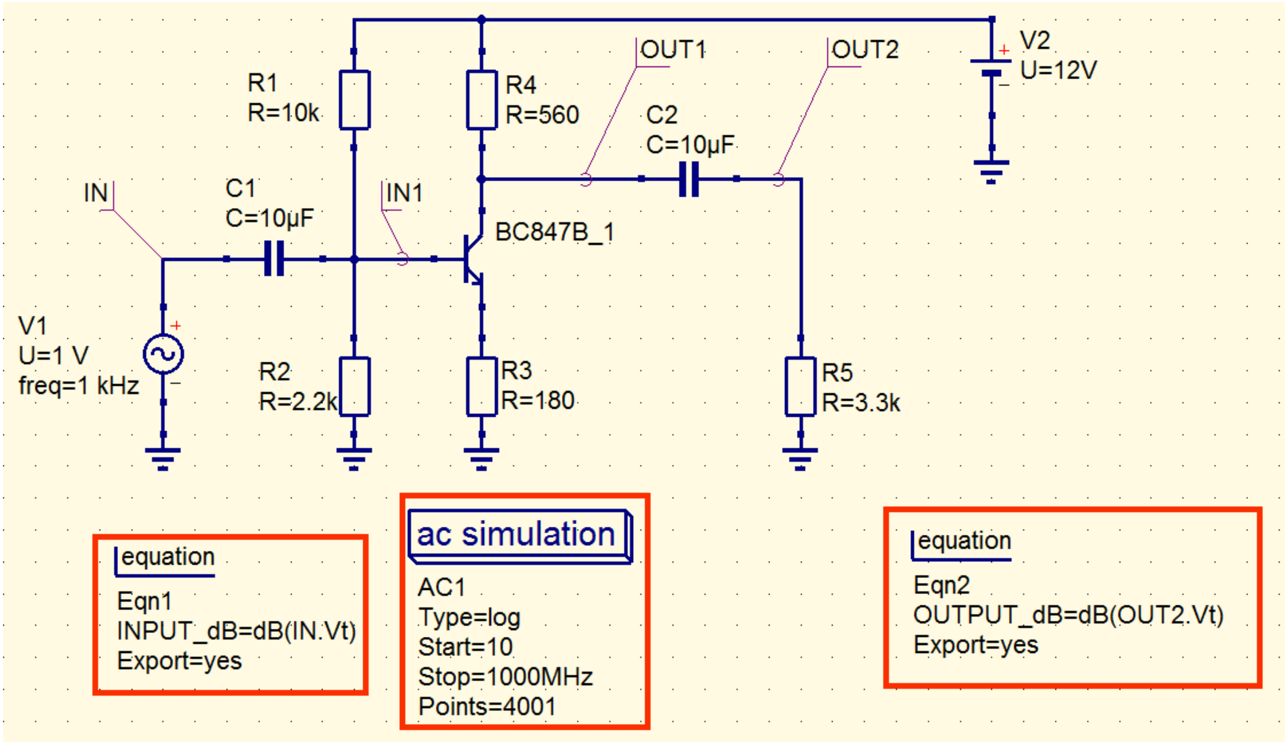
The right diagram is generated with the formula

**„dB(time2freqOUT2.Vt,3))"**

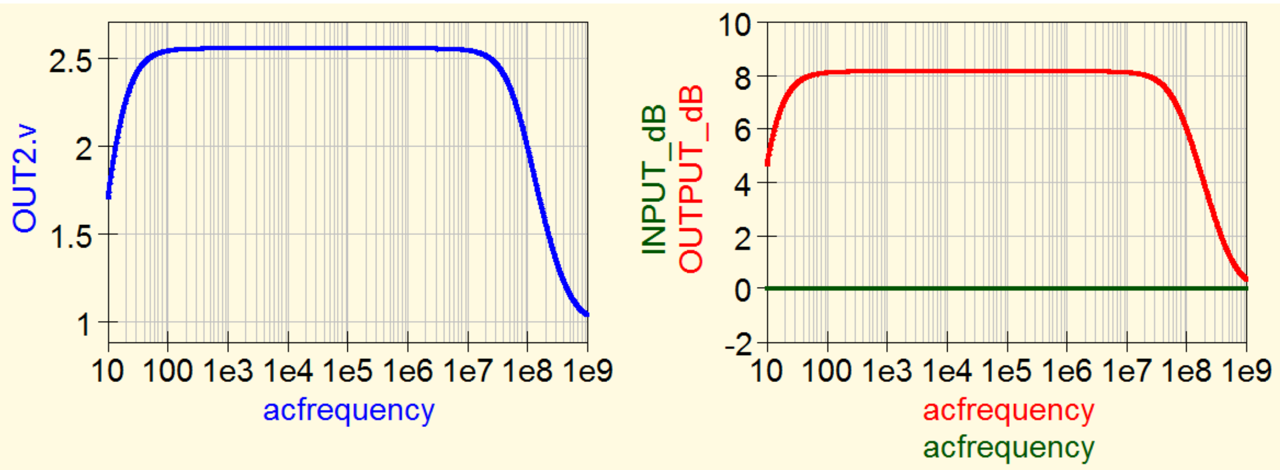
and uses a **Hanning Windowing** with index = 3. Thus the **dynamic range is increased.**
But **increased is also the width of every spectral line and the amplitude calibration is no longer correct** due to the „lost samples by the Windowing process".

## 6.3. AC Sweep

Use a logarithmic sweep from 10 Hz to 1 GHz with 4001 points per decade.
Add two equations to the schematic to convert the input and the output signal to „dBV“ (= dB, related to 1 V).




**Now show the output signal „OUT2.Vt“ versus frequency:**




**Linear presentation** **dB presentation**

## 6.4. Example with the Transistor BFR92A

### 6.4.1. We use a SPICE Model from the Internet

```
* Filename:   BFR92A_SPICE.PRM
* BFR92A SPICE MODEL
* PHILIPS SEMICONDUCTORS
* Date : September 1995
*
* PACKAGE : SOT23 DIE MODEL : BFR90A
* 1: COLLECTOR; 2: BASE; 3: EMITTER;
.SUBCKT BFR92A 1 2 3
Q1 6 5 7 7 BFR90A
* SOT23 parasitic model
               Lb  4 5 .4n
               Le  7 8 .83n
               L1  2 4 .35n
               L2  1 6 .17n
               L3  3 8 .35n
              Ccb  4 6 71f
              Cbe  4 8 2f
              Cce 6 8 71f
*
* PHILIPS SEMICONDUCTORS   Version:   1.0
* Filename:    BFR90A.PRM   Date: Feb 1992
*
.MODEL   BFR90A    NPN
+              IS = 4.11877E-016
+              BF = 1.02639E+002
+              NF = 9.97275E-001
+             VAF = 6.26719E+001
+             IKF = 3.20054E+000
+             ISE = 4.01062E-015
+              NE = 1.57708E+000
+              BR = 1.81086E+001
+              NR = 9.96202E-001
+             VAR = 3.36915E+000
+             IKR = 1.28155E+000
+             ISC = 2.79905E-016
+              NC = 1.07543E+000
+              RB = 1.00000E+001
+             IRB = 1.00000E-006
+             RBM = 1.00000E+001
+              RE = 1.16450E+000
+              RC = 2.32000E+000
+              EG = 1.11000E+000
+             XTI = 3.00000E+000
+             CJE = 8.90512E-013
+             VJE = 6.00000E-001
+             MJE = 2.58570E-001
+              TF = 1.54973E-011
+             XTF = 3.91402E+001
+             VTF = 2.15279E+000
+             ITF = 2.13776E-001
+             CJC = 5.46563E-013
+             VJC = 3.80824E-001
+             MJC = 2.02935E-001
.ENDS
```

**Step1:**
Start the Google and enter **„bfr92 spice model“**.

In the result list you'll find the NXP homepage and there a complete zipped collection of models.

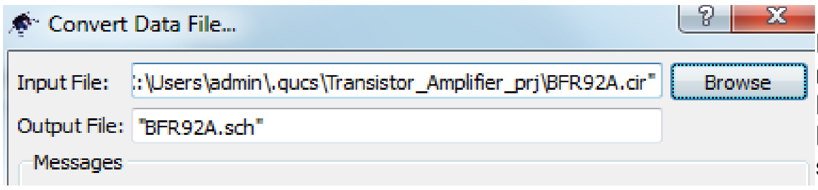
Extract the file **„BFR92A-Spice.prm**“ and open it. **Save it at once in your project folder as**

**BFR92A.cir**

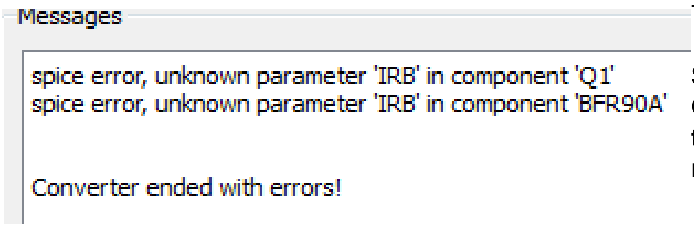
but do not forget to set your editor to „ANSI“ and „All files“.

**This is the complete NXP-File.**

**Step 2:**




Now start qucsstudio and open the menu „**Projects**“, followed by „**Import Data“**.
Browse for your file „**BFR92A.cir**“ and start the conversion.



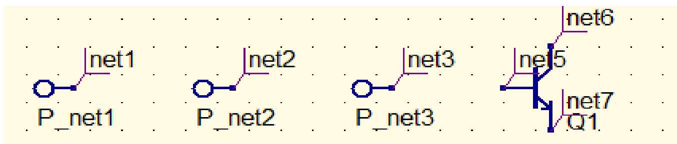

The Converter answers with this error message.

**Sorry:**
**Older converter versions have simply ignored the fact, that IRB is a non important detail of the model.....but progress...**

So please open the „**BFR92A.cir**“ file once more with a text editor, **delete this line concerning IRB** and save the file again. If you then start the conversion again, you'll get the message:

**Successfully converted file „BFR92A.sch“.**

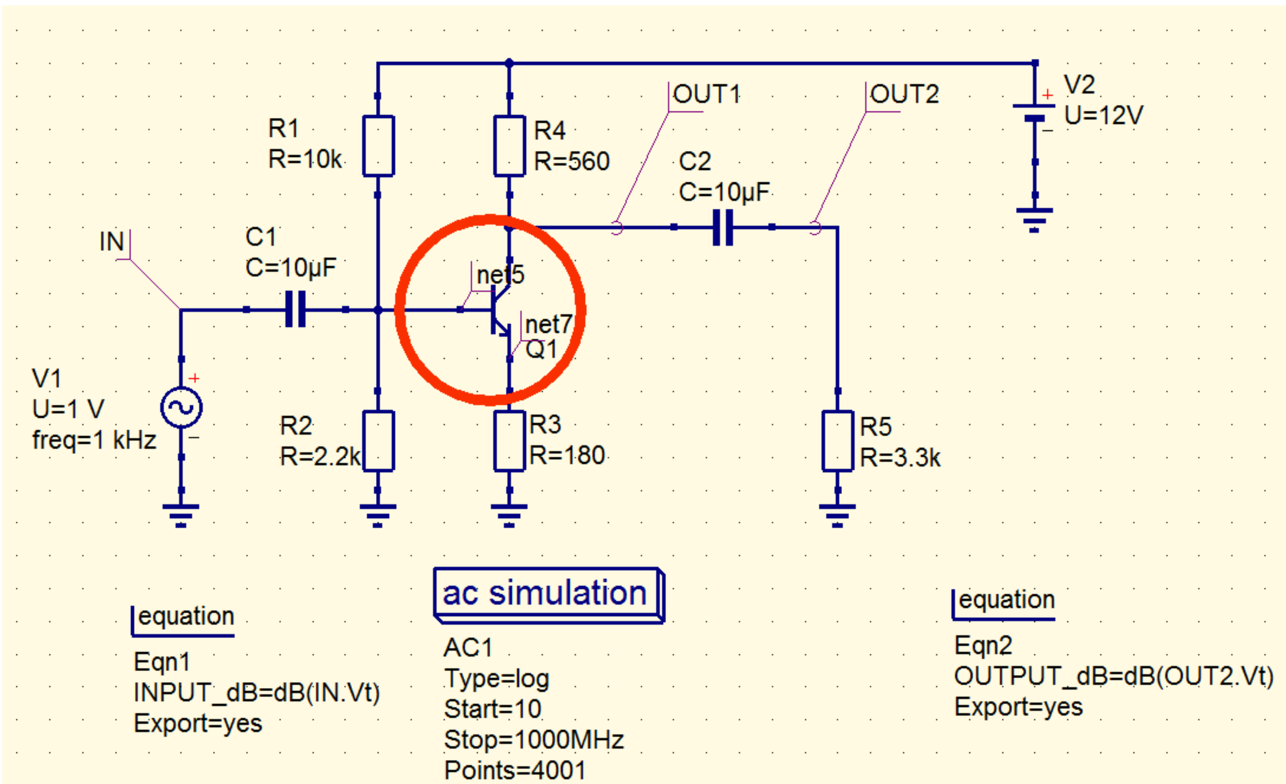
**Step 3:**

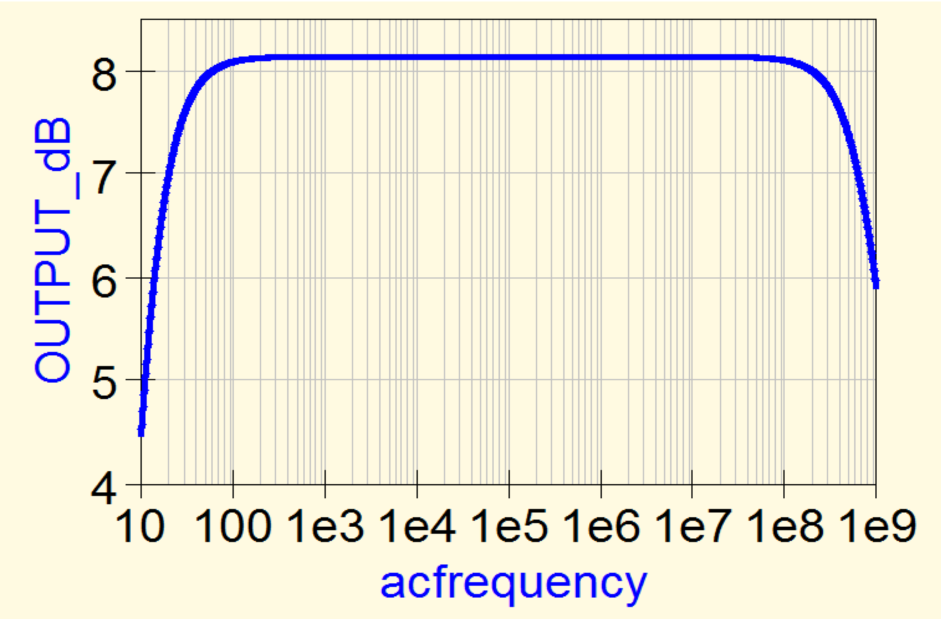



Open „**BFR92A.sch**“ and you'll see this screen.

You may delete the three offered ports but the transistor symbol is the really interesting device.

We open now the last transistor amplifier schematic and insert this new transistor symbol instead of the preceeding BC847 by „paste and copy“.






Now simulate...
...and you see what you want.

**But now you should yourself repeat all the simulations of the preceding chapters an exercise, like**

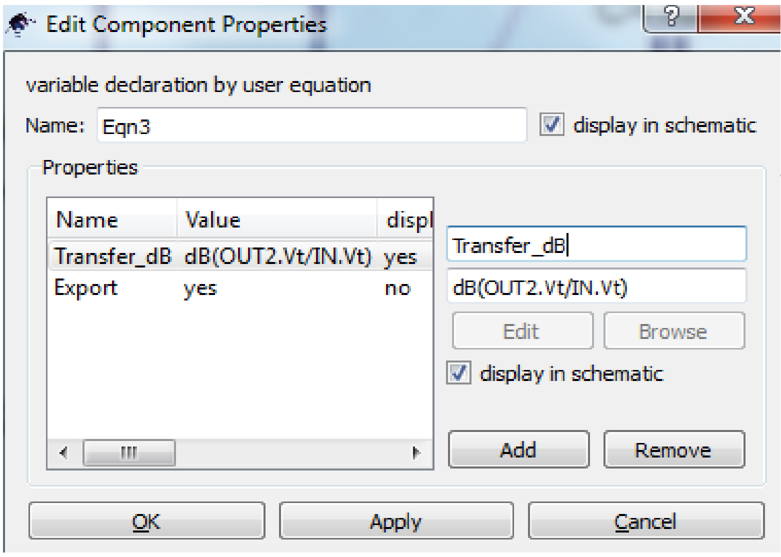
**DC Simulation**

**Transient Simulation using a 1 kHz sine wave, result presentation shows every voltage in the circuit**

**Frequency content of the output voltage, calculated by an FFT (with and without Hanning Windowing, index =3). Result presention in linear and dB form.**

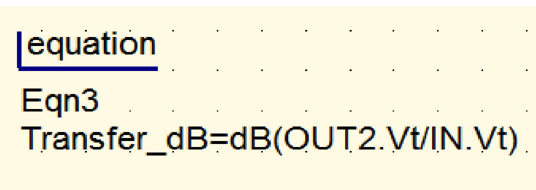
## 6.4.2. A final Test for You

**Task:**
**Calculate the Transfer-Function = (OUT2 / IN) in dB** for the range from 10 Hz to 1000 MHz in the following manner:




Go to the schematic of the circuit. Press the button for a new equation and enter the formula

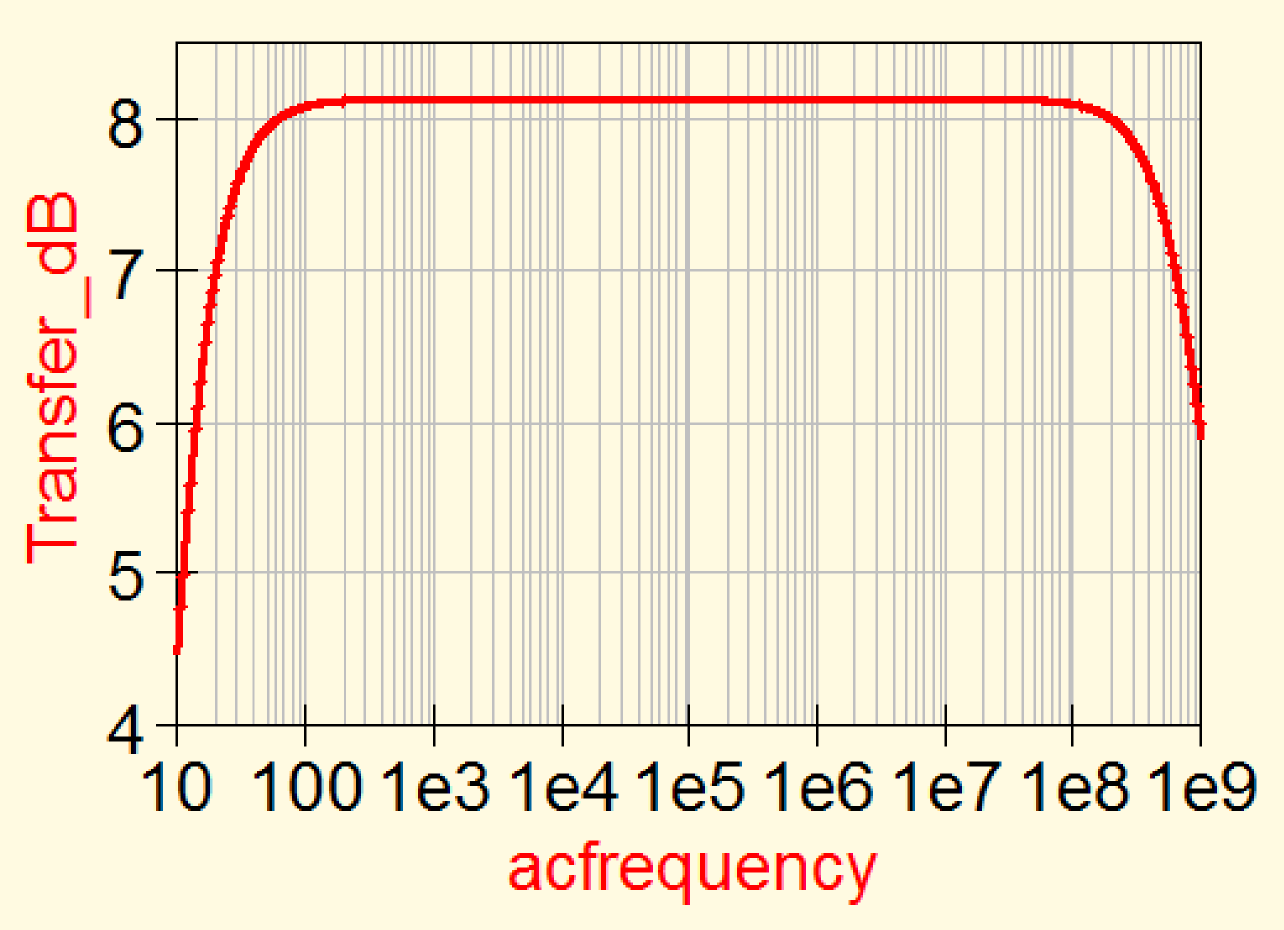
**Transfer_dB=dB((OUT2.v)/(IN.v))**




Now you must see this message beside your schematic.

Use now a logarithmic AC Sweep from 10 Hz up to 1000 MHz with 4001 points per decade and a Cartesian diagram for the result.

Set the x-axis of the diagram to „logarithmic“ and you should get this presentation:

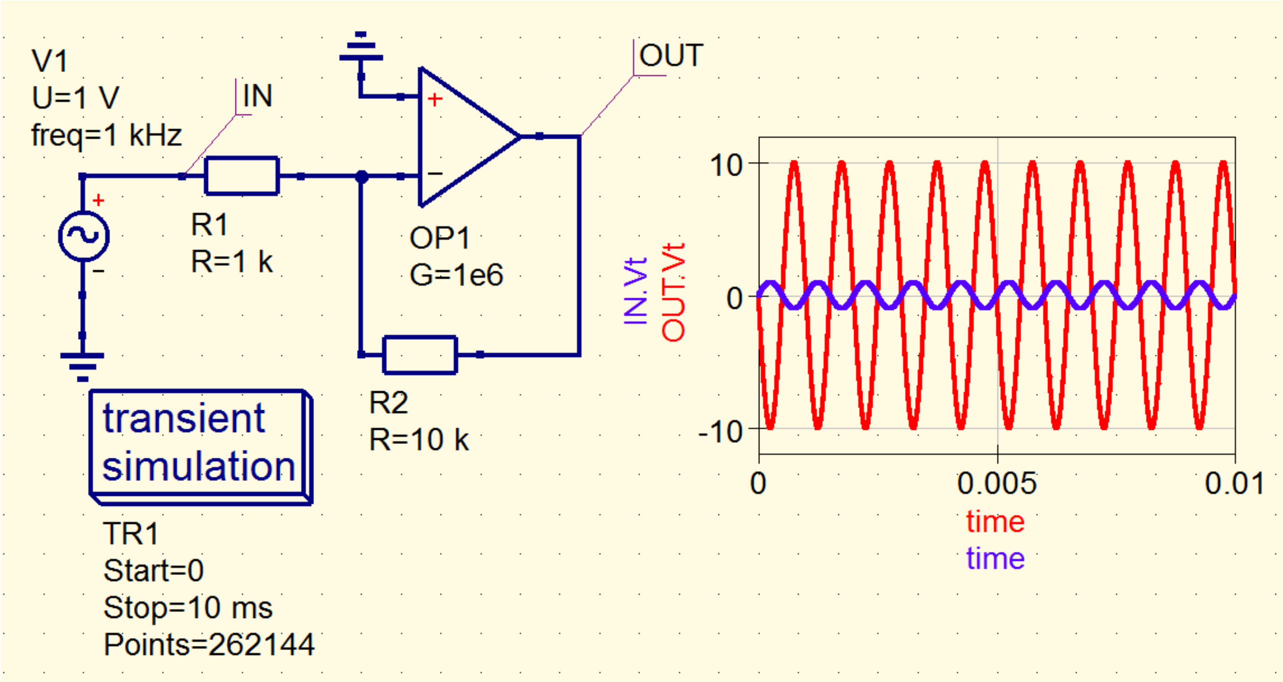


Transfer_dB
8
7
6
5
4
10
100
1e3
1e4
1e5
1e6
1e7
1e8
1e9
acfrequency

# 7. Operational Amplifiers

## 7.1. Inverting Amplifier

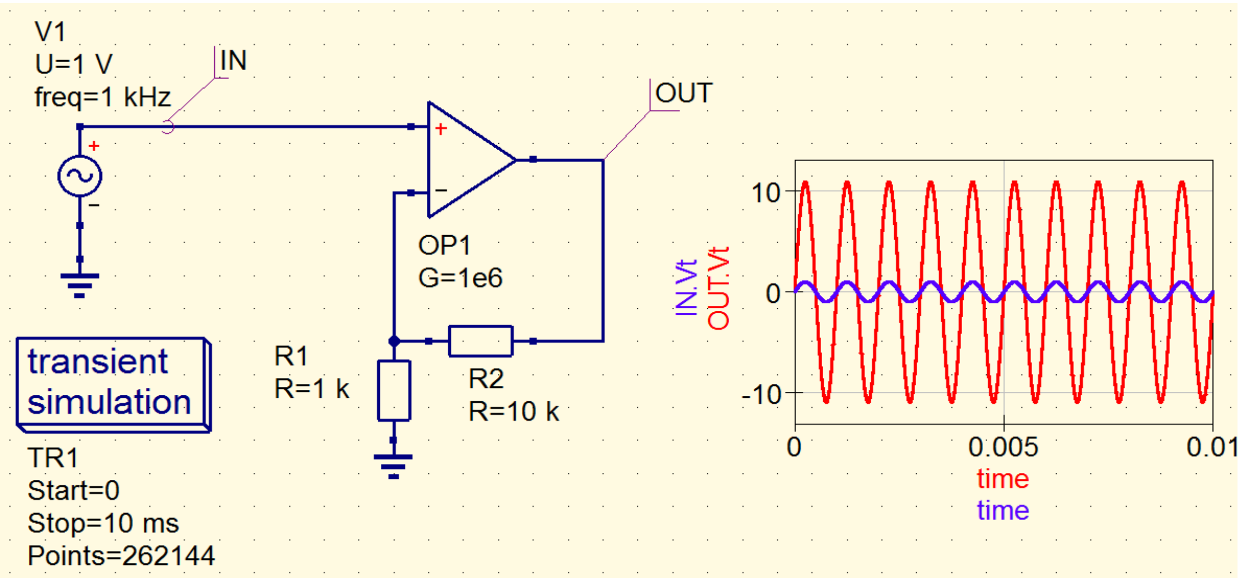
No problem – you find the ideal operational amplifier as „Opamp“ in „components / system components“.




The gain of this circuit is **G = -(R2 / R1)**

With R1 = 1k and R2 = 10k you get a gain of 10 including a phase shift of 180 degrees. For an input voltage of 1V you find an output voltage of -10V.

## 7.2. Non Inverting Amplifier




Gain is now **G = +(R1 + R2) / R1 = 11**. Input voltage = 1V and Output voltage = 11V are in phase.

## 7.3. The Analog Adder

Analog signals can be added with the Inverting Amplifier. In this circuit the different input currents (here: current in R1 and in R2) are added in the Resistor R3. The voltage drop

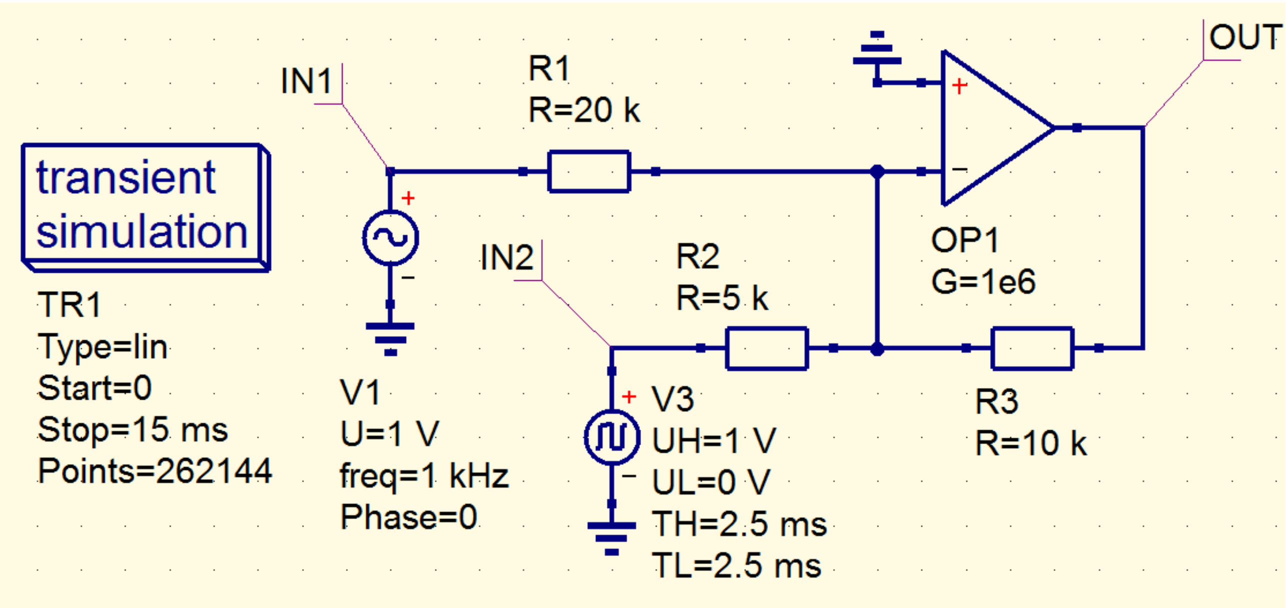
**UR3 = (IR1 + IR2) * R3**

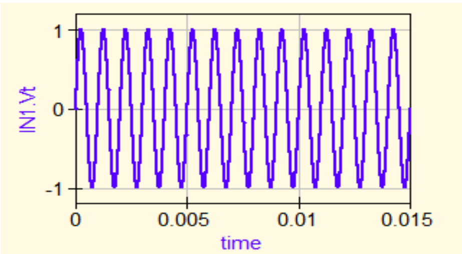
gives the (inverted) output voltage **Uout = -UR3**

Additionally you can use a different gain for every input voltage:

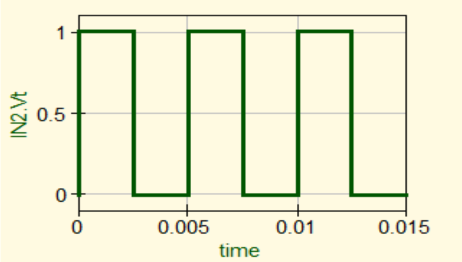
**For V1 (sine voltage) : gain = R3 / R1 = 10k / 20k = 0.5**

**For V2(pulse voltage) : gain = R3 / R2 = 10k / 5k = 2**

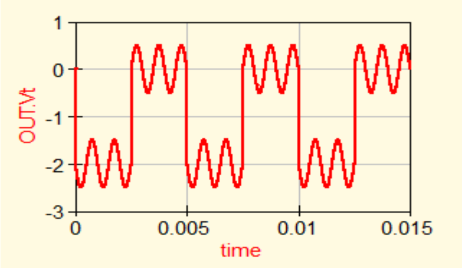



This is the input sine wave V1 with a peak value of 1 V and a frequency of 1 kHz




The pulse voltage V2 has a minimum value of 0 V and a maximum value of 1 V. Frequency is f = 50 Hz




This is the (inverted) output voltage.

Please check the different gain values for the two input signals....

## 7.4. Importing an OPA Spice Model from the Internet

This is practice – not every development task can be solved by the usage of an ideal OPA! In the meantime you get OPSs working up to 500 MHz.....

**Step 1:**

We look for the zipped SPICE model auf the TL072, manufactured by Texas Instruments (= TI). Uip it put and save it. So it does look like:

```
* TL072 OPERATIONAL AMPLIFIER "MACROMODEL" SUBCIRCUIT
* CREATED USING PARTS RELEASE 4.01 ON 06/16/89 AT 13:08
* (REV N/A)      SUPPLY VOLTAGE: +/-15V
* CONNECTIONS:   NON-INVERTING INPUT
*                | INVERTING INPUT
*                | | POSITIVE POWER SUPPLY
*                | | | NEGATIVE POWER SUPPLY
*                | | | | OUTPUT
*                | | | | |
.SUBCKT TL072    1 2 3 4 5
*
  C1   11 12 3.498E-12
  C2    6  7 15.00E-12
  DC    5 53 DX
  DE   54  5 DX
  DLP  90 91 DX
  DLN  92 90 DX
  DP    4  3 DX
  EGND 99  0 POLY(2) (3,0) (4,0) 0 .5 .5
  FB    7 99 POLY(5) VB VC VE VLP VLN 0 4.715E6 -5E6 5E6 5E6 -5E6
  GA    6  0 11 12 282.8E-6
  GCM   0  6 10 99 8.942E-9
  ISS   3 10 DC 195.0E-6
  HLIM 90  0 VLIM 1K
  J1   11  2 10 JX
  J2   12  1 10 JX
  R2    6  9 100.0E3
  RD1   4 11 3.536E3
  RD2   4 12 3.536E3
  RO1   8  5 150
  RO2   7 99 150
  RP    3  4 2.143E3
  RSS  10 99 1.026E6
  VB    9  0 DC 0
  VC    3 53 DC 2.200
  VE   54  4 DC 2.200
  VLIM  7  8 DC 0
  VLP  91  0 DC 25
  VLN   0 92 DC 25
.MODEL DX D(IS=800.0E-18)
.MODEL JX PJF(IS=15.00E-12 BETA=270.1E-6 VTO=-1)
.ENDS
```

**But you must not use this file in this manner for your qucsStudio work! So open it with your text editor and start your work to avoid nasty comments of your program.**

You must

a) **delete every line which starts with a „star"** (*)

b) Every line in the model description starts with an „empty space". That is not good – **please delete them all.**

c) When the work is done. Save the file as „**TL072.cir**" in your actual project folder (and pay attention that the editor is set to ANSI and „All files"...

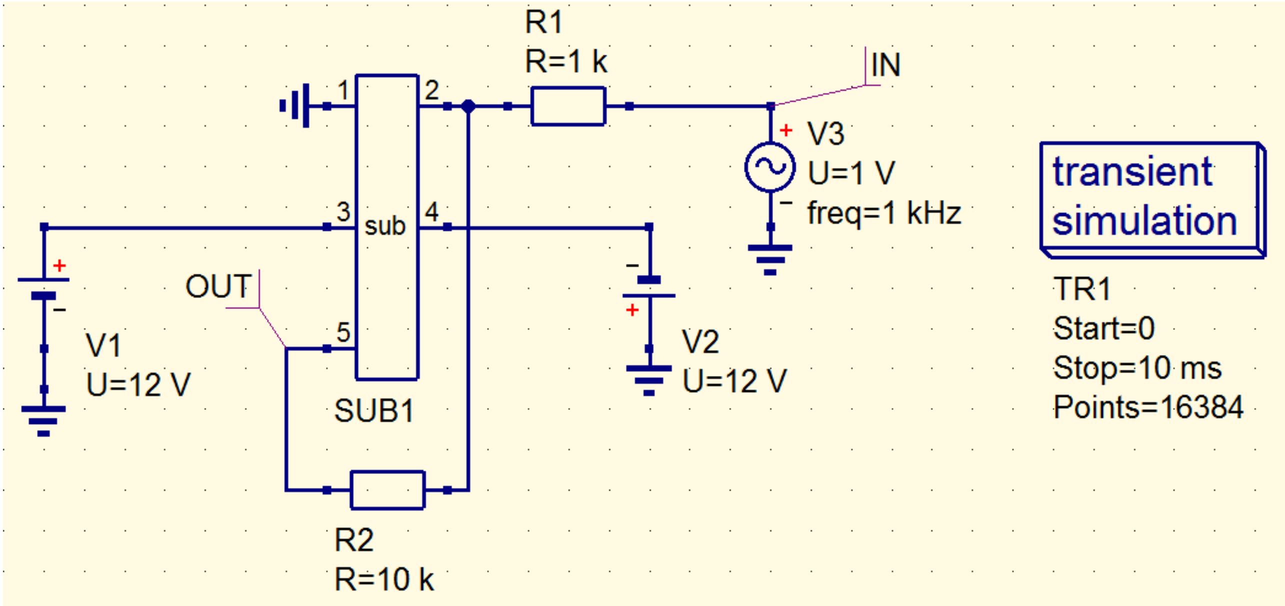
This is the goal:

```
.SUBCKT TL072     1 2 3 4 5
C1    11 12 3.498E-12
C2     6  7 15.00E-12
DC     5 53 DX
DE    54  5 DX
DLP   90 91 DX
DLN   92 90 DX
DP     4  3 DX
EGND 99   0 POLY(2) (3,0) (4,0) 0 .5 .5
FB     7 99 POLY(5) VB VC VE VLP VLN 0 4.715E6 -5E6 5E6 5E6 -5E6
GA     6  0 11 12 282.8E-6
GCM    0  6 10 99 8.942E-9
ISS    3 10 DC 195.0E-6
HLIM  90  0 VLIM 1K
J1    11  2 10 JX
J2    12  1 10 JX
R2     6  9 100.0E3
RD1    4 11 3.536E3
RD2    4 12 3.536E3
RO1    8  5 150
RO2    7 99 150
RP     3  4 2.143E3
RSS   10 99 1.026E6
VB     9  0 DC 0
VC     3 53 DC 2.200
VE    54  4 DC 2.200
VLIM   7  8 DC 0
VLP   91  0 DC 25
VLN    0 92 DC 25
.MODEL DX D(IS=800.0E-18)
.MODEL JX PJF(IS=15.00E-12 BETA=270.1E-6 VTO=-1)
.ENDS
```

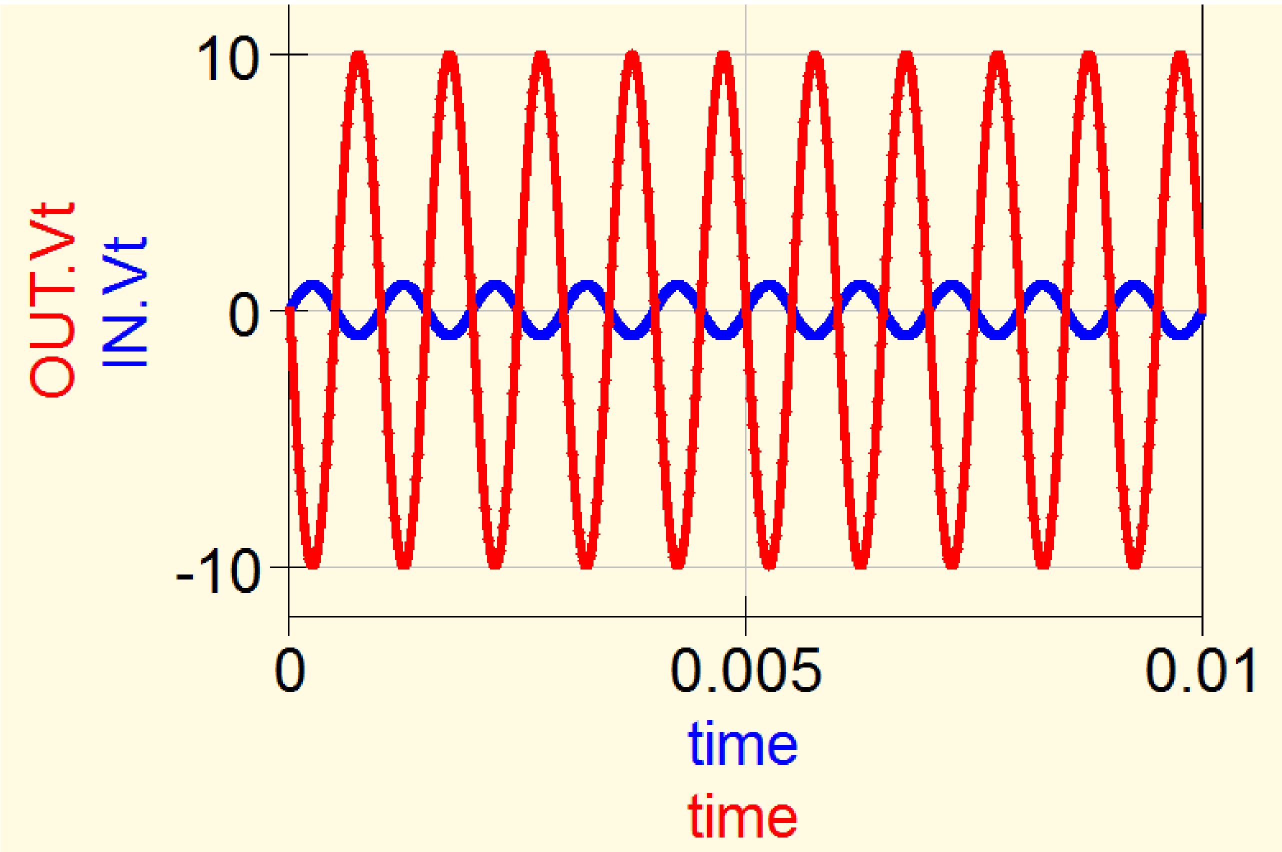
**Step 2:**
Start the program and change to your OPA project. Open the „Project" menu and go to „Import Data". Set the correct path to the „TL072.cir" file and start the conversion.

**Step 3:**
You get a new schematic file named „**TL072.sch 5-Port**". Open it and look for the „Sub-circuit" symbol with 5 pins (...normally hanging on the mouse cursor after one or two seconds when clicking on „TL072.sch" in the project list).

Place it in a new schematic and **draw an inverting amplifier with an input resistance of 1kΩ and a gain of 10.**




**The input voltage is a sine wave (peak value = 1 V, frequency = 1kHz). Use 2 supply voltages (+12V and -12V). Simulate for 0....10 ms using only 16384 points (to reduce the calculation time which has increased due to the complicated OPA Spice model).**



## 7.5. Active Filter using OPAs

Active filters use OPAs and are widely used – not only in the audio range! These are the advantages:

- -- no coils needed – only R and C
- -- much cheaper - coils are expensive...
- -- very small PCBs when using R and C and OPA as SMD versons
- -- Gain is adjustable in a wide range
- -- using negative feedback reduces distortions
- -- in the meantime usable up to more than 100 MHz

**To design such an active filter circuit use an online calculator in the Internet (e. G. from Burr Brown or Microchip or Texas Instruments).**

| **Example: Sallen Key LPF with order = 4** |
|---|

Search for an online calculator and enter the following properties:

**Filter:** **Low Pass Filter**

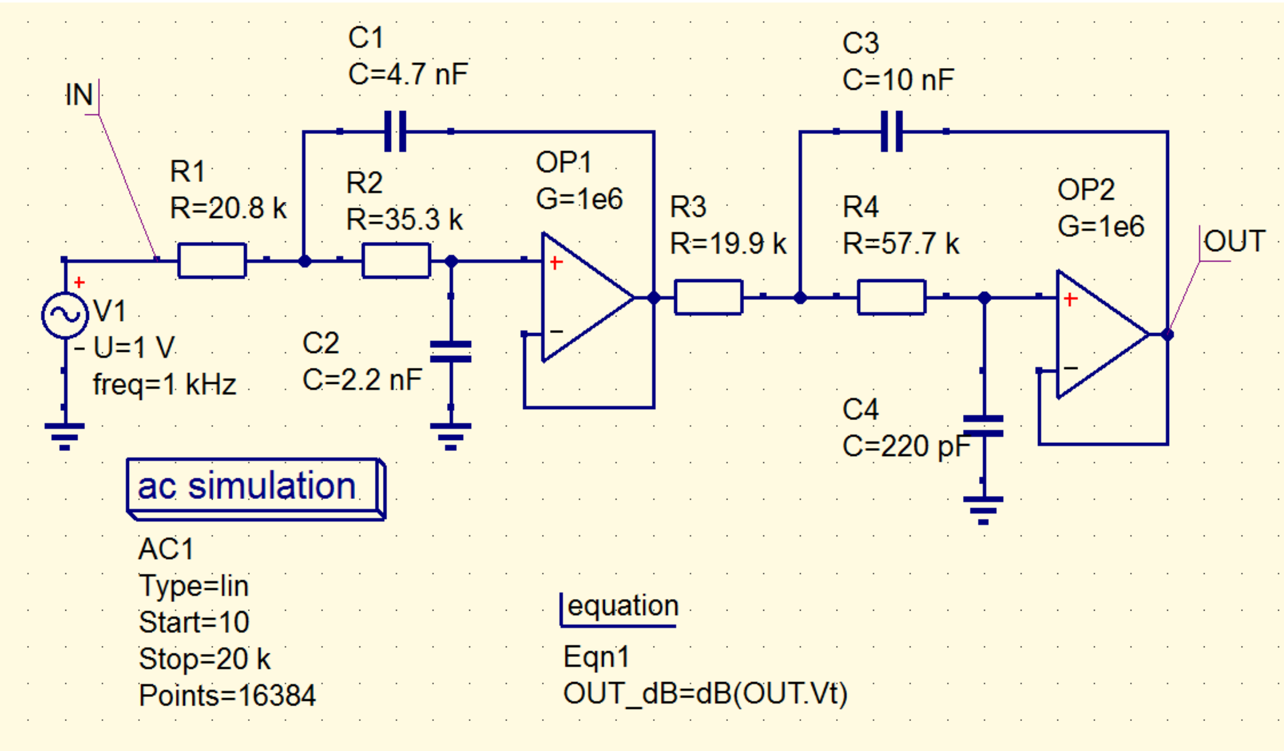
**Filter Type:** **Chebychev**

**Passband Corner Frequency:** **3400 Hz**

**Passband Ripple of the attenuation** **0.5 dB**

**Filter Order:** **N = 4**
(Determines the attenuation gradient when leaving the pass band.

This is the schematic with ideal OPAs.
Use an AC sweep from 1o Hz to 20 kHz. Do not forget the equation to calculate the output voltage in dB.



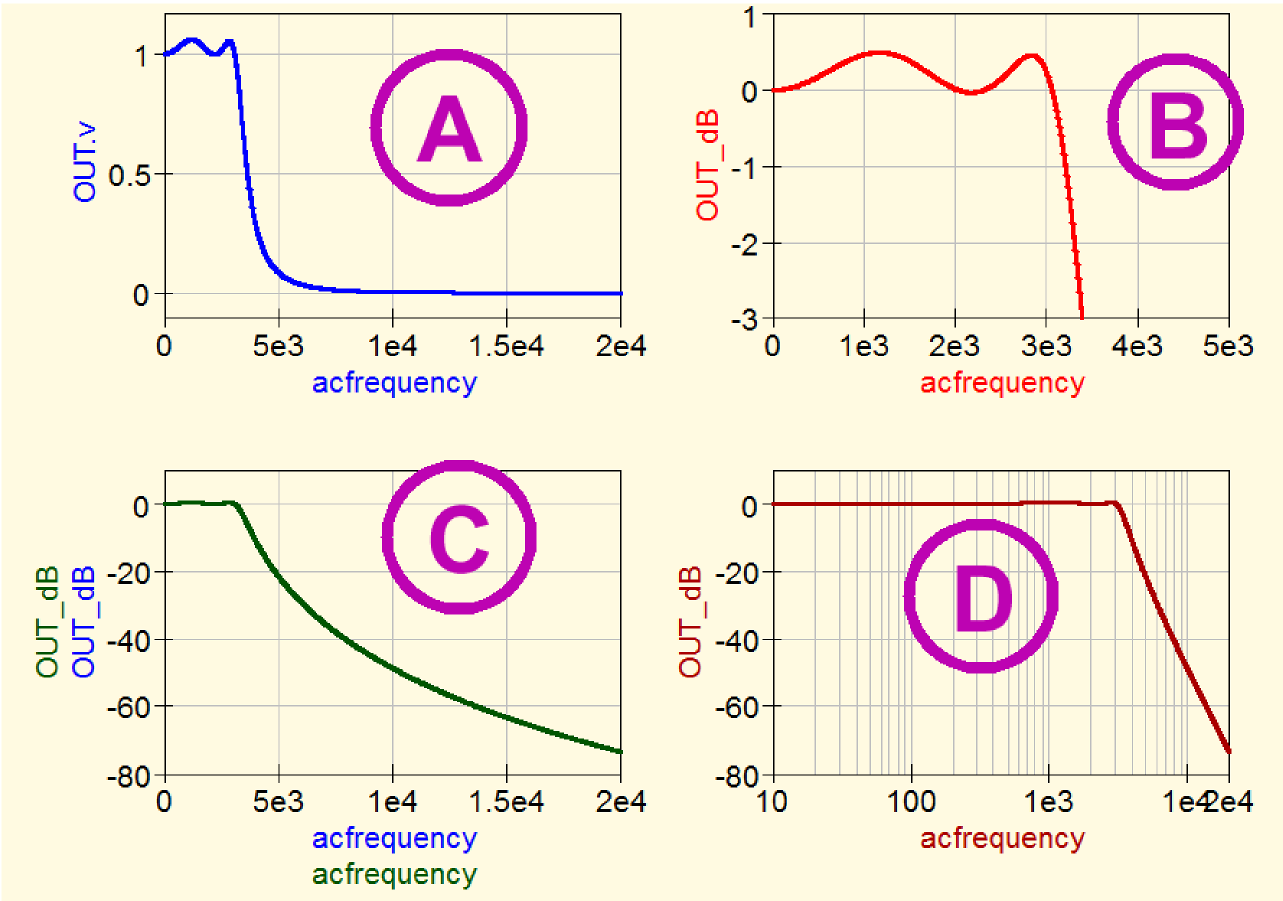
Present the result for OUT.Vt in four different manners:

A) Both axis linear. X-axis up to 20 000 Hz

B) Output in dB from -3 to +1 dB. x-axis linear up to 5 kHz.

C) Output in dB, scaling from -80 to +10 dB. Frequency axis linear up to 20 000 Hz.

D) Output in dB, scaling from -80 to +10 dB. Logarithmic frequency axis from 10 Hz to 20 000 Hz.

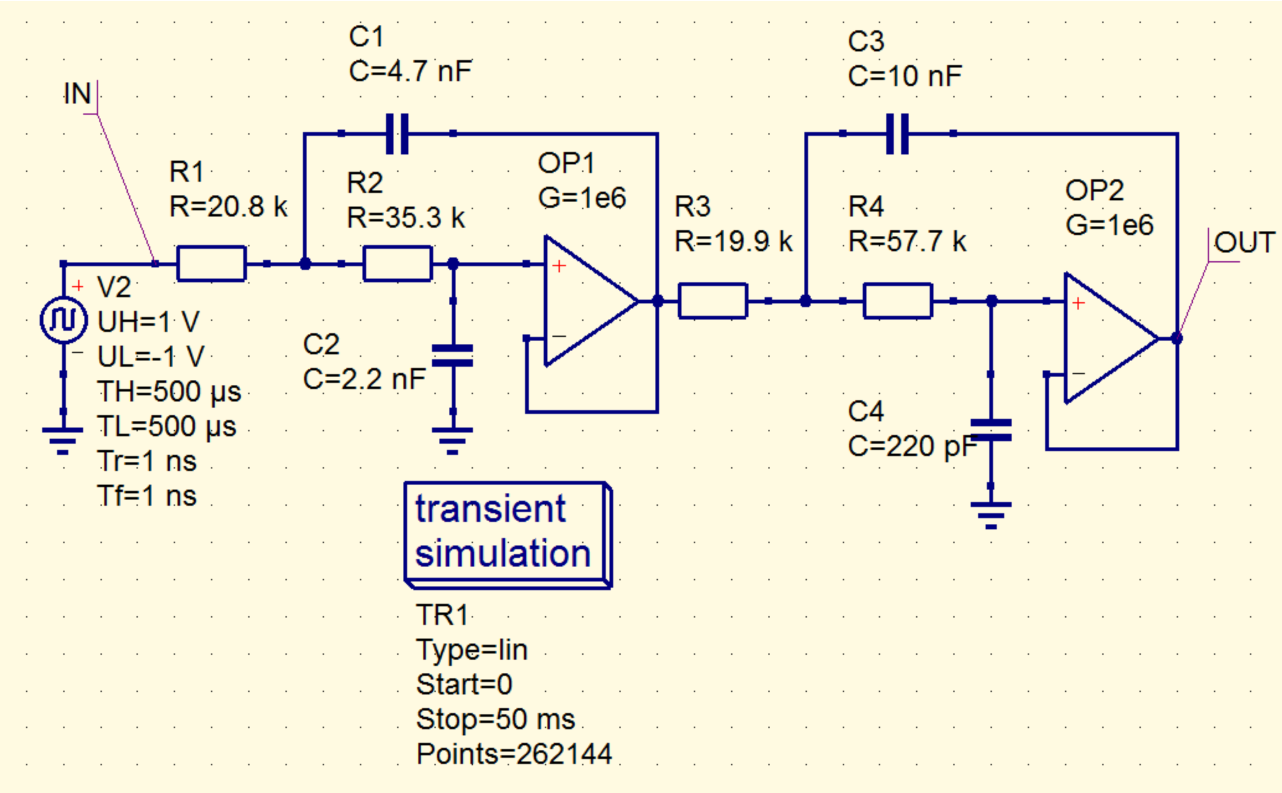


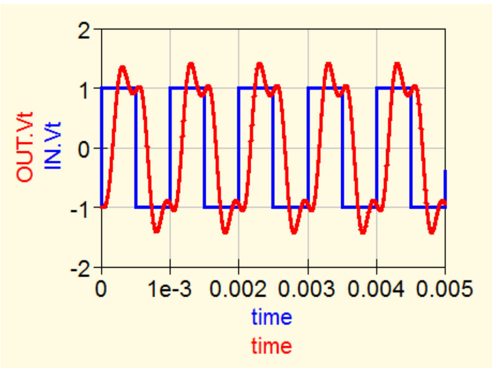

Now we want to have a look at the filtering in the Time Domain (= Transient simulation).

Please use a rectangular voltage with the following properties to feed the input<.

| | |
|---|---|
| **Frequency f =** | **1kHz** |
| **Minimum voltage value =** | **-1V** |
| **Maximum voltage value =** | **+1V** |
| **HIGH time =** | **500µs** |
| **LOW time =** | **500µs** |
| **rise resp. fall time** | **1ns** |

Simulate from 0 to 50 ms and use 262144 points (for a good FFT quality).
A simulation time of 50 ms gives a frequency resolution of **b = 1 / 50 ms = 20 Hz.**

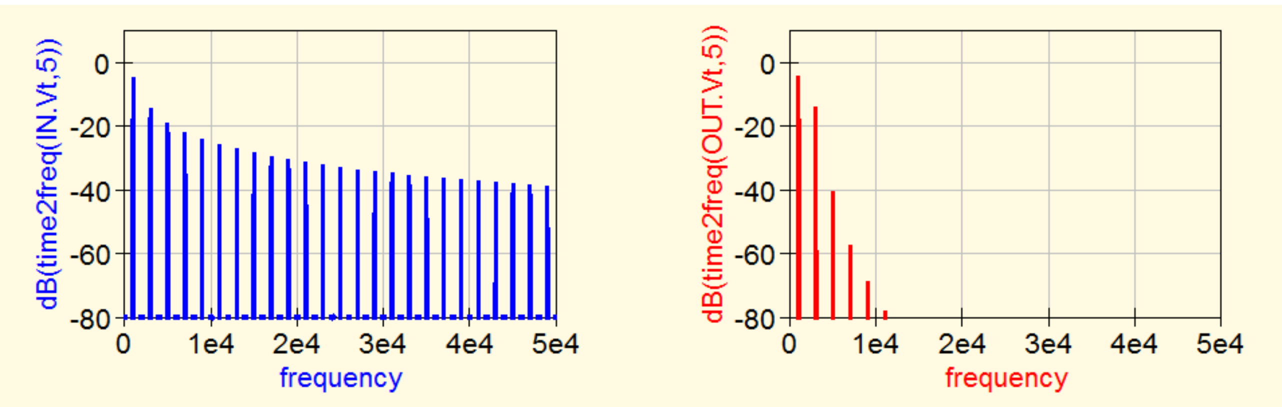



Here you can see the input and the output voltage for a time range from 0....5 ms.

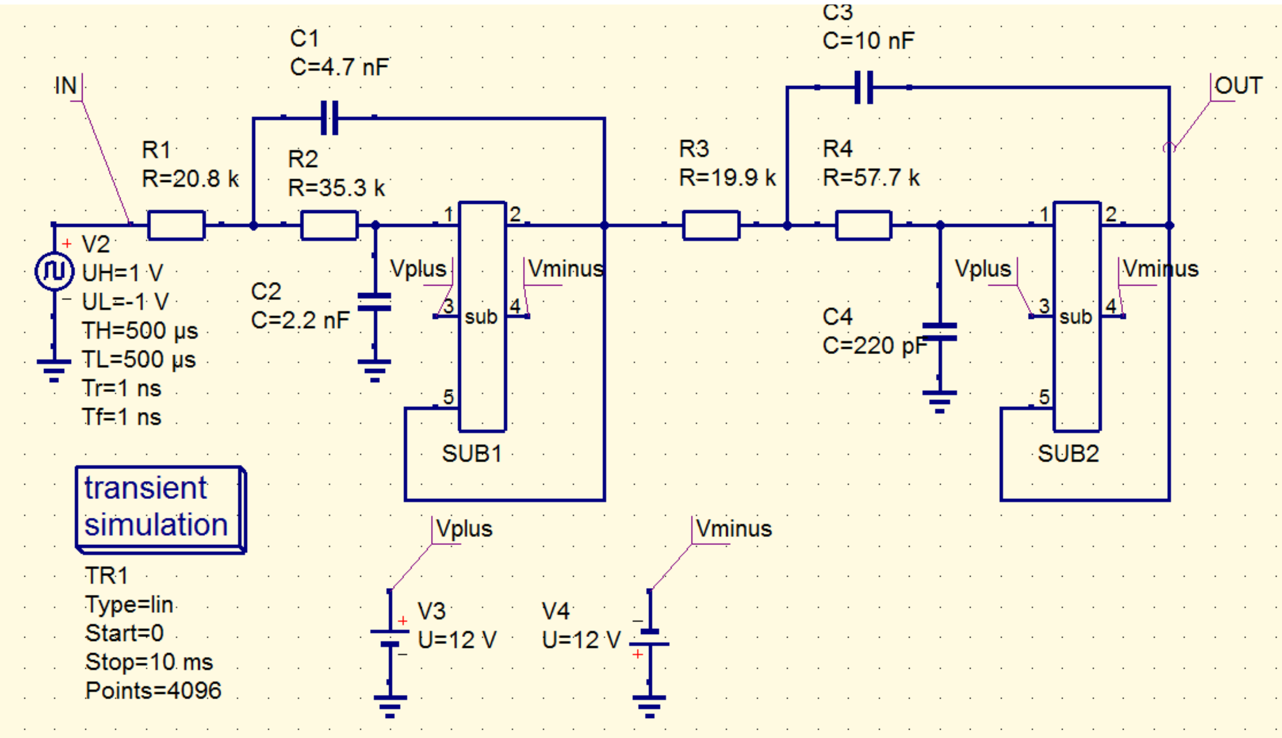
But the filtering result is much better examined in the Frequency Domain using a dB scaling.

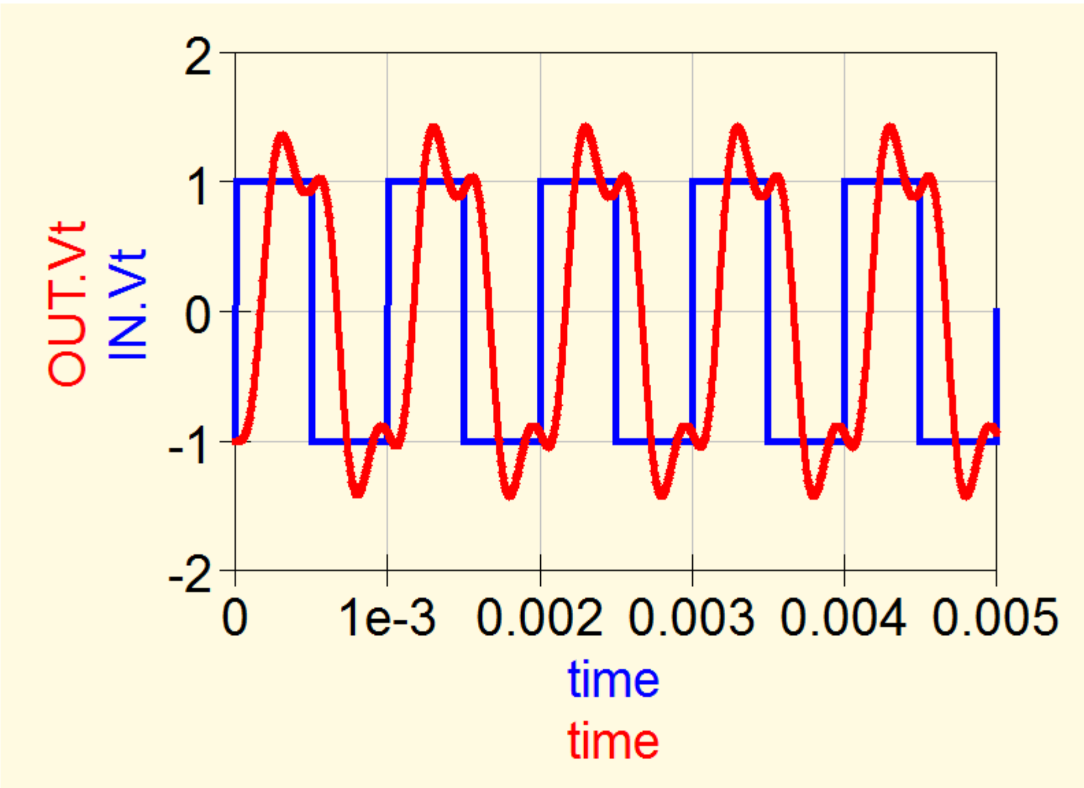
Use a FFT with Blackmann Windowing (index = 5) and show the frequency range from 0....50 kHz for the filtering result:



**And now:**
**the same procedure once more using the TL072 Spice model.**
(See chapter 6.4)

**Caution:**
**Please reduce the number of points to 4096 due to the increased calculation time (caused by the complicated converted TL072 model)!**




No surprise....

Some tips for the practical realization

a) You must not use resistor values greater than 1 MΩ when designing the filter schematic. This gives stability problems and increases noise levels.

b) Do not use capacitor values below 10 pF or greater than 1µF, but look for low losses and small tolerances. Use mica, polystyrene or polypropylene types.

c) Never use electrolytic capacitors for filter applications (= low Q, high tolerances).

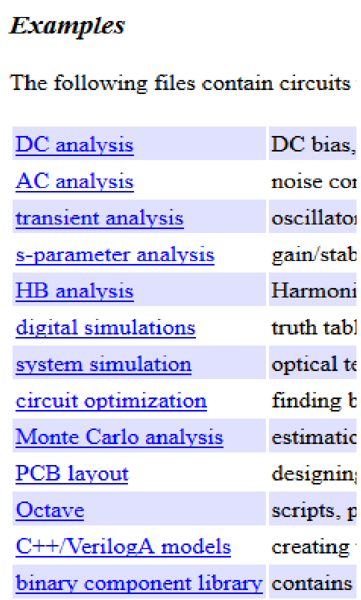
Also an other problem seems not to be well known:
At the output of every OPA in the circuit a feedback capacitor is connected. But a capacitor has a 90 degree phase shift between current and voltage - and this is a really problem for some OPA types, because most oft them use push pull versions to save quiescent current. Due to this fact (= a huge current demanded at a moment with a very small input voltage at the output stage) a piece of the output voltage is missing around the „zero voltage crossing moment". This effect is essential for capacitor values greater than 100 nF.

Curing this effect is very simple.
Connect an additional resistor with **1k….3kΩ from the OPA output to ground. This compensates the „capacitor phase shift effect" because the total phase shift is now below 90 degrees.**

# 8. Simulation of Digital Circuits

## 8.1. Examples coming from the qucsstudio Homepage

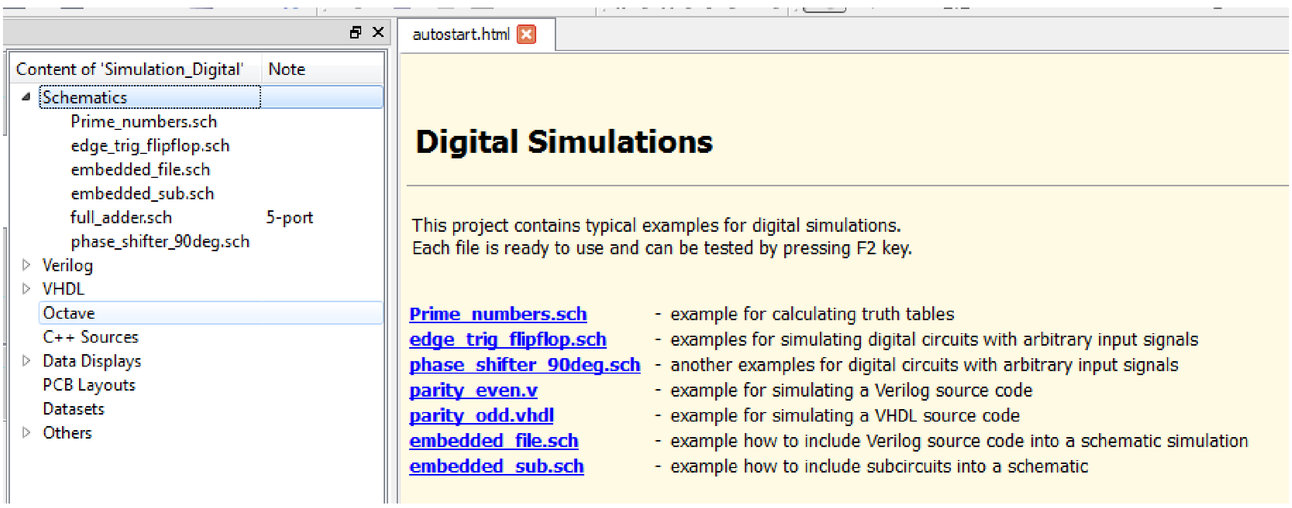
**Step 1:**




You find this list of examples in the qucsstudio homepage. Download them all and save them in your „**qucs**“ **project folder**. They come as **„*.qucs“ files“.**

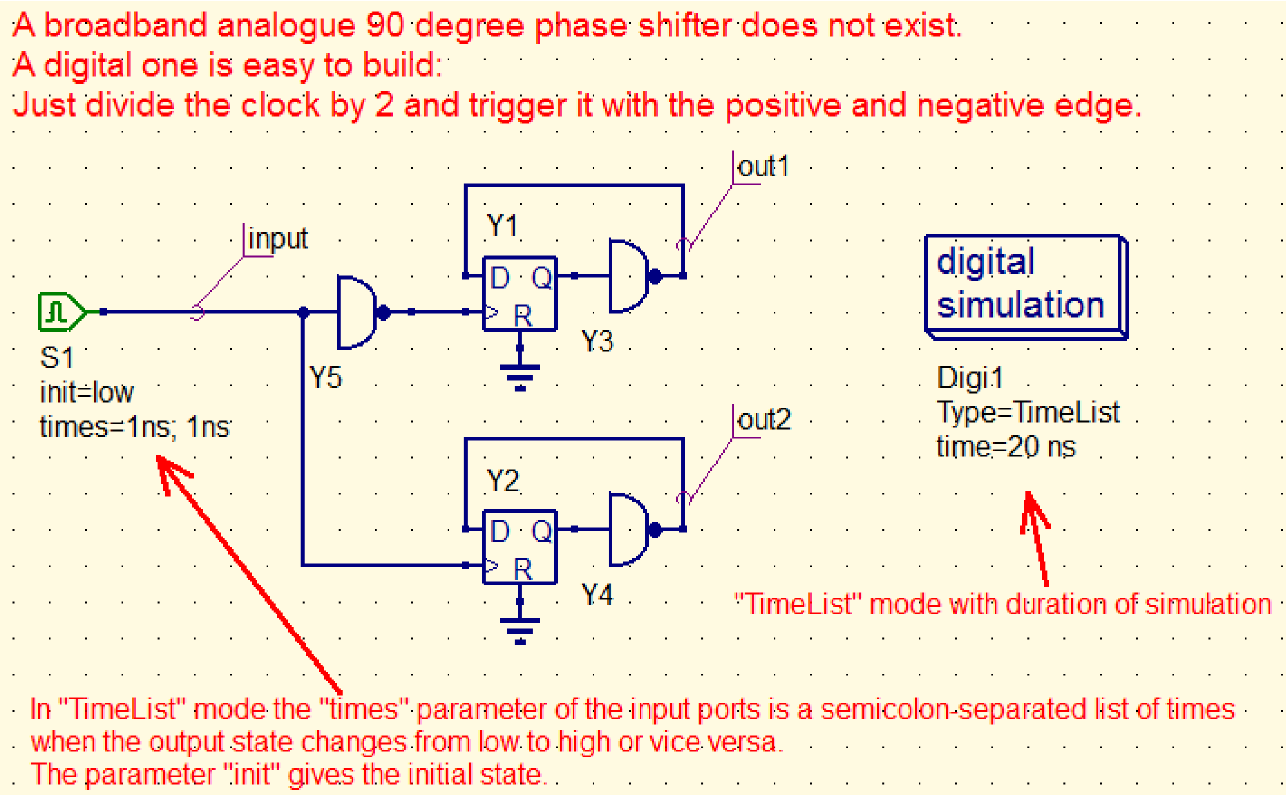
**Step 2:**
Now start qucsstudio and open the „**projects**“ menu (= left vertical side of your screen).
Change into „***project / extract package***“ and choose „**digital simulations**“. The extraction result is a new schematic file named „**Simulations_Digital.sch**“. Please open it.

**Step 3**:
Here we see what we want:




Let us use the file „**phase_shifter_90deg.sch**“ and open it.

## 8.2. The example „phase_shifter_90deg.sch“




You find all necessary parts in „**components / digital components**“:

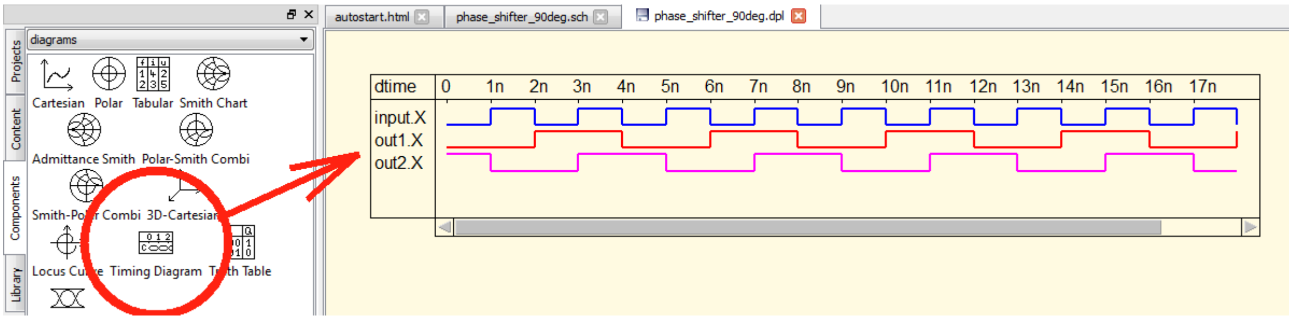
a) „S1“ as a source for digital signals

b) „Y3“, „Y4“ and „Y5“ as inverters

c) „Y1“ and „Y2“ as D-flipflops

**Please read now the information given to the source and the digital simulation.**

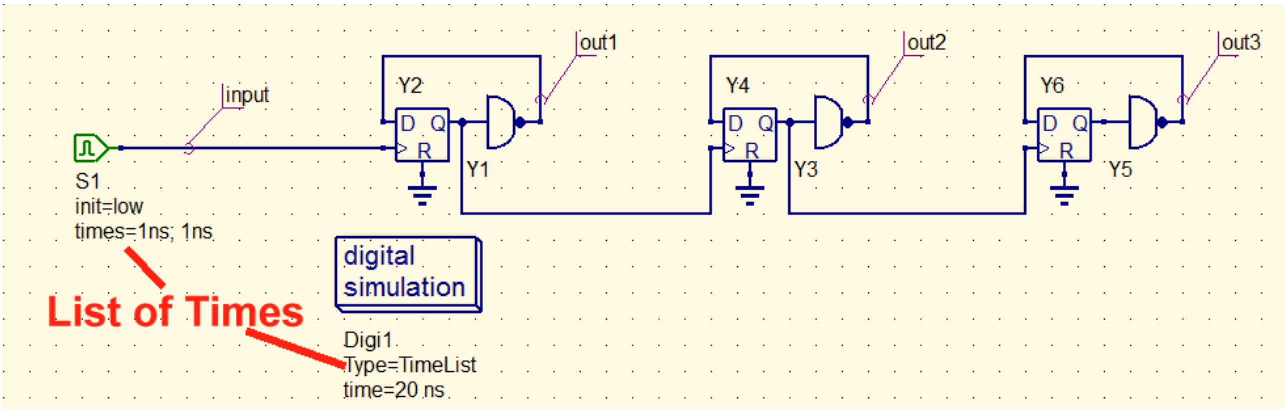
Then simulate and a timing diagram opens automatically. Edit the properties and set the line width to „2“




You find the used riming diagram in the left red circle.I

## 8.3. A Three Stage Frequency Divider using D Flip flops

Sinply make a copy of the last schematic („phase shifter) and delete one flipflop symbol. Then make two copies of the last flipflop and past them in a straight line with the remaining first. Delete all text and draw the following schematic. Save the new schematic as „frequency_divider.sch". Do not forget to set new labels.

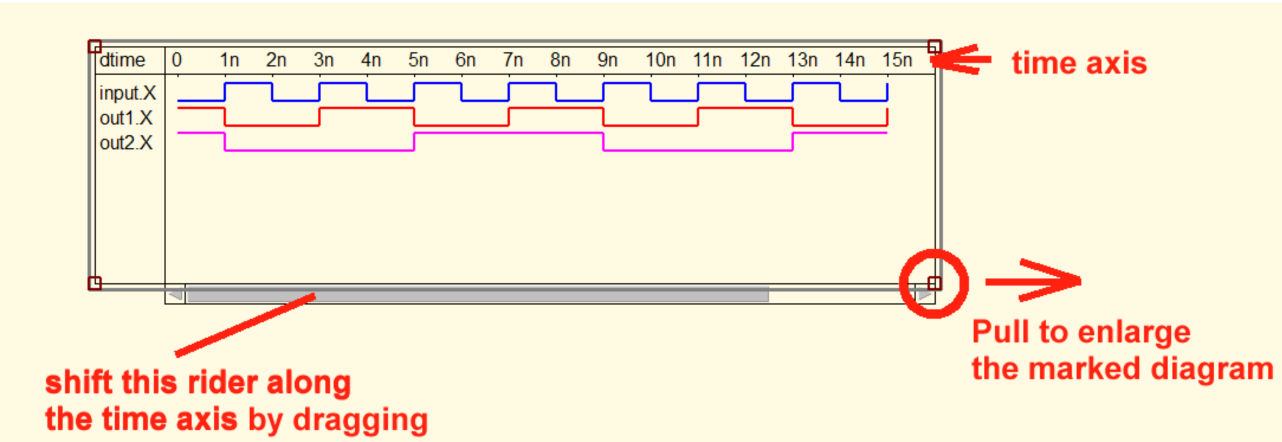



„**List of times**" is not complicated to understand:
The first value is the **delay time for the start**
Separated by a semicolon follows the „**system tick**" = the time interval between two changes of the input level

**Simulation time is 20 ns.**

„**init = low**" means that we start with „LOW" level at the input.

This is the simulation result and some information are necessary:



# 9. Working with S Parameters

## 9.1. Fundamentals

For a complete communication system you need a lot of components like transmitters, receivers, filters, couplers, amplifiers, antennas..... Thus the principle of power matching is important to

**a) transfer the maximum of power from one stage to the next**

**b) thus to guaranty the maximum of signal to noise ratio and quality**

This is guarantied by the application of the **same characteristic impedance in the complete system**. Normally

**Z = 50Ω**

is used.

Every stage and component is now described by **S Parameters (= scattering parameters).**
**Let us begin with the input side.**
Connect a signal generator (internal impedance = 50Ω) to the input (= Port 1) of a component using an RG58 cable with Z = 50Ω. The output of the component (= Port 2) is terminated by a load resistor with Z = 50Ω. Now the maximum available power travels as „**Incident Wave**“ (voltage amplitude $U_{incident}$ = Uo / 2) from the generator output to the components input. If now the input impedance differs from Z = 50 Ω you don't have the perfect match and so an echo (= reflected wave = surplus energy) travels back on the cable to the generators output jack.
Now calculate the ratio of reflected wave voltage amplitude to the incident wave voltage amplitude and you get the **Input Reflection S11** (...which will often be given as **20*log(|S11|) in dB**).

**S11** is an abbreviation and means:

**S** = „Scattering Parameter
**1** = reflected signal measured at Port 1
**1** = the incident wave travels from the generator in direction to Port 1 with an amplitude of $U_{incident}$ = Uo/2

Attention:

| |
|---|
| **S11 = input reflection = informs about the difference between the system impedance and the input impedance of the component. (…if the impedance does not differ a lot form Z = 50 Ω you get a very small reflection and so a low negative dB value for \|S11\|)** |

Please remember:

a) **Perfect match means R = Z = 50 Ω and in this case no echo will be found. This gives an |S11| magnitude of „zero“ = „minus infinite dB“**

b) **Open circuit or short circuit as input impedance causes a total reflection of the incident wave** because no power is consumed by the input. **This gives S11 = +1 (for an open circuit) and S11 = -1 (for a short circuit).**
**The magnitude of |S11| will in this case be „zero dB“**

S11 is the only available parameter for „**Oneports**" (like antennas, resistors, diodes, capacitors etc)
A „**Twoport**" (= amplifier, filter attenuator, coupler...) has an output port (= Port 2) and so an output signal (caused by the incident wave at Port 1) exists. So another S Parameter named „S21" is defined and named „**Forward Transmission".**

**S21 = „Forward Transmission" means the power gain of the stage in dB while the output (Port 2) is terminated by Z = 50Ω. The output voltage amplitde is measured, related to the incident wave's voltage amplitude (arriving at the input = Port 1). The ratio is "S21" and |S21| can again be expressed in dB.**

After this the stage or component is „**turned by 180 degrees**". The generator cable is now connected to the output (Port 2) and produces an incident traveling from the generator to Port 2.
Port 1 must now be terminated by **Z = 50 Ω.** In this case you can define the last two S Parameters:

**S22 = output reflection = tells you whether the output resistance differs from the characteristic system impedance of Z = 50 Ω. Then you get an echo which does not exist for perfect power matching ( Z = R = 50 Ω).**
**For perfect output matching the magnitude of S22 is „zero" and |S22| in dB shows an extremely high negative value.**

**S12 = reverse transmission regards the part of the incident power (arriving at Port 2) which can be measured at Port 1 due to internal feedback effects. This feedback voltage amplitude measured at the 50 Ω – termination at the input is related to the incident voltage amplitude at Port 2.**
**The magnitude of S12 for a good amplifier should be near to zero ( = a high negative dB value), otherwise this amplifier could self oscillate**

==============================================================================

## 9.2. Practical Example: Chebychev Low Pass Filter with a Passband Corner Frequency of 11 MHz

### 9.2.1. Fundamentals of Filtering

Filters are used to suppress signals in a defined frequency range. All other frequencies should not be attenuated. Please distinguish:

**Low Pass Filters (= only low frequencies can pass, middle and high frequencies are attenuated)**

**High Pass Filters (= only high frequencies can pass, middle and low frequencies are attenuated) (**

**Band Pass Filters (= only a small band of frequencies can pass, all other signals are attenuated)**

**Band Stop Filters (= only a small band of frequencies is attenuated, all other signals can pass).**

To design such filters you can use free „filter calculators" from the Internet. The filters are then designed as „**Two ports with a characteristic system impedance of Z = 50 Ω".**

But please distinguish the different **filter types**:

a) **Bessel type**
The most important property is „extremely low phase and group delay distortion" for the correct transmission of rectangular or pulse voltages. This causes the disadvantage of a very „lazy and tired" increase of the stop band attenuation

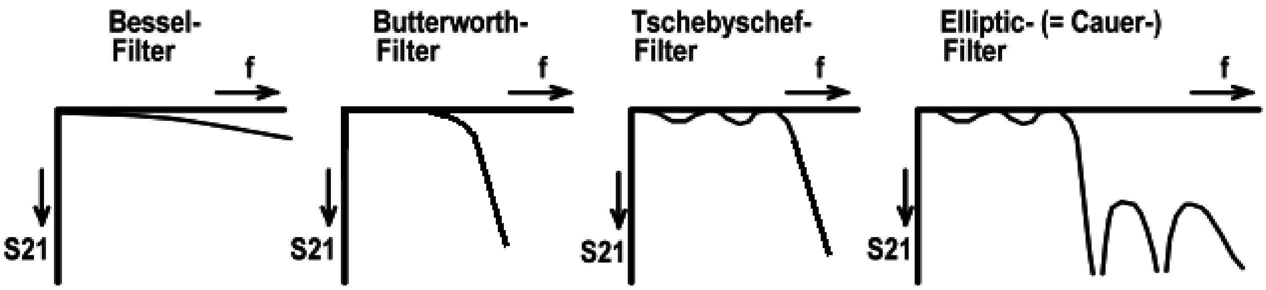
b) **Butterworth type**
The transition from the pass band to the stop band is a little bit improved and not so lazy as for a Bessel filter with the same filter order. But this decreases already the quality of the group delay in the pass band).

c) **Chebychev type**
**The change („edge“) from pass band to stop band is sharper as for Bessel or Butterworth filters. But you pay for that with a „S21 ripple in the pass band“. Thus the distortions of the group delay increase rapidly when approaching to the pass band corner (= cutoff) frequency.**
**Main application:**
Suppression of the harmonics when using a signal generator.

Stop band attenuation for this 3 filter types increases in monotonic manner (= S21 decreases monotonically) up to infinite.

d) **Elliptic filters (= Cauer type)**
Identical pass band ripple of S21 as Chebychev filters of the same filter order, but the pass band – stop band - transition is increased. Thus the stop band attenuation „comes back to a minimum value and increases then again“.

Here you have S21 for the four filter types of the same filter order.

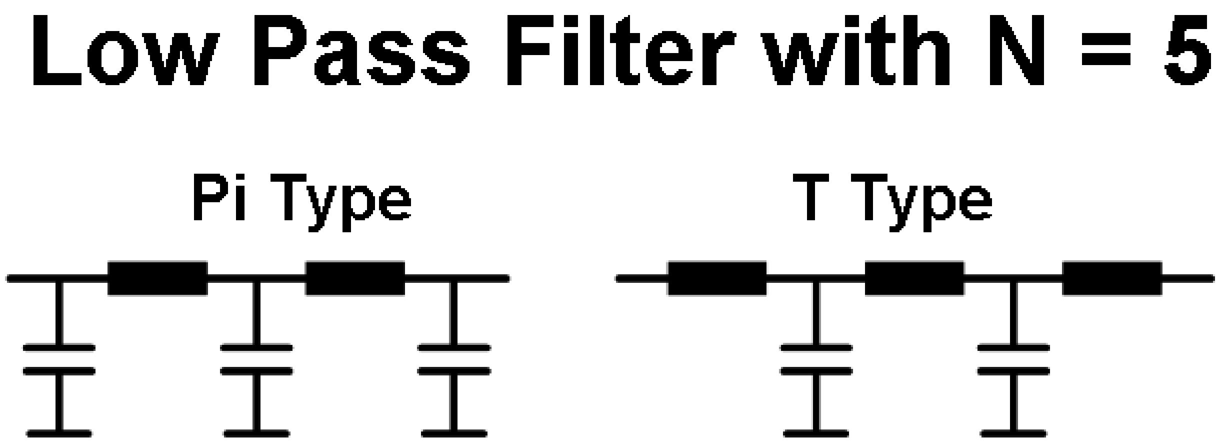



For a sharper increase of the attenuation transition from pass band to stop band you must use a **higher filter order (= degree of the polynome which describes the filters properties**).

This filter order can easily seen when regarding a low pass filter:

**The filter order „N“ equates to the number of used components!**

Let's have a look at a LPF with order N = 5. Five parts are used, but you can arrange them in **„T“ or in „π“ connection** (remember: coils are more expensive and so the „π“ connection is preferred in practice).




**….and never forget to terminate each filter side with the characteristic system impedance (Z = 50Ω)!**

An information:
At the pass band **corner (= cutoff) frequency** S21 has dropped to a value which is **3 dB reduced.**
**But for Chebychev filters and Elliptic filters sometimes another cutoff frequency version is used. This version is called „ripple corner frequency“ and at this point the attenuation is for the first time higher than the maximum ripple value.**

Also never forget:
for Band Pass Filters and Band Stop Filters two corner frequencies are existing....a lower and a higher point and the difference between them is called „bandwidth“...

## 9.2.2. LPF Specifications

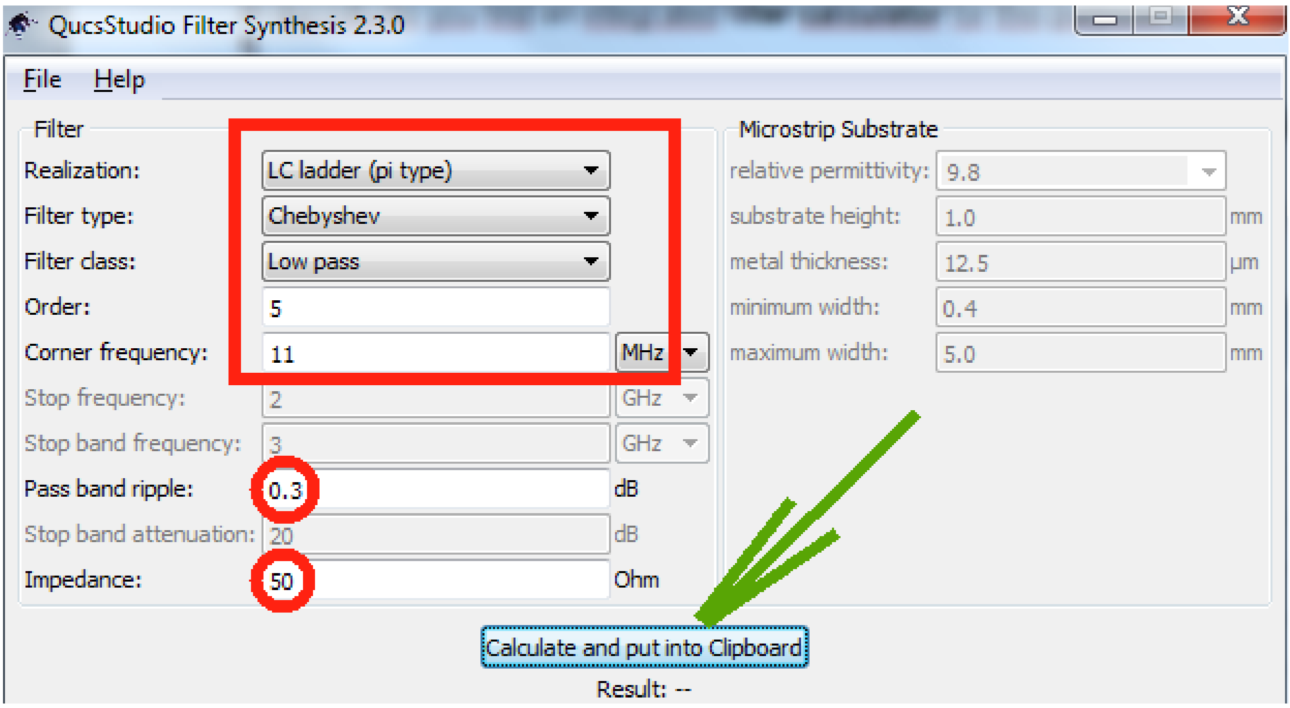
Let us design a LPF with Z = 50 Ω and N = 5 for a Communication System. This LPF shall suppress all signals higher than the FM Intermediate Frequency with f = 10.7 MHz.

| | |
|---|---|
| **Characteristic system impedance:** | **Z = 50 Ω** |
| **Filter type:** | **Chebychev** |
| **Filter order:** | **N = 5** |
| **Pass band ripple:** | **0.3 dB** |
| **3dB corner frequency:** | **11 MHz** |
| **Quality factor of the used coils:** | **Q = 70 at 11 MHz** |

In qucsstudio you find an integrated f**ilter calculator** for this purpose.

## 9.2.3. qucsstudio Filter Calculator Application and S Parameter Simulation of the LPF

Open a new page in your project. Then go to „**Filter Synthesis**“ in the „**Tools**“ menu and enter all values in the following mask:

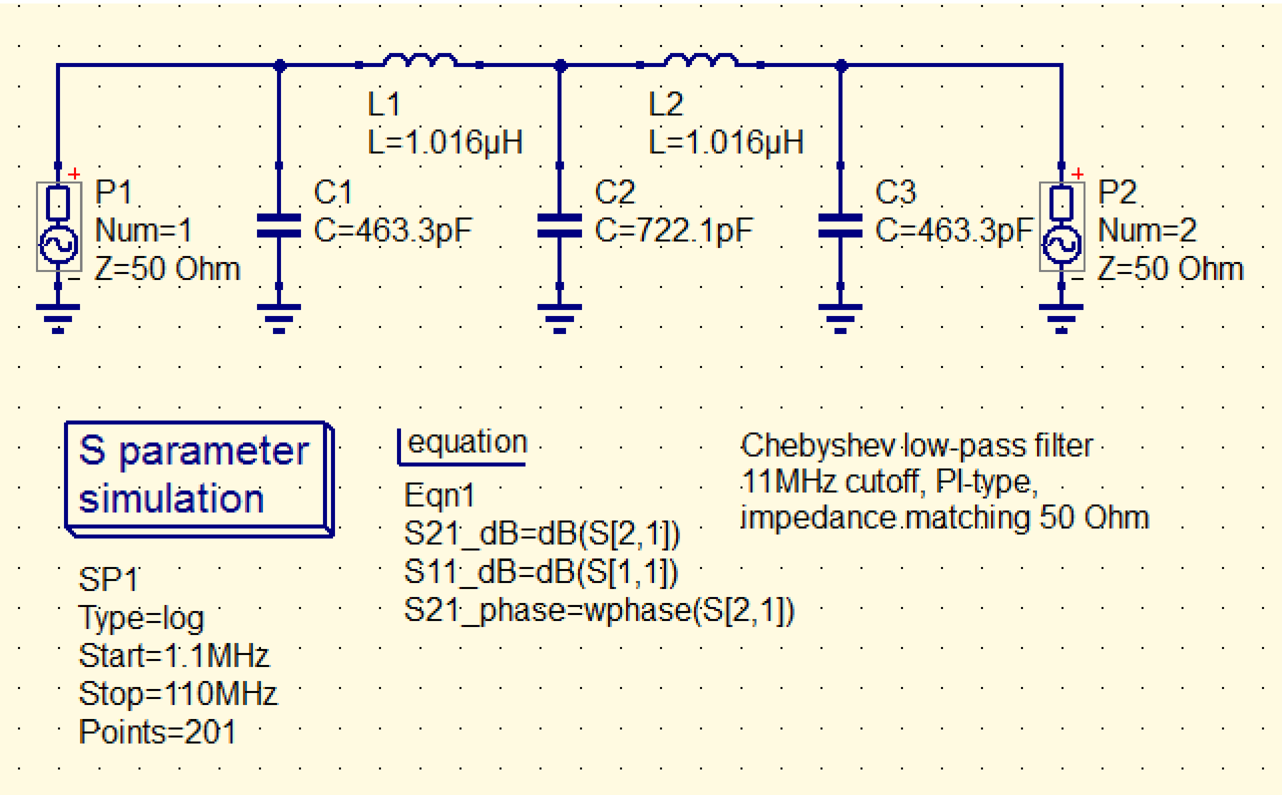


You have to enter:

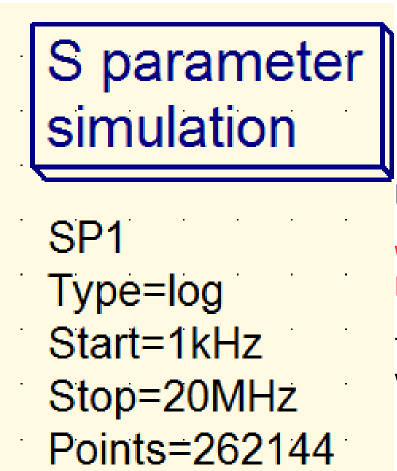
| | |
|---|---|
| **Realization** | **= LC Ladder (pi type)** |
| **Filter Type** | **= Chebyshev** |
| **Filter Class** | **= Low pass** |
| **Order** | **= 5** |
| **Corner frequency** | **= 11 MHz** |
| **Ripple** | **= 0.3 dB** |
| **Characteristic impedance** | **= 50 Ohm** |

Terminate the action by pressing the indicated button to put it into the clipboard.

**Attention:**
**The complete schematic (including all preparations for the simulation) is now stored in the clipboard!**
So copy it into a new schematic file and you will see this screen:




For your information:
The two ports come as „**power sources**" from „**components / sources**".
The equations were automatically prepared by qucsstudio.

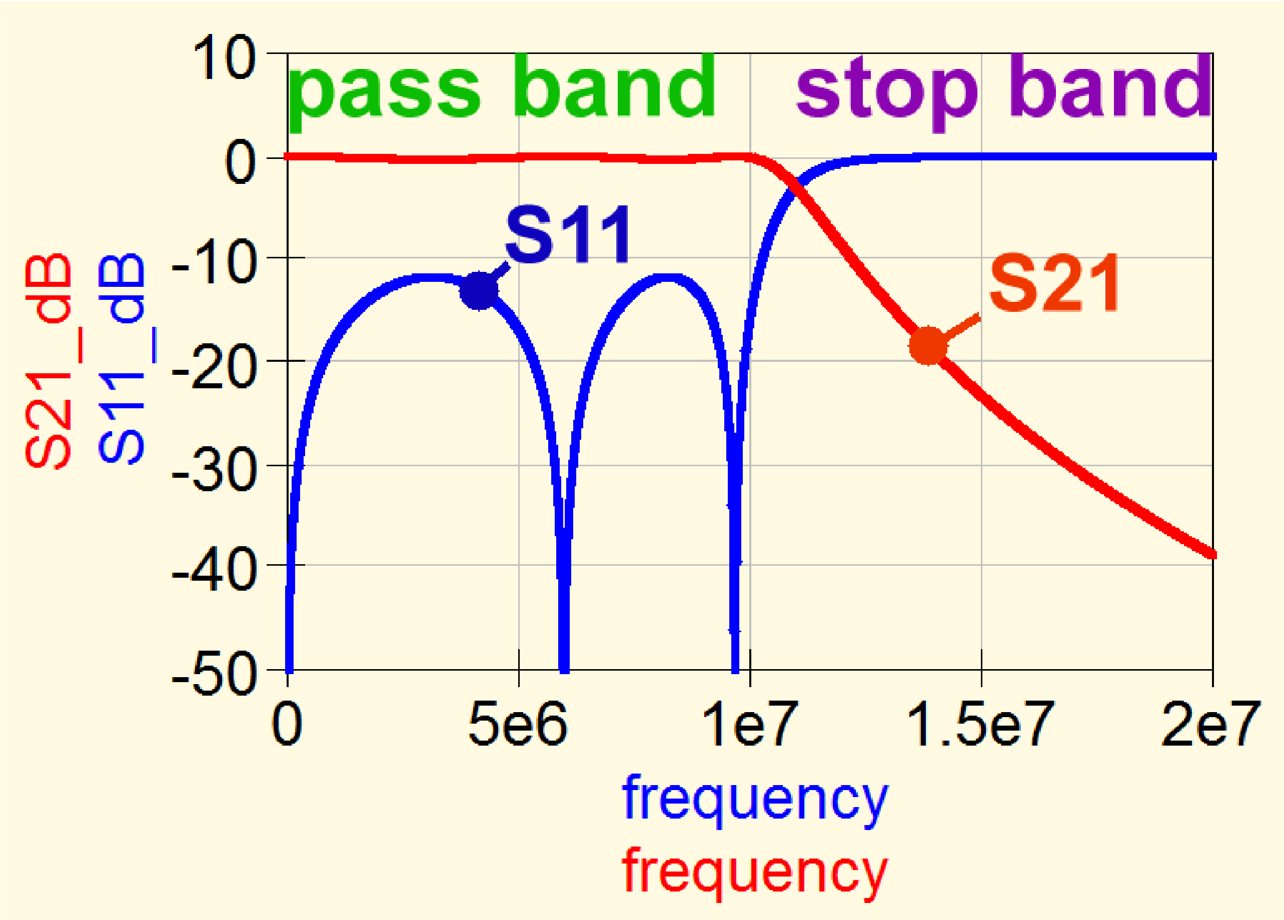



Please change the sweep settings to a **start at 1 kHz and a stop at 20 MHz.**

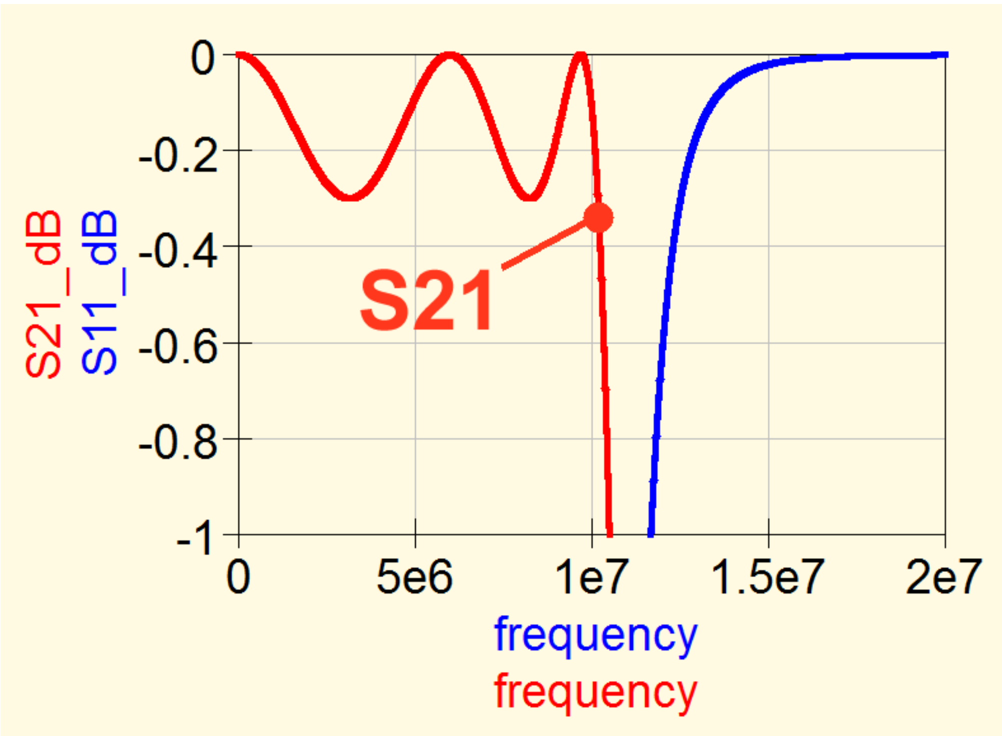
**Warning:**
**Never start at Zero! This produces hysteric messages on the screen....**

Then simulate and set the **vertical axis to a scaling from -50 ….+10 dB with a tick of 10 dB.**

That it is...




But for better information use a **vertical scaling from -1 to 0 dB with a tick of 0.2 dB** to demonstrate the **Chebychev ripple** of S21:

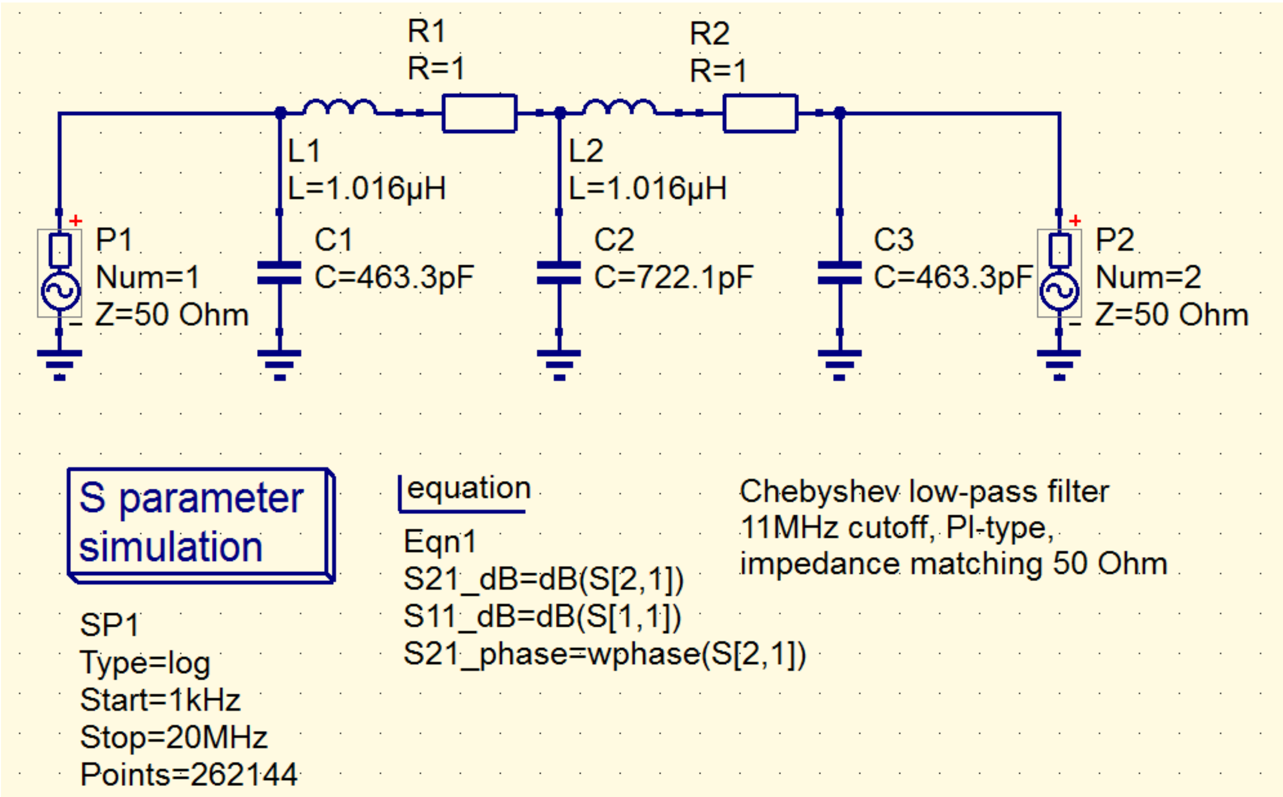


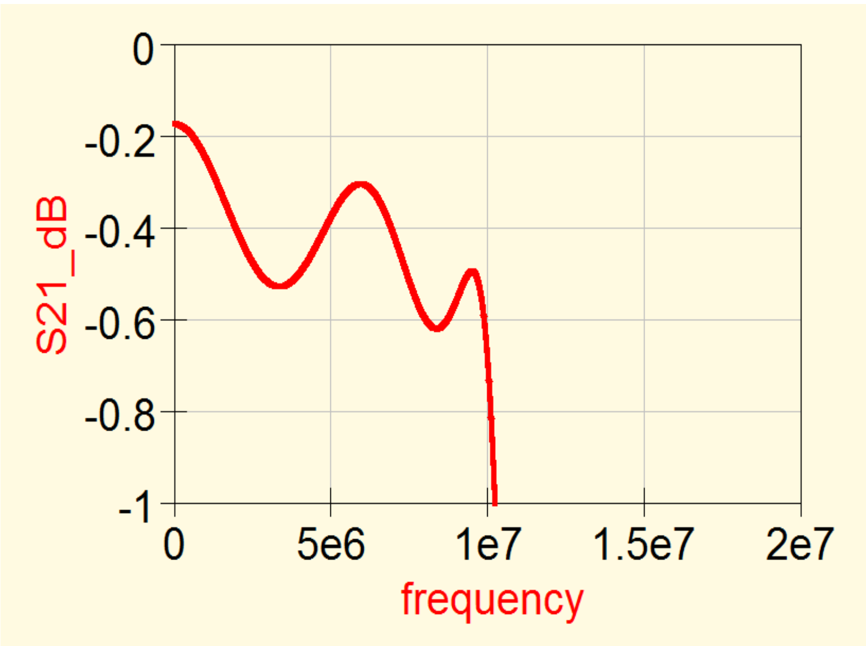
### 9.2.4. LPF Simulation including Coil Losses

Do you remember the notice in the filter specifications: „measured quality factor of the coils is Q = 70 at 11 MHz“?

**These losses are included in the simulation by a series resistor for every coil.**

**$R_r = X_L / Q = 2\pi fL / Q = 2\pi * 11MHz * 1{,}016\mu H / 70 = 1\Omega$**

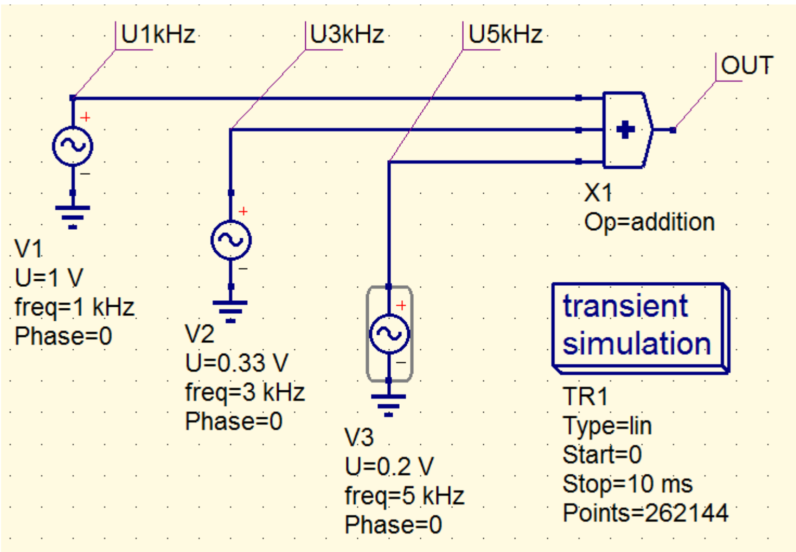



The result is „**an additional linear increase of attenuation with frequency in the pass band**“.

As you see in the illustration, S21 has already a value of -0.5 dB at the cutoff frequency....

## 9.2.5. The Secret of the Group Delay




Normally not only one single sine wave is transmitted. Look at AM or FM transmissions, at QUAM, at Pulse Signals., at music etc.

**If the curve form of the signal must not be changed during transmission, you have to pay attention to this rule:**

**Not only the relative amplitudes of all spectral lines must be correct at the end of transmission. Also all phase relationships between the spectral lines must be the same as at the start of the transmission.**

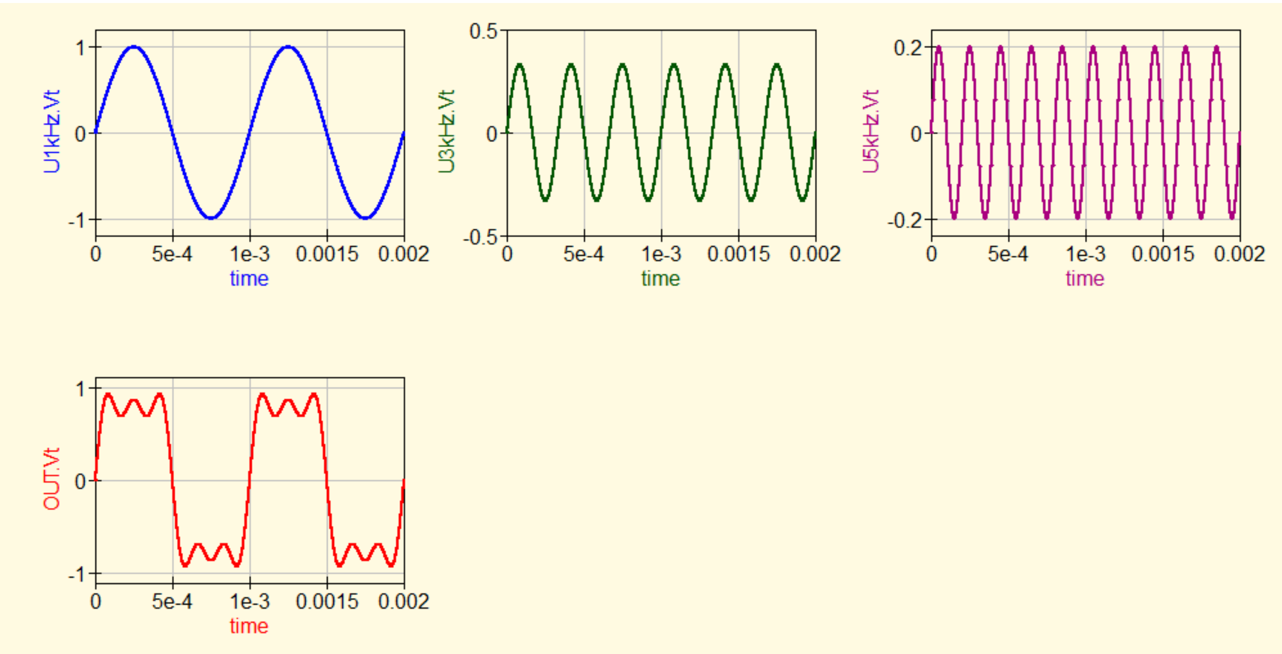
Let us simulate a symmetric rectangular voltage with only three spectral lines, which are added together.

We use
**a) the fundamental frequency with U1kHz= 1V and a frequency f = 1kHz**
**b) the third harmonic with U3kHz = 1/3 x (U1kHz) and the frequency f = 3 x f1 = 3kHz**
**c) the fifth harmonic with U5kHz = 1/5 x (U1kHz) and the frequency f = 5 x f1 = 5kHz**

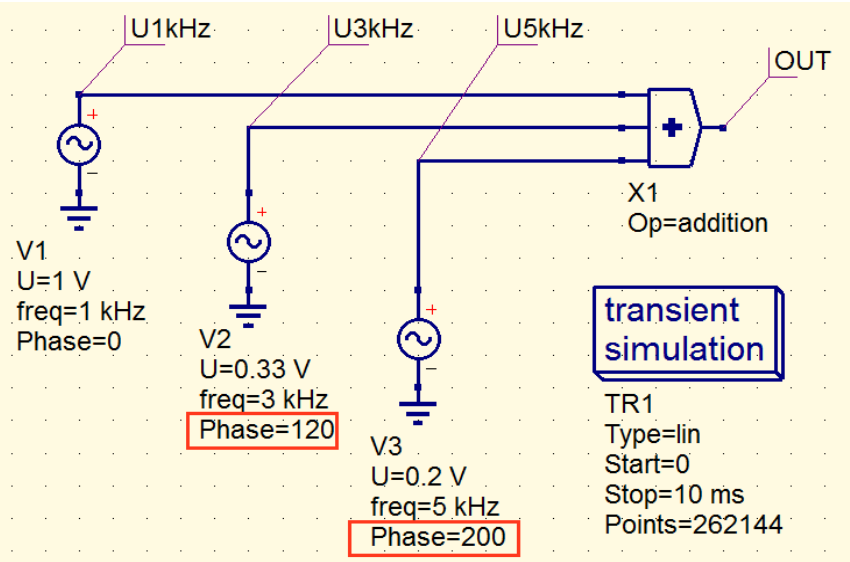
**All signals have the same start value (= zero) for the phase.**

**(The adder can be found as „Operation“ in „Components / System components“. Set the property to „three inputs“).**

Show the simulation result for 0...2 milliseconds:

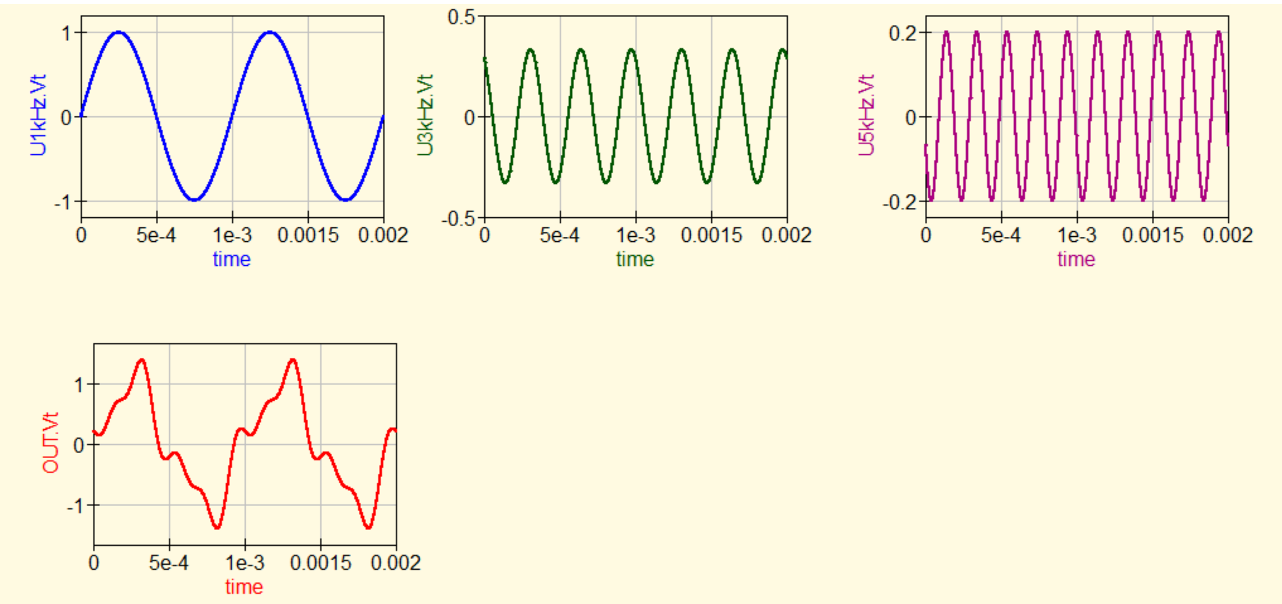



**The curve of the rectangular voltage at the output can be recognized without any difficulty.**




Now we let the amplitudes untouched but shift the phase of each harmonic.

The result is a completely altered curve form and the rectangular curve can hardly be recognized.




This means:
**If a signal travels through a communication system then the phase relationships of the spectral lines must he held constant – but an identical time delay for all spectral lines is allowed!**

In other words:

**Only a constant group delay guaranties the same correct curve form at the output.**

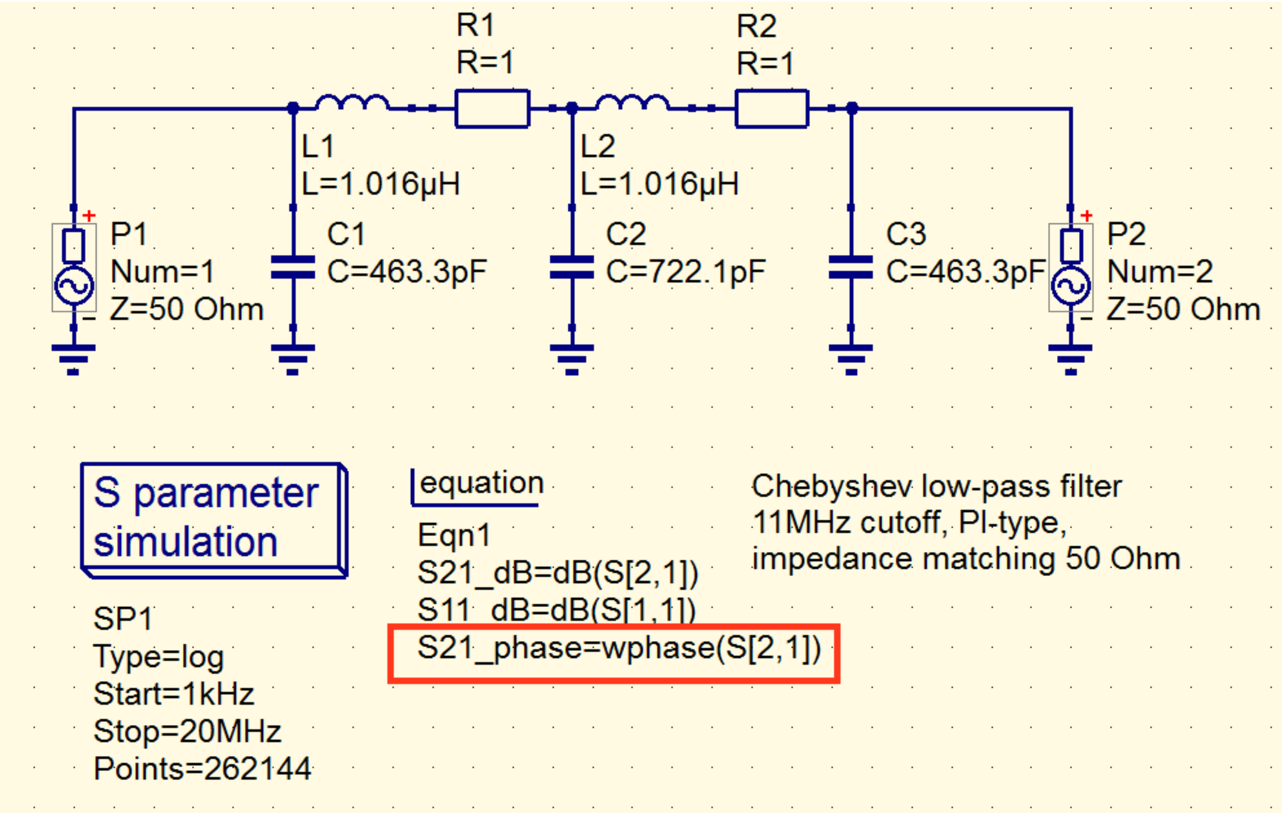
For specialists:

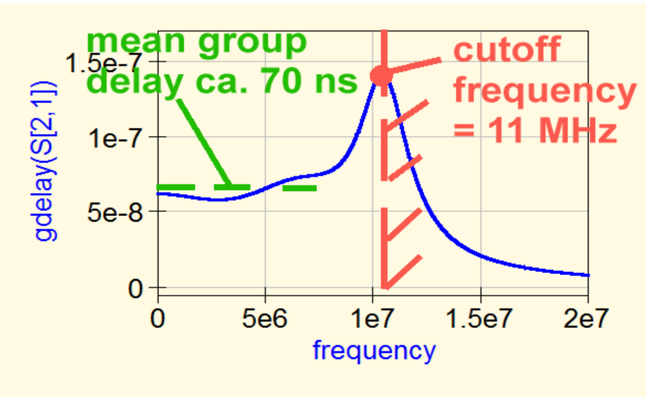
**A constant group delay means that the phase difference between input and output increases linearly with the freqency.**

We test this fact with our well known „LPF including coil losses“ of the last chapter. Very important is, that quscstudio has already prepared the phase simulation by the equation

**S21_phase=wphase(S[2,1])**

With this information it is possible to compute the group delay.




Use a cartesian diagram after the simulation and enter

**gdelay(S[2,1])**

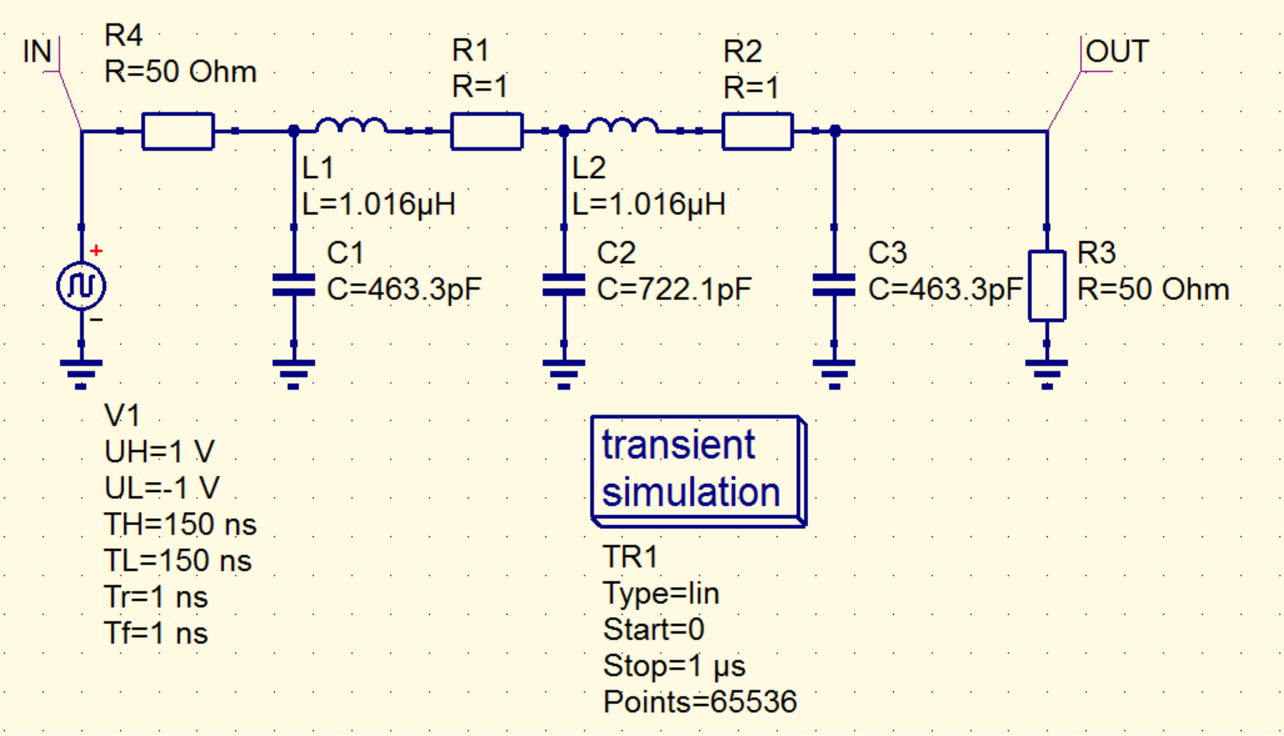
as graph property. That gives this result.

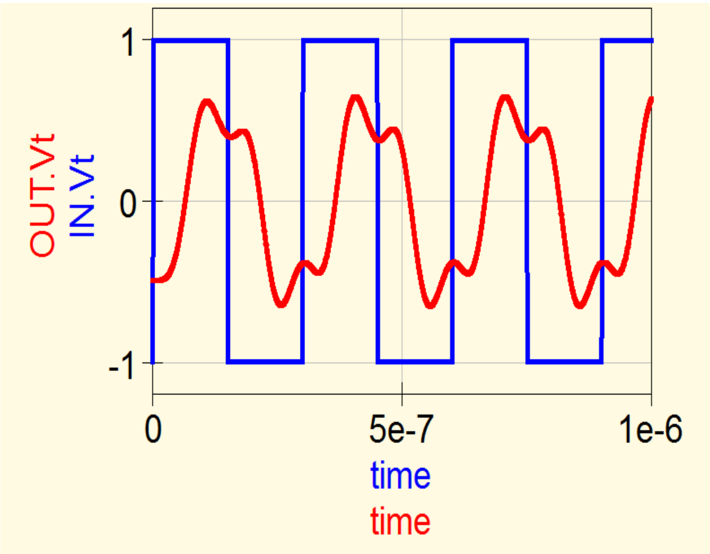
The mean group delay value is ca. 70 ns up to 75 % of the cutoff frequency. Then follows an abrupt rise.

Which effects causes this in practice?
Please feed our LPF with a rectangular input voltage and the following properties:

| | |
|---|---|
| **Uhigh** | **= +1V** |
| **Ulow** | **= -1V** |
| **Thigh** | **= 150 ns** |
| **Tlow** | **= 150 ns** |
| **Tr and Tf** | **= 1ns** |

Simulate for 0....1 microsecond with 65536 points.


IN
R4
R=50 Ohm
R1
R=1
R2
R=1
OUT
L1
L=1.016µH
L2
L=1.016µH
C1
C=463.3pF
C2
C=722.1pF
C3
C=463.3pF
R3
R=50 Ohm
V1
UH=1 V
UL=-1 V
TH=150 ns
TL=150 ns
Tr=1 ns
Tf=1 ns
transient simulation
TR1
Type=lin
Start=0
Stop=1 µs
Points=65536


1
0
-1
OUT.Vt
IN.Vt
0
5e-7
1e-6
time
time

The result.

# 10. The Smith Chart

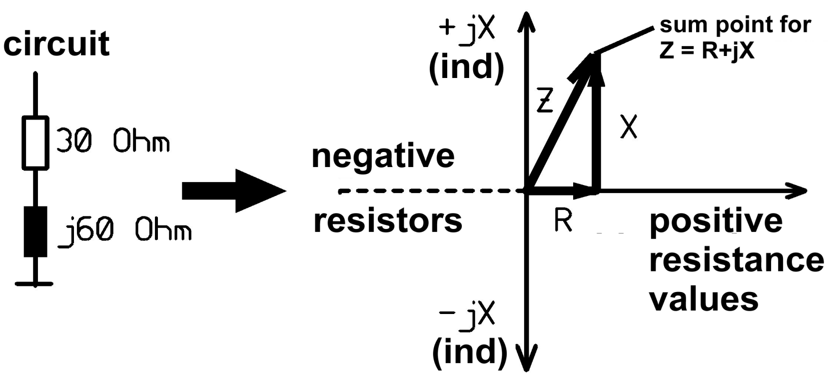
## 10.1. Preview

In all examples in which we simulated S parameters only „magnitude“ and „phase“ could be seen. But if you want to reduce reflections you are interested in the reasons for imperfect matching (mostly: capacitive or inductive influence). So you need another diagram to show a value range from Zero to Infinite for the real and for the imaginary axis. This can be done by a **coordinate transformation** and the result is the **Smith Chart.**

## 10.2. How to create a Smith Chart?

Connecting a resistor and a coil or a capacitor in series gives a phase shift between current and applied voltage. So you have to pay attention when calculating the total impedance due to the 90 degree phase shift between the voltage across the resistor and the voltage across the admittance.

Let us regard a series connection of a resistor with R = 30 Ohm and a coil with j60 Ohms. Now you can use an “IQ” diagram with two rectangular axis. The value of the resistor (30 Ohms) is marked on the positive horizontal axis and the reactance value of j60 Ohms on the vertical positive axis. To get the total impedance of the connection you have to “add the values geometrically” using Pythagoras law.




If we use an “IQ” diagram then we find the ohmic resistors on the horizontal ( = in phase) axis and the reactances on the vertical (= quadrature) axis.

Now we are interested in the “sum point” and have two possibilities to determine it:

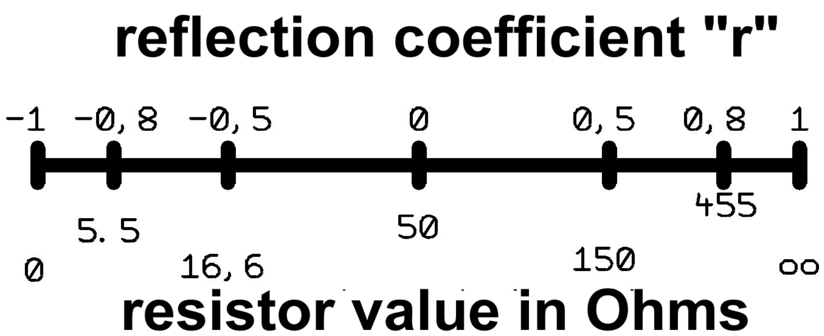
a) The point is given by two information “**Real part” and “Imaginary part”** (Z = 30Ω + j60Ω) or

b) the point is given as “**Magnitude and Phase”** of Z

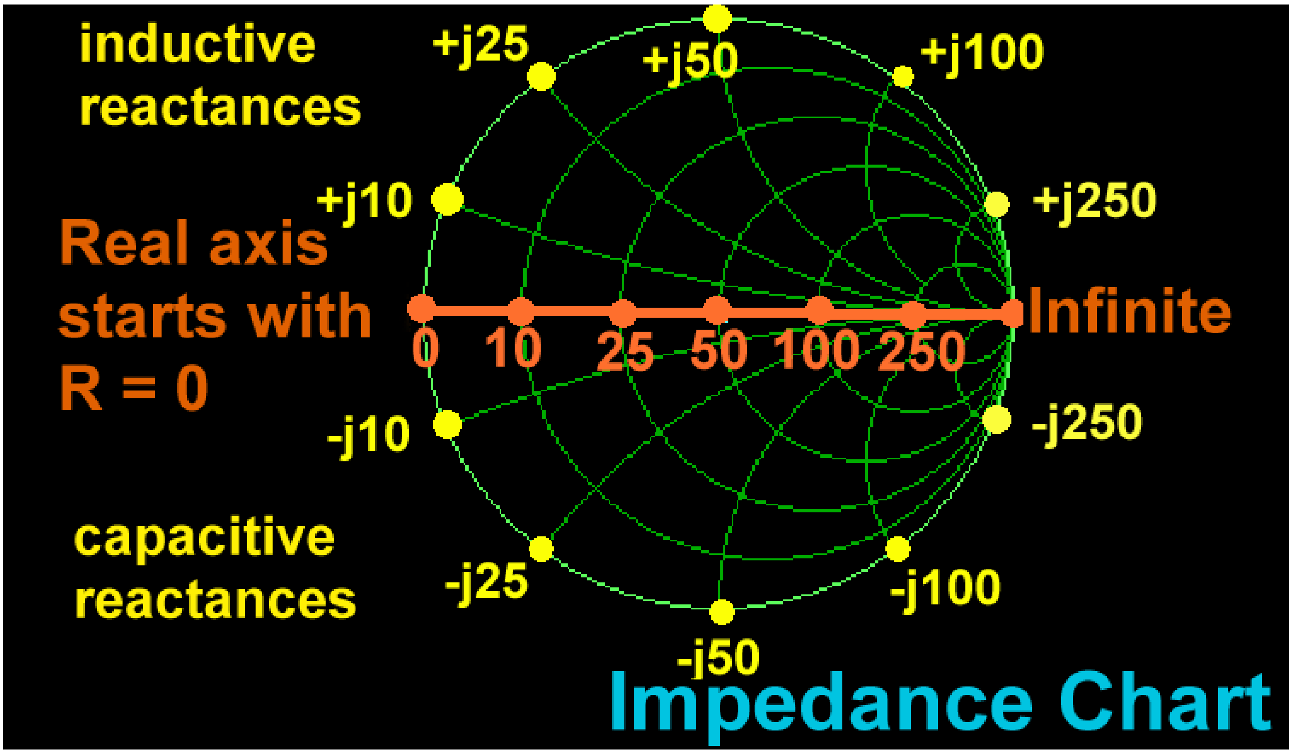
But in practice we have values for R or jX between Zero (= short circuit) and +-Infinire (= open circuit) and so we have to look for another axis scaling.

| |
|---|
| A diagram with the reflection coefficient “r” is used and this gives a “coordinate Transform” of the indicated resistance and admittance values at both axis derived from the formula $r = \frac{Z_{Last} - Z}{Z_{Last} + Z}$ |

So this does look like for the horizontal (= real) axis for a characteristic system impedance of 50 Ω:




**You find a linear scaling for “r” from “-1” (= short circuit, R = 0) running over Zero (= perfect match at 50 Ohms) to “+1” (= open loop, R = infinite).** And that is what we need...
Important:
**The admittance values with 90 degrees phase shift are now found on circles** (due to the coordinate transformation).:



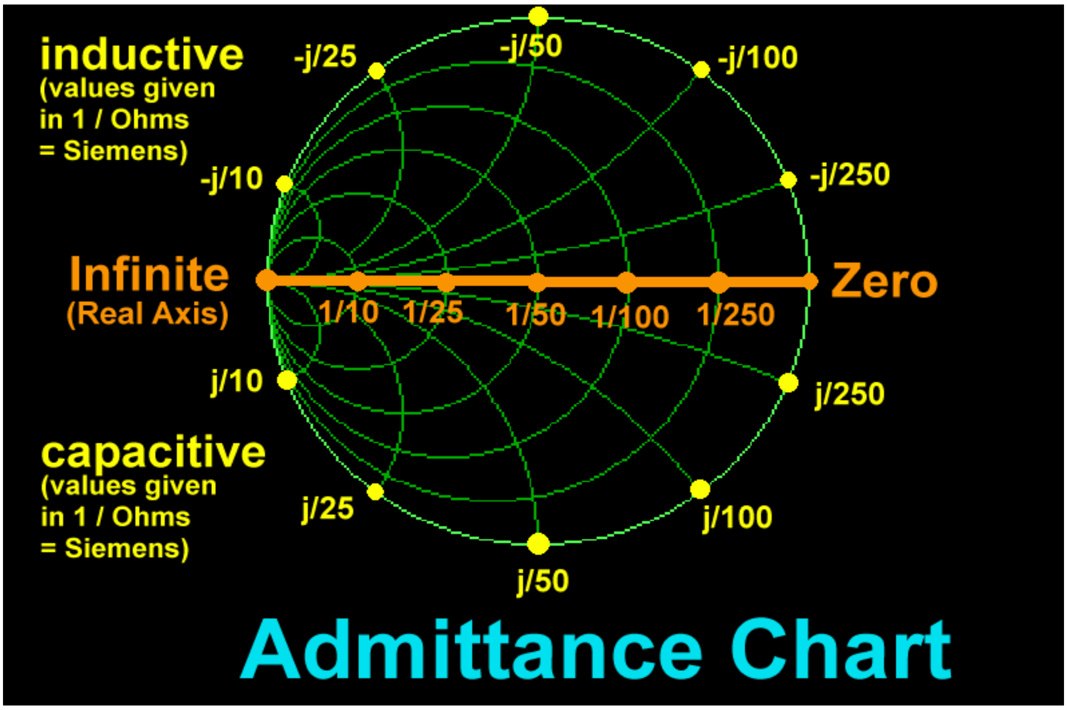

Remember:
a) The **center of the diagram means "no reflection"** (r = 0) due to R = Z = 50Ω = perfect match

b) The **start of the diagram at the left hand side means "r = -1"** and this equates to a short circuit = 0**Ω**

c) The **end of the diagram at the right hand side is the point for "r = +1"** and this equates an open circuit.

d) Every point anywhere in the diagram is a resistor and an admittance connected in series..

Very often you need a **parallel connection of resistance and admittance.** For this purpose the **smith chart scaling is mirrored horizontally – but the curves for S11 or S22 stay unchanged!**



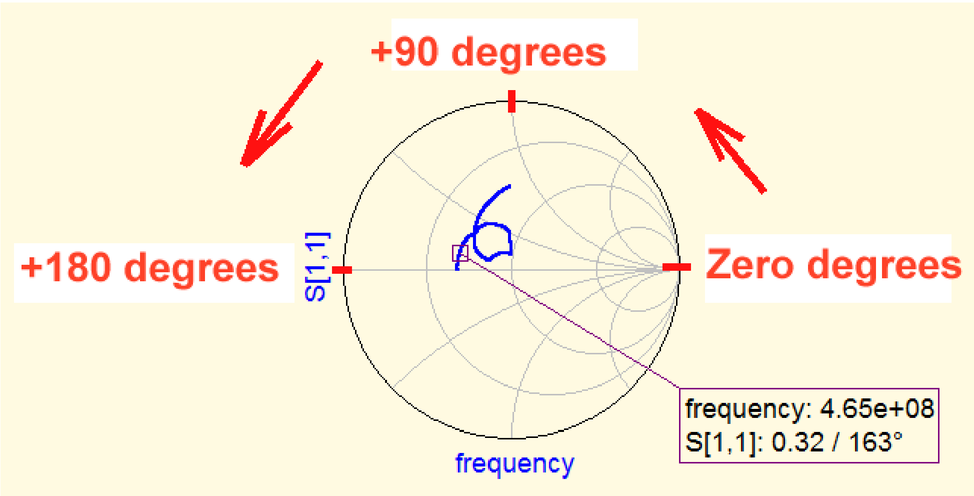
## 10.3. Nice little Traps: Degree Information in the Smith Chart

The degree information in a Smith chart can be irritating due to the fact that two different things are mixed:

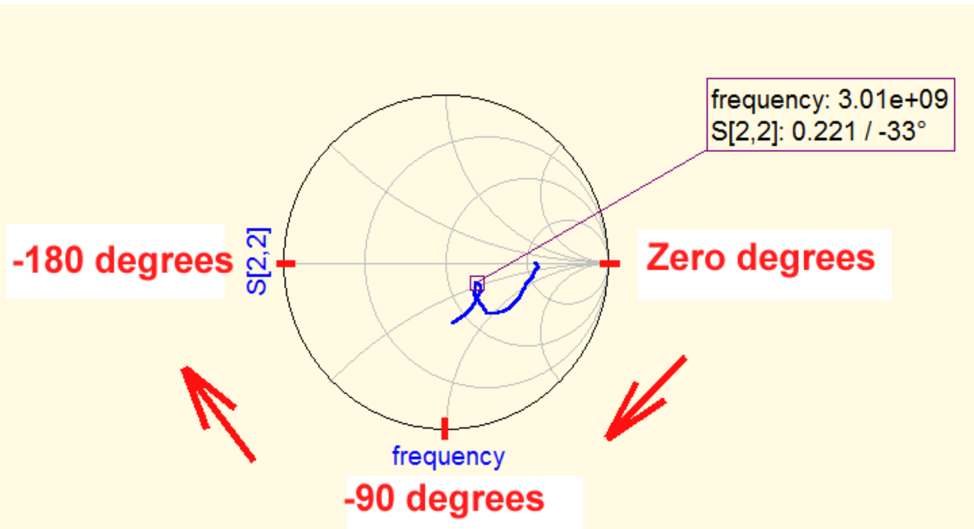
a) An Input Reflection S11 or an Output Reflection S22 can be regarded as an impedance given as a series connection. For an inductive circuit you move only in the upper half of an Impedance Smith Chart and the phase shift will always be between Zero and +90 degrees.

For capacitive impedances you get phase shifts between Zero and -90 degrees for the complex impedance and you move always in the lower half of the Impedance Smith Chart.

b) But the structure of the Smith Chart is based on the reflection coefficient "r" (mostly given as **magnitude and phase**) and now you have to pay attention:




An S parameter curve for S11 of an MMIC equates to the input reflection coefficient as magnitude and phase. But now you find the phase angle for Zero degrees at the right hand side of the diagram and **the scaling runs counter clockwise from Zero to +180 degrees.** This is valid for an inductive behavior of S11.




In comparison you see here S22 of the same MMIC. The curve moves only in the lower half of the Impedance Smith Chart and this means capacitive behaviour for the output impedance. In this case the phase starts again with "Zero" at the right hand side and **the scaling runs clockwise from zero to +180 degrees in the lower half of the Impedance Smith Chart.**

The question **"but how can I get the exact real part and imaginary part of values my impedance?"** will be answered in the following two chapters.

## 10.4. Determining the Real and the Imaginary Part of the Input Impedance as Series Connection using the Impedance Smith Chart

Let us use an well known and “classic” MMIC in an example (= INA03184) and simulate the S parameters. On the web you can find a file named “**a03184ia.s2p**“ in the Agilent Avago homepage for this purpose – or copy the following list and save it as “a03184.s2p” (attention: always save it as ASCII file...).

==========================================================

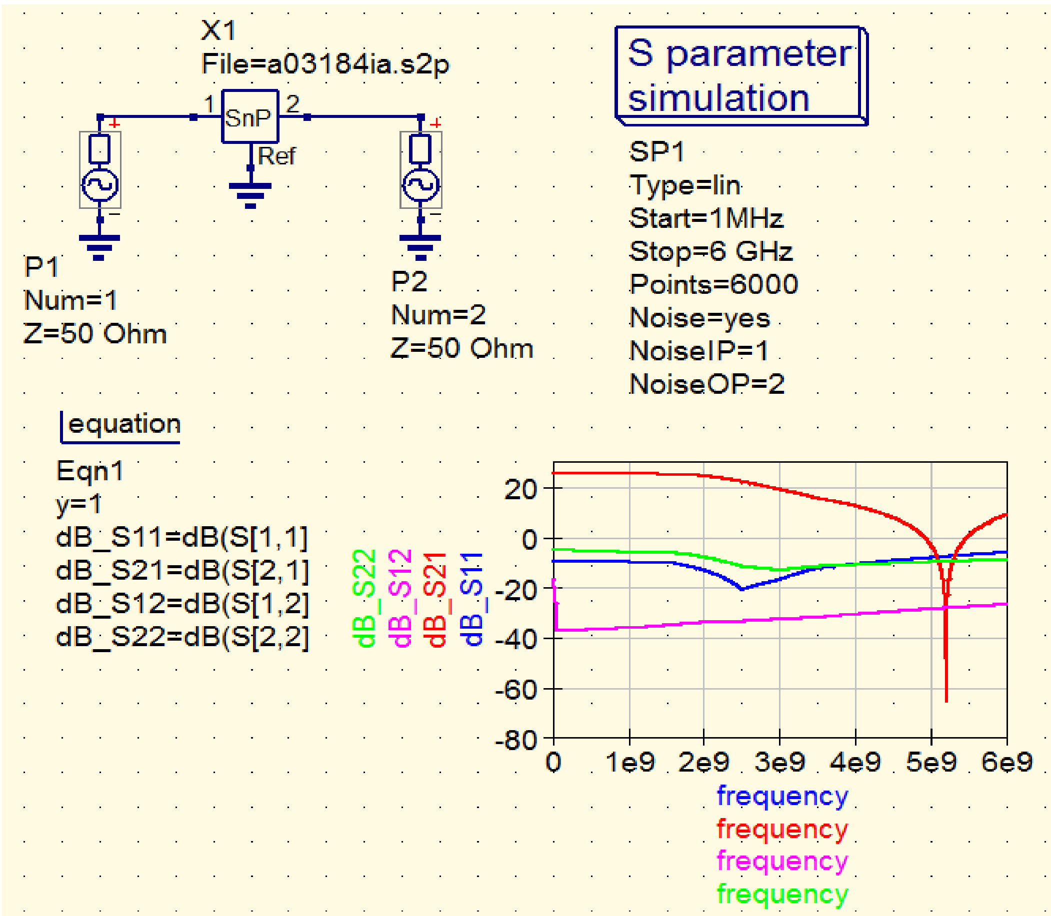
**! INA-03184 S PARAMETERS**
**! Id = 10 mA LAST UPDATED 07-22-92**

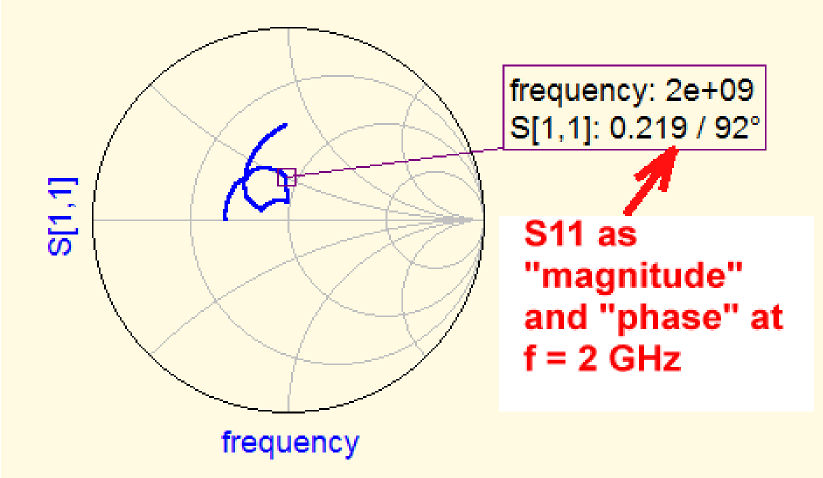
**# ghz S ma r 50**

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| **0** | **.32** | **180** | **19.2** | **0** | **0.14** | **0** | **0.55** | **0** |
| **0.05** | **.32** | **179** | **19.14** | **-3** | **.014** | **3** | **.55** | **0** |
| **0.10** | **.32** | **176** | **19.05** | **-7** | **.014** | **4** | **.57** | **-3** |
| **0.20** | **.32** | **172** | **19.05** | **-14** | **.014** | **6** | **.55** | **-5** |
| **0.40** | **.32** | **165** | **18.78** | **-29** | **.014** | **10** | **.53** | **-11** |
| **0.60** | **.32** | **158** | **18.71** | **-43** | **.015** | **11** | **.51** | **-14** |
| **0.80** | **.32** | **151** | **18.53** | **-57** | **.015** | **13** | **.51** | **-17** |
| **1.00** | **.32** | **144** | **18.18** | **-72** | **.016** | **21** | **.50** | **-20** |
| **1.20** | **.30** | **135** | **18.27** | **-86** | **.016** | **25** | **.50** | **-23** |
| **1.40** | **.31** | **126** | **18.10** | **-102** | **.017** | **30** | **.49** | **-29** |
| **1.60** | **.30** | **117** | **17.92** | **-117** | **.018** | **38** | **.48** | **-34** |
| **1.80** | **.26** | **102** | **17.49** | **-135** | **.019** | **44** | **.45** | **-41** |
| **2.00** | **.22** | **92** | **16.62** | **-153** | **.020** | **49** | **.40** | **-50** |
| **2.50** | **.09** | **91** | **12.88** | **168** | **.021** | **57** | **.26** | **-48** |
| **3.00** | **.14** | **160** | **8.79** | **134** | **.023** | **65** | **.22** | **-33** |
| **3.50** | **.24** | **151** | **5.92** | **108** | **.025** | **69** | **.26** | **-33** |
| **4.00** | **.29** | **139** | **4.18** | **87** | **.029** | **81** | **.28** | **-43** |

==============================================================

Now the S parameter simulation can be done for the frequency range from 0....6 GHz using this circuit:





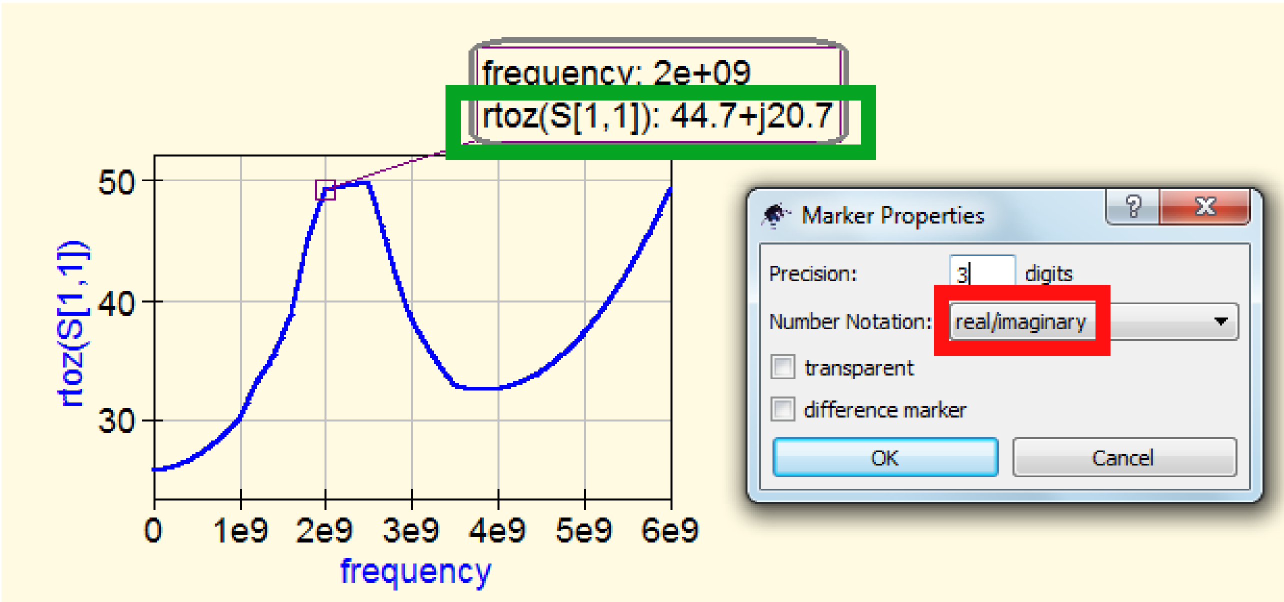

Afterwards we open an „**Impedance Smith Chart**“ to plot **S11**.
The actual and interesting frequency shall be f = 2 GHz and is marked by a frequency marker from the tool bar (“dragged” to this point).

Then we open a Cartesian diagram and use for the Graph Properties the following line:

**rtoz(S[1,1])**

We add a new marker for this curve, mark it and drag it to 2GHz. With a right mouse click on the marker text field the properties can be opened an the number notation set to „**real / imaginary**“. This gives as result this series connection for the input impedance:

**44.7Ω + j20.7Ω**



## 10.5. Determining the Real and the Imaginary Part of the Input Impedance as a Parallel Connection using the Admittance Smith Chart

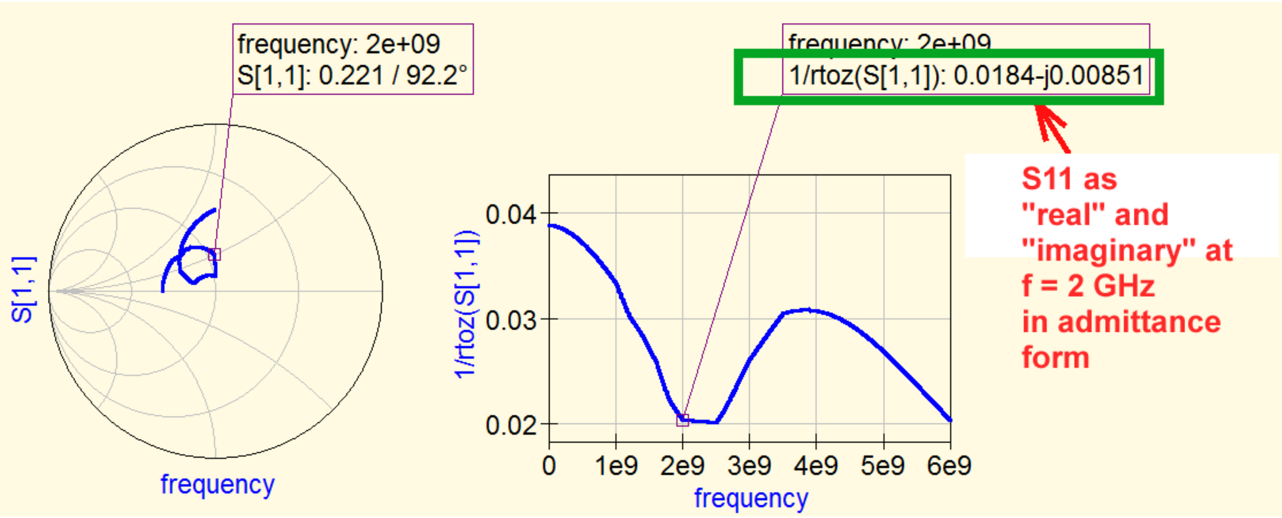
**Task:**
**Show S11 in an Admittance Smith Chart.**

**Mark the S11 point at f = 2 GHz by a frequency marker**

**Determine the real and imaginary part values of the Input Admittance.**

Solution:
At first get an Admittance Smith Chart to show the S11 curve. Then use a frequency marker from the tool bar to mark the frequency f = 2 GHz.

In a new Cartesian diagram enter **1/rtoz(S[1,1])** as Graph Properties to calculate the input admittance:




In the right hand graph we can see:
The input impedance can be regarded as a resistor and an inductance in parallel. The part values are

**real part = 0.0184 Siemens (= 54.34 Ω)**

and

**imaginary part = inductive admittance**
**= -j0.00851 Siemens = 117.5 Ω**

**This value equates to an inductance of 9.3 nH at f = 2 GHz.**

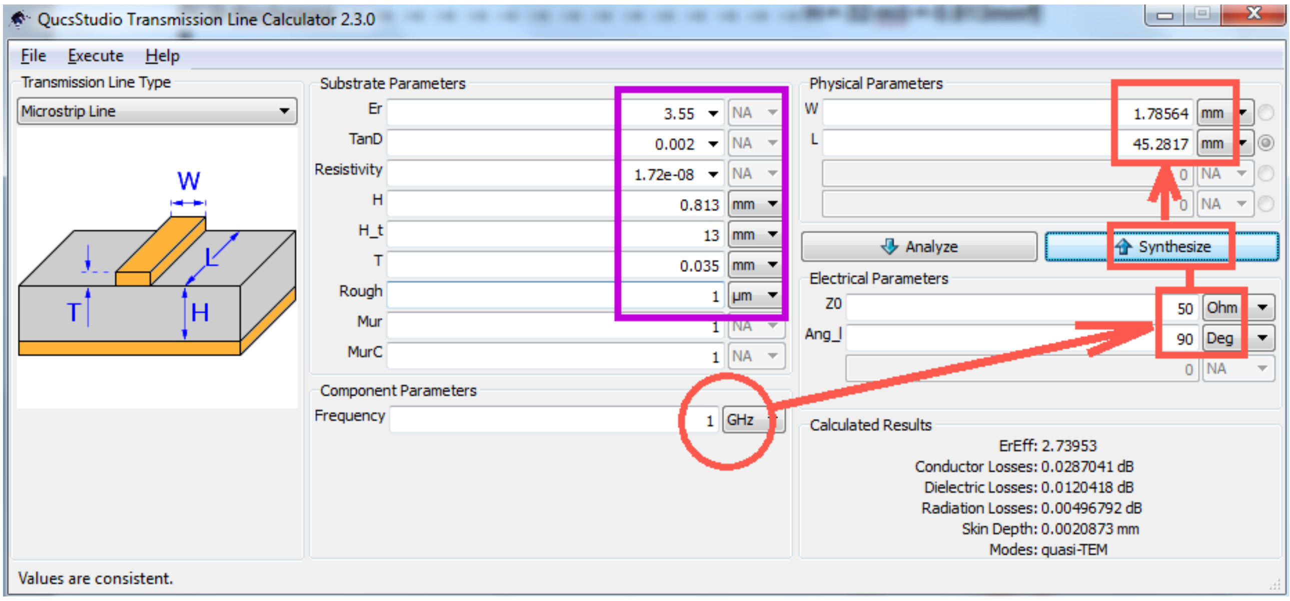
Please check this procedure...

# 11. Microstrip Lines

In a communication system we need for a correct signal transmission always cables and lines with an exact characteristic impedance of Z = 50Ω. On the PCBs (= printed circuit boards) these lines are realized by „microstrips“. This means that the lower side of the PCB is totally covered with copper. On the upper side a small strip realizes (together with the substrate of the PCB and the bottom copper plane) the desired transmission line. The characteristic impedance ist a function of the strip width and the thickness and dielectric constant of the PCBs substrate. The relationships are complicated and thus a good „line calculator“ will do the best design job for you. In qucsstudio you find such a „line calculator“ in the „Tools“ menu.

Normally I use the excellent and non expensive PCB material „RO4003“, fabricated by Rogers. These are the properties of the used material:

| | |
|---|---|
| **Dielectric constant** | **Er = 3.55** |
| **Tangent loss** | **TAND = 0.002** |
| **Copper resistivity** | **1.72e-8** |
| **PCB thickness** | **H = 32 mil = 0.813mm** |
| **Distance to the upper case cover** | **H_t = 13mm** |
| **Metal coating** | **copper with thickness = 0.035mm** |
| **Roughness of the copper plating** | **Rough = 1µm** |

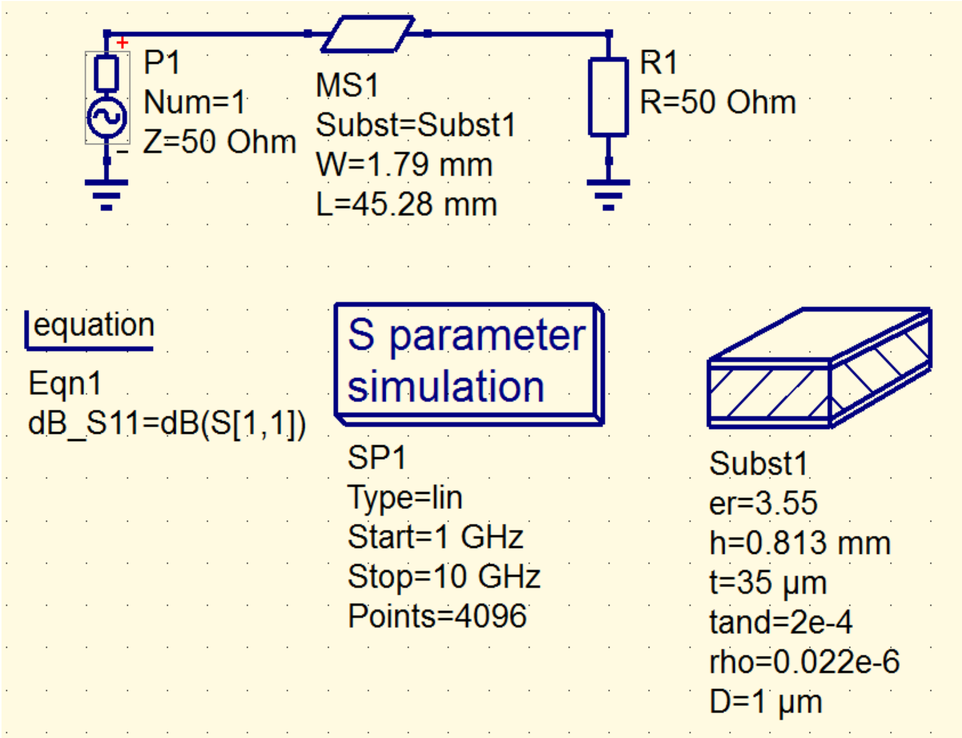



First switch the calculator to „**Micro strip Line**“. Then enter the line properties in the pink square as given.

Enter a **design frequency of 1 GHz**, followed by an **electrical length (= angle) of 90 degrees and a $Z_0$ of 50Ω**. Click on „**Synthesize**“ and you get the line dimensions:

| | |
|---|---|
| **Width** | **W = 1.79 mm** |
| **Length for 90 degrees at 1 GHz** | **L = 45.28 mm** |

Now we want to check the accuracy of the design.
Connect a port (= Power source) to the line's input and terminate the end of the line by a resistor of 50Ω.
Then simulate S11 (linear and in dB).

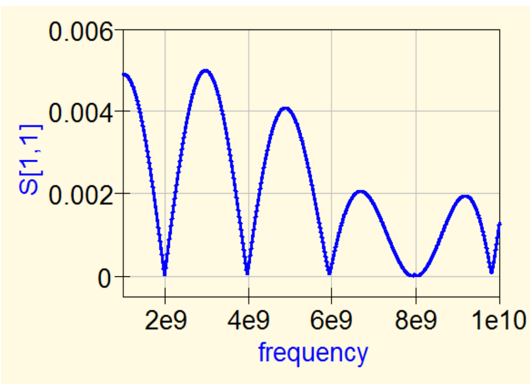


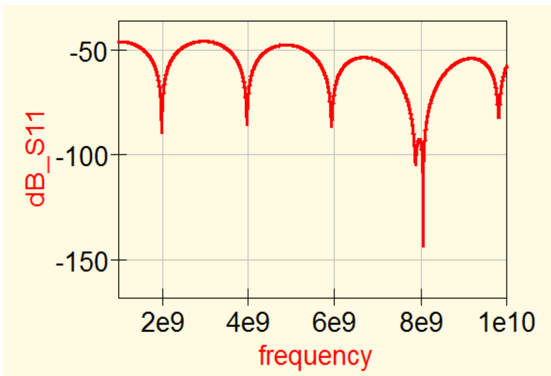

Therefore you need a „**microstrip MS1“ symbol** and a „**Substrate Subst1“ symbol** from „**components / transmission lines**“.

**Enter all the properties of the line and the substrate as given in this illustration.**

**Write the equation to calculate |S11| in dB.**

The simulation result looks really good.
In the linear presentation of S11 (left picture) we find magnitude values smaller than 0.5% up to 10 GHz.



# 12. Practical Project: MMIC Wide Band Amplifier up to 3 GHz

## 12.1. Touchstone Files (S2P Files)

Every Semiconductor Manufacturer offers these files for each produced part. They are a collection of the part's properties measured with a vector network analyzer in a given operating point.

MMIC means „**Monolithic Microwave Integrated Circuit**“ = a complete ready to go amplifier for the microwave range on a chip. This is a **Two port** with an input and an output – thus you get four S Parameters in your Touchstone File: in the range

**S11 / S21 / S12 / S22**

for increasing frequency.

Here comes an example for the ABA52563. You find this list at the end of the ABA52563 data sheet and you have to copy and to past this list into a new file of your text editor. Delete all unnecessary details to get exactly this form and **save it in a new next project („MMIC_3GHz“) as „ABA52563.S2P“.**

```
!     ABA-52563 S PARAMETERS

# ghz s ma r 50
0.05  0.01 146.6     12.10 -2.6      0.03 0.3     0.15 -2.4
0.10  0.01 134.0     12.11 -4.8      0.03 -0.3    0.15 -5.1
0.20  0.01 -40.6     12.16 -9.6      0.03 0.1     0.15 -9.6
0.30  0.01 -53.2     12.19 -14.5     0.03 1.2     0.15 -13.0
0.40  0.02 -56.7     12.19 -19.5     0.03 2.4     0.14 -15.7
0.50  0.03 -141.5    12.26 -24.8     0.03 1.0     0.15 -15.7
0.60  0.03 -128.1    12.24 -29.8     0.03 3.1     0.15 -17.6
0.70  0.04 -127.5    12.21 -34.9     0.03 4.3     0.15 -20.3
0.80  0.04 -126.7    12.18 -39.8     0.03 6.1     0.15 -22.5
0.90  0.05 -123.9    12.16 -44.7     0.03 7.4     0.15 -24.2
1.00  0.05 -125.0    12.13 -49.7     0.03 11.7    0.15 -26.4
1.20  0.05 -123.4    12.10 -59.6     0.03 10.8    0.15 -29.4
1.40  0.06 -127.4    12.05 -69.4     0.03 12.4    0.15 -32.4
1.60  0.06 -133.8    12.04 -79.6     0.03 13.0    0.15 -35.3
1.80  0.06 -136.7    12.00 -89.8     0.04 14.7    0.15 -37.8
2.00  0.07 -142.5    11.94 -100.4    0.03 14.3    0.15 -38.3
2.20  0.07 -143.9    11.87 -111.2    0.04 16.7    0.15 -37.8
2.40  0.08 -146.1    11.75 -121.9    0.04 16.2    0.15 -37.3
2.60  0.09 -148.4    11.56 -133.2    0.04 17.3    0.14 -36.9
2.80  0.09 -149.5    11.33 -144.5    0.04 15.6    0.14 -36.4
3.00  0.10 -152.7    10.95 -156.1    0.04 15.8    0.13 -35.9
3.20  0.10 -158.7    10.51 -167.5    0.04 15.6    0.13 -35.4
3.40  0.11 -163.2    9.97 -178.7     0.04 15.5    0.13 -34.9
3.50  0.11 -167.6    9.67 175.9      0.05 16.0    0.13 -34.6
4.00  0.12 165.9     8.25 150.6      0.05 12.0    0.13 -33.4
4.50  0.16 138.3     6.98 126.3      0.05 12.7    0.14 -37.1
5.00  0.19 122.8     5.71 105.0      0.06 9.5     0.12 -48.4
5.50  0.25 112.3     4.85 86.7       0.07 6.0     0.12 -63.0
6.00  0.30 99.3      4.14 70.4       0.07 1.0     0.11 -83.5
```

**Details:**

a) Lines starting with **„!“** are comments and will be ignored.

b) In the line starting with **„#“** you find all information about the details of the parameters:

**ghz s m a r 50**

**„ghz“** means: all lines in the following tabular start with the **frequency value in GHz**.

**„m“** and **„a“** say that every parameter is given as „**magnitude**“ **and „angle“**

**„r 50“** tells you that you are working with a real characteristic impedance of 50 Ohms.

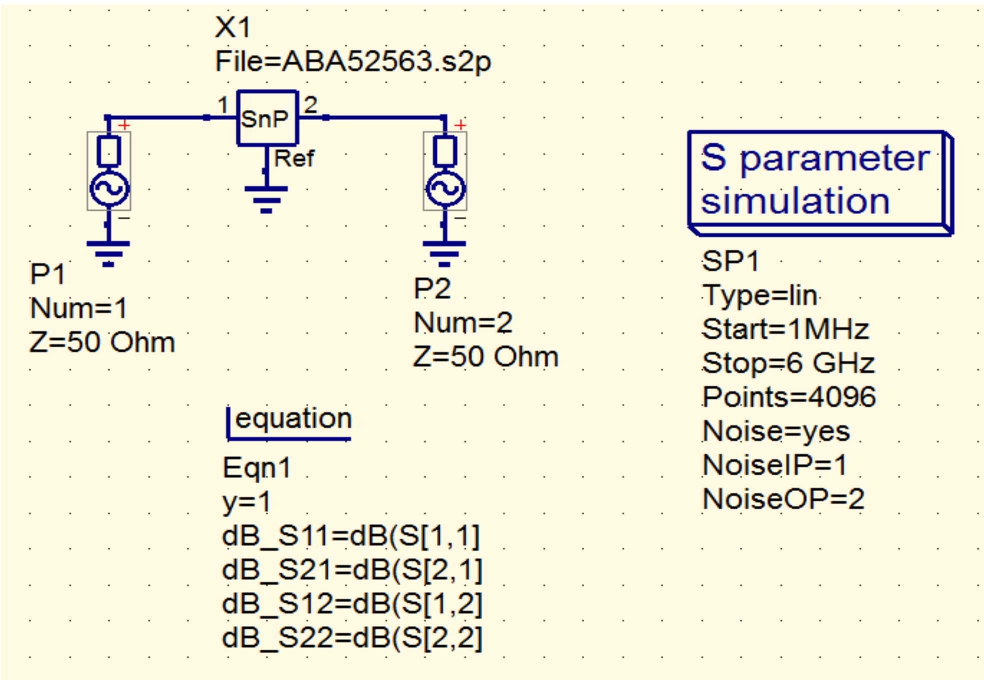
The range of the parameters from the left to the right in the tabular is

**S11 S21 S12 S22**

==================================================================

## 12.2. Simulation of the S Parameters for the ABA52563 up to 6 GHz

Therefore you need this schematic:




a) The **Ports P1 and P2** can be found as „**power sources**“ in „**components / sources**“

b) **The Two Port X1** comes as „**S parameter file**“ from „**components / devices**“

c) „**S parameter simulation**“ is located in „**components / simulations**“

d) And for the **dB calculation** we need again the well known equation

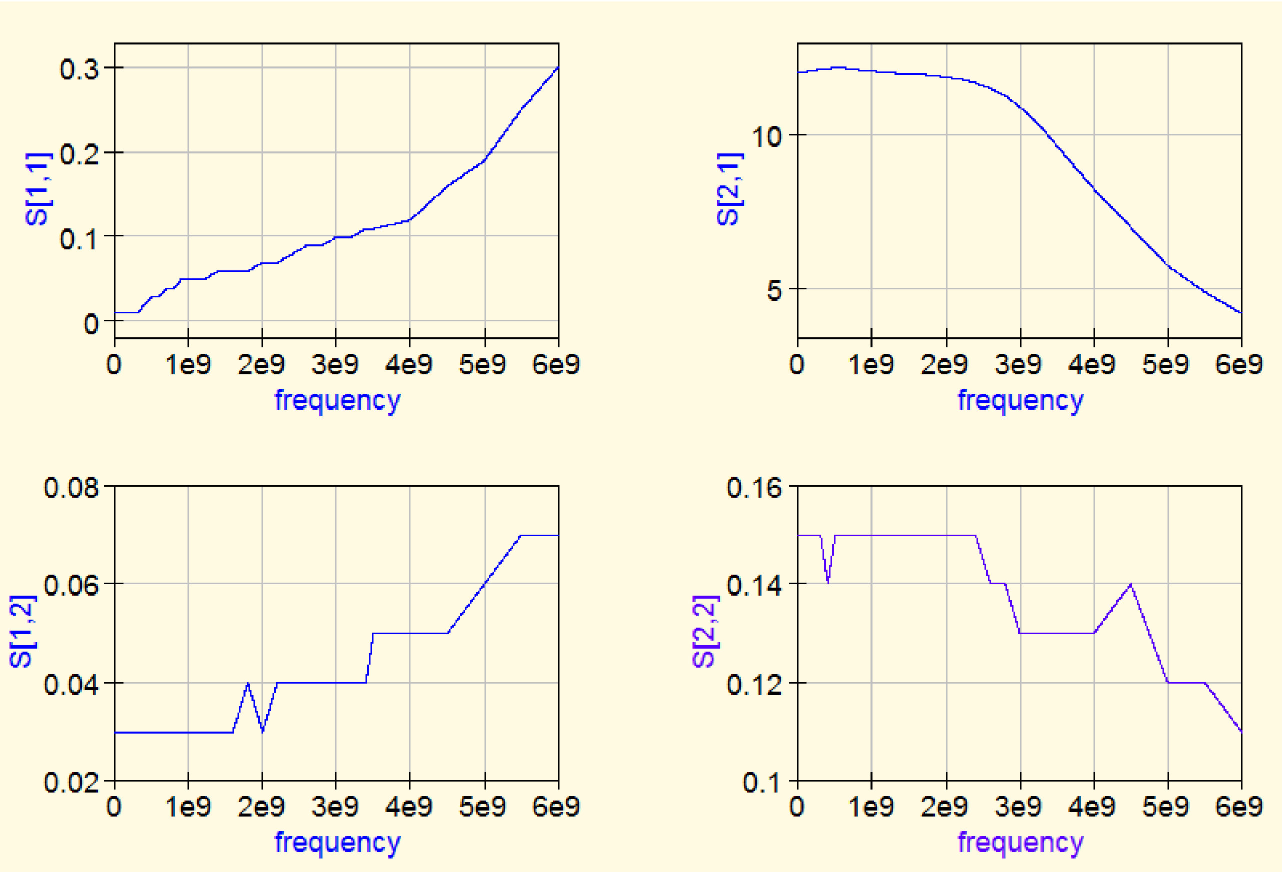
Right click on the symbol of X1 to edit the properties. Then enter the name of your prepared „**ABA52563.S2P**“ S parameter file.

Simulate by using a sweep from 1 MHz to 6 GHz using 4096 points.

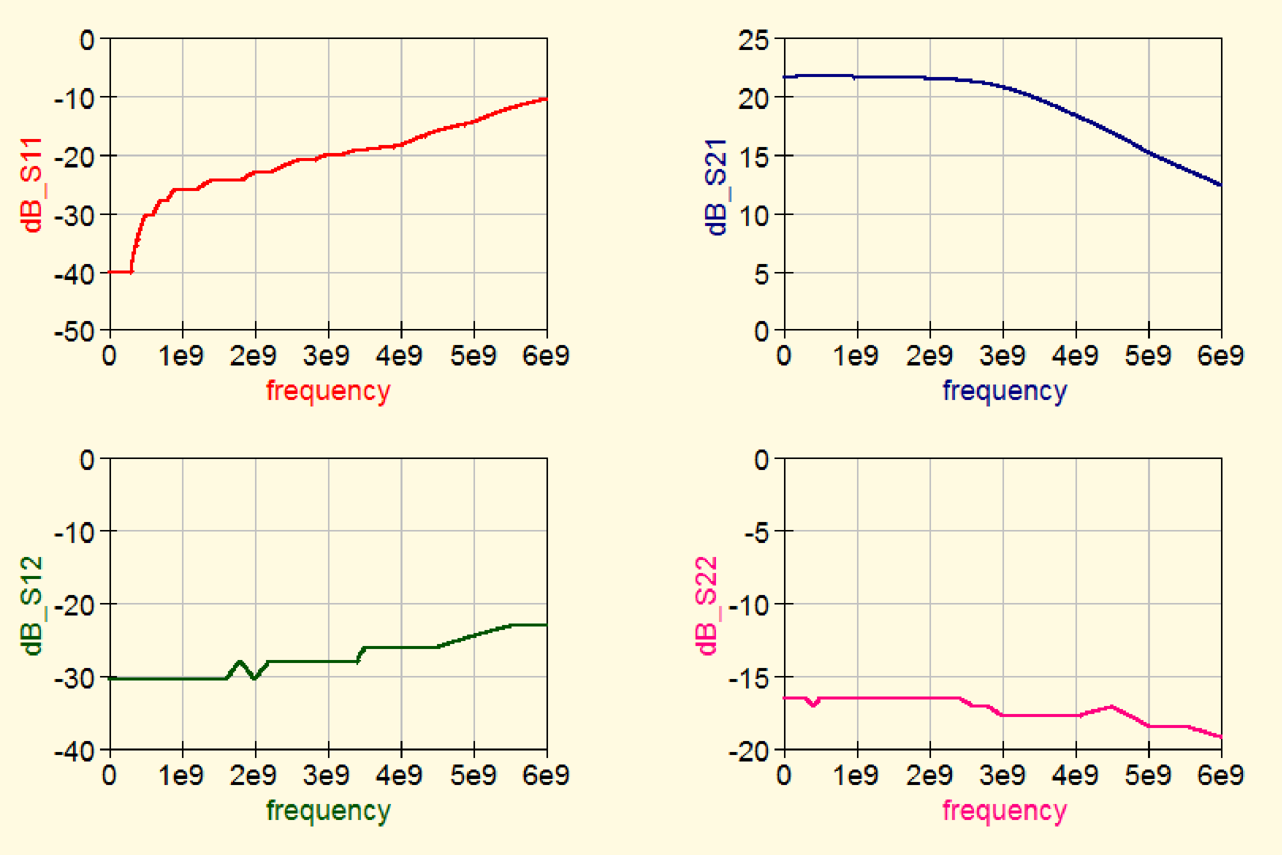
| frequency | S[1,1] | S[1,2] | S[2,1] | S[2,2] |
|---|---|---|---|---|
| 1e6 | 0.01 / 159° | 0.03 / 0.888° | 12.1 / -0.444° | 0.15 / 0.246° |
| 2.46e6 | 0.01 / 159° | 0.03 / 0.87° | 12.1 / -0.508° | 0.15 / 0.167° |
| 3.93e6 | 0.01 / 158° | 0.03 / 0.853° | 12.1 / -0.573° | 0.15 / 0.0878° |
| 5.39e6 | 0.01 / 158° | 0.03 / 0.835° | 12.1 / -0.637° | 0.15 / 0.00868° |
| 6.86e6 | 0.01 / 157° | 0.03 / 0.818° | 12.1 / -0.702° | 0.15 / -0.0704° |
| 8.32e6 | 0.01 / 157° | 0.03 / 0.8° | 12.1 / -0.766° | 0.15 / -0.15° |
| 9.79e6 | 0.01 / 157° | 0.03 / 0.783° | 12.1 / -0.831° | 0.15 / -0.229° |
| 1.13e7 | 0.01 / 156° | 0.03 / 0.765° | 12.1 / -0.895° | 0.15 / -0.308° |
| 1.27e7 | 0.01 / 156° | 0.03 / 0.747° | 12.1 / -0.96° | 0.15 / -0.387° |
| 1.42e7 | 0.01 / 156° | 0.03 / 0.73° | 12.1 / -1.02° | 0.15 / -0.466° |
| 1.56e7 | 0.01 / 155° | 0.03 / 0.712° | 12.1 / -1.09° | 0.15 / -0.545° |
| 1.71e7 | 0.01 / 155° | 0.03 / 0.695° | 12.1 / -1.15° | 0.15 / -0.624° |

The first task is to print the four S parameters in a tabular.

Then use four Cartesian diagrams with linear vertical axis to print the four S parameters



But very important for the practical work is the **S parameter presentation in dB**:




**Important:**

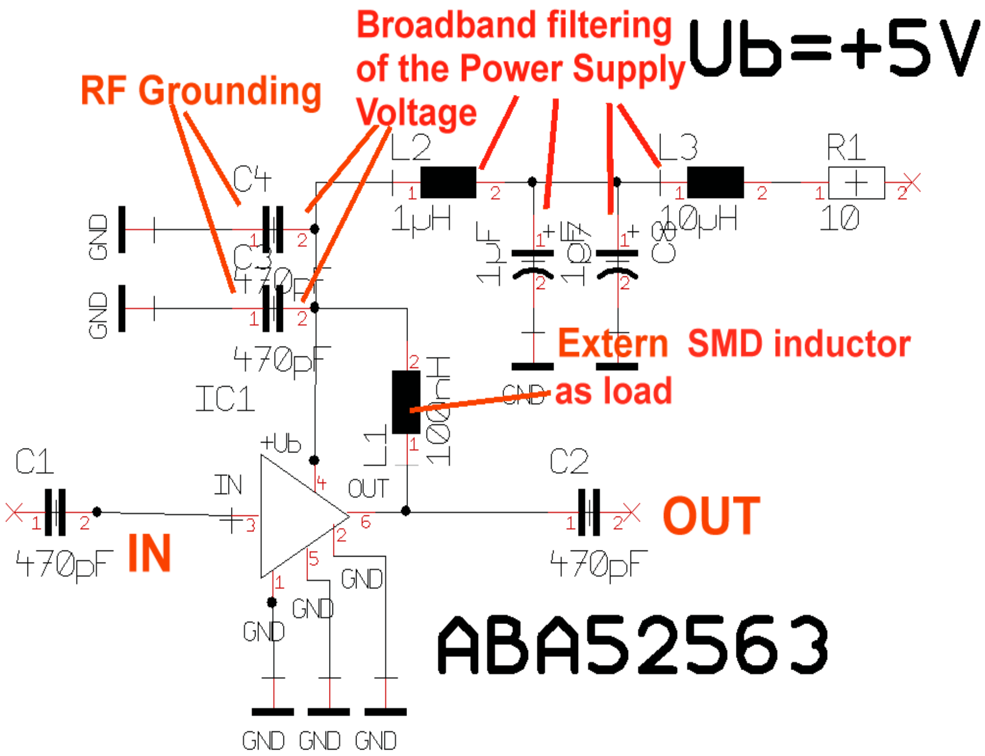
The **Gain** (represented by S21) is nearly constant up to 3 GHz.

The **input reflection** (represented by S11) is up to 3 GHz smaller than -20 dB and thus the input resistance not far from exactly 50Ω in this range.

**Feedback** (represented by S12) is also smaller than -20 dB up to 6 GHz. So we are hopefull that no self oscillation will occur.

Only the **output reflection** S22 gives a small reason to worry, because the value lies only between -15 and -19 dB.

## 12.3. Check of the complete practical Amplifier Circuit




The practical circuit for the PCB looks very different in comparison to the simulation schematic.

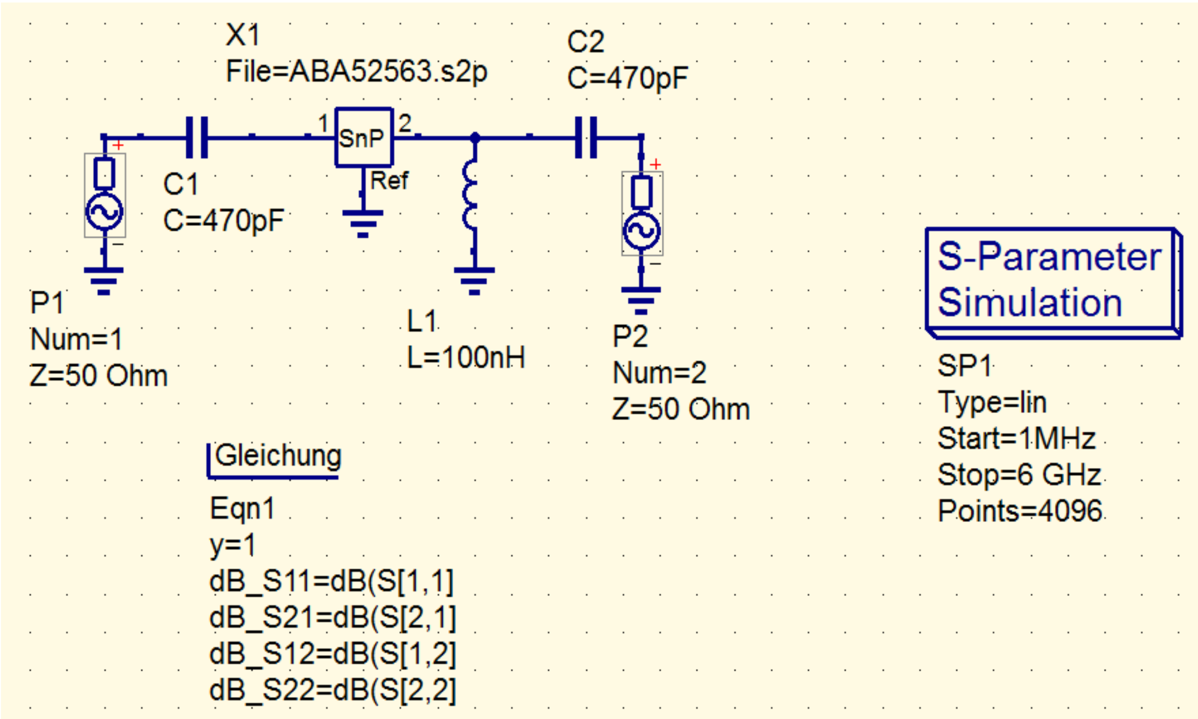
But don't worry:
we are going to explain all confusing details:

a) **All capacitors with values higher than 1 nF are short circuits for RF.**

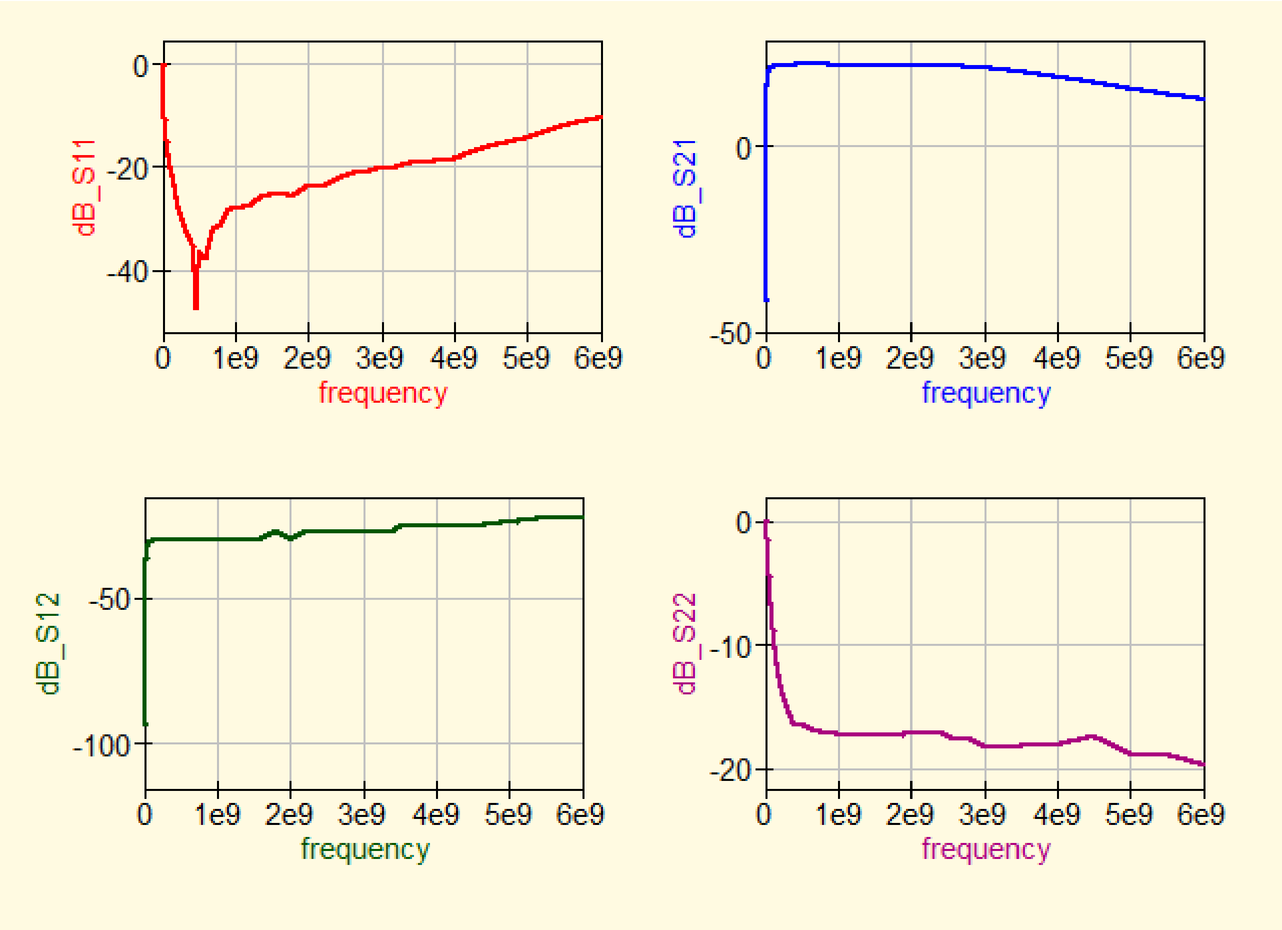
b) **Also the power supply is grounded for RF.**

c) **Only coils and capacitors with small values must be taken in account for the simulation.**

So you can now simulate this simple schematic:



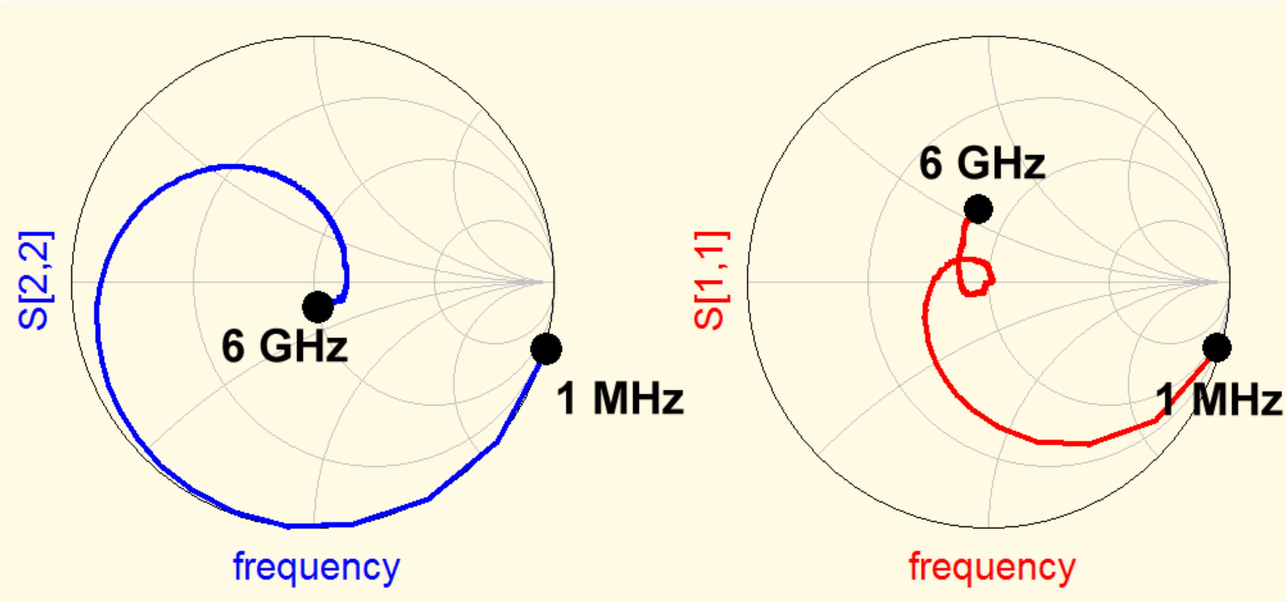
Here come the S parameters in dB:




**At very low frequencies S21 decreases due to the decreasing impedance of the inductance L1 at the output. Also S22 changes in the same manner: towards „zero Hertz“ we get an impedance of the inductivity of zero Ohms – thus S22 shows „total reflection“ with a value = „1“ = „0 dB for |S22|“.**

## 12.4. S11 and S22 of the MMIC presented in the Smith Chart

No problem: simulate and then use the Smith chart diagram as given in „components / diagrams“.
Open the property menu and enter S11.
Repeat the procedure for S22. Use two different colors.

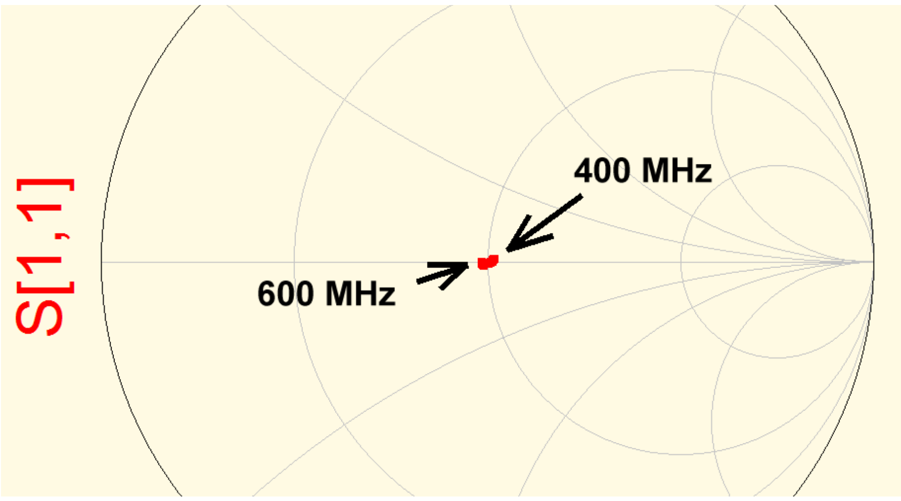



Remarks:

**a)** All curves **turn clockwise with increasing frequency.**

**b)** Both curves start near the „infinite“ point due to the increasing impedance of the coupling capacitors at the input and out when the frequency decreases.

**c)** **For S22 the curve stays long on the circumference of the Smith chart (= total reflection) but walks in direction to „zero“ due to the very low impedance of the 100 nH inductance at low frequencies.**




S11 crosses the center of the Smith chart between 400 and 600 MHz (= perfect match).

To see the details repeat the sweep for this small frequency range.

## 12.5. The Amplifier in the Real World

### 12.5.1. The PCB (= Printed Circuit Board)

There are only few but obligatory rules for a successful design of microwave circuits on a PCB:

a) Use a good RF substrate like Rogers RO4003 or 4350B

b) Use a PCB with upper and lower side covered with copper

c) The lower side is always the ground plane

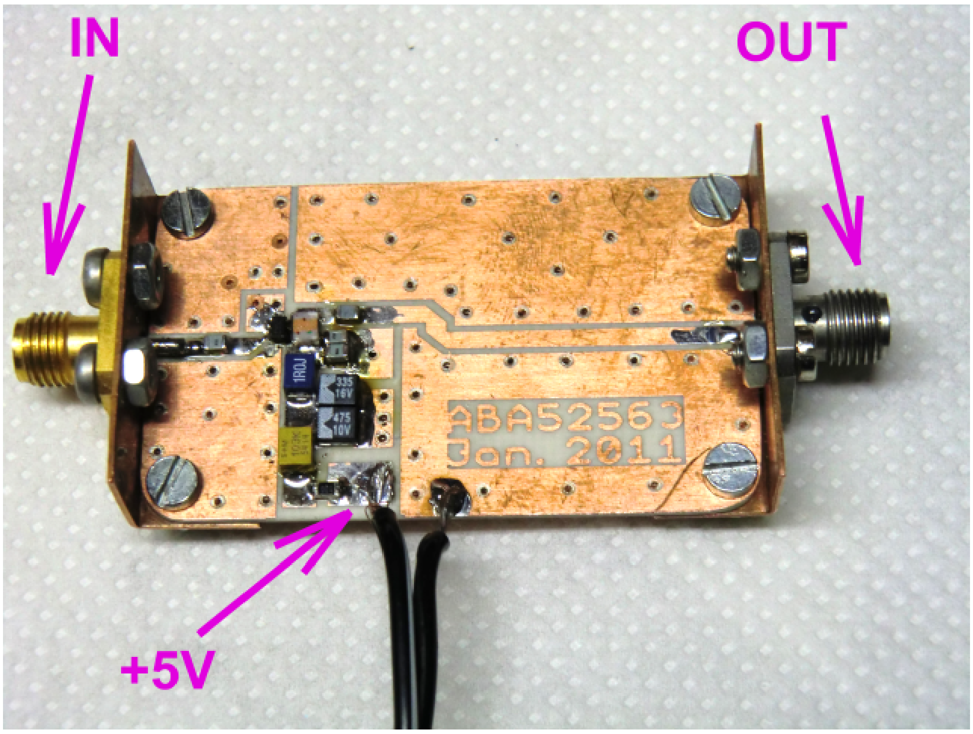
d) Use micro strip line or „guided co planar wave guides“ for your RF connections on the upper plane

e) If you need a „ground connection“ for a part of your circuit use as much via as possible for your circuit point down to the ground plane (...every via is a little inductor with increasing Z when frequency is increasing). So a lot of vias in parallel connection reduce the total inductance).

f) Divide your circuit in „separated grounded islands“ to avoid self oscillating. Never connect an input island to an output island...

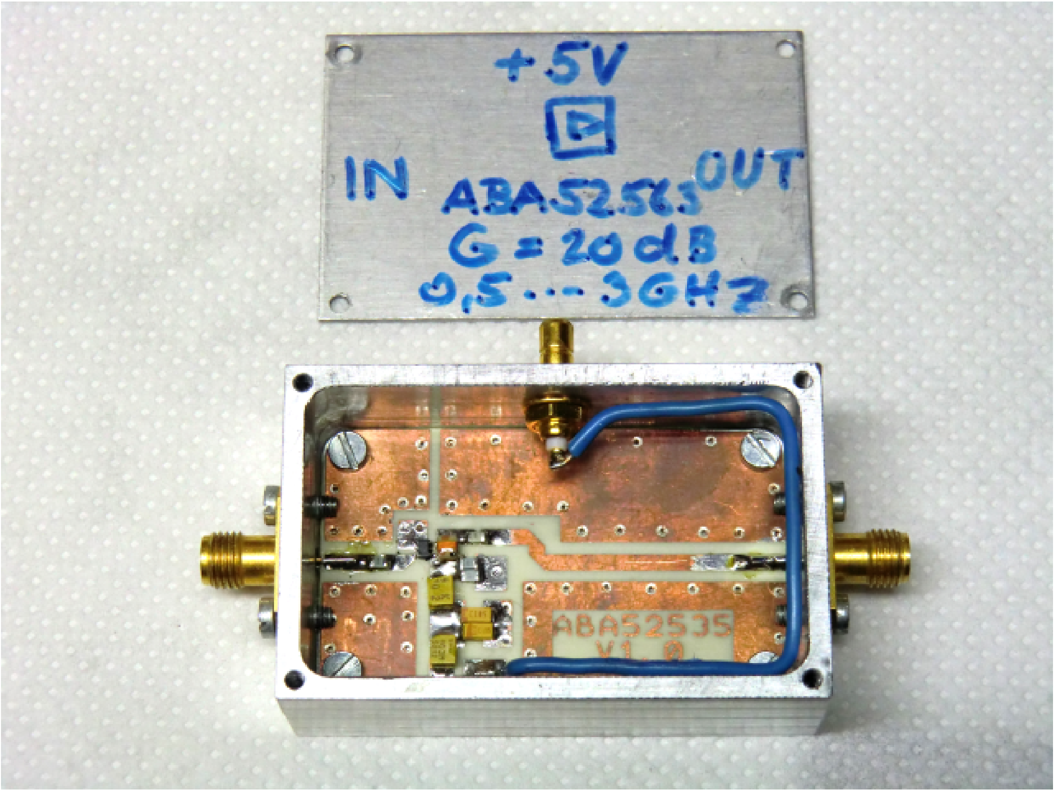
g) For self made via use small copper rivets (with a diameter of 0.6...0.8 mm). Do not use massive wires instead of via (= metallized holes)! Due to the skin effect a massive wire says „off limit“ to the current because this current is only flowing at the wire surface and if you solder this wire in your circuit the path for the current is interrupted...The poor current must look for another (= mostly longer) way from the upper side to the ground plane and that causes an several GHz suddenly an additional attenuation of up to several dBs....or self oscillation if there are phase shifts caused by inductive components on the new way...

Here you can see the complete board ready for testing. Please be aware of the via and the grounded islands and so the input isolated from the output, the „grounded co planar wave guide“ from the amplifier output to the SMA socket...



The SMA sockets are screwed to copper angles, the inner connector of each SMA socket „lies on the micro strip line“ and is connected to the line by soldering. This gives minimum reflections for the first test and runs without any problems up to 8 GHz..

When all is OK then the PCB disappears in a milled alumina case with cover.
Supply is fed to the circuit by a SMB jack.




If somebody asks: „why is the amplifier shifted so much in direction of the input side“:

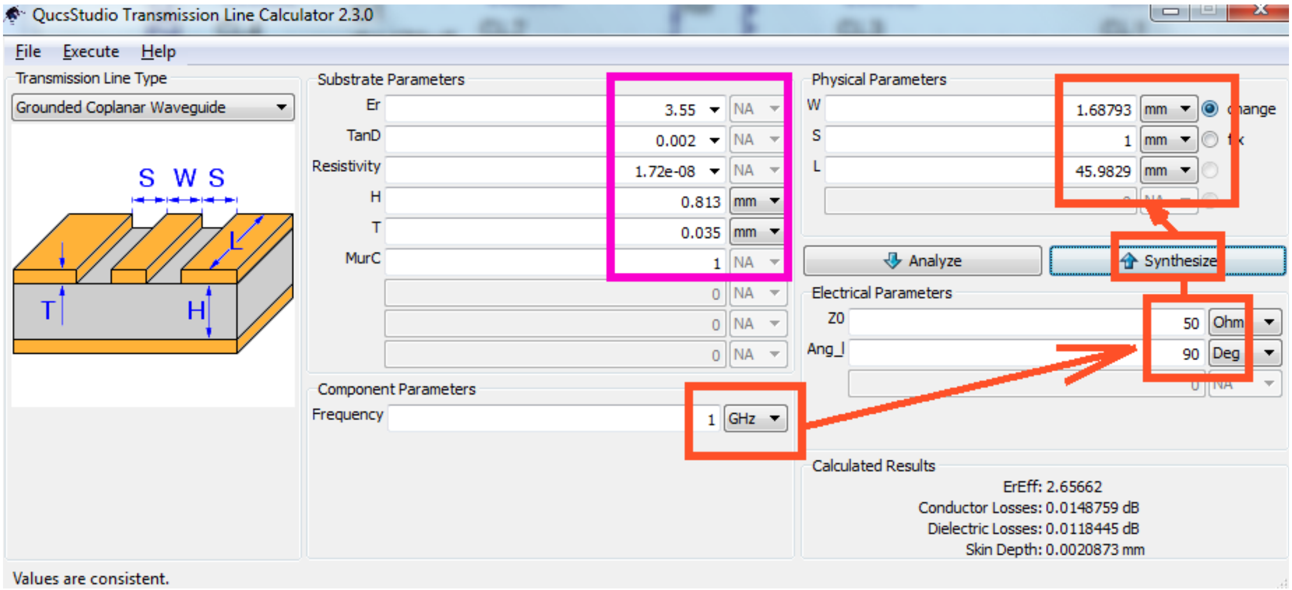
a) The input micro strip must be as short as possible, because their attenuation in dB increases the Noise Figure NF of the stage by the same value.

b) To achieve a total gain of more than 40 dB you could add an identical amplifier stage on the right hand side (S21 of one stage is ca. 21 dB).

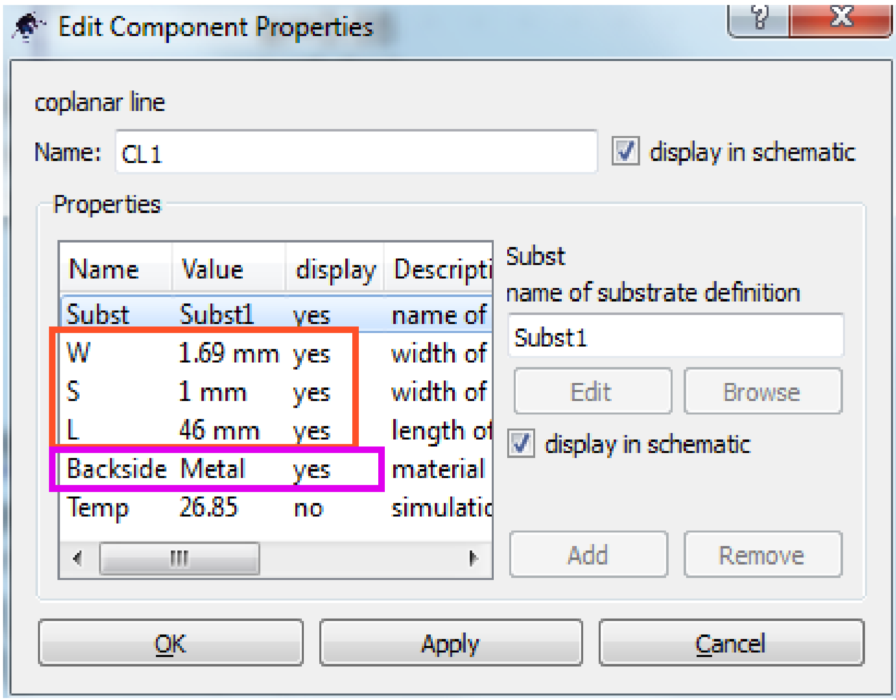
## 12.5.2. Design of the „Grounded Co - Planar Wave Guide“

We still need to complete the simulation the properties of the „grounded co planar wave guide“ with Z = 50 Ω. This wave guide starts at the input socket, is interrupted for the input coupling capacitor by a „gap“, an another gap for the MMIC and a third gap for the output coupling capacitor.
(..the shielding by the ground plane at the left and at the right reduces the sensitivity to undesired couplings or radiations).

Please open „Tools“ and „line calculation“, then switch to „Grounded Co planar Wave guide“.
First enter the substrate properties in the pink frame, followed by „1 GHz“ for the design frequency.
Now enter ***Z = 50Ω*** and an electrical length of **90 degrees** for the line.
Before clicking on „Synthesize“ enter a **„Separation S = 1 mm** for the gap at the left and the right of the central conductor (upper right red frame).
Simulation result is a central line width of **1.69 mm**.




Do not hesitate to test the result by a short simulation. Connect a Microwave Port to the line's input and terminate the line's end by a resistor of 50Ω. Sweep from 1 GHz to 6 GHz.

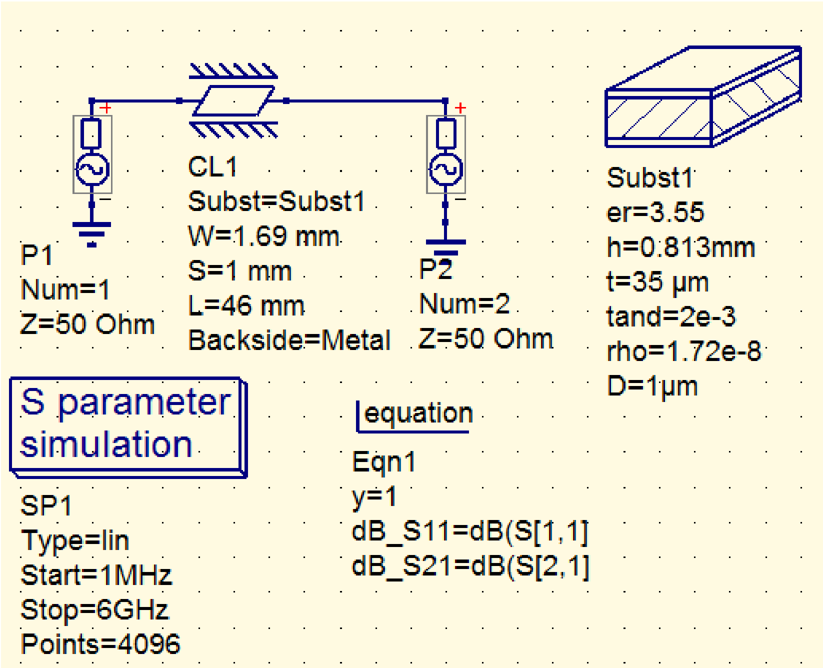



Look for the **„Coplanar Line“ in „Components / Transmission Lines“.**

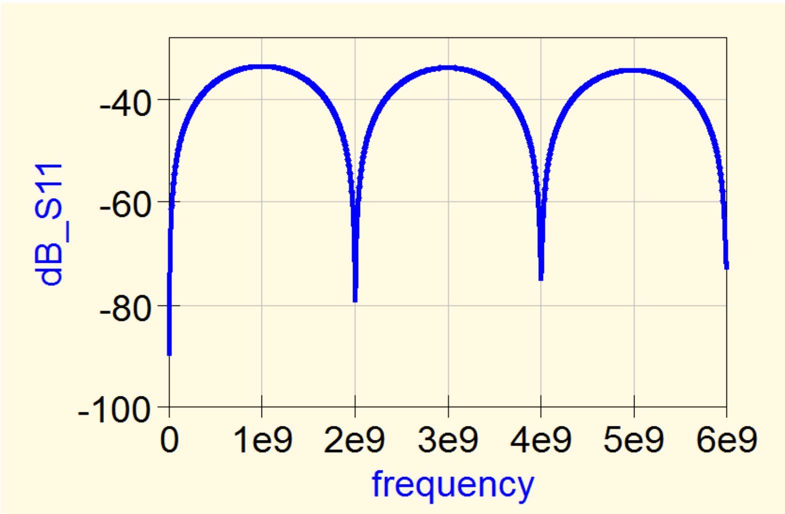
But when editing the properties don't forget not only to enter the line dimensions and the separation „S“, but also to set

**Backside Metal = yes**

Make all properties visible.

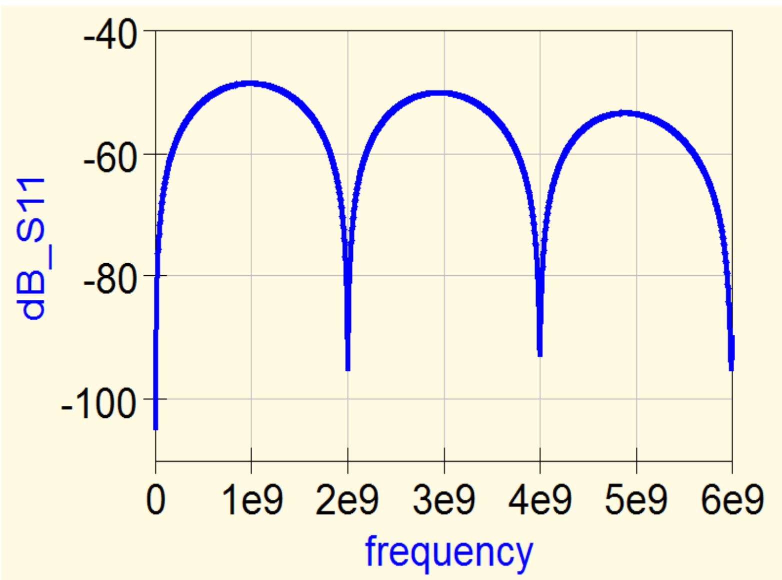



This is the simulation....




...and this is the result for S11.

Not perfect – could be better, because S11 rises up to -35 dB.




But if you choose a **line width of 1.74 mm instead of 1.69 mm,** you reach the goal. S11 is now smaller than -50 dB.

## 12.6. The complete Circuit

### 12.6.1. S Parameter Simulation

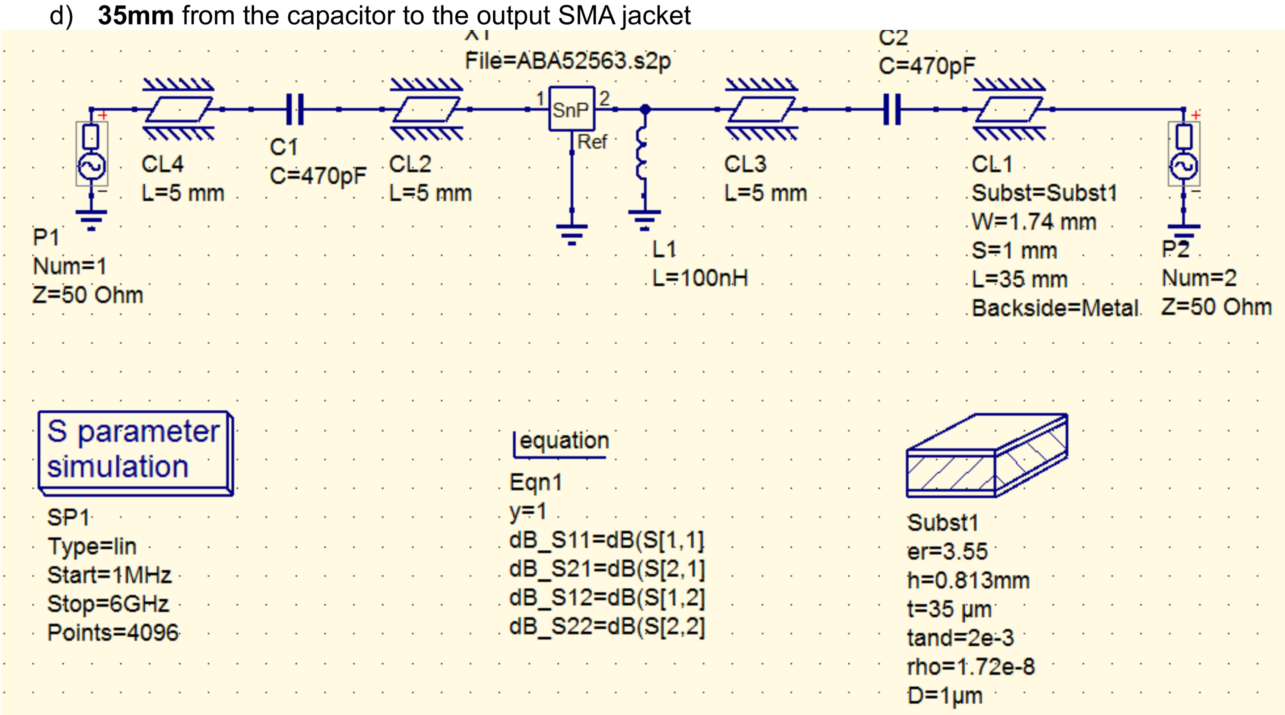
We open the simulation schematic of chapter 11.3 and save it under a new name.
Then four grounded co planar wave guides with the following lengths must be inserted:

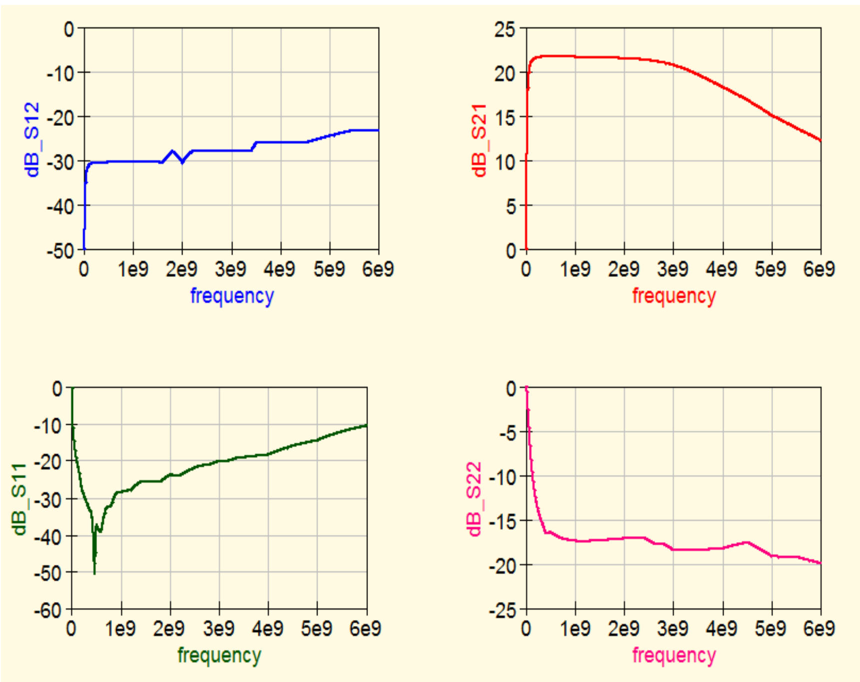
a) **5 mm** from the input SMA socket to the first decoupling capacitor with 470 pF

b) **5mm** from this capacitor to the input of the MMIC

c) **5mm** from the output of the MMIC to the output decoupling capacitor with 470 pF

d) **35mm** from the capacitor to the output SMA jacket




Use all line and substrate data value of the last chapter.

Then simulate from 1 MHz to 6 GHz.

Result:
no difference to the last simulation.

## 12.6.2. The Rollet Stability Factor „k“

A very important property of an amplifier is safety against self oscillating. This check can be done by calculating the „Rollet Stability Factor from the S parameters of the circuit. This is a little bit complicated – but our PC will do the job.

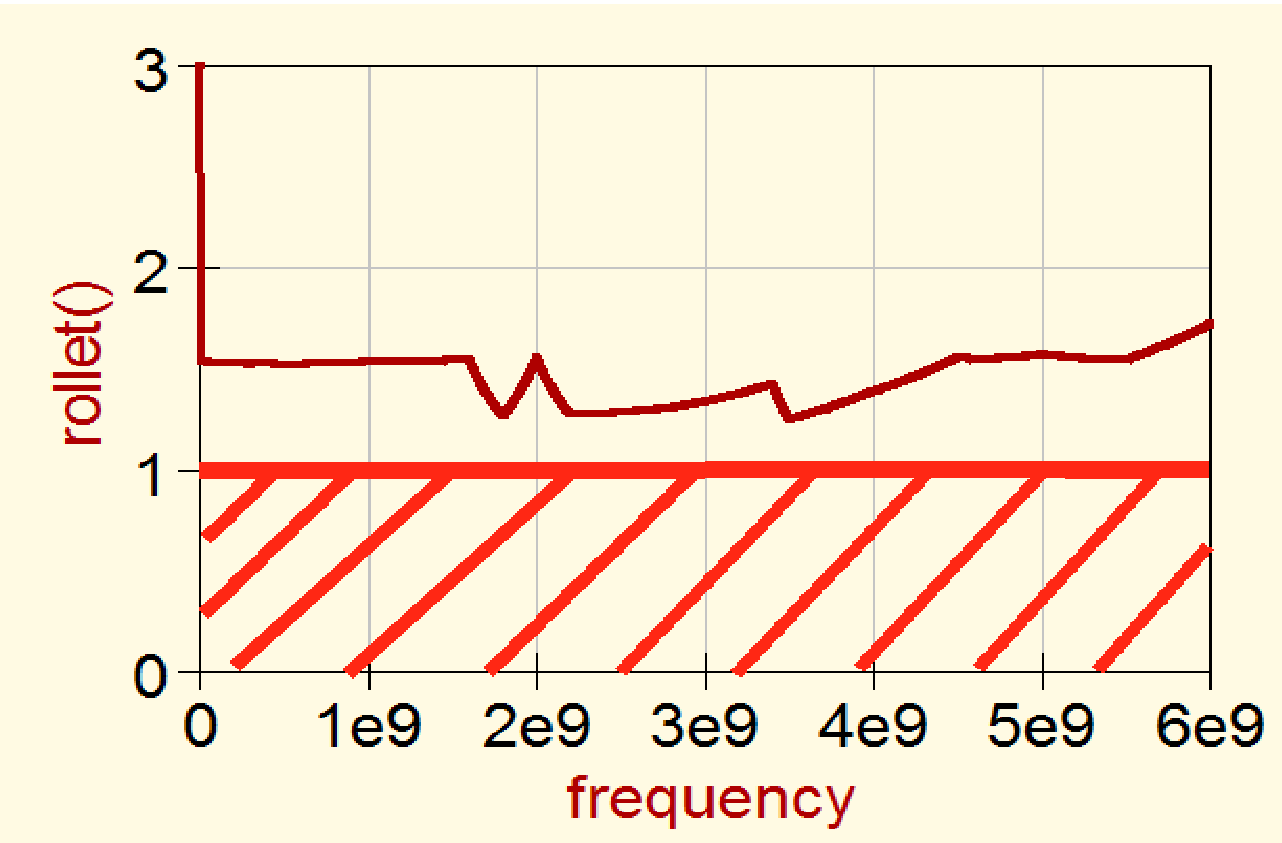
There exists a simple rule:

**This stability factor „k“ must always (in any case and at every frequency) be greater than 1 to avoid self oscillating!**

So simulate, open a Cartesian diagram and enter as property:

**„rollet()“**

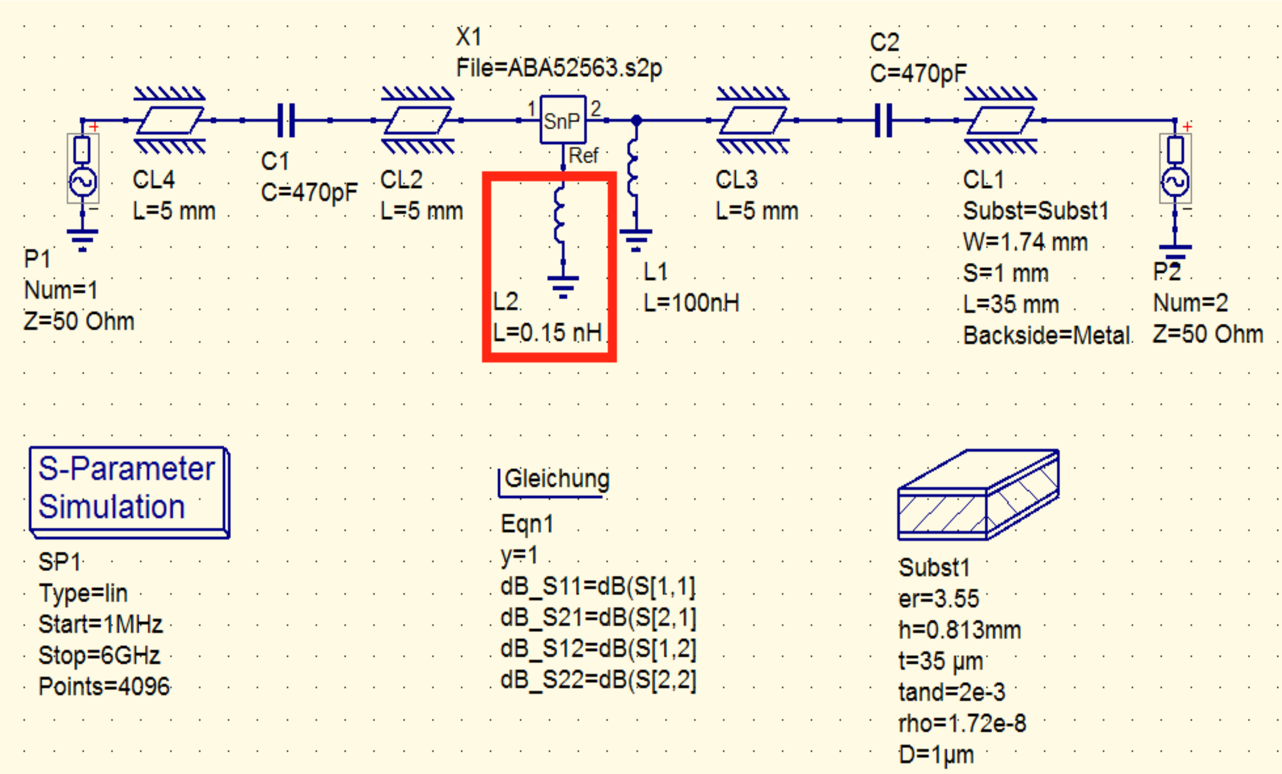
After pressing „Apply“ you get this diagram and you see, that everything is OK up to 6 GHz (= k is always greater than 1).

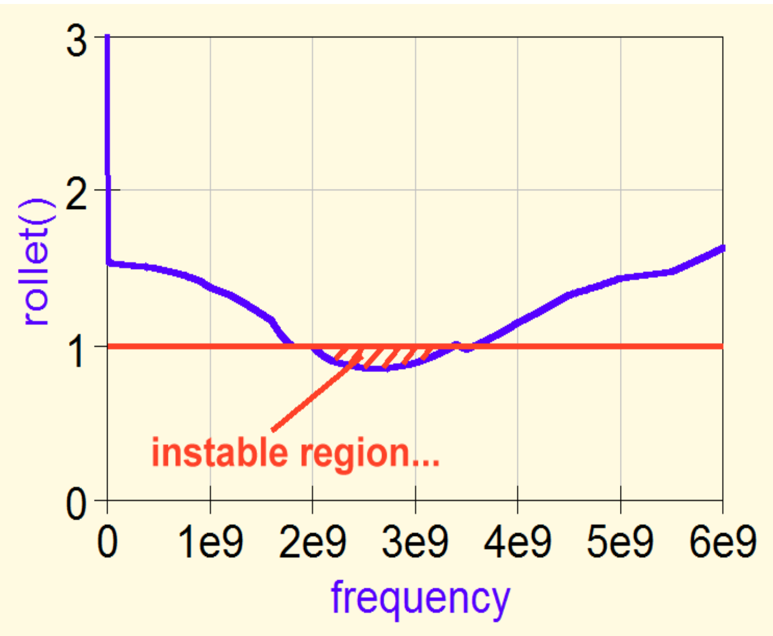


**A usual reason for self oscillating is the connection from the „Ref" -point of the MMIC to ground by the collection of via**

**If this connection is not realized by as many via as possible, k will fall below 1....**

Let use simulate for 7 used via in parallel. Each via equates an inductance of ca. 1nH for a PCB height of 1 mm. So the complete parallel via connection is an inductance of **L = 0.15 nH**




This says the simulation... and the first built up prototype oscillated at 2.2 GHz.

The problem was cured by increasing the number of via at each side of the MMIC.D

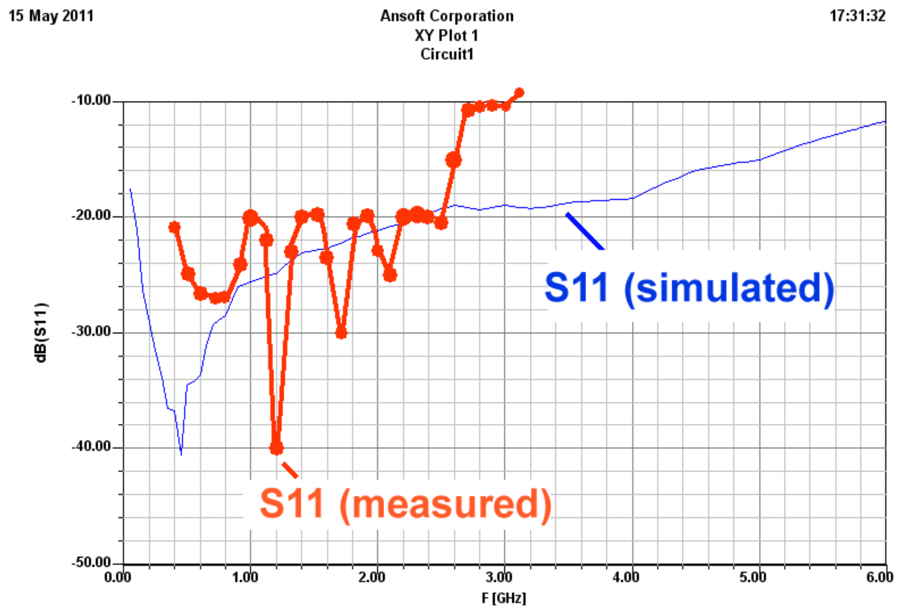
**For further information and exact knowledge of critical impedance values at the input or output side you have to simulate and analyze the „Stability Circles"**.

We will deal with that in a later project.

## 12.6.3. Hour of Truth: what says the Vector Network Analyzer?

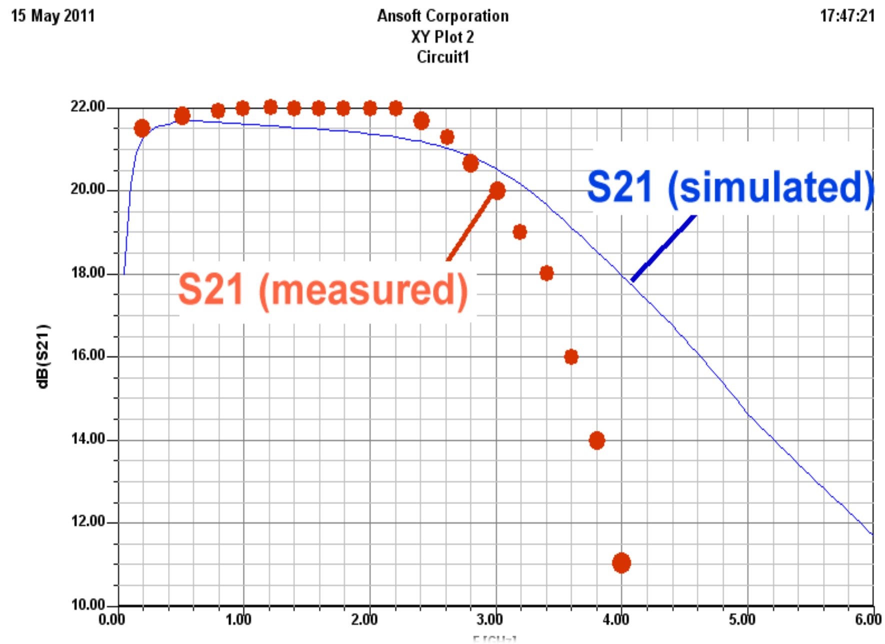
This chapter is part of a published article in the german „UKW Berichte“ journal.
The simulations were done with the free software „Ansoft Designer SV“

**Let's start with S11:**




Up to 2.5 GHz simulation and measuring do coincide.

But the desired specification (smaller than -10 dB) is reached in this range.



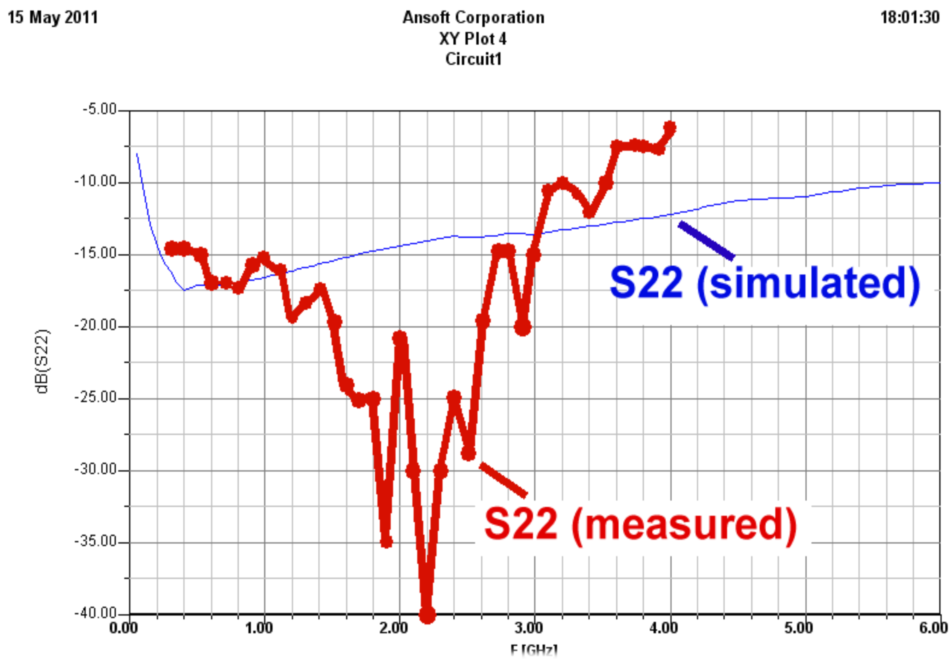

**S21** shows deviations to the simulation due to the self resonance of the 100 nH inductance at the MMIC output at 2.2 GHz.

For higher frequencies this inductance is only a simple capacitor with decreasing impedance.

**The inductance value should in this case be reduced, but this increases automatically the lower cutoff frequency.**

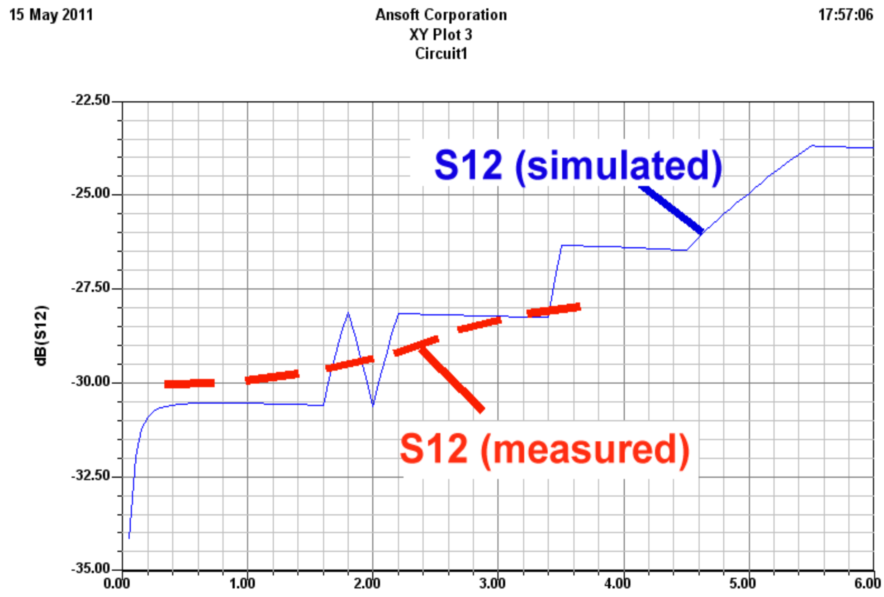
**The measured lower cutoff frequency (= corner frequency) was f = 44 MHz for a gain reduction of -3dB**

And now the output reflection **S22**:




Very nice and convenient...

The last parameter „**S12**" is also OK:




That was it!

Now the PCB is ready for production.
And the alumina case does not effect the measured properties.

# 13. A Low Noise Amplifier for 1 Ghz...1.7 GHz with a maximum Noise Figure of 0.4 dB

## 13.1. Overview

The development of integrated microwave amplifiers (MMIC's) improves permanently their properties. Today the construction of an LNA is very interesting  when regarding the advantages:

| |
|---|
| Input and output already matched to 50Ω-Betrieb<br><br>The noise figure NF has now dropped below 0.5dB in a 50Ω system<br><br>The gain for one stage is typically S21 = ca. 20dB at 2 GHz<br><br>Only a minimum of additional external parts necessary |

But you should also know the difficulties and problems:

| |
|---|
| These SMD are very small (= package with 2mm x 2mm, but with 8 pads = 4 on two opposite sides plus a little extra ground pad in the center on the lower side.). Thus PCB design and soldering need efforts and attention<br><br>Only SMD 0603 parts (or smaller versions) are used.<br><br>The transit frequency has meanwhile raised up to 10 GHz or more. So the stability of the design is an important point.<br><br>To achieve the high transit frequency value of 10 GHz you need a high quiescent current in every stage (typically 50 mA) and you must stabilize it.<br><br>The thickness of the PCB must be reduced to 0.25mm to avoid undesired propagation modes for the signals traveling on the the board. Additionally a lot of via should be applied for grounding to avoid self oscillation. These via cannot longer be realized by silver plated rivers. |

This amplifier shall be used in the 23 cm band (= ca. 1300 MHz) and for the reception of Meteosat weather satellite signals at 1691 MHz. Thus the following properties are desired in the frequency range from 1 to 2 GHz:

**Noise figure :        maximum 0.4 dB**

**gain (S21):            ca. 20 dB**

**Absolute stability (k higher than 1 up to 10 GHz)**

The Agilent Avago part „MGA-635P8" does fulfill these specifications.

## 13.2. Starting with an Application found in the Data Sheet

Have a look at the data sheet and two application notes found in the Internet. There you find application examples for 2500 MHz and 3500 MHz using the same fundamental circuit and the same layout with the necessary modifications [1] [2] [3].

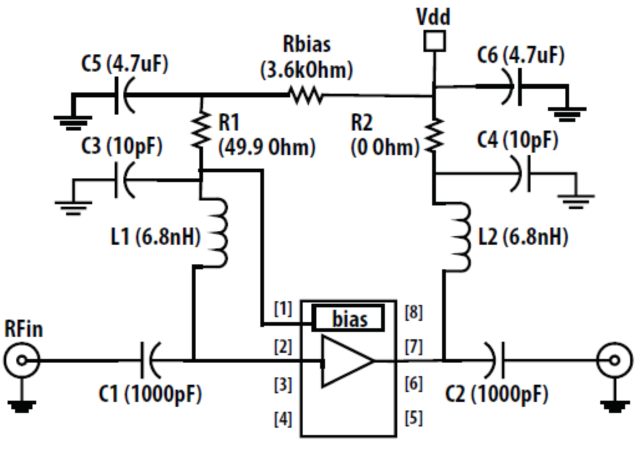


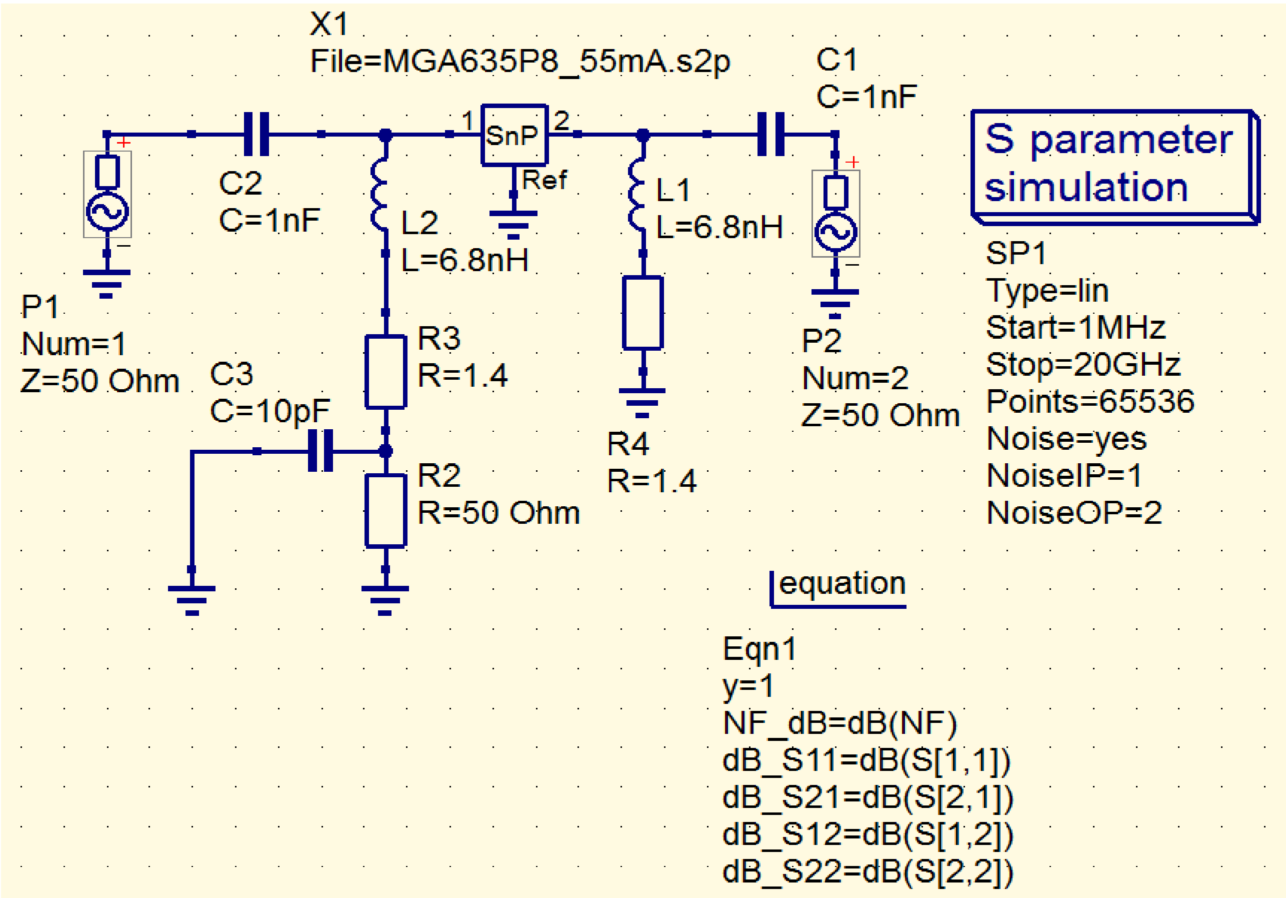
Figure 6. Demo Board Schematic Diagram

This is the schematic.

The circuit works with a supply voltage of Vdd = +5 V and contains in the MMIC case a GaAs pHEMT cascode amplifier.
Pin 1 is the bias pin and with an Rbias = 3.6 kΩ the correct operating point (I = 55 mA) is set.

A great problem of HEMT parts is the stability a low frequencies = the problem of self oscillating. This can be corrected with a simple trick: the resistor R1 Is more and more active at pin 2 for low frequencies because the impedance of L1 decreases and the impedance of C3 increases with decreasing frequency. Thus the self oscillating tendency is reduced and suppressed. Additionally a low inductance of L2 (6.8 nH) reduces the gain with decreasing frequency.

The resistor R2 with a value of zero Ohms is ignored and then the circuit simulated.

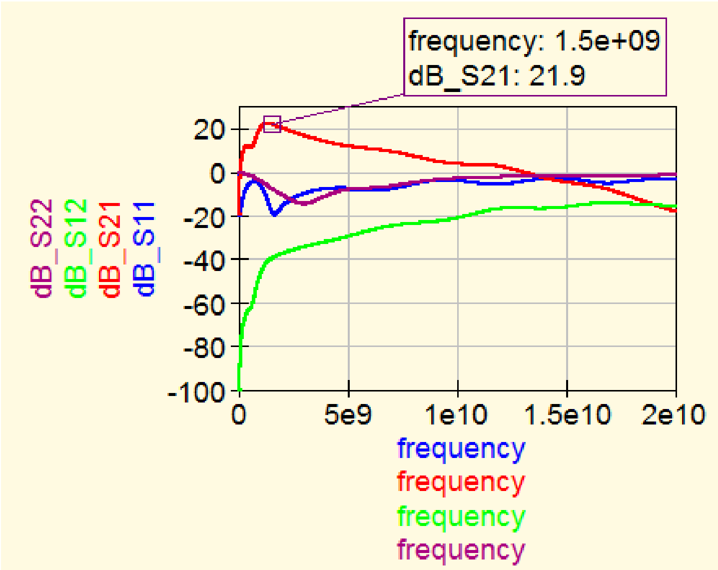



The S parameter file „**MGA635P8_55mA.S2P**“ comes from the **Agilent Avago homepage.** Copy it into a new folder for this project.
Inductance L1 and L2 (6.8 nH) are SMD 0805 versions with a quality factor Q= 30 at 1 GHz. The losses are represented in the simulation by a series resistor of 1.4 Ω.

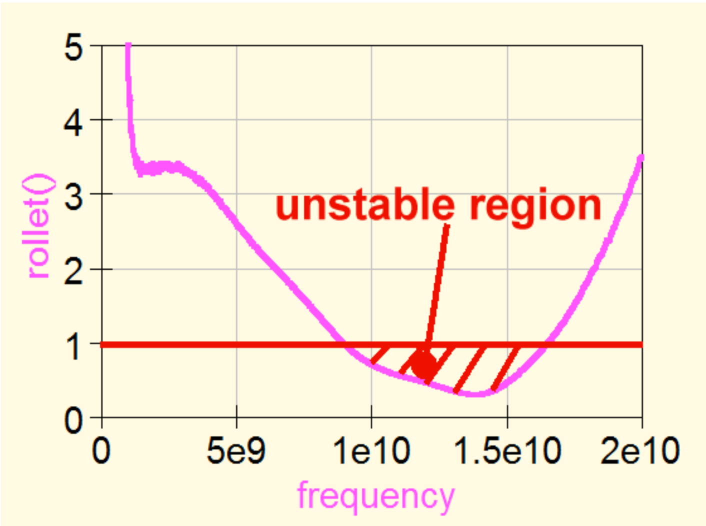
S parameter conversion to “dB” is done by the following equations:
**dB_S11=dB(S[1,1])**
**dB_S21=dB(S[2,1])**
**dB_S12=dB(S[1,2])**
**dB_S22=dB(S[2,2])**

and the Noise Factor calculation is already prepared.

Simulation range is 1 MHz to 20 GHz (...because this is the highest frequency in the S parameter file).




This is the S parameter collection of the circuit. The marker is set to 1.5 GHz and shows a gain of S**21 = 21.9 dB.**

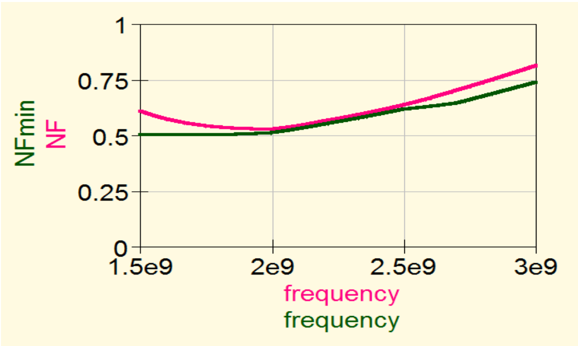



Also the stability is checked up to 20 GHz.

But the development engineer at Avago was not very careful: he ignored to check the frequency range above 8 GHz and so we find an unstable region between 8 and 17 GHz...

```
! Noise Parameters
! Freq      FMIN     GAMMA OPT      Rn @
! (Hz)      (dB)    Mag       Ang   50 ohm
1.900       0.40     0.200     95.5        0.0482
2.000       0.42     0.206     96.4        0.0552
2.200       0.49     0.205    113.2        0.0456
2.500       0.58     0.216    128.8        0.0472
2.700       0.62     0.214    163.5        0.038
3.300       0.82     0.292    172.7        0.038
3.500       0.77     0.289    174.7        0.0426
```

The S parameter file contains also some information for the simulation of the minimum and the actual Noise Figure. So open a Cartesian diagram to show the simulation of the two parameters in the frequency range from 1.9 to 3.5 GHz.




It seems that the circuit is optimized for the WLAN frequency range at 2.4 GHz.

Thus some efforts will be necessary to get the circuit optimally running in the range from 1.... 2 GHz.

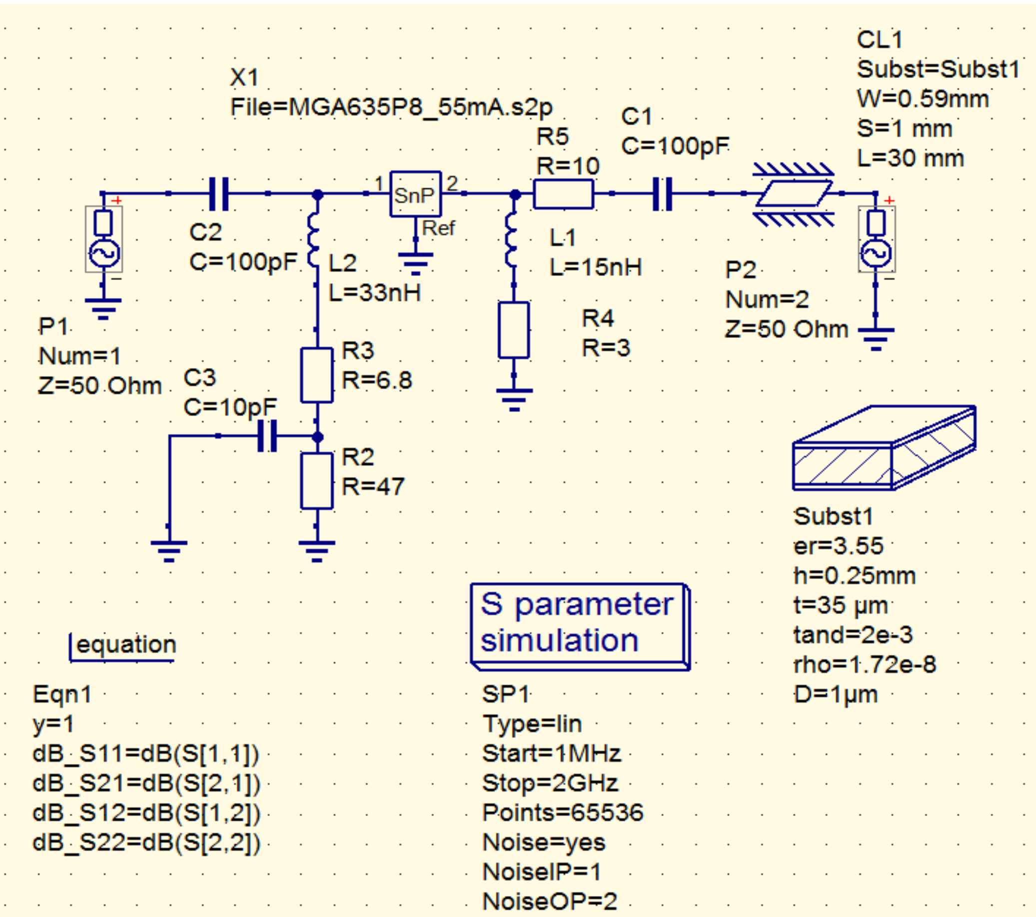
## 13.3. Modifying the Amplifier Circuit for the 1....1.7 GHz Range

The inductance values of L1 and L2 were step by step increased and the new simulation result of the Noise Figure NF examined.
R3 and R4 represent the inductor losses.
C3 had also to be changed.

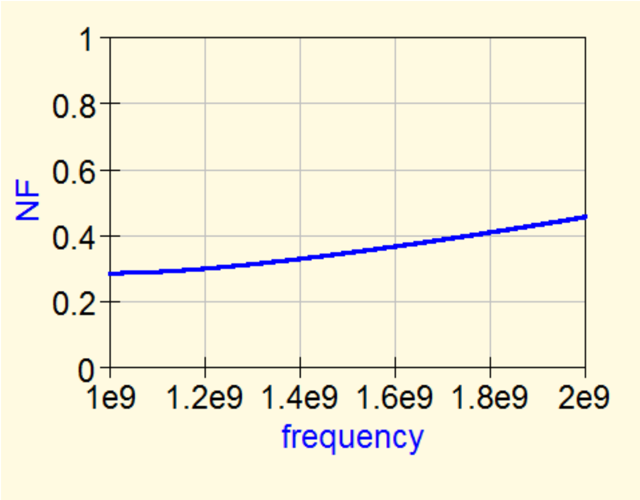
A minimum value of NF = 0.4 dB was achieved with **L1 = 33 nH / L2 = 15 nH.** For sufficient stability a small resistor with 10 Ω was added to the output pin.
Due to these actions the gain (= S21) fell to 20 dB at 1.7 GHz.
Also C1 and C2 had to be reduced to 100 pF to increase the lower cutoff frequency up to 1 GHz.
For the “Grounded Co planar Wave guide“ with Z = 50 Ω at the output the qucsstudio line calculator proposed

**a line width of 0.59 mm and**
**a gap of 1mm at each side of the central line**

for a Rogers RO4003 PCB with a height of 0.25 mm. The necessary line length of this PCB is 30 mm.

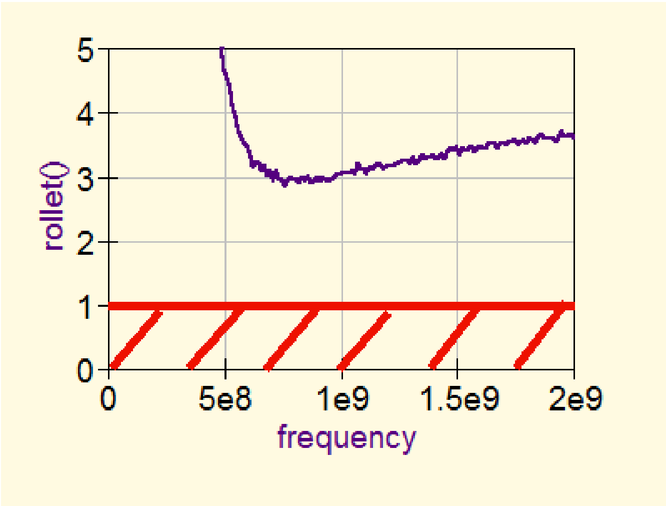



This is the final simulation schematic.



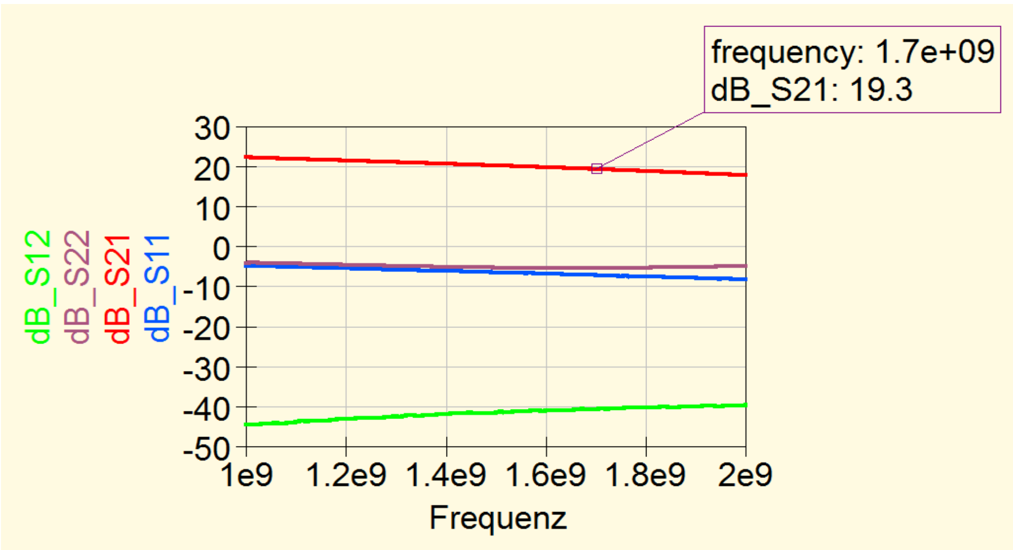

This is the simulated noise for f = 1...2 GHz.

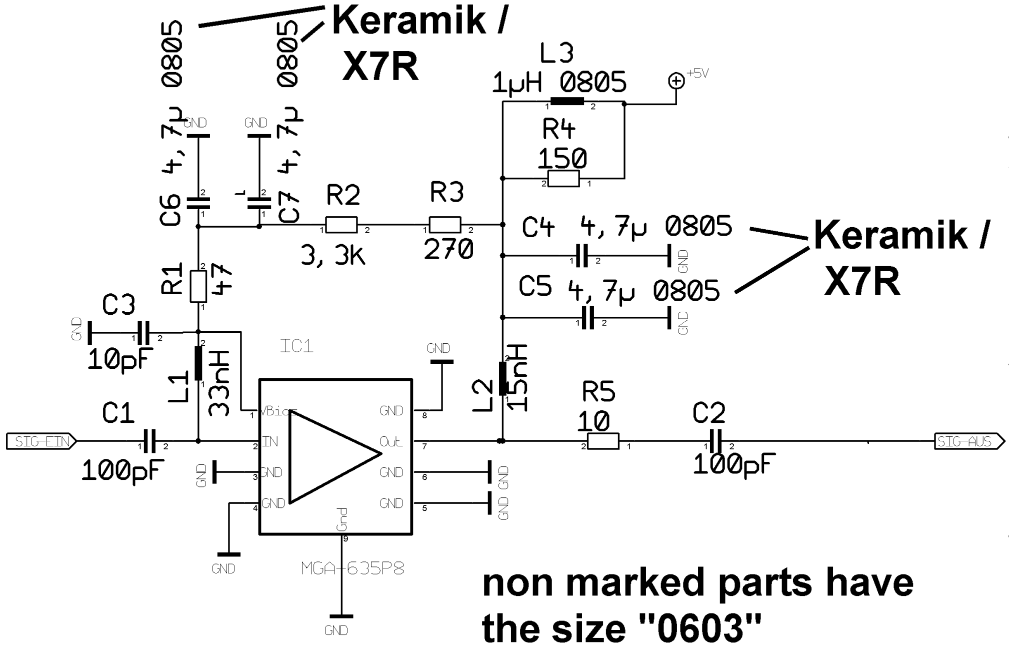
Up to 1.7 GHz the Noise Figure NF is smaller than 0.4 dB




Also the stability is no problem up to 10 GHz (= k is greater than 1)

These are the S parameters from 1....2 GHz. Sorry, but S21 does (with 19.3 dB) not reach the foreseen goal (= 20 dB). But lower noise is more important...

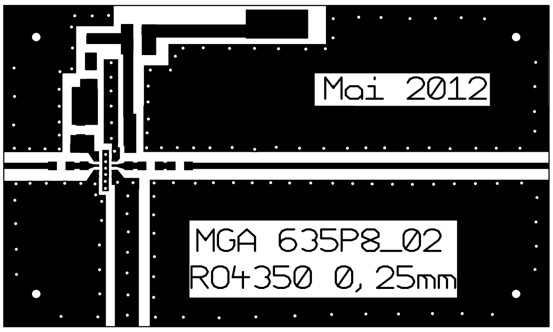





The realized circuit includes broadband filtering for the supply voltage.

A Rogers RO4350B PCB is used instead of RO4003 material (...RO4350B is an improved version of RO4003, but non flammable).
PCB height is 10 mil = 0.254 mm, copper plating is 35µm on both sides.

Outer dimensions of the PCB are 30 mm x 50 mm.



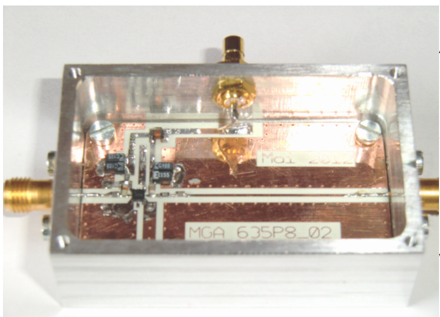

The “grounded coplanar waveguide from the input in direction to the output can easily be recognized, also the “gaps” for the input and the output coupling capacitor, the MMIC and the 10Ω resistor to improve the stability.

The central ground pin at the lower side of the MMIC case demands an own “grounding island” with a width of 0.6mm and a lot of via.

Every point of the schematic which must be grounded is located on an own “grounded by via” island to avoid self oscillating. Via diameter is 0.3 mm.

## 13.4. The Prototype

Due to the very thin PCB (height = 0.25 mm) a professionally manufactured and plated through version was necessary. This was perfectly done by an Enterprise at Munich after mailing the Gerber Plot of the design, but 4 PCBs cost 235 Euro...
Still some remarks to the Rogers material RO4350B in comparison to RO4003.
The additives for the property “not flammable” reduce the electrical quality a little bit:
At 10 GHz you get a loss tangent of 0.0027 for RO4003 but 0.0037 for RO4350.
Also $\varepsilon_r$ increases a little – but please download the data sheet from the Rogers homepage for exact information.

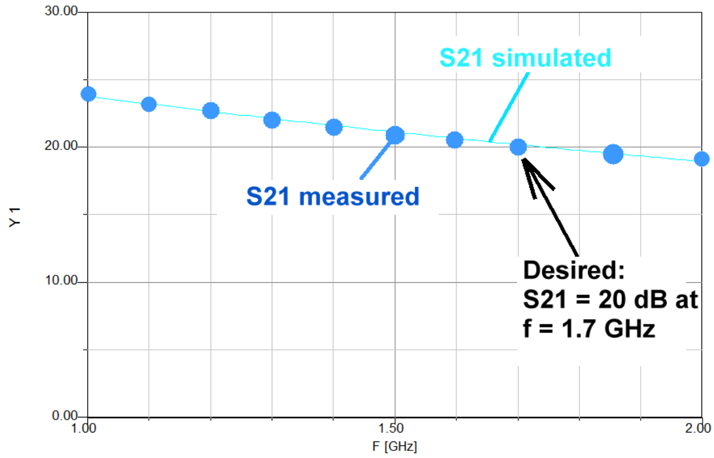


And this is the final result after installing the PCB in the milled alumina case (outer dimensions: 35 mm x 55 mm).
On the upper side you see the SMB jack for the Power Supply voltage of +5V
O

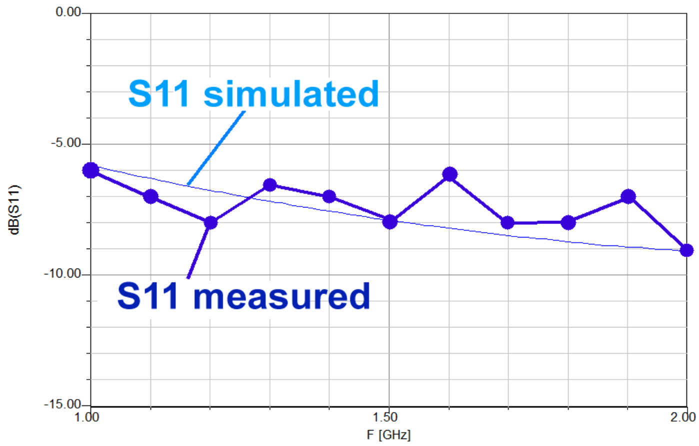
Input and output are realized by SMB jacks with flat center conductors which are directly soldered to the microstrip lines (...gives a minimum of reflection).

## 13.5. Measured S Parameters

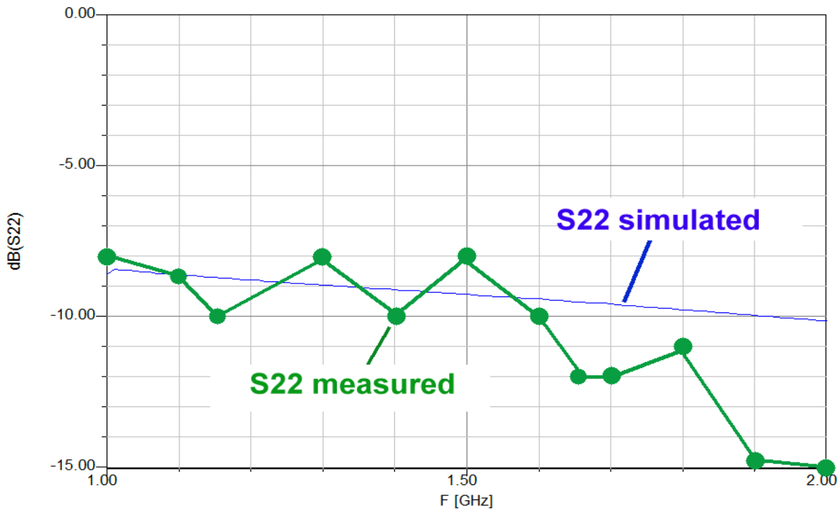
I used the well known Vector Network Analyzer hp8410 and the S parameter test set (hp8745A). But a 20 dB attenuator pad was connected to the input of my amplifier to avoid overload.




**S21: no comment.**




**S11** is as good as desired

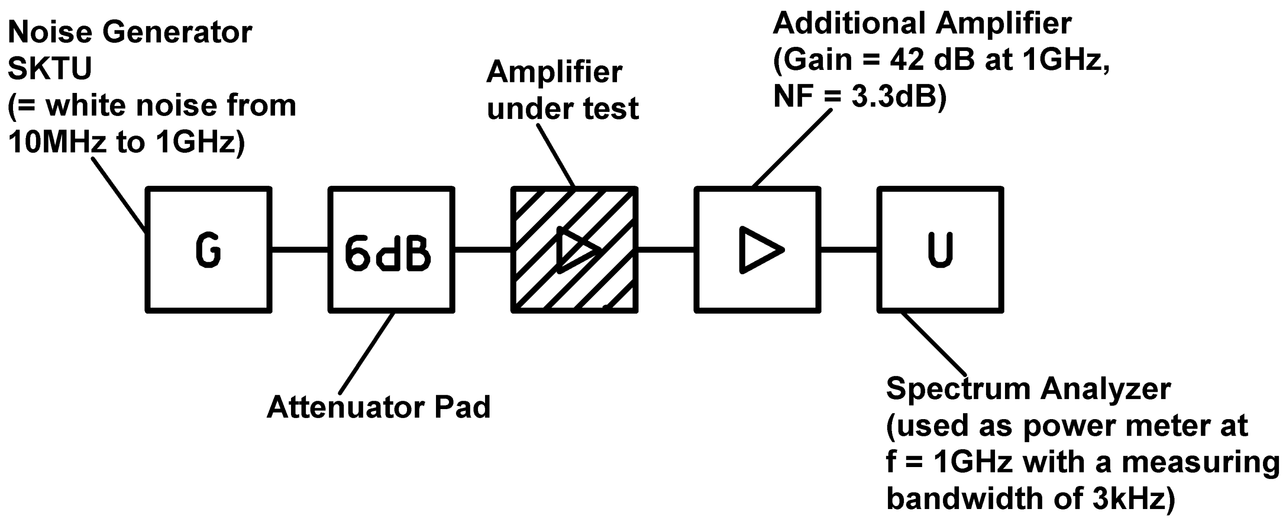



**S22** is better than simulated for frequencies higher than 1.5GHz

**S12** was measured to ca. -44dB in the frequency range between 1…..2 GHz and is so a little bit smaller than simulated.
Therefore no picture is necessary...

## 13.6. The Noise

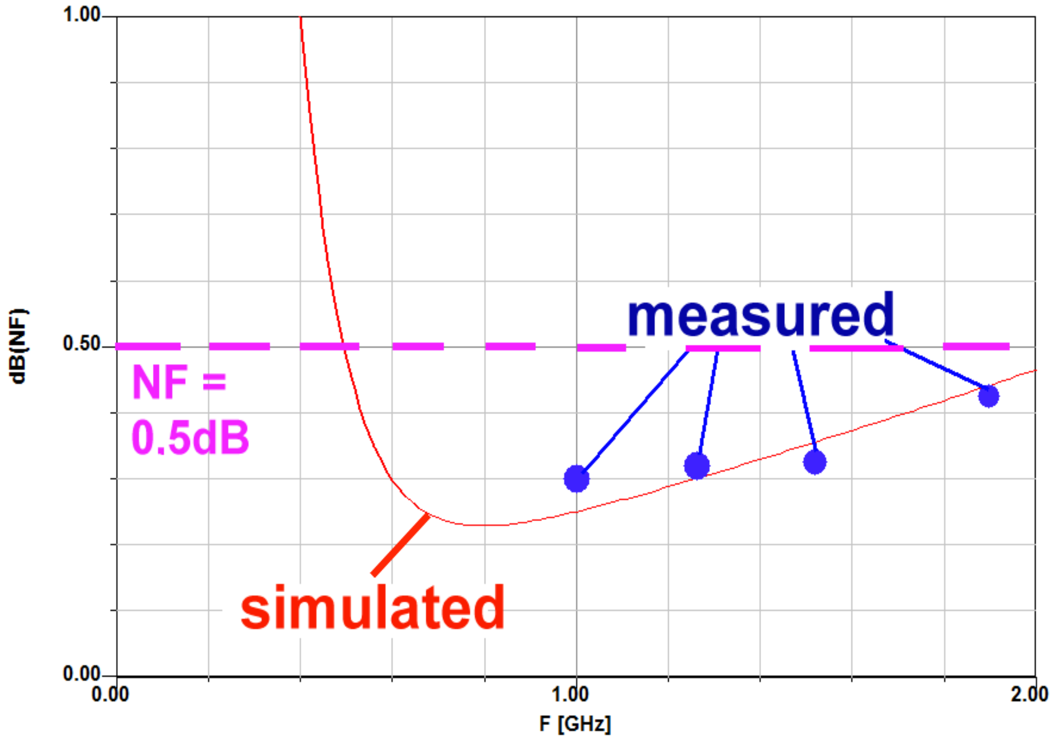
Not every electronic homeworker has a Noise Figure Meter in his workshop. Especially for such small noise figures of 0.4 dB the equipment is very expensive. So it was necessary to try some historic measurement principles with the material found in the own cave workshop for the frequency of 1 GHz. This “measuring set” was used:




The principle is well known and not complicated:
Increase the noise level (delivered by the SKTU) as long as the **output noise power (indicated by the power meter) doubles (= increase of 3 dB).**
Then read the generator power (indicated on the generator instrument) and reduce this value by the 6 dB of the attenuator pad.

### The result is a NF value of ca. 0.4...0.5 dB

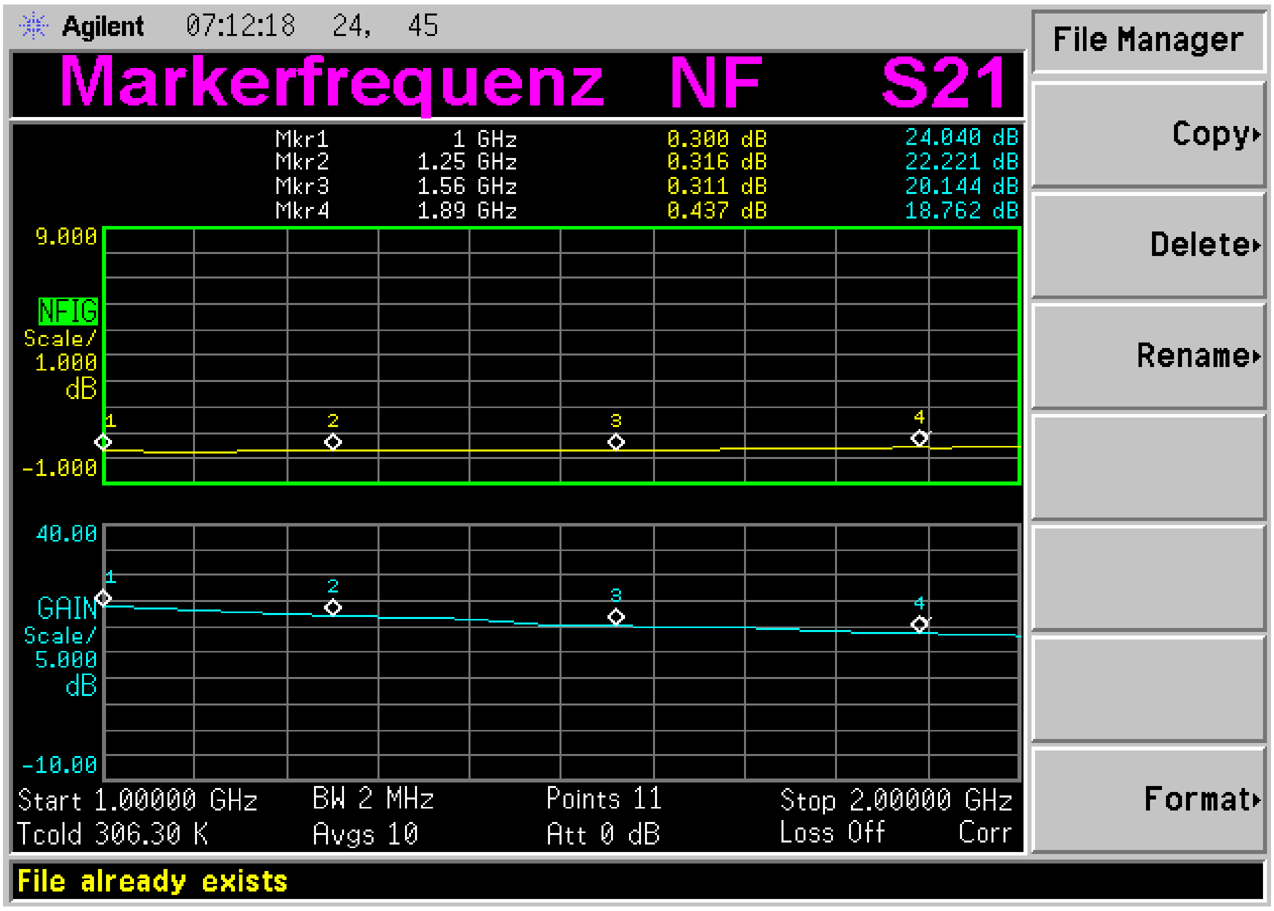
**But more accurate measurements can only be done by friends which own a modern noise figure test set!**




Ulli Kafka (with his own enterprise “Eisch Elektronik”) did this job and mailed me this result with the comment “a fine and top amplifier”.

**You can see that the noise figure starts with a value of 0.3 dB at 1 GHz and at 1.7 GHz exactly 0.4 dB are reached.**

This protocol was attached to the email containing all interesting information.




For specialists a further information found in the data sheet of the **MGA-635P8 for I = 55 mA at 2.5 GHz:**

| | | |
|---|---|---|
| **Output IP3** | **=** | **+35.9 dBm** |
| **P1dB_out** | **=** | **+22 dBm** |

If you now think about the properties of a two stage version....there is enough room left on the right half of the PCB for a second stage....

Literature:

[1]: Data sheet and S parameter filed for the MGA-635P8 coming from the Avago Technologies homepage

[2]: Application Note of Avago: “MGA-635P8 GaAs ePHEMT MMIC 2.5 GHz Low Noise Amplifier with Superior Noise and Linearity Performance”

[3]: Application Note of Avago: “MGA-635P8 GaAs ePHEMT MMIC 3.5 GHz Low Noise Amplifier with Superior Noise and Linearity Performance“

# 14. RF Mixer

## 14.1. Introduction

Remember:

**An Audio Mixer adds two (or more) signals, but an RF Mixer will multiply them!**

**Thus an RF Mixer is a „Threeport“ with two input ports and one output port.**

By the multiplication process the frequencies of the input signals are changed (= frequency conversion) without damaging the information content.

The names for the ports are:

a) **RF = radio frequency signal input with small amplitude**.

b) **LO = local oscillator signal input. This signal is necessary to convert the frequency. The amplitude of the LO signal has always a high value.**

c) **IF = intermediate frequency output. Here we find the conversion result.**

The multiplication process of the input signals is based on this mathematical law:

$$\cos(\alpha)\bullet\cos(\beta)=\frac{1}{2}[\cos(\alpha+\beta)+\cos(\alpha-\beta)]$$

Regarding the right side of the equation we find:

**At the IF output both input signals have disappeared! You will now find the sum and the difference of both frequencies as new signals.**

In practice you have to distinguish between three different mixer groups:

a) **True multipliers**

b) **Additive mixers circuits**

c) **Switched mixers. This can be single balanced or double balanced mixers which use a rectangle voltage as LO signal**

Important:
**"True Multipliers" work with the highest possible linearity and try to apply the mentioned mathematical law as perfect as possible (with a minimum of distortion).**
But this is only possible up to several MHz.
This mixer type accepts equal signal amplitudes on both inputs.
--------------------------------------------------------------------------------------------------------------

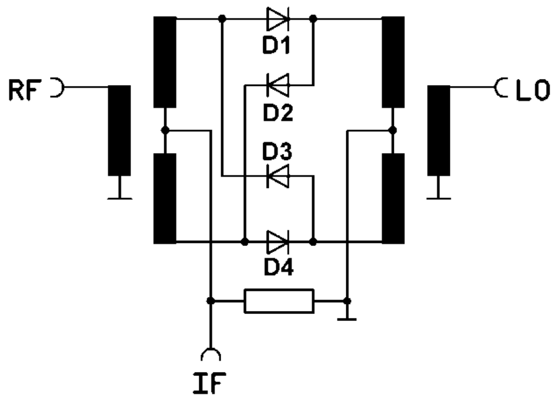
**An "Additive Mixer**" is a very simple affair (e.g. a diode or a transistor or a FET). Here the big LO signal shifts the "operating point" along the nonlinear U / I curve of the part. Thus an added very small RF signal will vary its amplitude due to the different "gradients" of the curve.....and this is like multiplication of RF and LO signal.

**Advantage:** a simple and cheap method. Used up to 100 GHz and more.

**Disadvantage**: linearity problems and distortions for RF signal amplitudes higher than 1 Millivolt. A huge "forest" of undesired lines in the output spectrum with increasing RF amplitude.
-------------------------------------------------------------------

**"Switched Mixers**" are the working horses in the Communication Technique. They are realized as passive or active mixer circuits.

The best and well known **passive mixer is the DBM (= double balanced mixer) or "ring modulator".** The circuit uses 4 fast Schottky diodes and two transformers. Every transformer consists of 3 equal winding. Two winding are in series connection to realize the secondary side with 3 tabs.




Principle:
Diodes D2 and D4 (in series connection) **form an electronic switch** which is "ON" for the positive polarization of the LO signal. Thus the RF signals runs from the primary winding of the left transformer over the lower half of the secondary winding to this switch. It continues its way over the two "switched – on" diodes to the upper and lower end or the secondary winding of the right transformer. 50% of the RF current enter at the upper end into this transformer, the other 50% at the lower end.
But: in opposite directions and so the magnetic fields will cancel in the transformer and he is "not existing for the RF signal". At last the total current reaches the IF output and we can say, **that the RF signal is multiplied by "+1"**

For the negative half of the LO signal diode D1 and D3 form now the electronic switch and the RF signals runs over a path using the upper half of the RF transformers secondary winding. But the voltage polarity is inverted and this means, **that the RF signal is multiplied by "-1".**

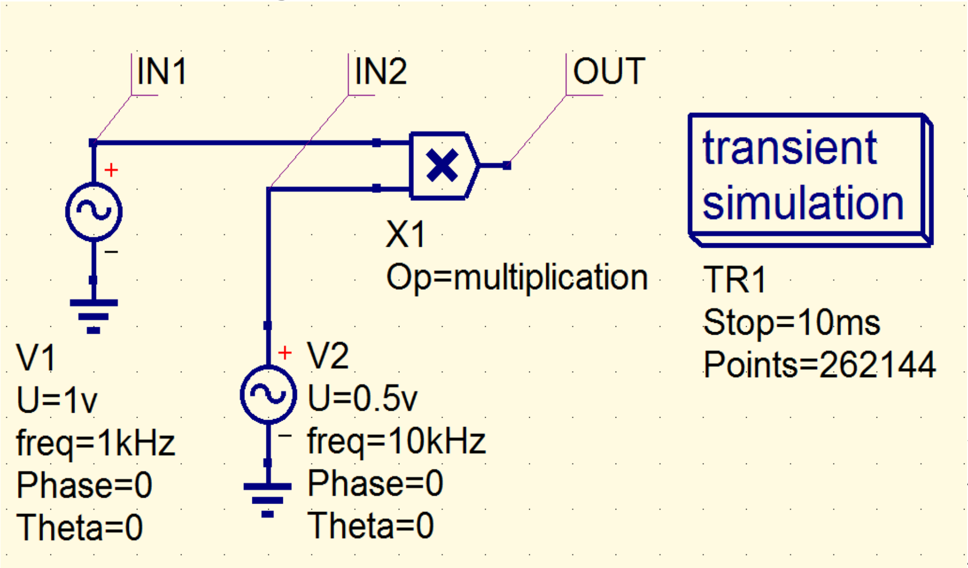
**So the RF signal is multiplied by a rectangle voltage (frequency = LO frequency, amplitude = "+1" and "-1") as desired.**

***Advantages:*** extreme low distortions. Usable from DC up to a lot of GHz. Needs no supply voltage. Standard industry part and thus not expensive.

***Disadvantages***: because no power supply is used, the LO source must deliver the complete energy to switch the diodes ON and OFF. And you should switch as fast as possible – that demands high LO power levels.

No gain possible – only always a typical attenuation of 5....8 dB. This decreases the signal to noise ratio in your system.

## 14.2. Simulating a true Multiplier

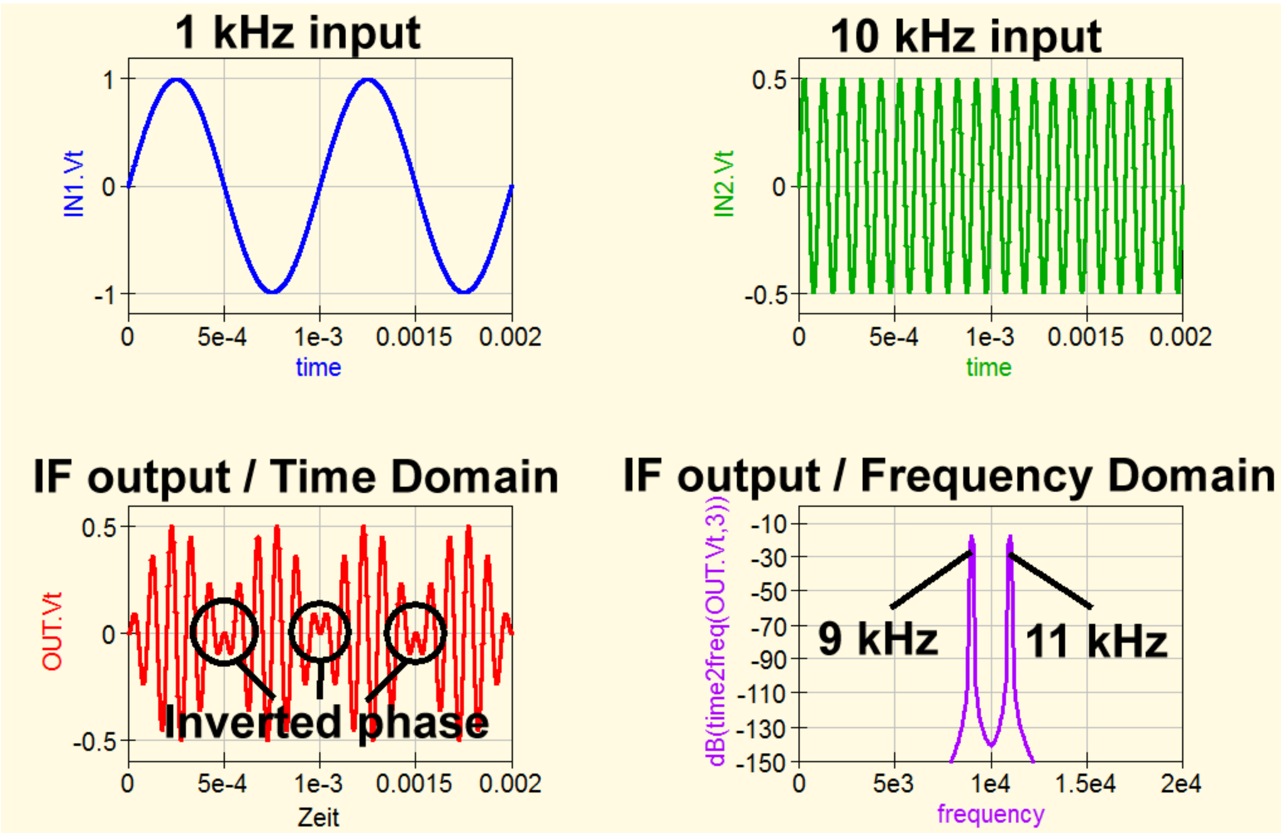



That is no problem because in “**components / system components**” you find the “**Operation**“ part.

Edit the properties and set its function to “**multiplication**“ and “**2 entries**”.

Multiply now an sine voltage V1 (amplitude = 1V / frequency = 1 kHz) by another sine voltage (amplitude = 0.5V / frequency = 10 kHz).

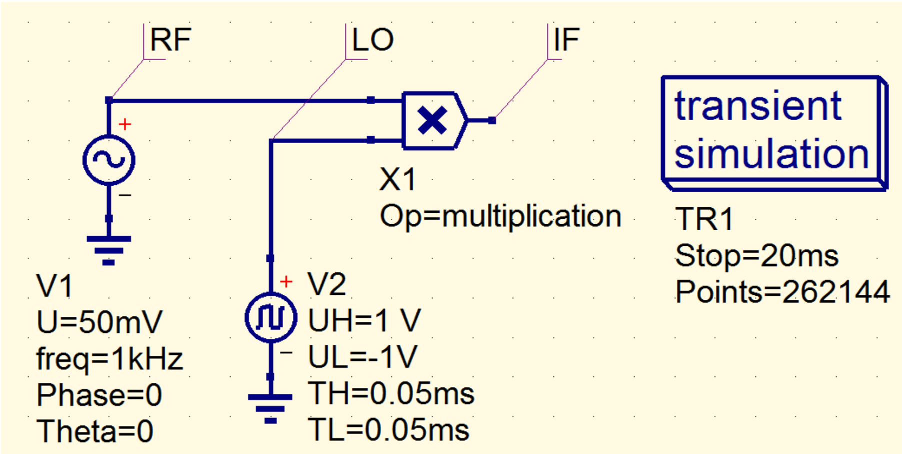
Simulate with the given settings and show the result:




At the IF output you can either choose a Time Domain or a Frequency Domain presentation.
In the Time Domain you see the “phase inversions” caused by the LO signal.
In the Frequency Domain the truth of the mathematics (= generation of sum and difference frequency) is proved.

### 14.2.1. The Multiplier as Switched Single Balanced Mixer

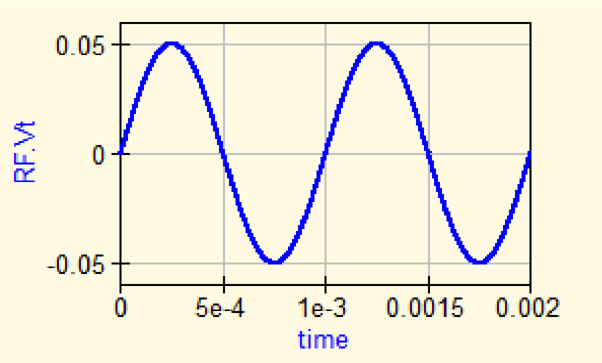
Therefore you need a rectangular voltage for the LO signal.



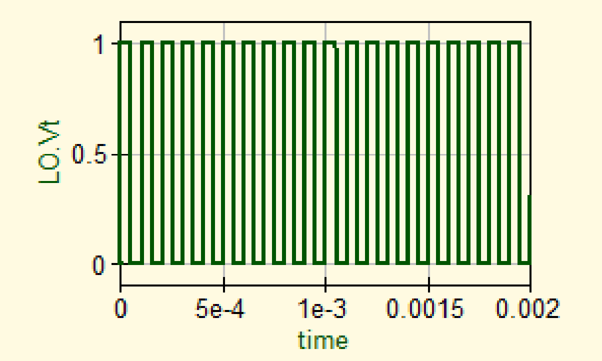

a) The **RF signal** is (like often in reality) a sine signal and has only a small voltage amplitude (lower than 50 mV). Its frequency shall be converted. (Information: you get minor distortions only with such a small amplitude value or less).

b) The „**LO**" signal (= Local Oscillator Signal) is a symmetrical rectangular voltage with a maximum amplitude of "+1 V" and a minimum amplitude of "0 V).

c) The output signal is named „**IF**" **(= Intermediate Frequency).**

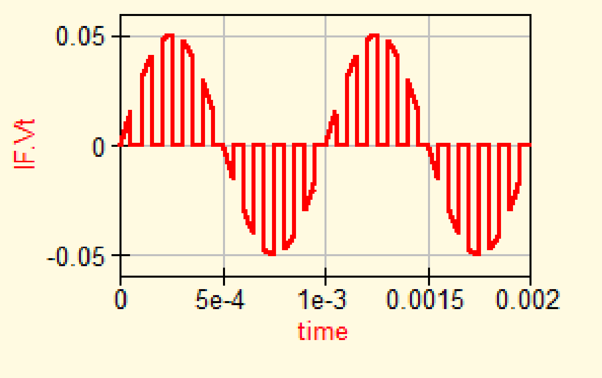



The RF signal with a peak voltage value of 50 mV and a frequency of f = 1 kHz....

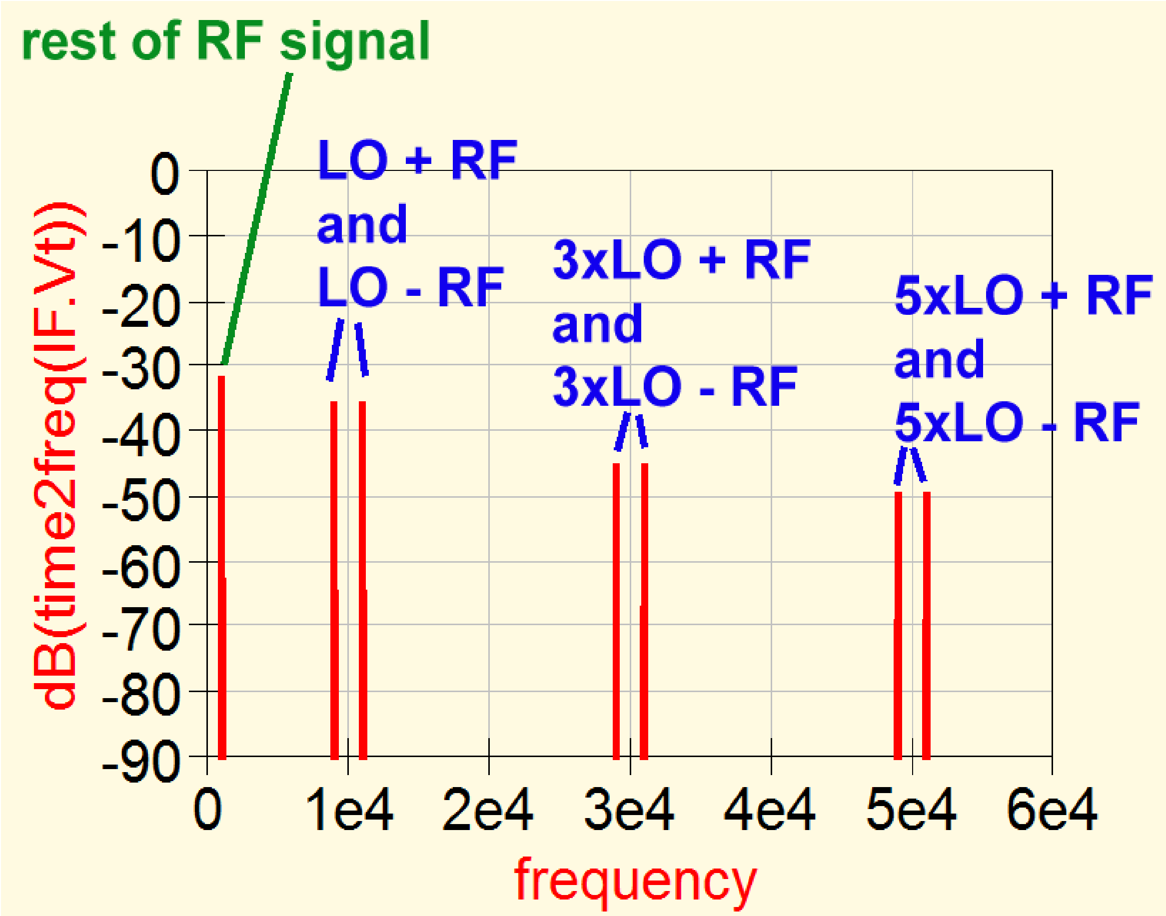



….is by the LO signal acting by an ON / OFF - switch with a frequency of 10 kHz....

….permanently switched ON and OFF.




It will be interesting to examine this signal in the frequency domain.




We find the rest of the RF signal at 1 kHz with a reduced amplitude caused by the switching process.

Then couples of line follow with increasing frequency, located at **10 kHz / 30 kHz / 50 kHz** etc.
The reason is simple:

**The LO signal is a symmetrical pulse signal and consists (due to Fourier) of the fundamental frequency, the triple fundamental frequency etc. (= harmonics).**
**Every harmonic is now multiplied by the RF signal of 1 kHz and the multiplication result is in every case the sum and the difference frequency of both frequencies.**

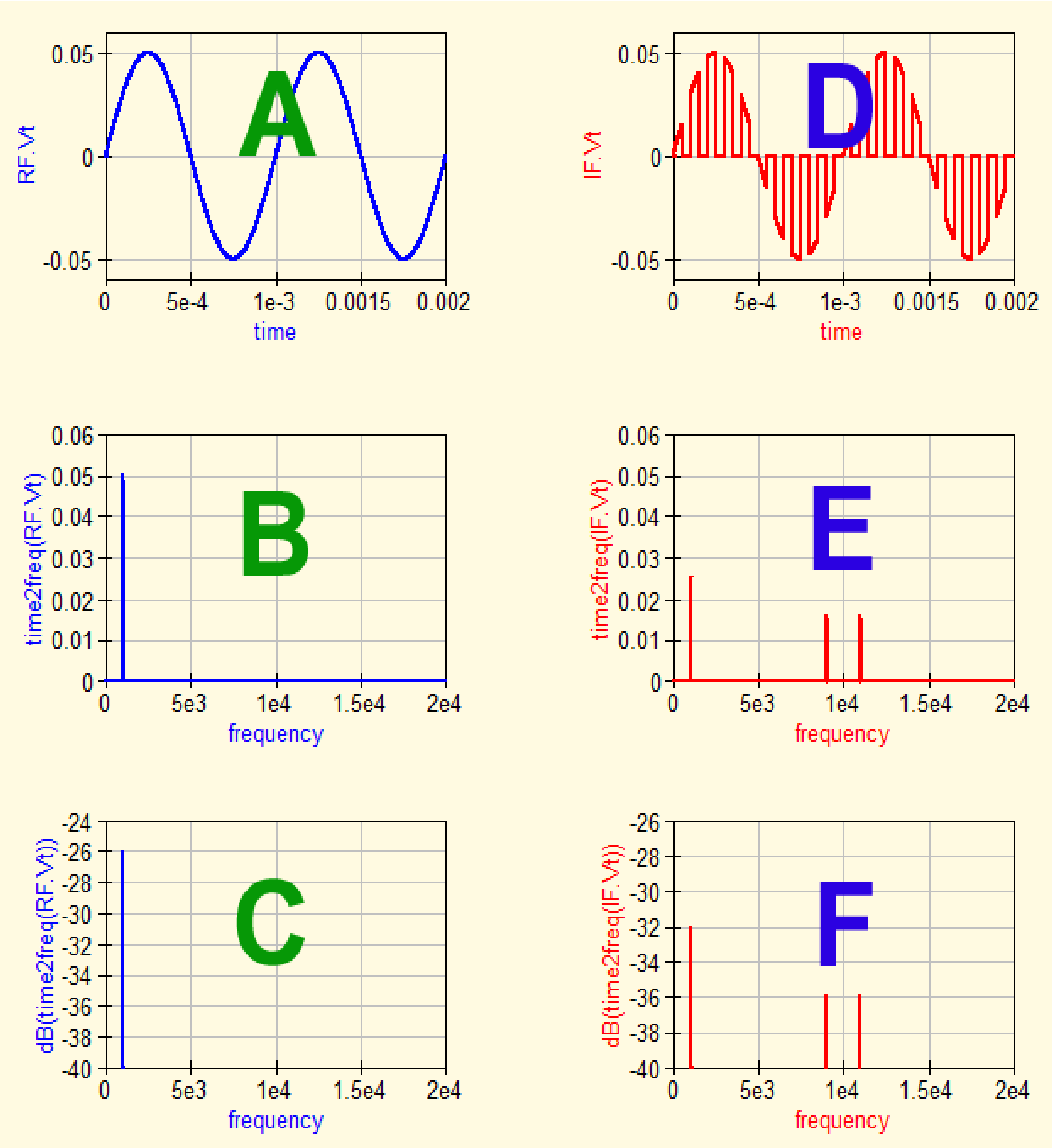
**The LO signal is suppressed at the output.**

On the next page we want to answer to the following questions:

a) What is the amplitude of the RF signal at the output in comparison to the amplitude value at the input?

b) What is the attenuation in dB for the first two side band frequencies „LO – RF“ and „LO + RF“?

Solution:




The left column is for the RF signal: illustration "**A**" shows the signal in the Time Domain with a **peak value of 50 mV at f = 1 kHz**.

**Illustration "B"** is the FFT result given as linear presentation. You find the same peak value of **50mV.**

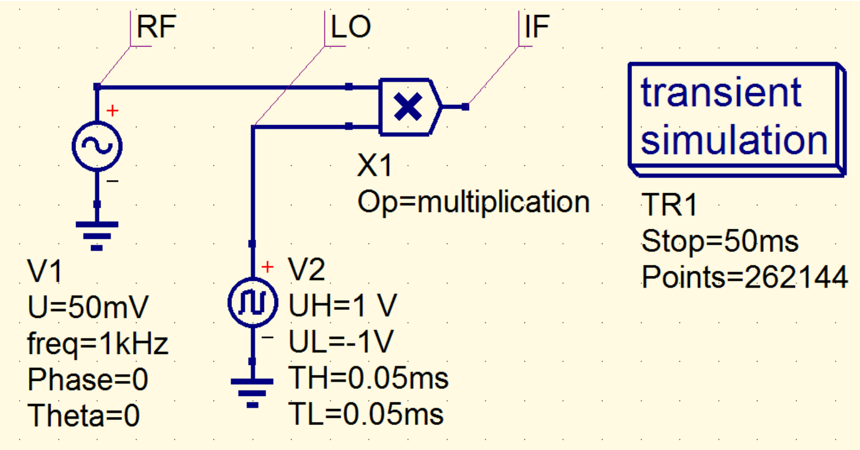
**Illustration "C"** is the presentation of "B" in dB, referred to "1 Volt. Thus we read an amplitude of **-26 dBV** (….say: **-26 dB Volt**).

At the **IF output of the mixer** (= illustration **"D"**) the RF signal peak value is reduced by 50% due to the ON / OFF switching process. This is a value of 25 mV (**illustration "E"**) or a **reduction by 6 dB to -32 dBV (illustration "F"**).

Regarding the two generated side band frequencies with 9 kHz and 11 kHz in illustration "F" you find an amplitude difference in comparison to the RF input signal at the input of **10 dB.**

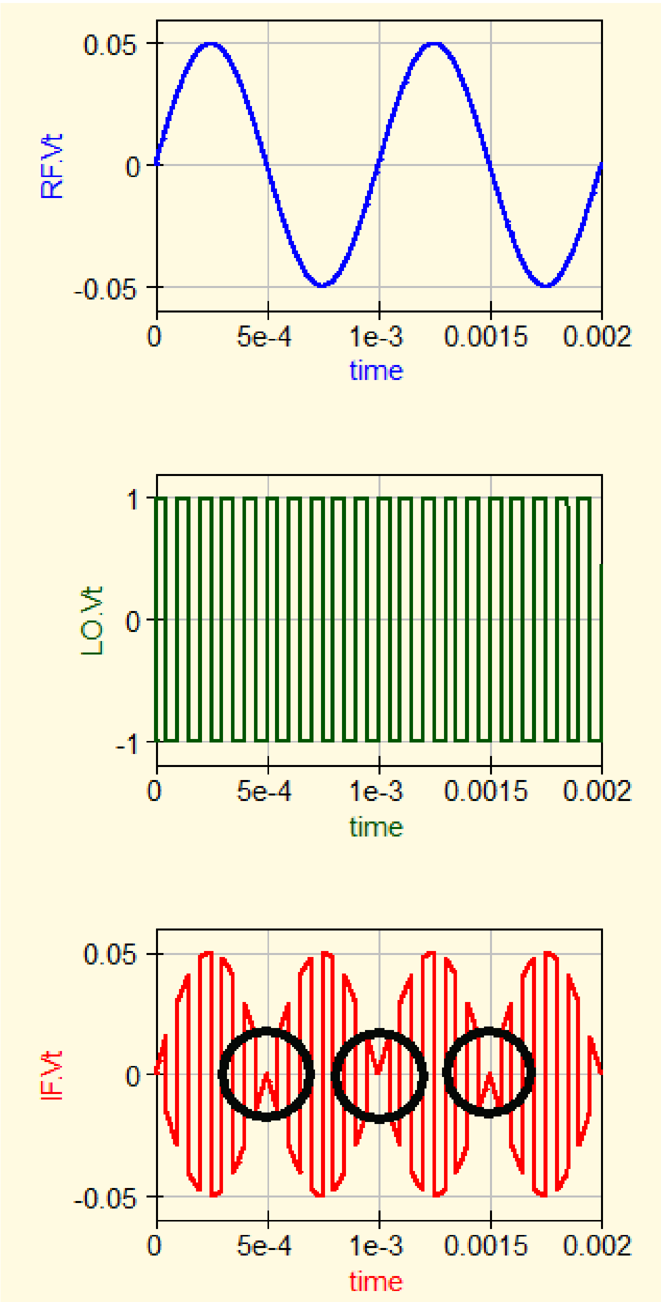
**So the conversion loss of such a "Single Balanced Mixer" is 10 dB.**

## 14.2.2. The Multiplier as Switched Double Balanced Mixer




Now a symmetrical rectangular voltage with an amplitudes of„+1V“ and „-1V“ is used.

**Thus the multiplier acts now as a “permanent inverter” instead of an ON / OFF switch.**




Here you can see what “permanent inverting” means.

And at the IF signal you can admire the “phase inversion”.

On the following site the signals “RF” and “IF” are analyzed and compared in the Frequency Domain.

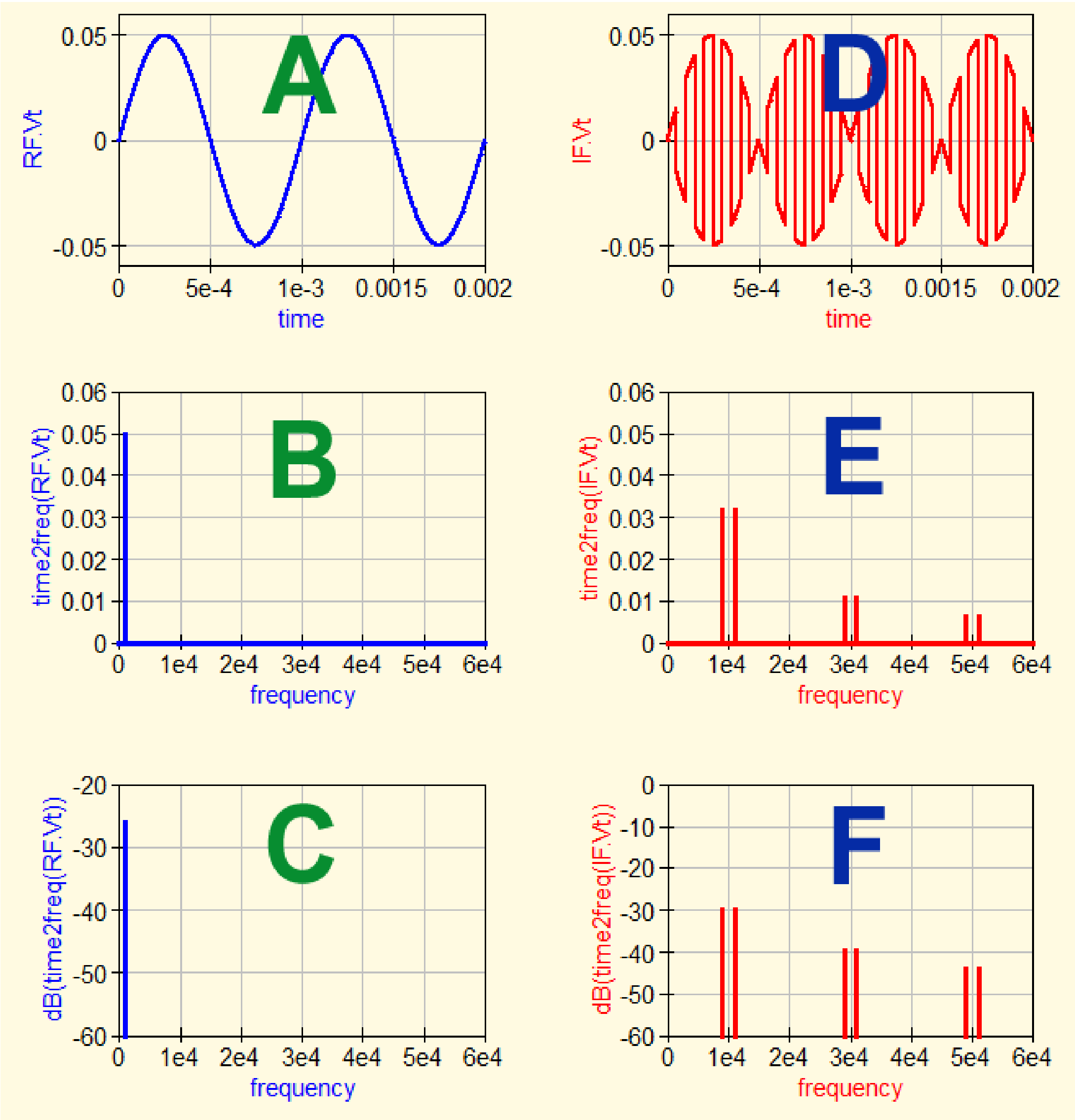



**Illustration "A"**: the peak value of the **RF signal** is **50 mV at f = 1 kHz**.

**Illustration "B"**: After the FFT you find this peak value of **50 mV at 1 kHz** also in the Frequency Domain.

**Illustration "C"**: Converted to dBV you get a value of **-26 dBV.**

**Illustration "D"**: this shows the permanent inverting of the RF Signal.

**Illustration "E"**: the peak voltage values of both side band frequencies 9 kHz and 11 kHz) are **32 mV.**
**RF signal and LO signal are suppressed in the IF spectrum!**

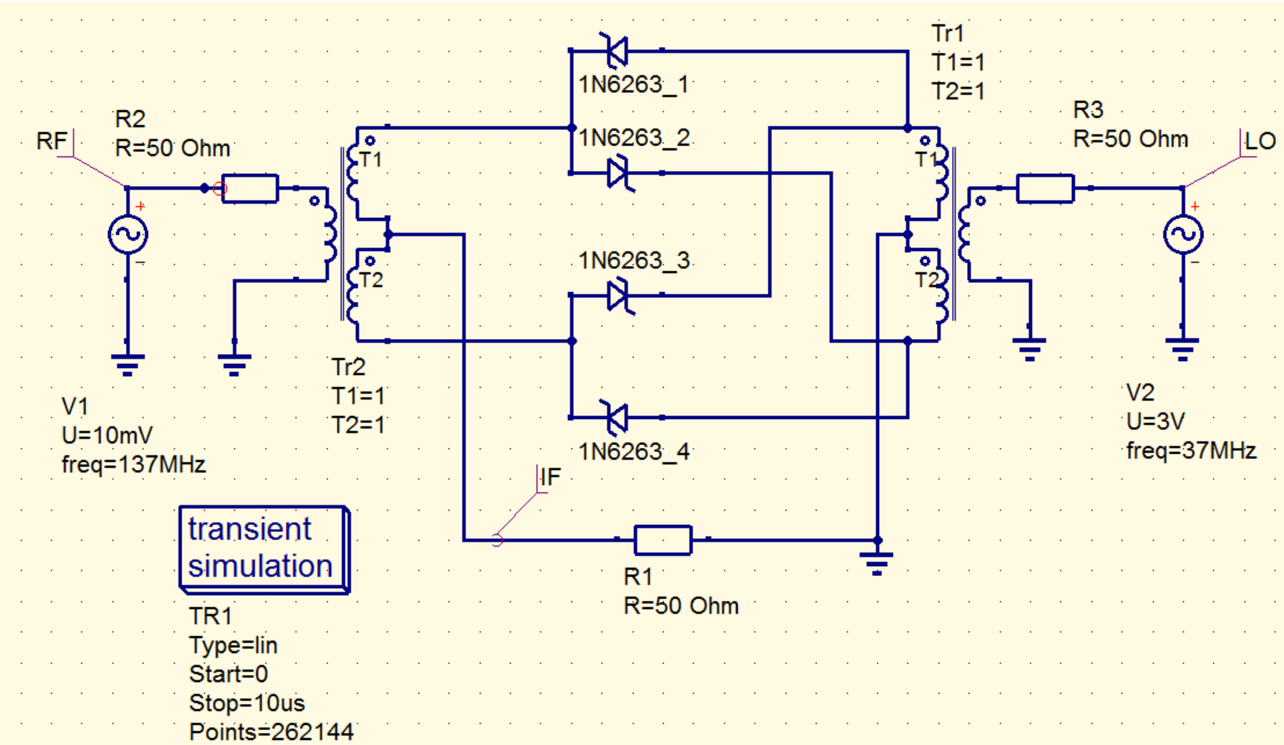
**And at every odd harmonic frequency we find the well known "couple of side band frequencies"**

**Illustration "F"**: the level difference between the RF signal at the input (f = 1 kHz) and the first side band frequencies (9 kHz and 11 kHz) at the IF output is **-26dB - (-30dB) = 4 dB**

**This is the conversion loss for the Double Balanced Mixer.**

## 14.3. Simulation of a true Double Balanced Mixer (= DBM = Ring Modulator)

Images sent by NOAA Weather Satellite can be received on **$f_{RF}$ = 137 MHz** as FM signals.
A converter for this purpose will at first amplify the antenna signal, then filter and at last convert to an IF frequency of **$f_{IF}$ = 100 MHz** using a DBM. The DBM circuit is a ring modulator and the incoming RF signal of 137 MHz is multiplied by an LO signal with 37 MHz. As LO signal a sine voltage with big amplitude is used instead of a rectangular voltage which gives nearly the same result.
At last a very selective Band Pass Filter with a pass band center frequency of f = 100 MHz and a bandwidth of only 3 MHz serves as filter for the difference frequency ($f_{IF}$ = 100 MHz) generated in the converter. This output signal is fed to an SDR (= software defined radio).
Here is the simulation schematic of the DBM:




The **RF Input** is fed by the received and amplified antenna signal. A peak value of **$U_{RF}$ = 10 mV / f = 137 MHz gives an incident wave of Uinc = $U_{RF}$ /2 = 5 mV = -46 dBV.**

The transformers can be found in **"components / lumped components / three winding Transformer"**. The transfer ratio between the winding is "1"**.**

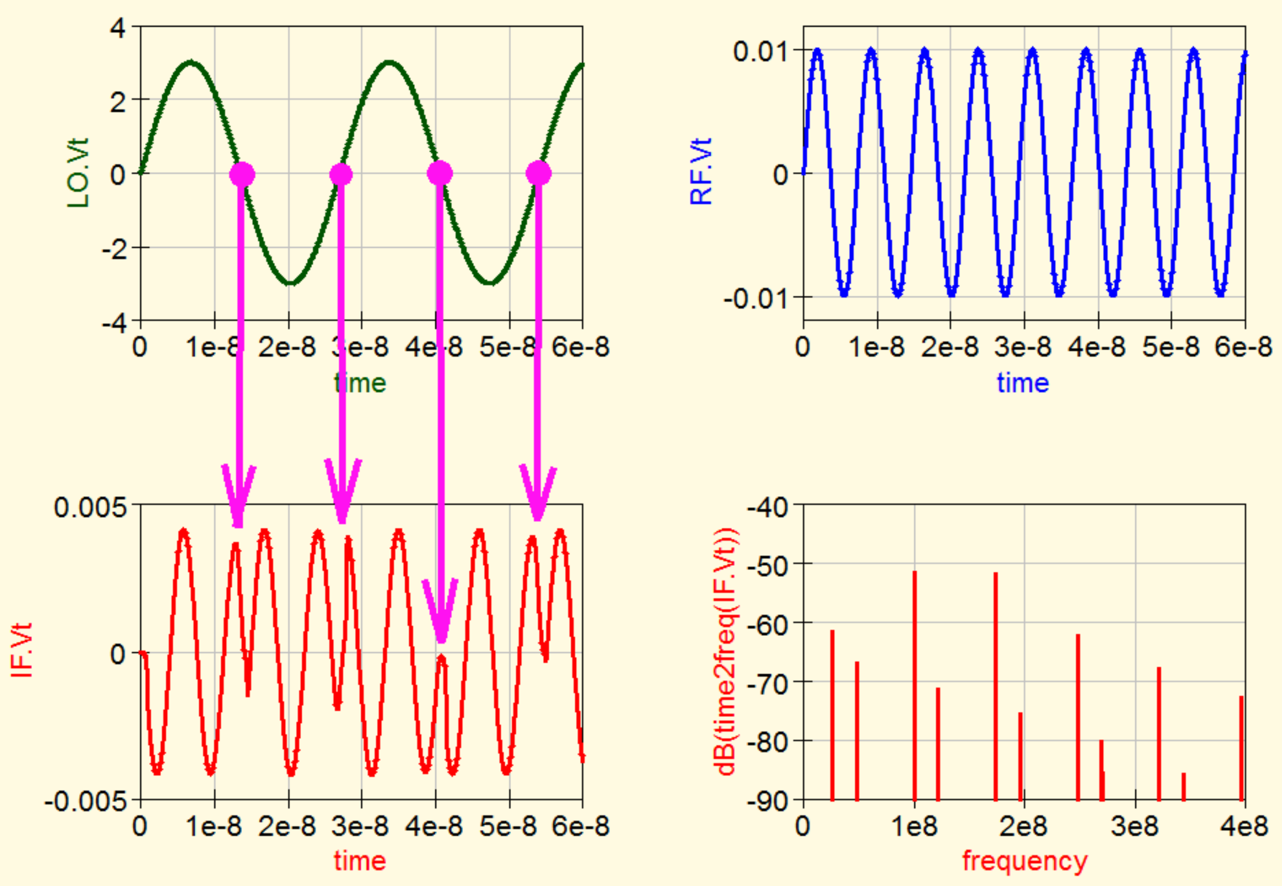
The **Schottky diodes (**1N6263) come from the qucsstudio Schottky diode library

**The LO-Signal** is a sine wave with a peak value of **3 V and a frequency of 37 MHz.**

The simulation runs from **zero to 10 Microseconds using 262 144 points for a successful FFT.**
The resolution in the Frequency Domain is now

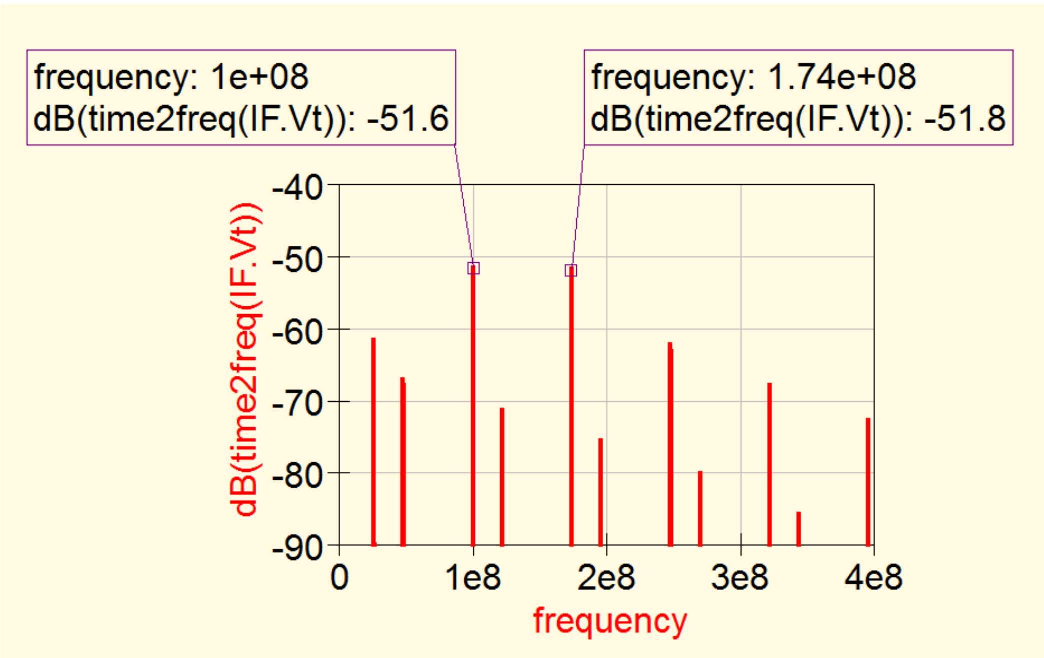
**1 / 10 µs = 100 kHz**

This is the result of the simulation:



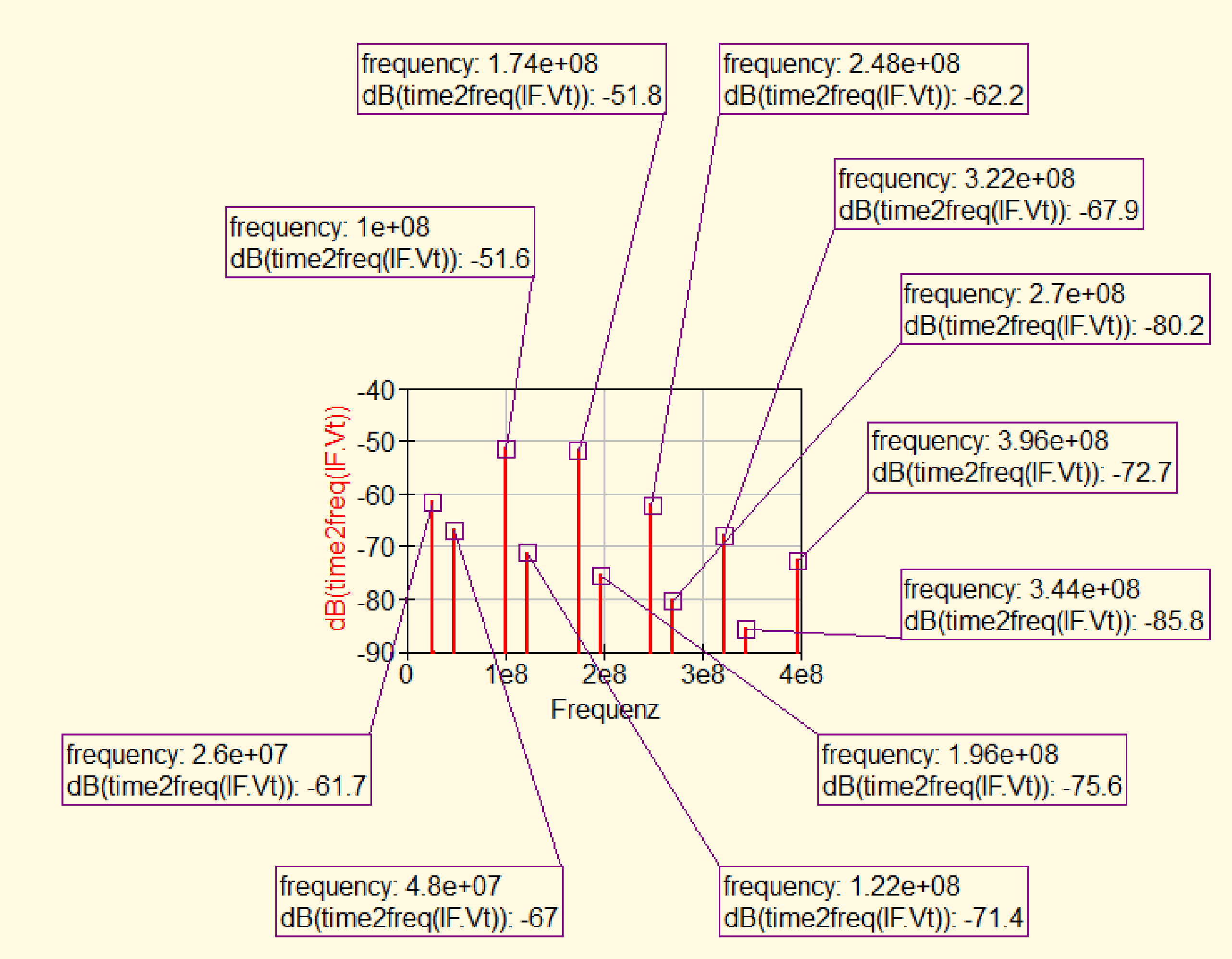

At the IF signal you can admire the “points of polarity inversion of the RF input” when the LO signal changes its polarity.

The IF spectrum is at first a little confusing and must be analyzed (used frequency range = 0....400 MHz)




At first we look for the frequency sum (**137 MHz + 37 MHz = 174 MHz)** and the frequency difference (**137 MHz – 37 MHz = 100 MHz**) of the RF input signal an the LO signal.
Use two frequency markers for this purpose.

Let us continue this game for all other lines in the IF spectrum:




**Task:**
**Identify the different "couples of spectral lines with the same amplitude" which are generated when harmonics of the RF signal are multiplied by odd harmonics of the LO signal!**

(To save time use a pocket calculator...)

Example:

**RF+3xLO = 137 MHz +3x(37 MHz) = 137 MHz + 111 MHz = 248 MHz**
**RF - 3xLO= 137 MHz - 3x(37 MHz) = 137 MHz – 111 MHz = 26 MHz**

**Task 2:**
**Determine the conversion loss of the RF signal when passing the mixer.**

**Solution**:
The levels of the frequency sum and difference in the IF spectrum are -51.6 and -51.8 dBV (= a mean value of **-51.7 dBV)**.

The incident wave of the RF signal at the input was **-46 dBV.**

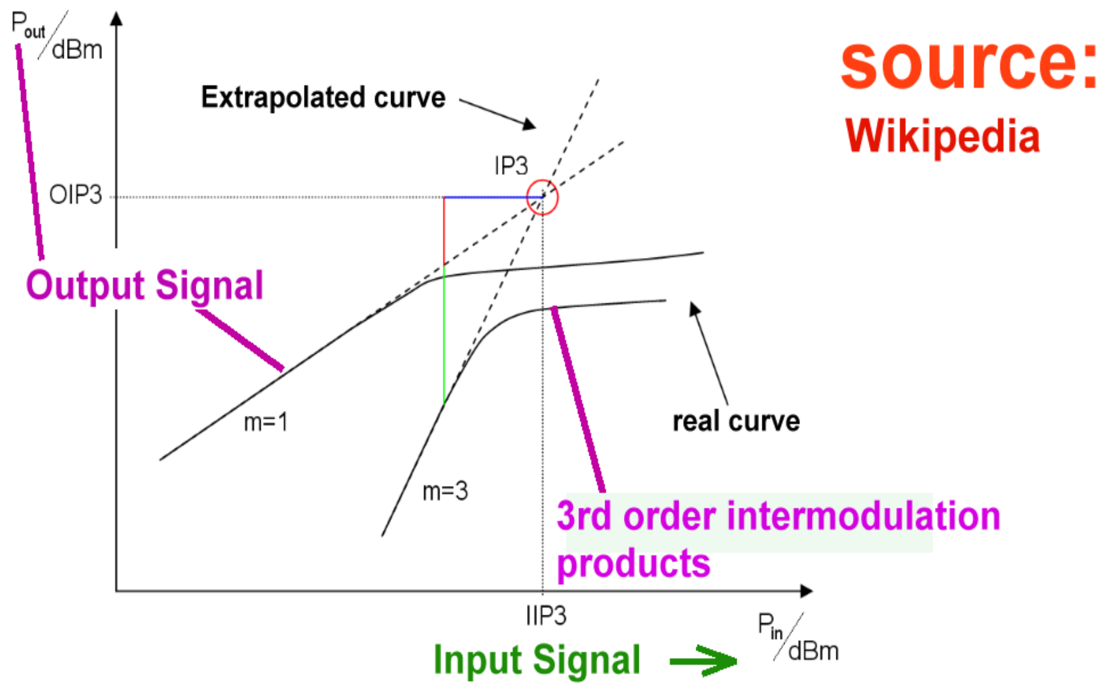
So the difference is **4.7 dB and this is exactly the conversion loss.**

## 14.4. The IP3 Point

The simulation of the „***Third Order Intercept Point***" informs about the nonlinearities and distortions of a stage when the input level increases.

### When reaching the IP3 (on the "Pout / Pin curve") the third order distortion products would be equal to the applied RF signal at the input. And this is not very good, because 3rd order distortion products have nearly the same frequency as the input signal.....

But:
this IP3 point is a **pure theoretical value** due to the fact of compression, followed by limiting, in the stage with increasing input levels.
That can be shown by this diagram:



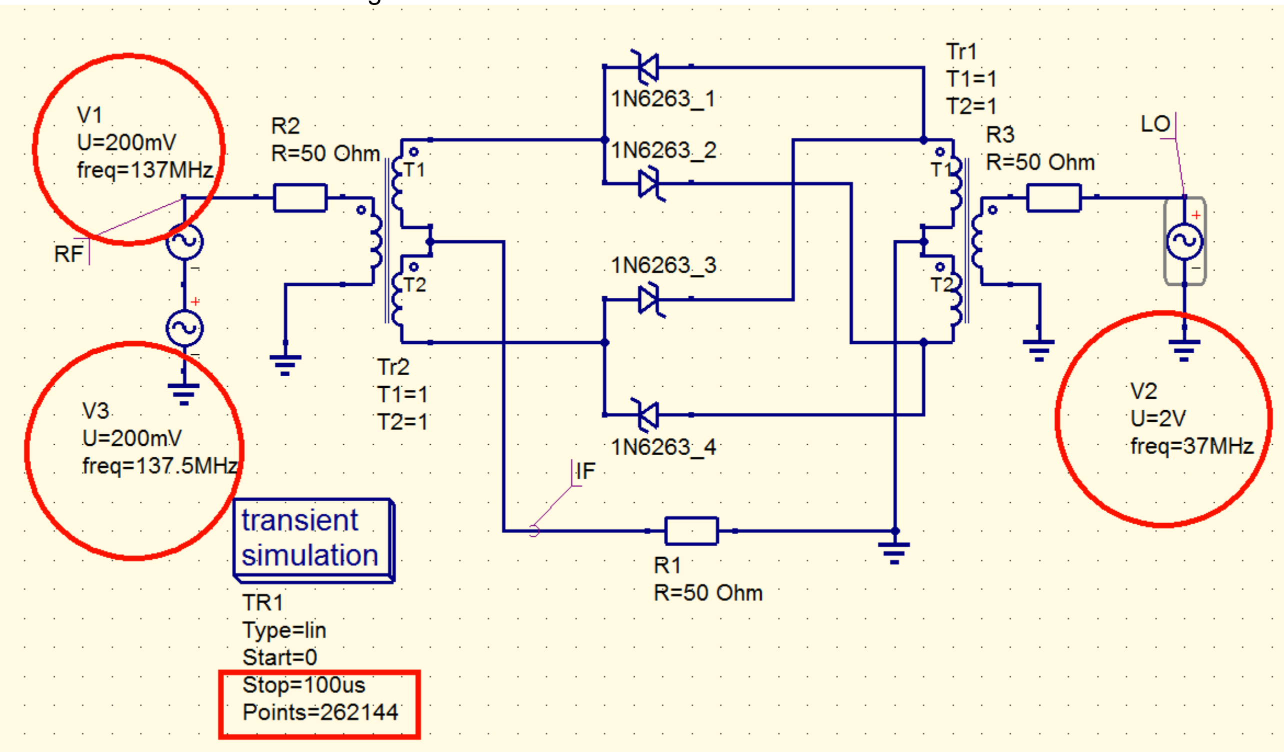

**But if you exactly know this IP3 point, then you can for every input or output level calculate the level difference to the undesired 3rd order intermodulation products!**
**(You have simply to solve two equations for straight lines....)**
**And so you can for this case define an intermodulation free dynamic range with the noise in the stage as "floor"**

## To determine the IP3 we feed the stage with two “in band signals” with the same amplitudes but a very small frequency difference.

**The Level is now increased and very soon you will see the 3rd order intermodulation products at every side of the couple of test signals. The frequency distance to the test signals is exactly the frequency difference of the test signals. But the IP3-products can't normally be eliminated by filtering!**

**(...There exist also 2nd order intermodulation products with higher amplitudes. But their frequencies are far away from the two test signals. So they don't disturb and can be eliminated by filtering....).**

Let us test this information using our DBM circuit.




A) At the input there are two voltage sources connected in series. The peak amplitude is 200 mV, the frequency difference is 0.5 MHz.

B) The peak LO level is reduced to 2 V

C) The simulation runs from **0 to 100 µs** to achieve a frequency resolution of **1 / 100µs = 10 kHz with 262 144 points**

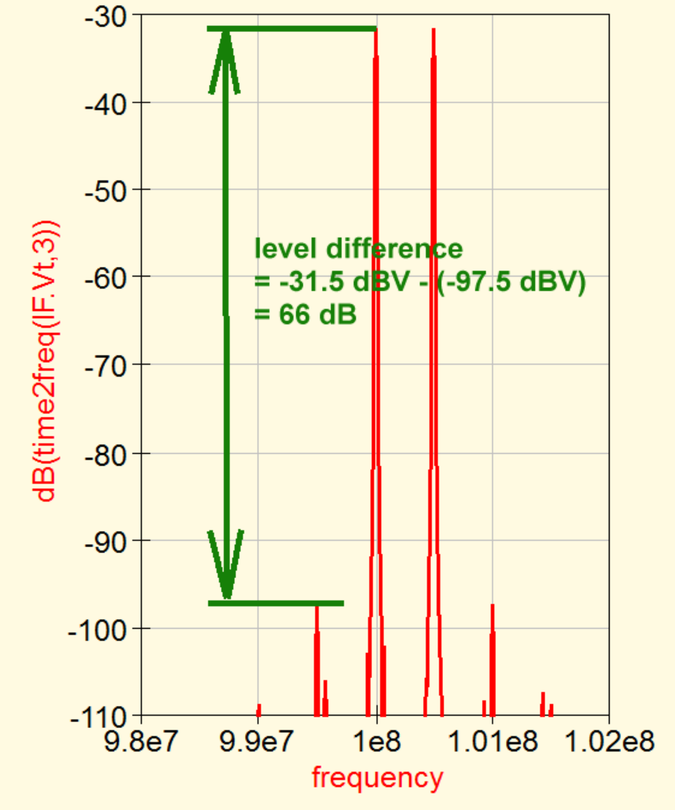
For the FFT some setting changes are necessary to get a good demonstration of this effect:

We use a **Hanning window (index = 3)** and show the spectrum only in the range from **98 MHz to 102 MHz with a frequency spacing of 1 MHz.**

The vertical axis is scaled in dB and the indicated range is **-30 to -110 dB**

The diagram is “vertically stretched” to get a better resolution.

The 3rd order intermodulation products can be found at 99.5 MHz and 101 MHz beside the two converted RF test frequencies at 100 MHz and 100.5 MHz.
The level difference between test signal and 3rd order product is **-31.5 dBV -(-97.5 dBV) = 66 dB**




Now you need the equation

**IP3 = RF Incident Power Level**
**+**
**0,5 x (Level Difference)**

Please pay attention:

**You must always use the incident wave for the input signal.** This wave must be calculated as power using the effective voltage value.

For our example with a peak voltage value of 200 mV:

**Uinc = 0.707 x (200 mV / 2) = 70.7 mV (effective value)**

This gives a power level in dBm of

**20 x log (70.7mV / 223 mV) = -10 dBm**

for a 50Ω System.

Now:

**IP3 = -10 dBm + 0,5 x (66dB) = +23dBm**

**That's all!**

# 15. Developement of a Narrow Bandpass = Coupled Resonator Type for a Center Frequency of 10.7 MHz

## 15.1. Information

We need often in the RF range a band pass filter with a very small bandwidth (= 1 to 3% of the center frequency). This can only be achieved by circuits using “transformations”. The theory is complicated but the design is simple due to good but expensive filter design programs.
Already since DOS times exist free filter calculators which can do this job very well. But DOS and modern WINDOWS versions....a special problem. And that is a pity because at DOS time a lot of free calculators for filters, couplers, antennas etc. was available without charge.

But modern free Windows based filter calculators cannot handle the design of a “Narrow Band Pass = Coupled Resonator Type”. Thus we solve the problem as follows:

a) We install a “**DOS Box**” which simulates a complete DOS operating system and can be closed when we work is done.

b) We install a “**DOS Shell**” for the easy use of the DOS system.

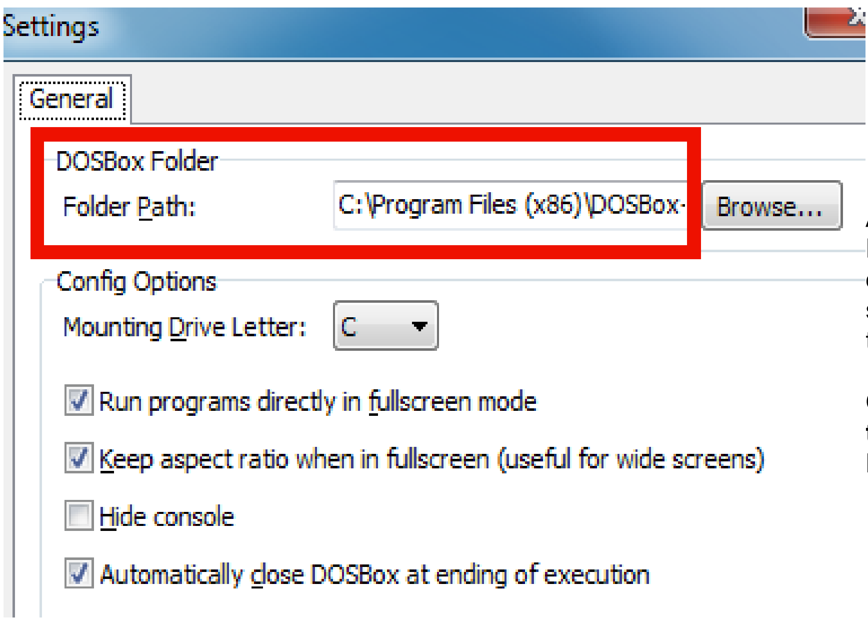
c) Then we start “**fds.exe**” (= abbreviation for filter design software....) and design the desired Narrow Band Pass Filter.

**Attention:**
**Starting with software version 2.4.0. the filter calculator of qucsstudio is now able to design this filter type. But as a simpler version and with less possibilities than fds.exe....at the end (= chapter 14.9) we will compare the qucsstudio design to the fds.exe design.**

**But at first: back to DOS!**

## 15.2. Installation of DOS Box, DOS Shell and fds.exe

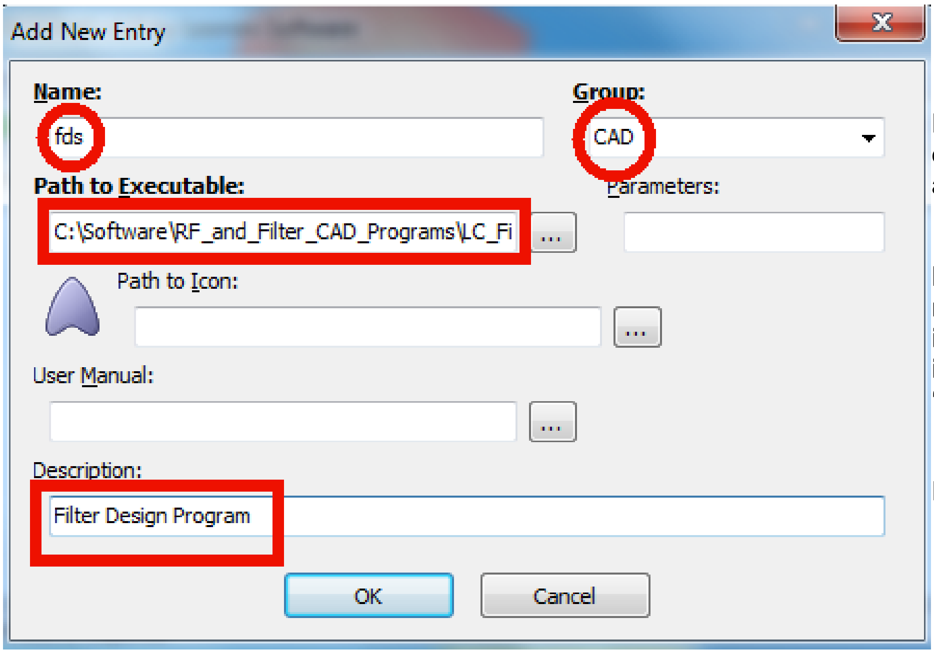
**You can download these 3 programs from the qucsstudio page in my homepage.**
(www.gunthard-kraus.de)




At first install the DOS Box and the DOS-Shell. Save “fds.exe” in an own folder named “filters”. Then start the DOS Shell by clicking on the icon on your screen.

**Open “Edit / Settings” and enter the correct path to the DOS Box.**

Then close the DOS Shell and open it once more. Open “**Edit**“ again but now chose “**Add new entry**“.




Here you see all necessary entries for the successful access to “fds.exe” on my PC.

**If you want then you can now delete the “DOS Box” icon. Only the DOS Shell icon is needed to start “fds”.**

Finished!

## 15.3. Specification of the 10.7 MHz Band Pass Filter

A frequency of 10.7 MHz is often used as IF in FM receivers. Lot of parts (like ceramic or crystal filters etc.) are available on the market and also modern techniques like digital signal processing are applicable. So for your own work or developments a collection of filters or amplifiers is a find thing because very often you have to suppress strong signals beside your 10.7 MHz signal.

Filter design starts always with the result of thinking = the specifications. This is a list of properties which must be fulfilled by the final product. And so it looks like:

| | |
|---|---|
| **Characteristic system impedance:** | **Z = 50 Ω.** |
| **Filter class:** | **Chebychev with a pass band ripple of 0.3 dB** |
| **Center frequency:** | **10.7 MHz** |
| **3 dB - bandwidth:** | **500 kHz** |
| **Pass band attenuation:** | **less than 6 dB** |
| **Attenuation slope:** | **a minimum attenuation of 70 dB at 9 MHz and at 12 MHz** |
| **wide band attenuation:** | **an attenuation higher than 70 dB up to 500 MHz** |

The PCB (dimensions: 30 mm x 50 mm) is mounted into a milled alumina case with cover.
Input and output use SMA jacks with the center conductor soldered to the microstrip lines on the PCB.
Substrate material is Rogers RO4003 with a height of 32 MIL = 0.813 mm, both sides copper plated (thickness = 35 µm, roughness = 2 µm).
The lower side of the PCB is the “infinite ground plane” and a lot of via are used as connection between the ground plane and the “islands” on the upper side which must be grounded. Silver plated rivets are used as via.

## 15.4. Strategy of Development

At first answers must be found to these questions:

**a)** Which filter type must be used?

**b)** Which filter degree is necessary?

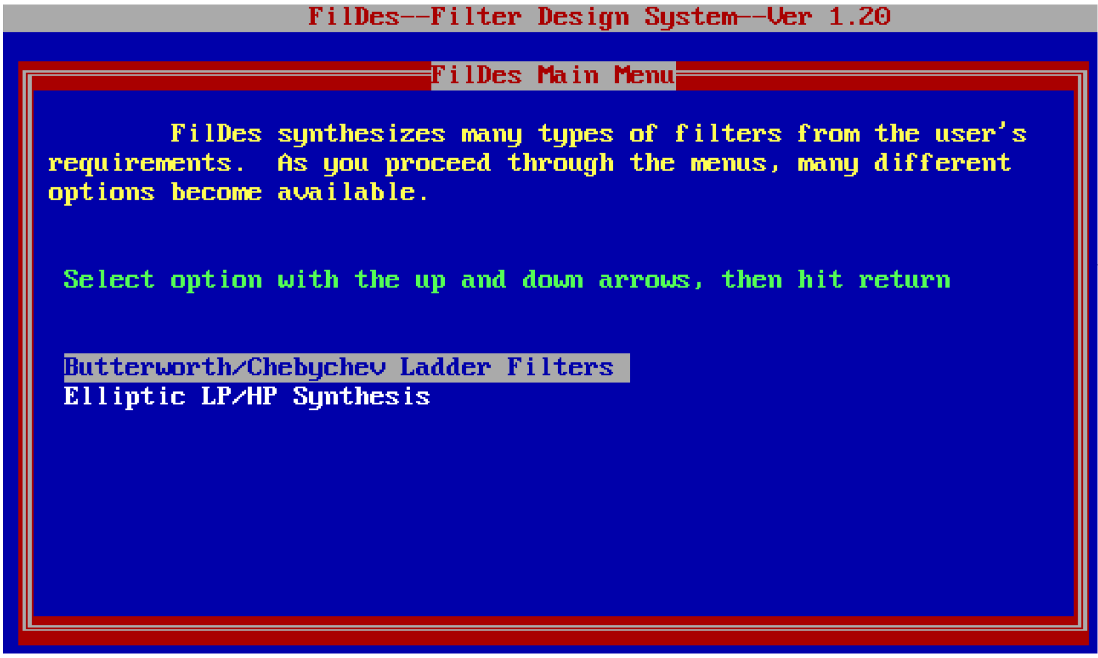
**c)** What is the minimum necessary quality factor Q for the coils to achieve the desired minimum pass band attenuation?

**d)** How must such coils be constructed to achieve the small size AND the necessary Q for the given alumina case?

**e)** How can the desired stop band attenuation of 70 dB up to 500 MHz be achieved?

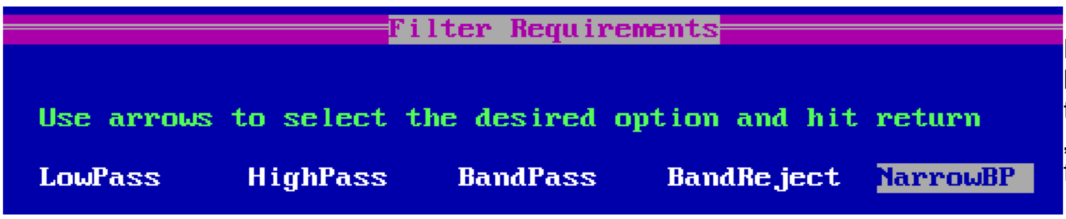
The "**Coupled Resonator Band pass Filter Typ" = "Narrow Band pass**" is well known to fulfill all these specifications. In this filter all inductors have the same value and the matching to the low source and load resistance (50 Ω) is done by a capacitive transformation.
So a first simulation is used to know which filter degree is necessary for the desired bandwidth and stop band attenuation. Also the necessary Q factor of the coils is tested out.
The fundamental questions are answered by the first design: a filter with a degree N = 3.
A second and final design (with a higher filter order) will follow.

## 15.5. Necessary Filter Type and Quality Factor

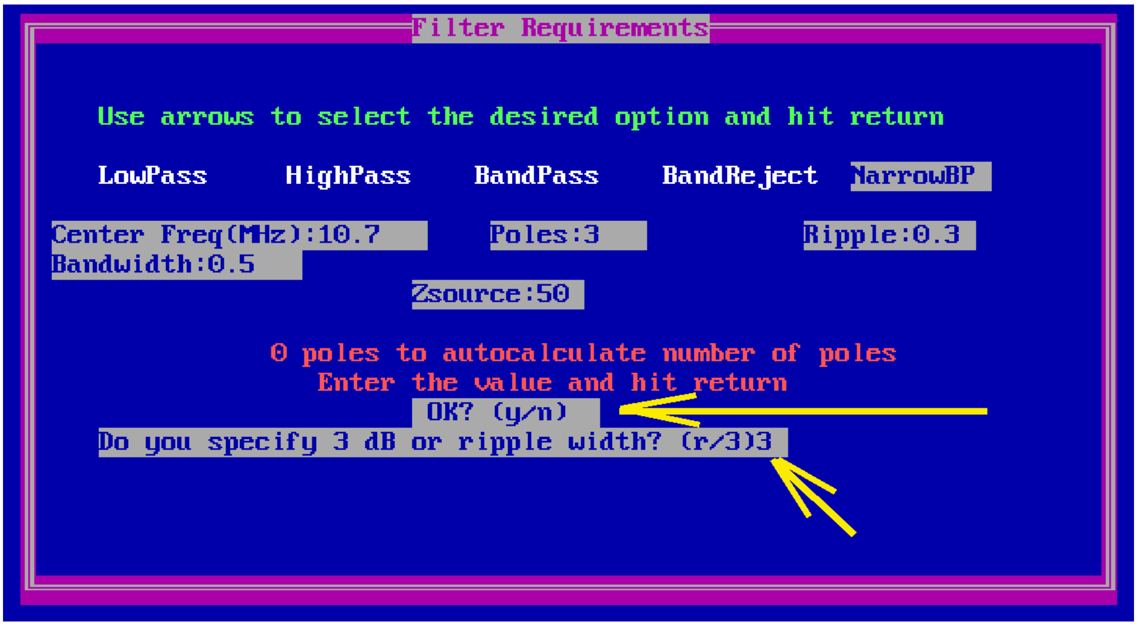



Start the **DOS Shell** followed by **"fds.exe"**

At first mark the line for the **"Chebychev"**-Filter and press OK.




Now use the horizontal arrow tabs to mark „**NarrowBP**", followed by OK.

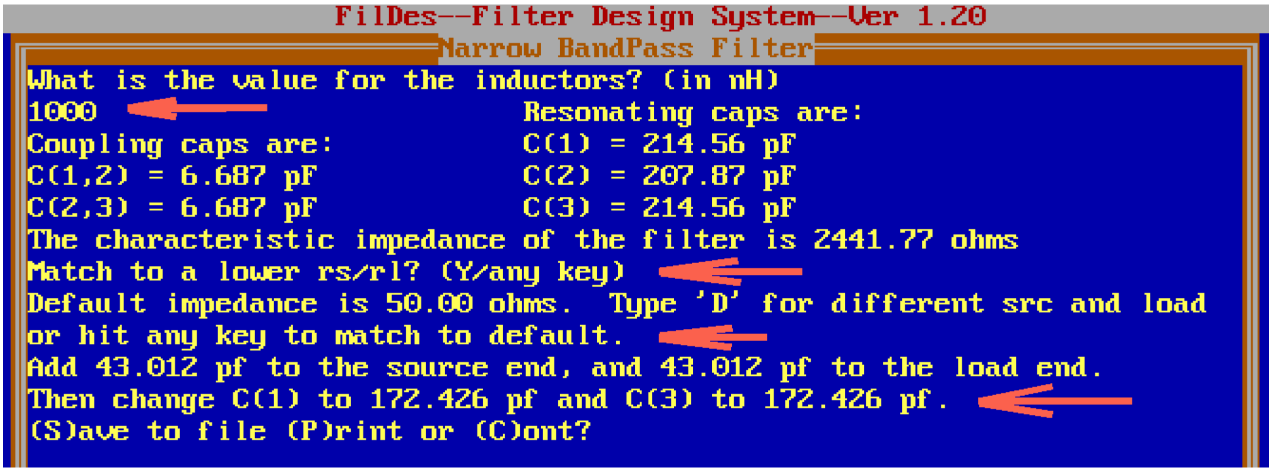



Enter now all the properties according to this mask (continue with “ENTER“ after every entry) :

**Center Frequency = 10.7**
**Poles = 3**
**Ripple = 0.3**
**3 dB or ripple width = 3**
**Bandwidth = 0.5**
**Zsource = 50**
**OK? = y**

In the next mask please enter:
**L = 1000 nH**
**Match to a lower rs/rl = y**

**Hit any key for default of 50 Ω**

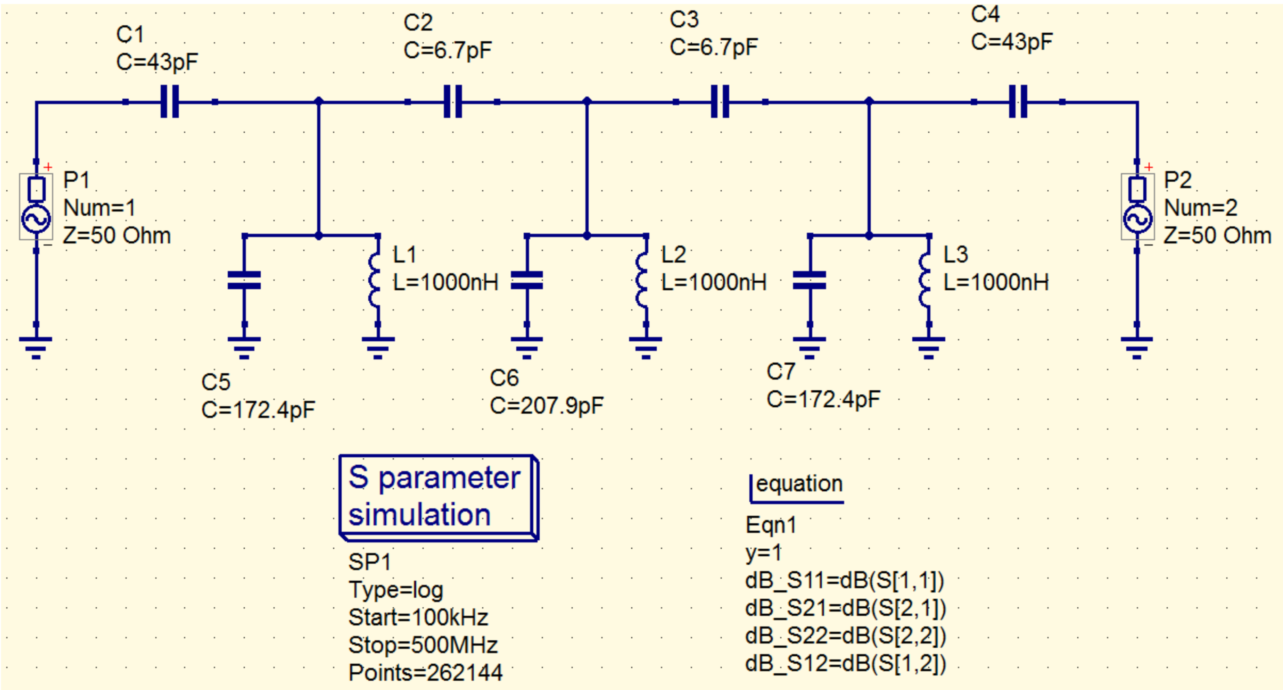


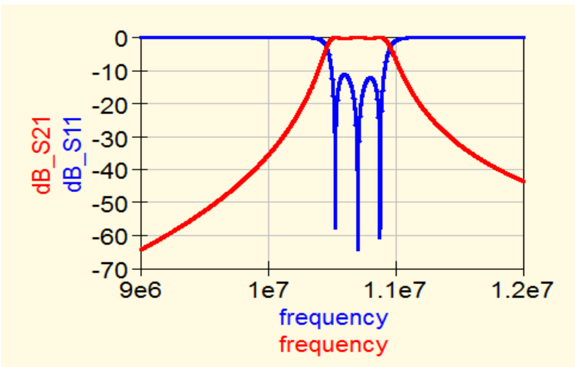

**Important information to the value of the inductor “L“, because you are responsible for your choice – and this choice should be good:**

**All inductors have the same value, but the impedance of this inductors must have a value between**
**50 and 100 Ω at the passband center frequency for good practical filter properties!**
**Thus an inductor of 1 μH = 1000 nH with an impedance of 63 Ω at 10.7 MHz is used.**

Further information, given by the program:
**Every side of the filter starts with an additional series capacitor of 43.012 pF. But due to this additional capacitance you must reduce the capacitor in the adjacent parallel resonant circuit to-172.46 pF.**

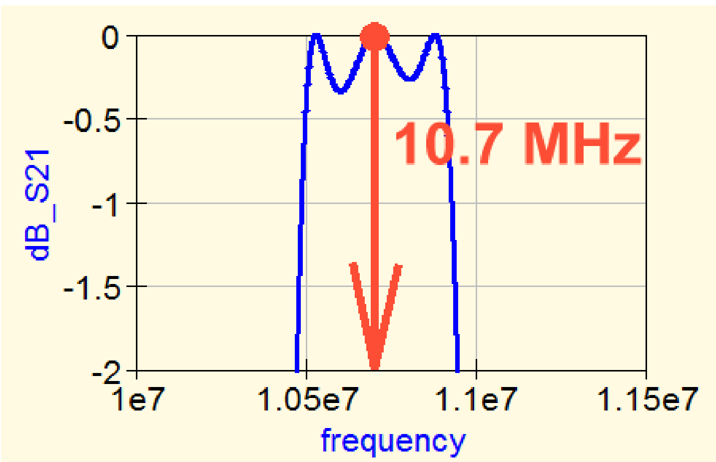
Now everything is prepared for the first qucsstudio simulation.
But please reduce the number of digits of every part value to "**1 behind the comma**"



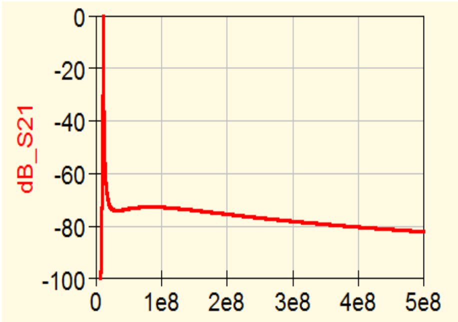

The simulation runs from 100 kHz up to 500 MHz.
But at first look at the pass band region and simulate S11 and S21 from 9....12 MHz.

**Sorry, but the stop band attenuation is not as good as desired (demanded: 70 dB at 9 MHz resp. 12 MHz).**




Here you can see the Chebyshev pass band ripple of 0.3 dB (with a small slope..

Not so bad...

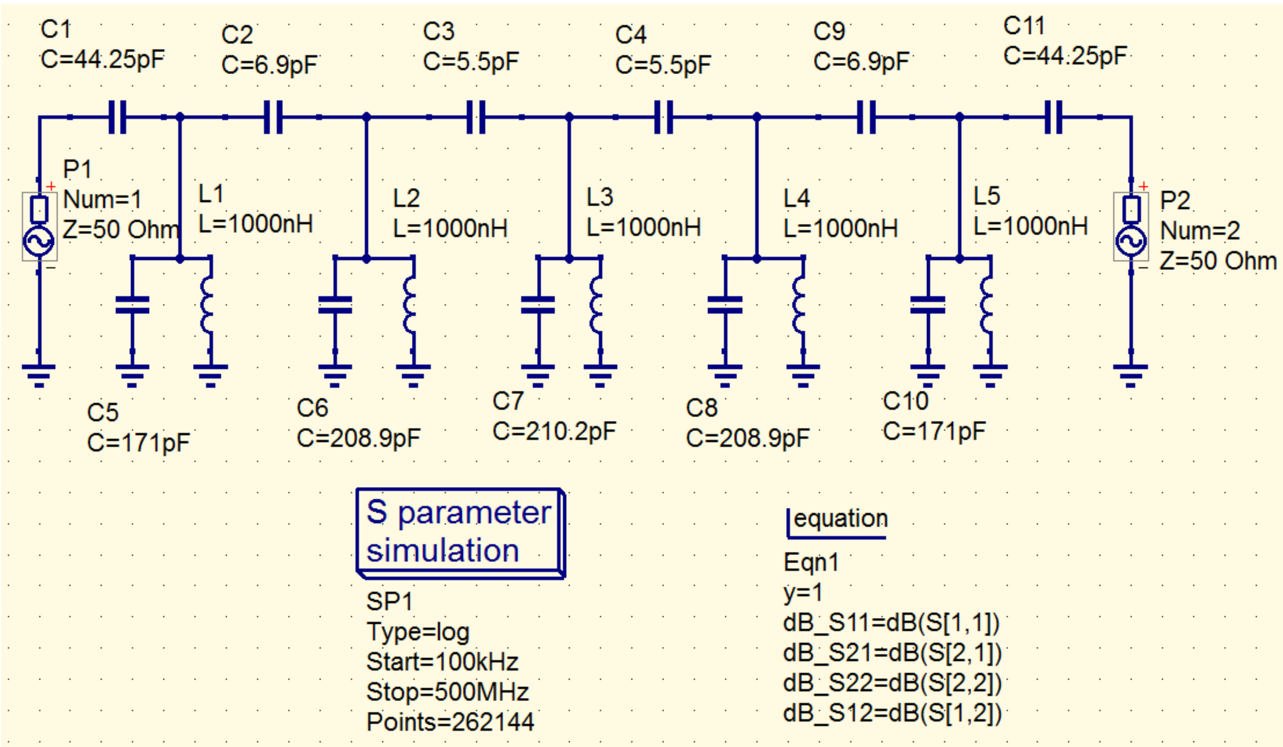



And now the stop band attenuation up to 500 MHz. This isn't convenient because only a value between 70 and 80 dB is achieved.

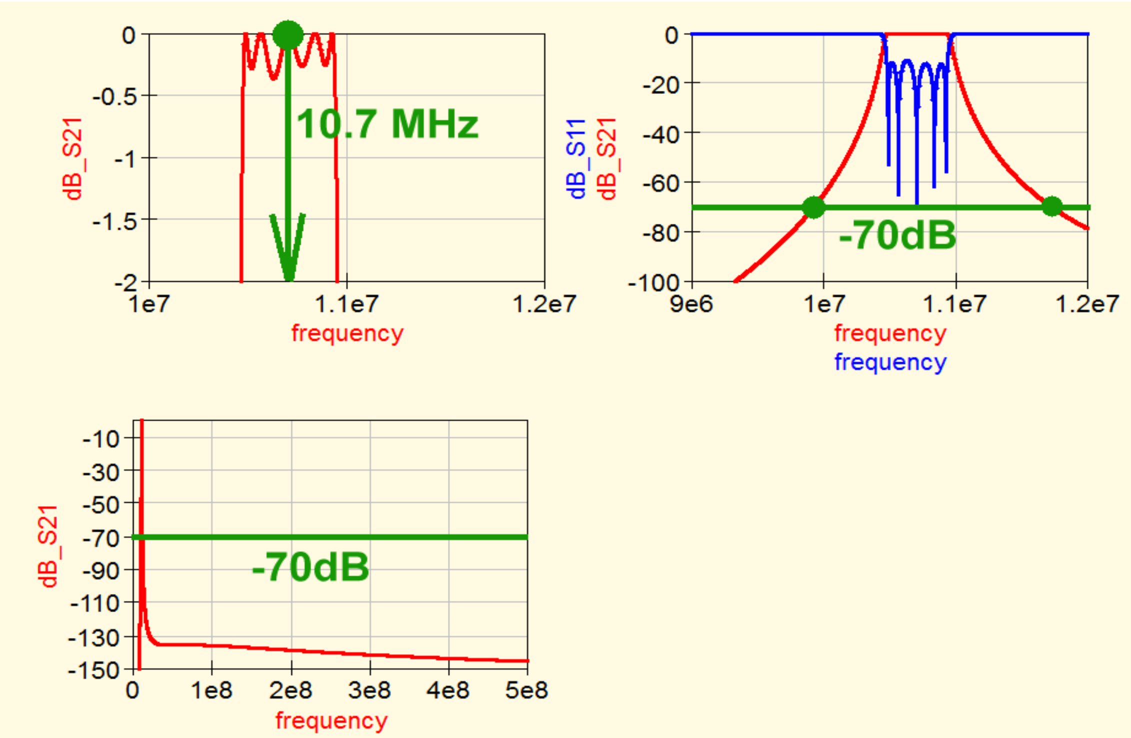
Thus a new design with a filter degree of **N = 5** must be done with “fds”.

```
What is the value for the inductors? (in nH)
1000                          Resonating caps are:
Coupling caps are:            C(1) = 214.37 pF
C(1,2) = 6.877 pF             C(2) = 208.86 pF
C(2,3) = 5.508 pF             C(3) = 210.23 pF
C(3,4) = 5.508 pF             C(4) = 208.86 pF
C(4,5) = 6.877 pF             C(5) = 214.37 pF
The characteristic impedance of the filter is 2309.75 ohms
Match to a lower rs/rl? (Y/any key)
Default impedance is 50.00 ohms.  Type 'D' for different src and load
or hit any key to match to default.
Add 44.251 pf to the source end, and 44.251 pf to the load end.
Then change C(1) to 171.075 pf and C(5) to 171.075 pf.
```

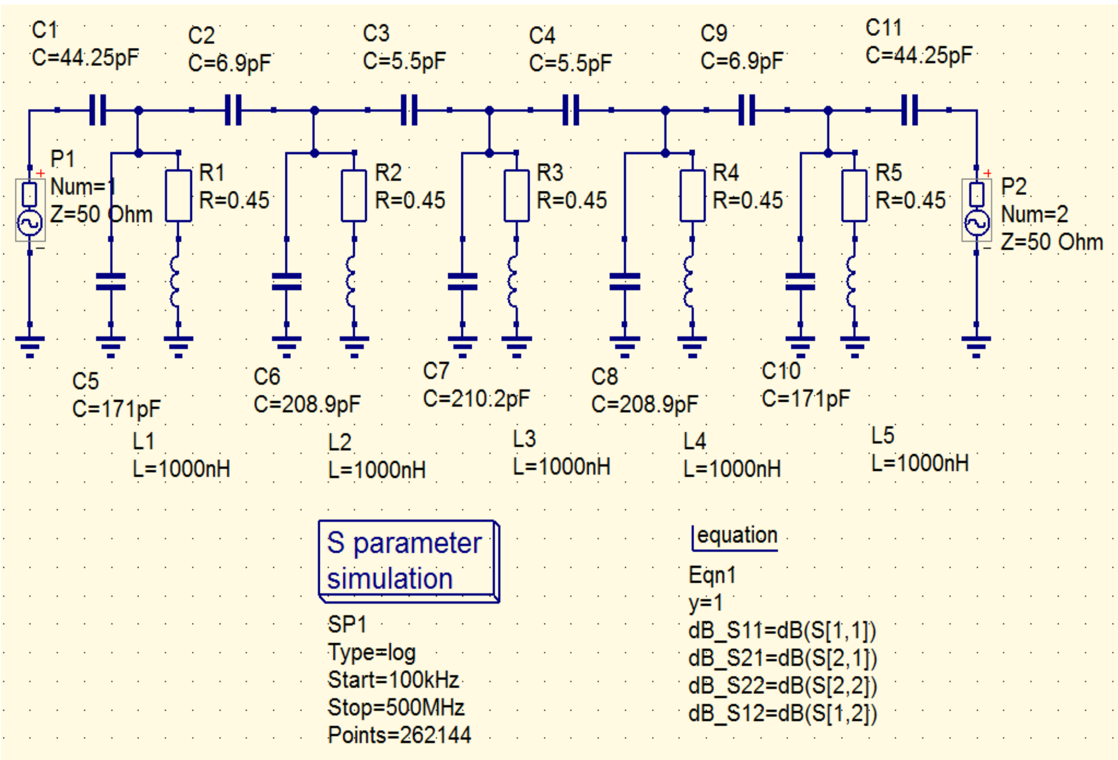
This is the new simulation schematic with rounded values:



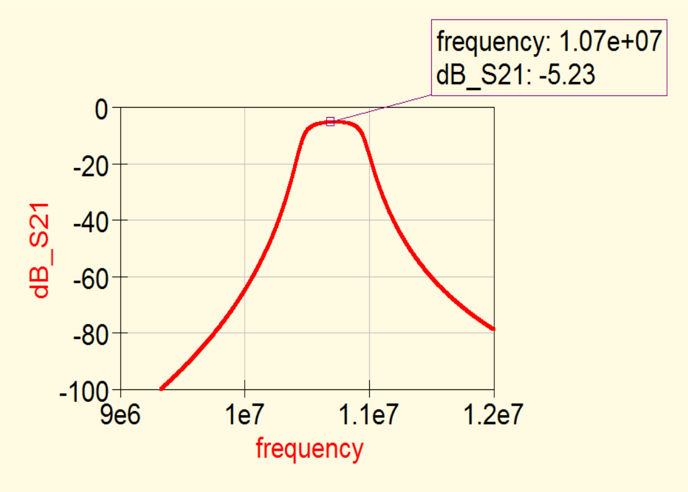

Here come the properties of the design:




Now the selectivity is everywhere OK.
But in the specifications you find the point: **"Maximum insertion loss = 6 dB at the pass band center frequency".**
The insertion loss is determined by the Q factor of the used coils.
A realistic maximum value (which can be achieved by the usage of Amidon ring cores) is **Q = 150.**
This gives for f = 10.7 MHz an L = 1µH a series resistor of
**$Rr = X_L / Q = 2 * \pi * 10.7MHz * 1\mu H / 150 = 0.45\Omega$**




This is the new simulation schematic (losses included).



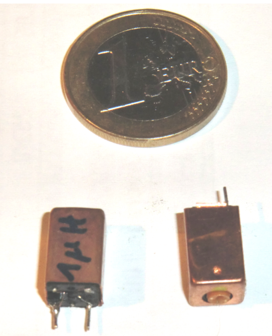

Using a frequency marker the determination of the insertion loss at 10.7 MHz is a joy.

**A value of 5.23 dB is very good and below the maximum of 6 dB.**

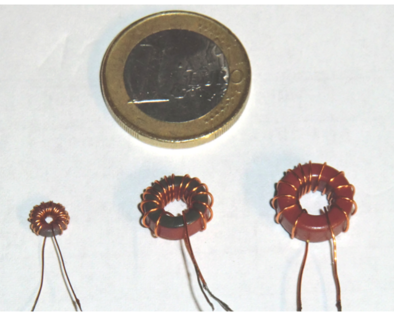
## 15.6. The Coil Fight

To find out Q factors at high frequencies is a special affair for expensive measuring sets. So an old Boonton RX Meter from 1960 was bought on the HAM Radio flee market and repaired. This is an impedance bridge for the frequency range from 10 MHz to 250 MHz and uses a test oscillator for a modified Schering bridge, followed by a single conversion super heterodyne receiver – and still equipped with tubes.

A measuring bridge is a fine affair – but needs brain efforts when using and some calculations with the pocket calculator to get correct results. If you try to measure an inductance you'll get a result scaled as "negative capacitance" with a maximum value of 100 pF.
Thus when checking L = 1 µH the frequency had to be increased up to 16 MHz to press the indicated "negative spare capacitance value" below "-100 pF.



Now the well known Neosid coils were tested. They need an area of 7.5 mm x 7.5 mm on the PCB and are well shielded. But all efforts caused the same result:
Only a maximum Q factor = 100 was achieved.



So the famous Amidon ring cores were used. They are cheap, show high Q values and due to the "closed magnetic field lines way" no additional shielding is necessary.

But you have (in the range up to 30 MHz) to decide between cores made from "ferrite" or from "fine powdered pure iron". Ferrite gives the higher permeability but suffers under early saturation and increasing non linearity at higher currents.

Fine powdered pure iron cores are a mixture of an isolating substrate and microscopic fine iron powder. Thus the saturation level is much higher combined with a high Q factor and low losses. This core type was used.

The first step is to order a small collection different core sizes (T20-2, T37-2 and R44-2) for experiments. The marking system is very simple:

**The first number is the outer diameter in 0.01 inch values. The number "2" tells us, that we have a material with an "optimum frequency range between 1 and 30 MHz". Additionally the ring is colored in red / brown for this material.**

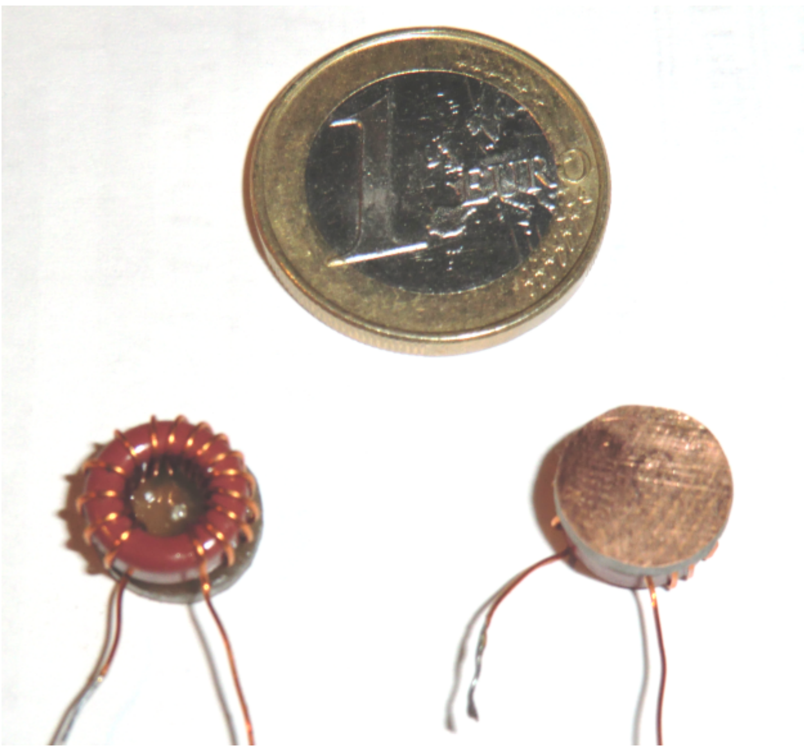
Now these three cores were wound with copper wire to achieve an inductance of 1µH an measured at 16 MHz with the following result:

| Core type | Outer diameter | Windings | Wire diameter | Inductance | Losses as parallel resistor | **Q factor at 16 MHz** |
|---|---|---|---|---|---|---|
| T20-2 | 5.08mm | 19 | 0,2 mm | 0.97µH | 12.9k | **132** |
| T37-2 | 9.53mm | 15 | 0.3 mm | 1.02µH | 18.8k | **183** |
| T44-2 | 11.2mm | 13 | 0.3 | 0.99µH | 15.9k | **159** |

The core version T37-2 gives the highest Q factor and was used. But how to fix the core on the PCB? We need

a) absolute mechanical stability to reproduce the values and

b) the high Q factor must not be reduced by the fixing.




This is the solution:
Small discs were cut out from a singled side copper plated PCB (material = FR4, thickness = 1.52 mm).
The core is glued to the non plated side using araldite.

This is a solution with a very high mechanical stability and the part can be soldered to the PCB. t

The most important question is now:

**what says the quality factor Q to this procedure?**

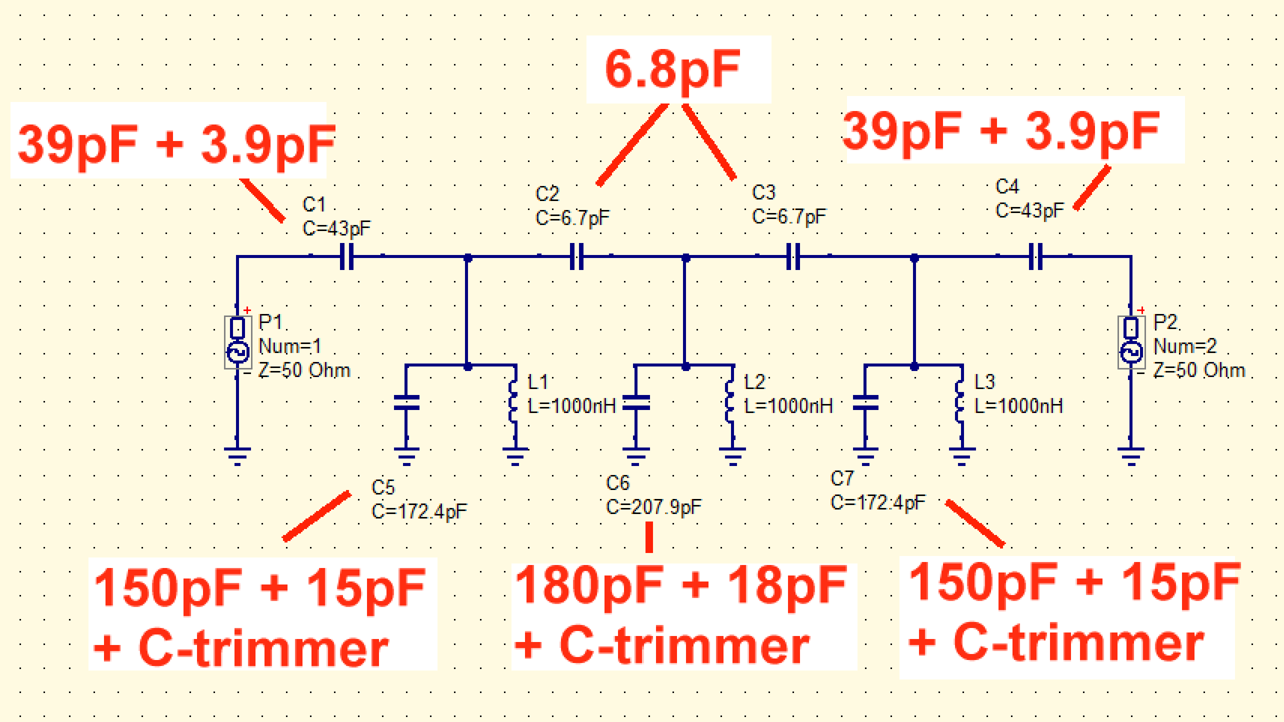
Here comes the result:

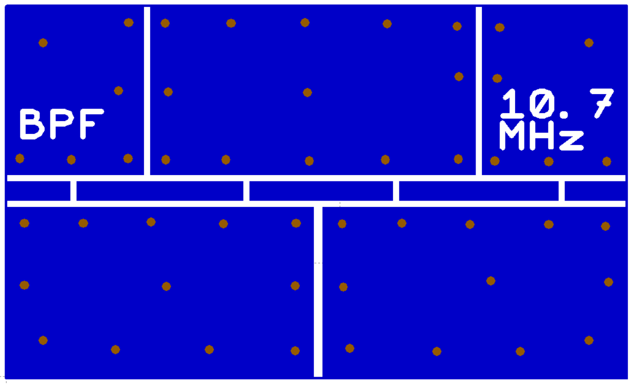
**The inductance values are 1.013 / 1.01 / 1.015 µH and the quality factors are 162 / 167 / 164.**

Thus the coil problems are regarded as to be solved.

## 15.7. The first Test: a Filter Prototype with N = 3

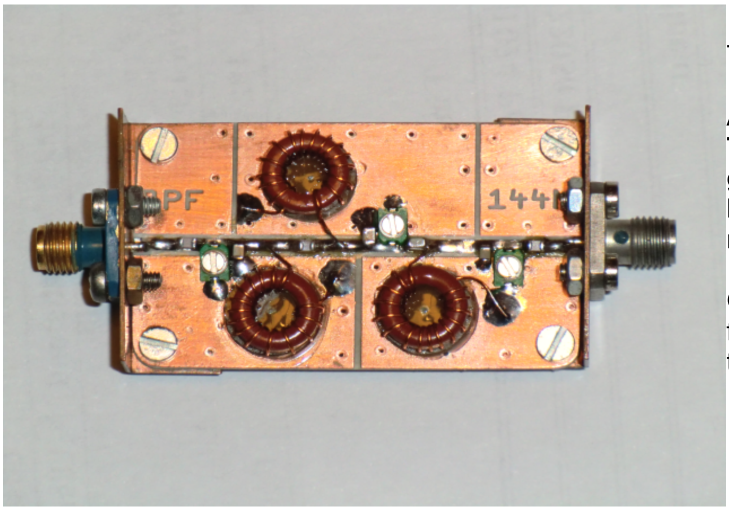
The calculated capacitance valued must at first be changed to “practical values in parallel” (attention: maximum deviation = 1...2%). Trimmers are used for fine tuning.




The prototype is realized on this PCB (dimensions: 30 mm x 50 mm).

The “grounded islands” with a lot of via connected to the “infinite ground plane on the lower side” can be identified

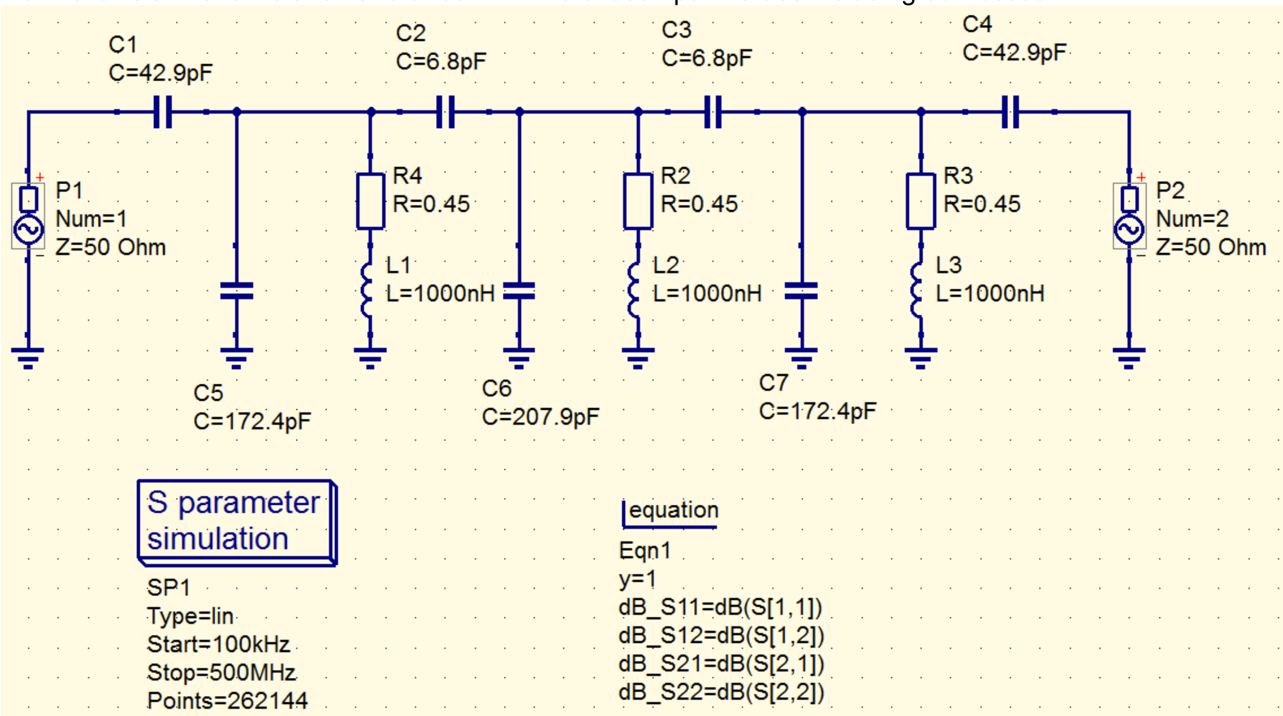


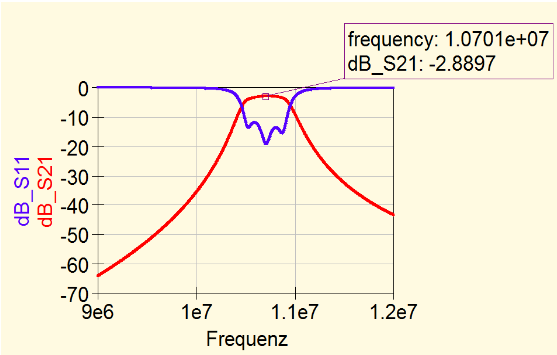
This is the finished prototype.

Attention:
**The rotors of the three C-trimmers must be grounded. When examining you find a nice but small marking on every trimmer for the rotor connection.**

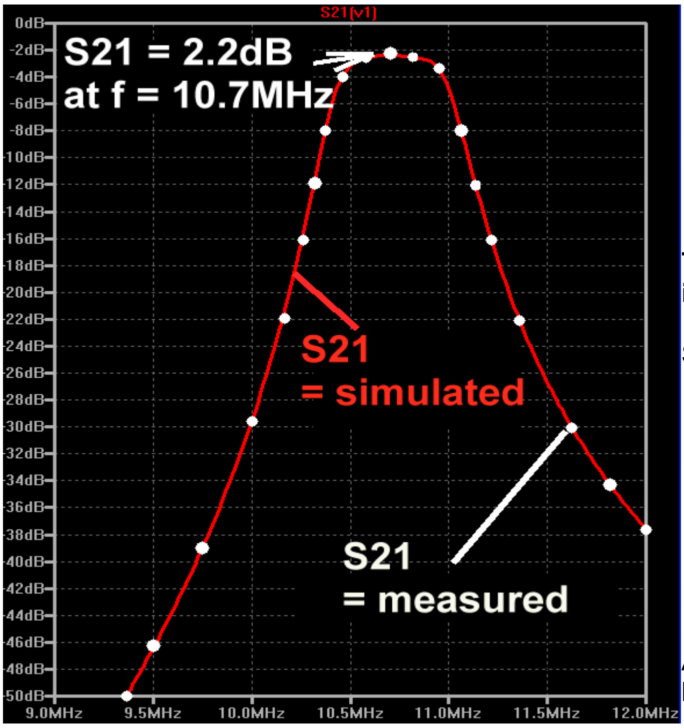
Otherwise you get already problems with the filter curve when approaching the screw driver to the rotor.....

Now follows a final simulation and a  est with the chosen part values including coil losses:



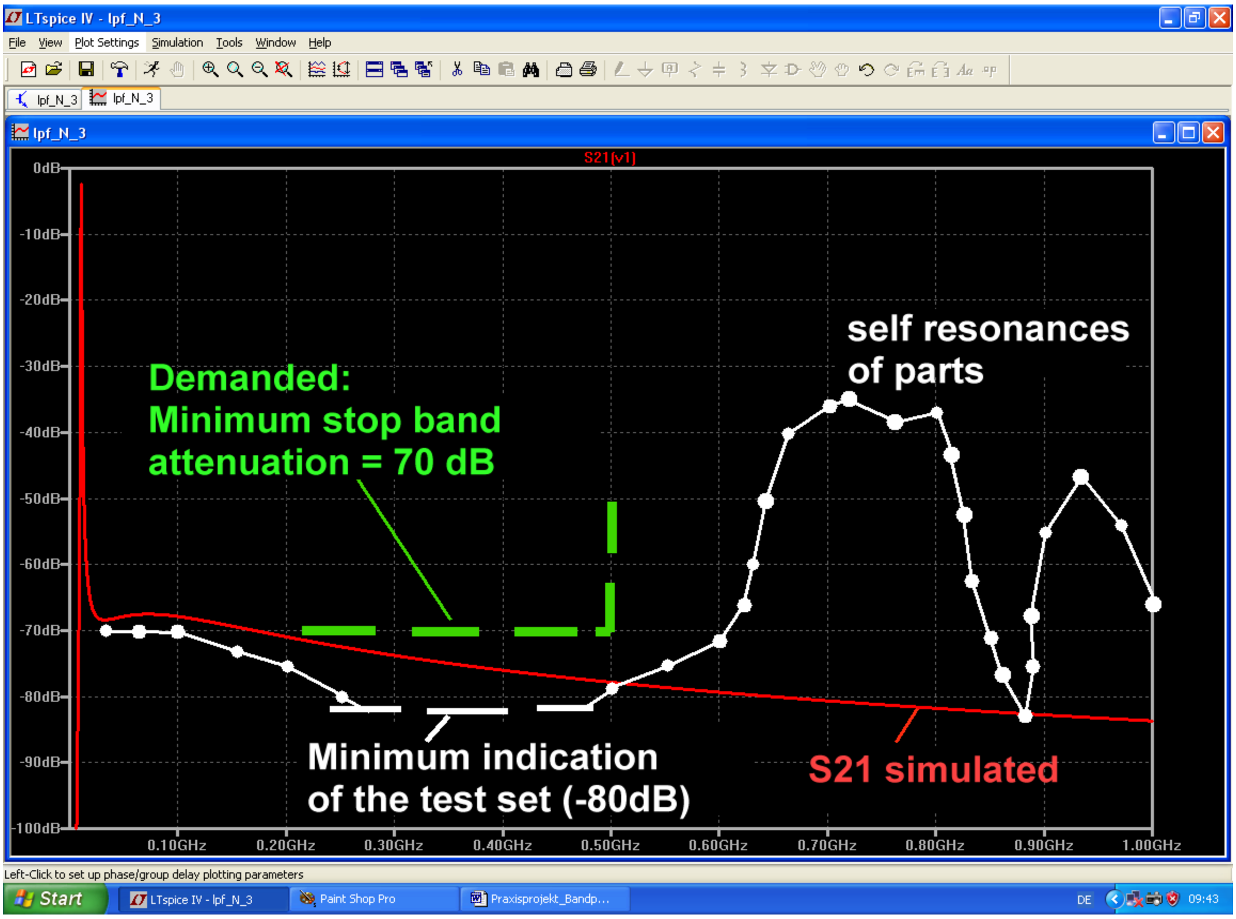

This diagram shows S11 and S21.
The calculated insertion loss is ca. **2.9 dB at 10.7 MHz.**
Also the "negative peaks of S11" are smaller than without losses.




**This is the measurement result. S21 is 2.2 dB instead of the calculated value of 2.9 dB.**

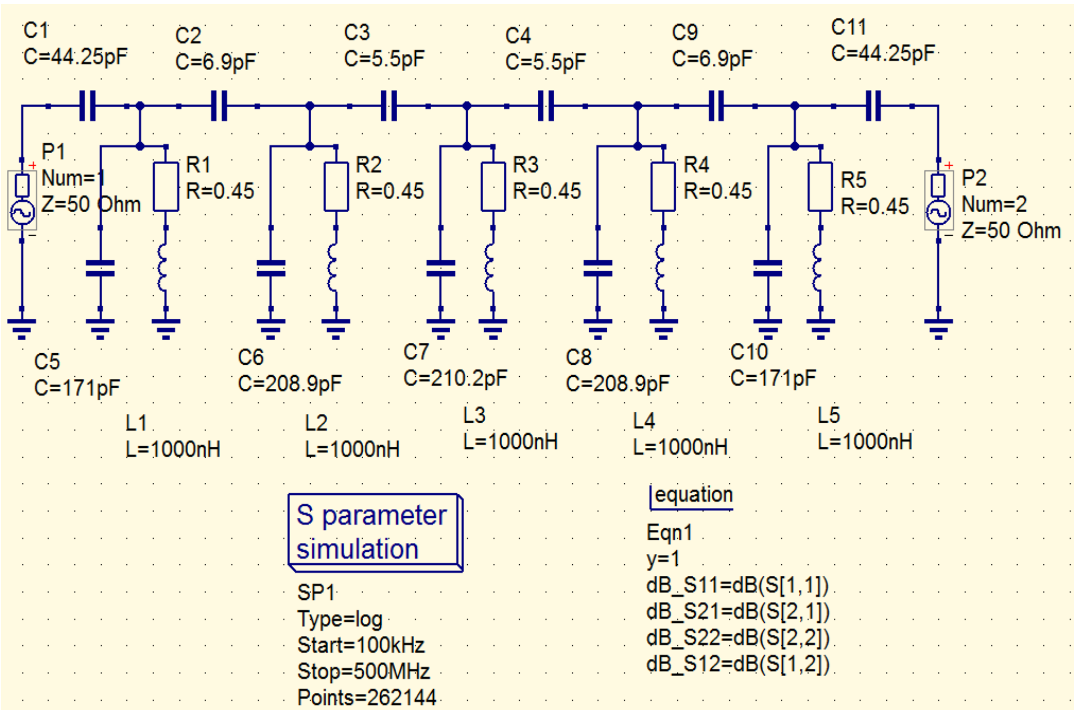
Simulation and realization are nearly similar.

And what says the wide band attenuation in the stop band?



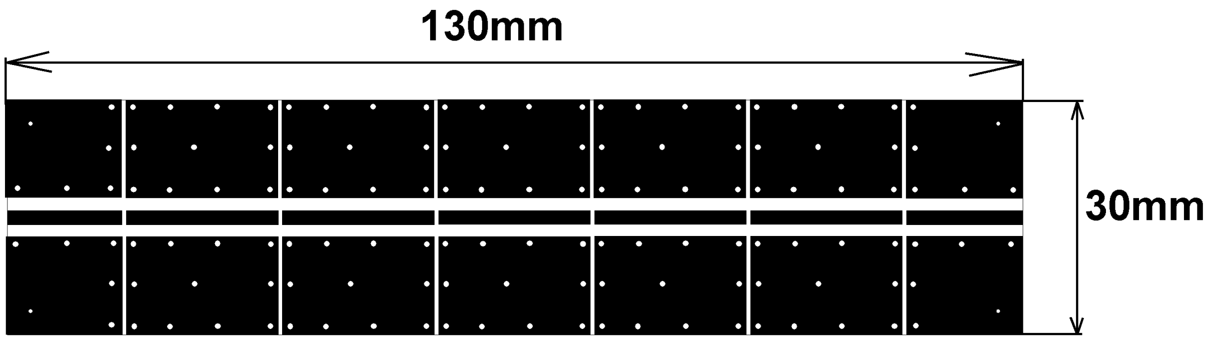

S21 measured up to 1 GHz shows that the specification “minimum stop band attenuation greater than 70 dB up to 500 MHz” is achieved.

## 15.8. The final Game = the 5 Pole Version

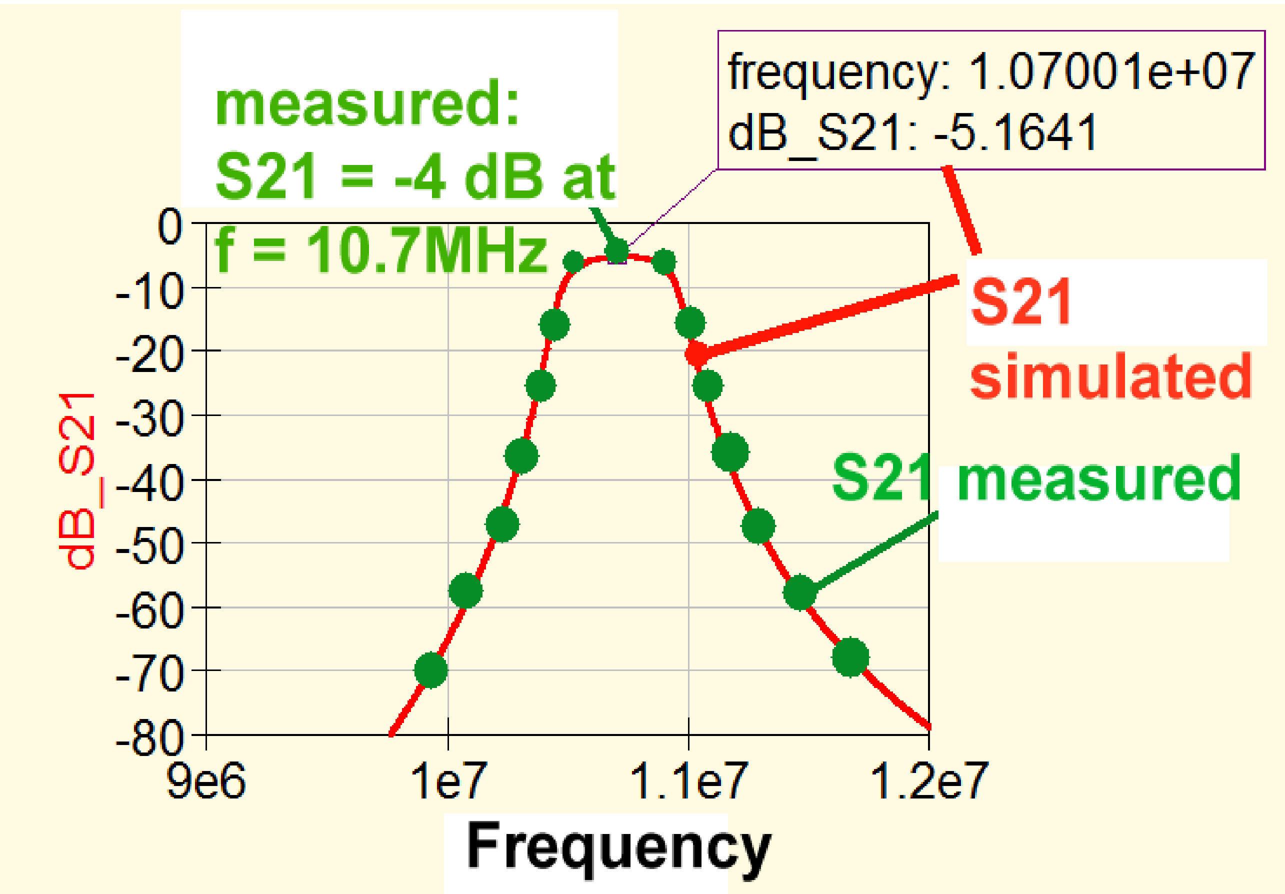



This is the circuit including coil losses:

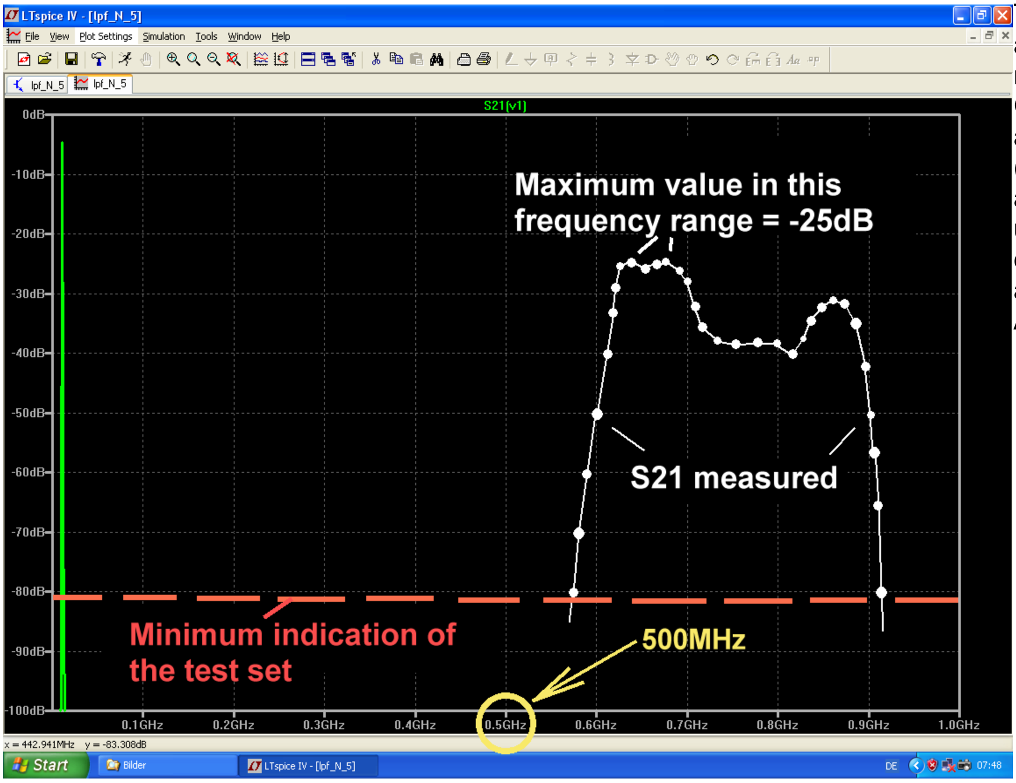
And this is the used PCB:




(Substrate = Rogers RO4003 / height = 2 MIL = 0,813 mm).
You see again the “islands grounded by a huge collection of via. Also the central microstrip (= grounded co planar wave guide) can be seen. Gaps are foreseen in the central microstrip line for the coupling capacitors. The microstrip line pieces between two gaps act up to 500 MHz as additional parallel capacitance to every resonant circuit. But this can be easily corrected by the used C-Trimmer in every circuit.

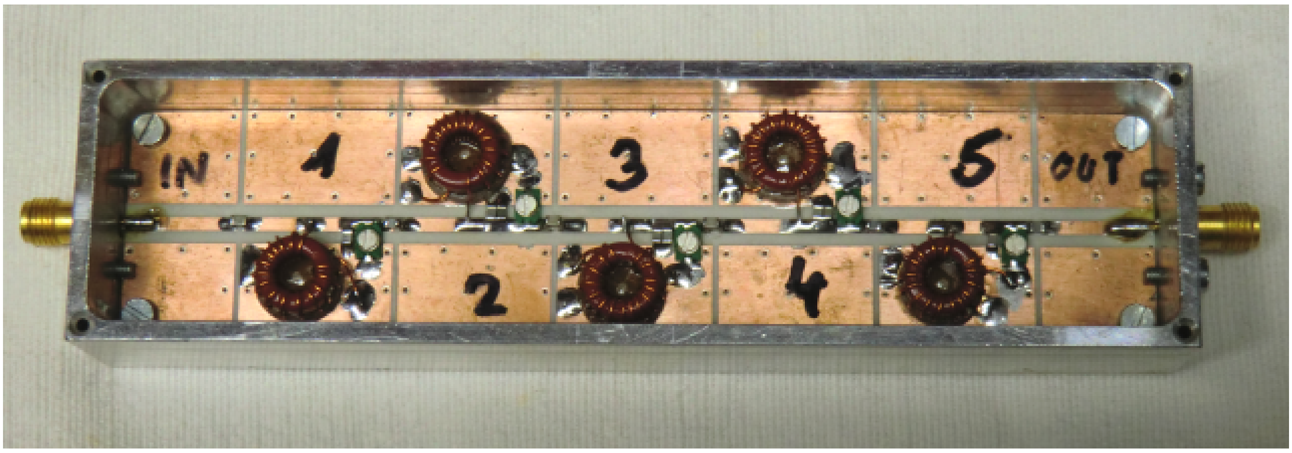


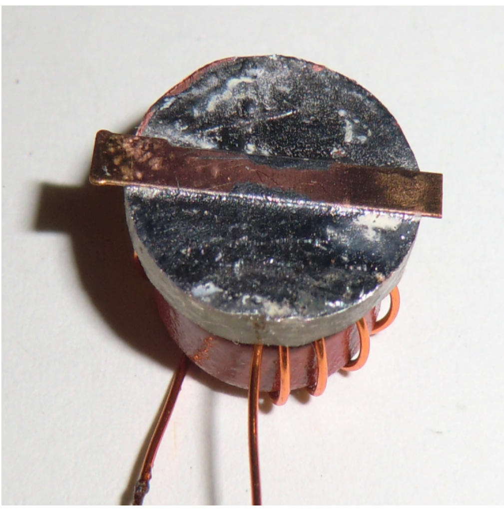

**S21 (and thus the pass band attenuation) is at the center frequency better than expected (-4 dB instead of -5.16 dB).**



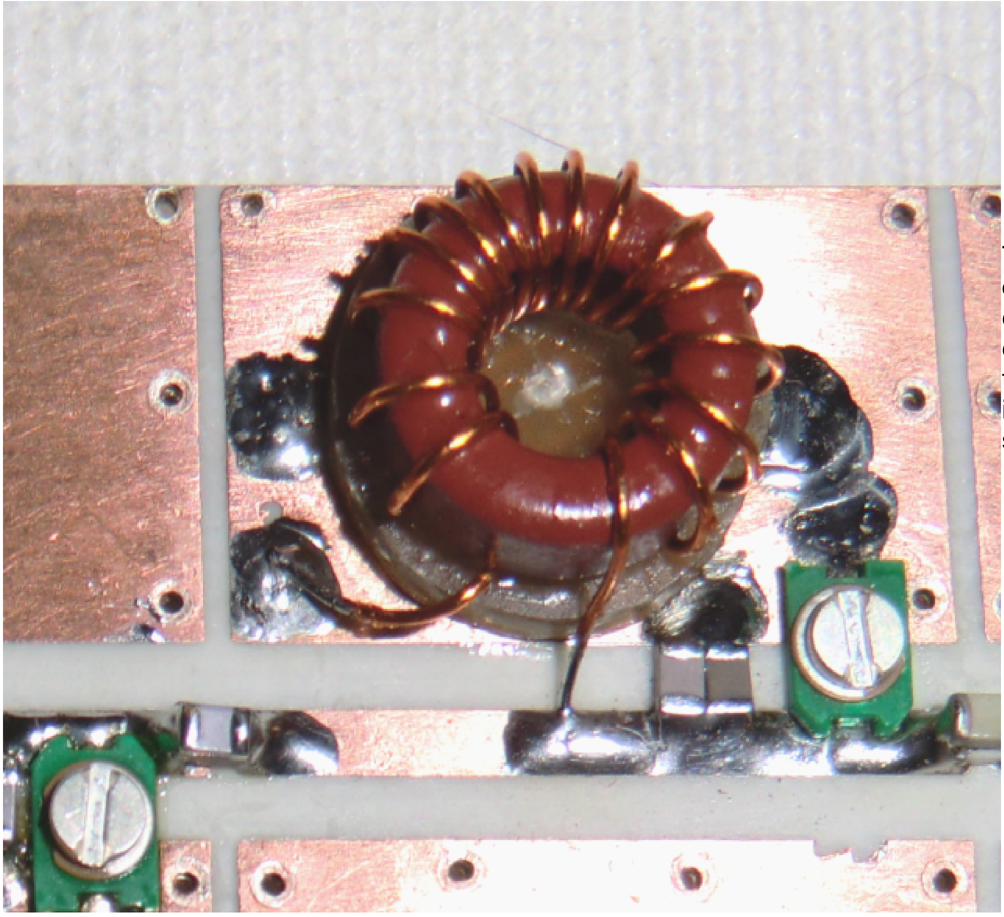

The stop band attenuation was measured up to 1 GHz and is as good as desired.
(The older network analyzer hp8410 was used and the result controlled by a Rohde and Schwarz Analyzer ZVRE)

This is the complete filter including the milled alumina case. But some improvements are included....



The stability of the ferrite ring core coils was improved by soldering a small copper strip to the copper plated side of the “carrier”. Thus we get “SMD parts” which can easily be soldered to the PCB and are fixed like a rock....



This illustration shows one of the 5 resonant circuits including all capacitors and the C trimmer. The mounting is really stable and not sensitive to vibrations.

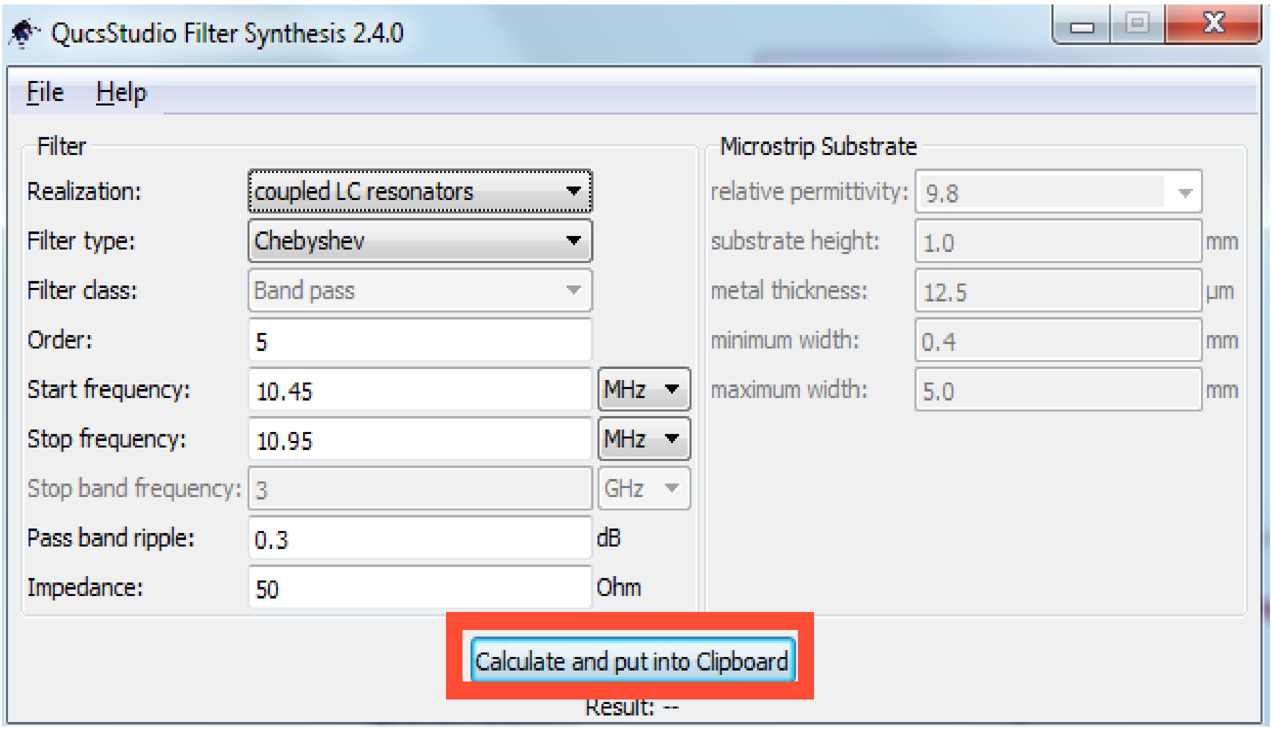
## 15.9. A Comparison: Filter Design using the new qucsstudio Filter Calculator (starting with version 2.4.0) against fds.exe

Let us examine the 5 pole version of the last chapter. If this works fine the efforts with DOS Box and DOS shell can be avoided.

Here come the used specifications of this version:

**Center Frequency = 10.7 MHz**
**Poles = 5**
**Ripple = 0.3**
**3 dB or ripple width = 3 dB**
**Bandwidth = 0.5 MHz**
**Zsource = 50**

Please start qucsstudio 2.4.0. and open the “Tool“ menu. There you'll find “Filter Synthesis“ and you can mark the **“coupled LC resonator**“:

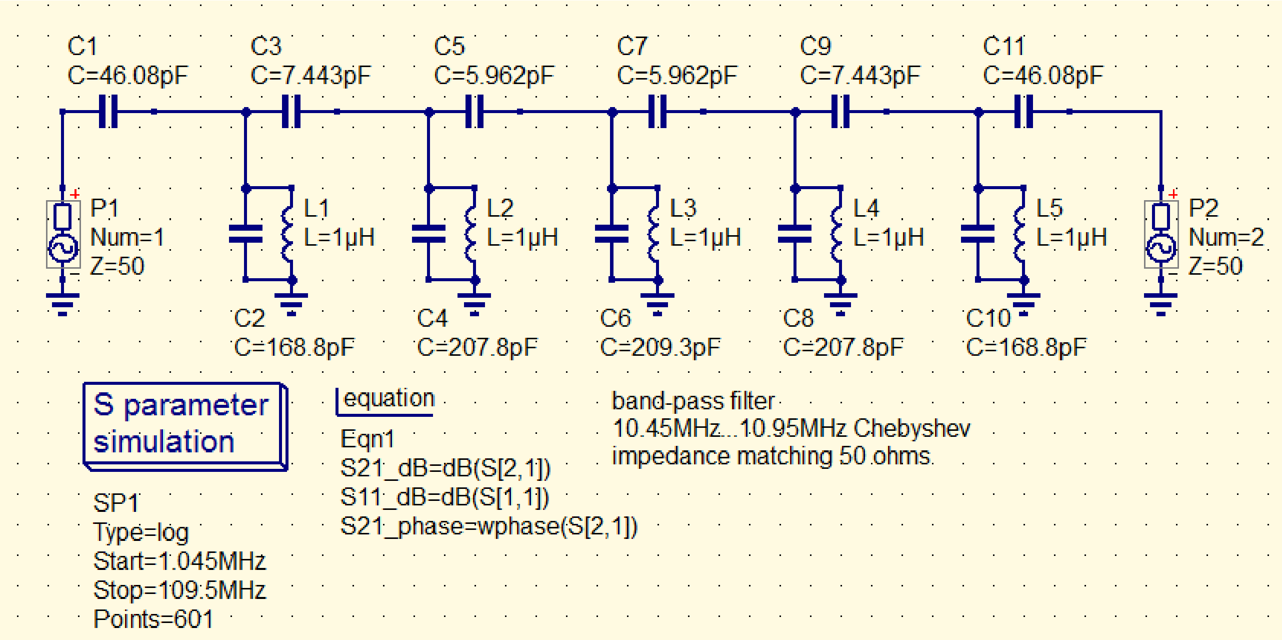



Entering the properties runs without problems.

But:
**The program cannot work with the bandwidth information: you must enter the lower and the upper corner frequency (= start frequency and stop frequency)!**

Then press the red marked “calculate” button. If the calculation was successful then you find the result in the Clipboard.
So open a new schematic and paste the Clipboard content to it.

This is your screen:



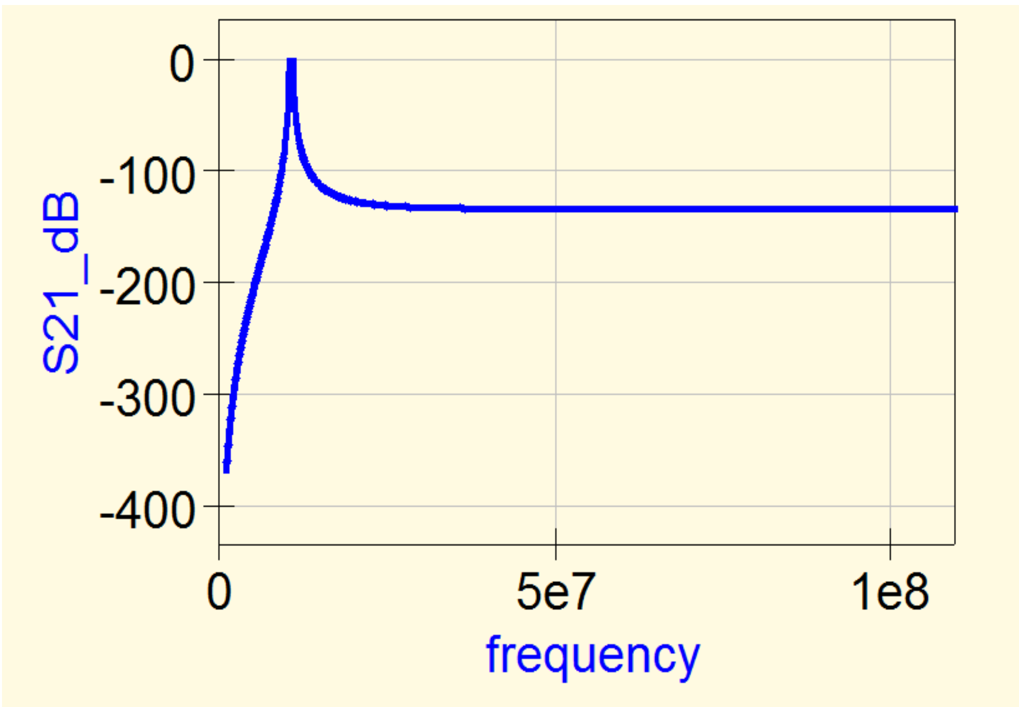

It seems that the qucsstudio calculator works with a little different method: the part values differ fa little from the fds design.

**Other differences to fds:**

a) You cannot choose the inductance – it is automatically set to an optimum value as mentioned in chapter 14.5. for this center frequency.

b) You cannot decide between ripple and 3dB bandwidth.

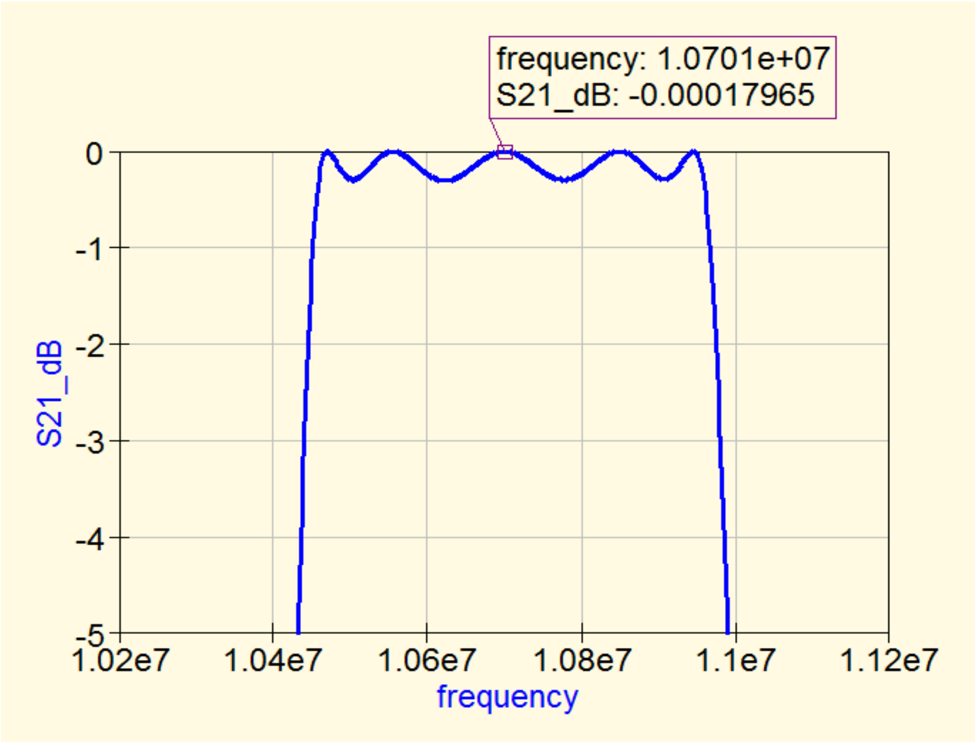
c) You cannot match to other values of the characteristic system impedance with 50 Ohms.




The simulation is already prepared – please simulate (after saving the schematic under a new name).

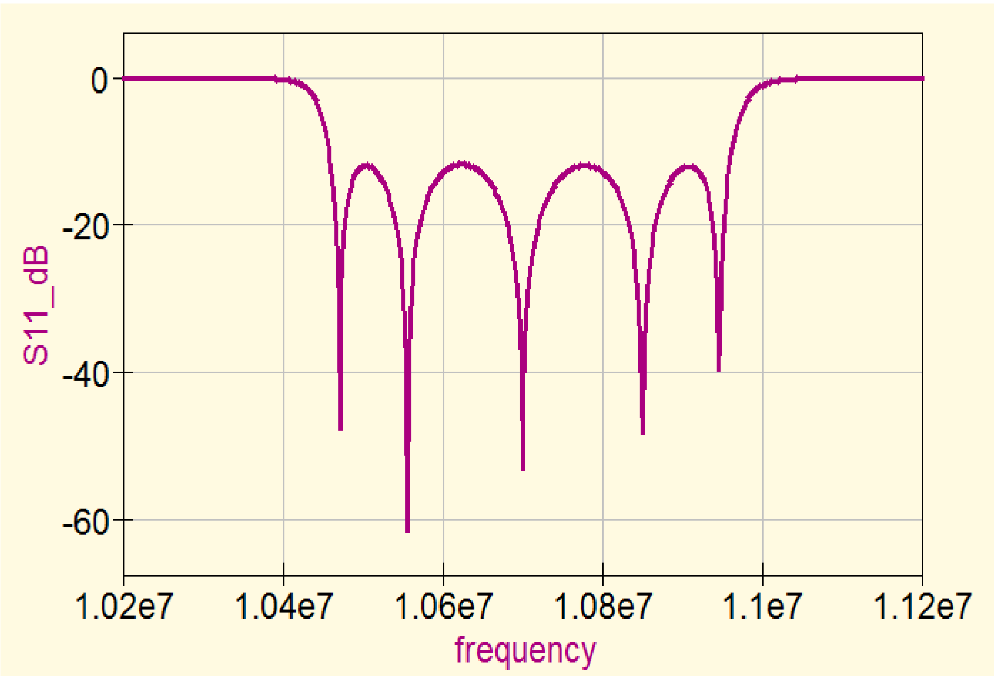
The result is the wide band attenuation up to 110 MHz with a value of ca. 130 dB.

But for an accurate demonstration and analysis of the pass band properties please increase the number of calculated points.
Simulate from 1  to 21 MHz with 20001 points and show the Chebyshev ripple in the pass band (vertical axis = 0....-5 dB / horizontal axis = 10.2 MHz to 11.2 MHz).




Use a frequency marker with 5 digits.

Also demonstrate the input reflection S11 for this frequency range in dB:




No problems....?!

# 16. Developing a WLAN (= 2.45 GHz) Patch Antenna for linear Polarization

## 16.1. Antenna Fundamentals

**A simple definition:**

**An antenna transfers electrical energy coming on 2 lines to the free space and radiates it as an electromagnetic wave.**

The free space can be regarded as a wave guide with a characteristic impedance of $120\pi = 377\ \Omega$.
And because electrical power is transported in the air, electric and magnetic field lines must always be in phase at a regarded point of the way (...this is characterized by the Poynting vector which means “power per area unit”).
But never forget: **the electric field and the magnetic field are perpendicular to each other and to the direction the plane wave is propagating.**

The permanent transmission of energy into space is represented by a radiation resistance at the antenna input and the power comes from an applied voltage source.

**The radiation resistance is depending of the antenna construction and type, from length and diameter of the antenna wires, the frequency, the environment, the ground properties....not easy to calculate!**

You must distinguish between narrow band and broad band antennas.
Narrow band antennas can be regarded as resonant circuits and this means:

d) They have an exactly defined resonant frequency and at this frequency only a pure resistor is the input impedance (= No capacitance or inductance can be measured). But the input resistor itself is a series connection of the radiation resistance, the losses of the antenna wires and cable losses, losses in the ground around the antenna....

e) If you change the frequency you get the same behavior as in a true resonant circuit: inductive and capacitive part do not longer cancel another.

f) An antenna is an open resonant circuit. Thus we find very strong electric and magnetic fields around the antenna (= Near Field Region), but a phase difference of 90 degrees exists between them and the resistor current responsible for the transmitted power (...as demanded for an inductance and capacitance). These strong near fields do not radiate but can be dangerous for living organism like men. These near fields **decrease with “distance$^3$” to the antenna.**

g) **With increasing distance from the antenna (thumb rule: minimum distance = 10 x wave length) we are in the pure far field region (=Fraunhofer region).** Now the electromagnetic field transmits energy into space and thus the electric and the magnetic field must be in phase (...but the field lines perpendicular to another)**.These far fields decrease only with (1 / distance)** and so a talk between Europe and America is possible – with very low transmitted power.

When traveling away from the antenna the energy is spread over an permanently increasing area. Thus the energy kept up by an antenna for the receiver is decreasing with the distance due to the **Friis formula**:

$$P_{received} = P_{transmitted} * G_{transmitting\_antenna} * G_{receiving\_antenna} * (\lambda / 4*\pi*d)^2$$

**„d“** is the distance between receiver and transmitter. “

“**λ” is the used wavelength and can be calculated from frequency and light velocity.**

**“G”** is the antenna gain.

## 16.2. Fundamentals of Patch Antennas

They use a PCB with both sides plated with copper. The lower side is the “infinite ground plane” and on the upper side we find a simple rectangle or square. The outer dimensions of the PCB must be ca. 3...5 mm greater than the patch dimensions or more.

– The patch length is responsible for the resonant frequency and must be approx. **0.49 x (λ / 2)** (..the exact value will come from the simulation...)

– the patch width is responsible for the bandwidth and the radiation resistance.

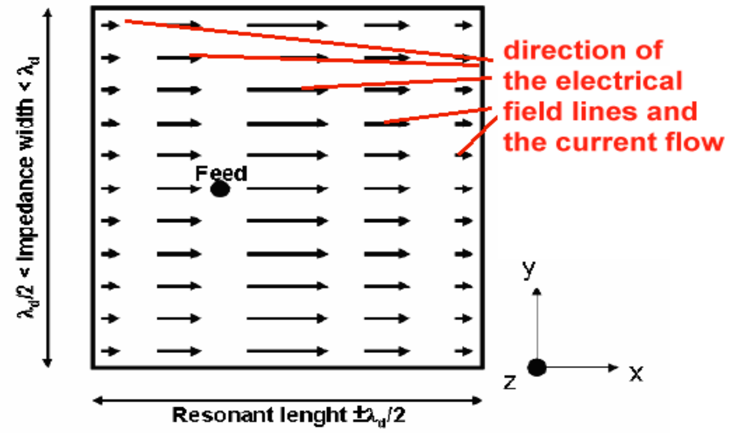
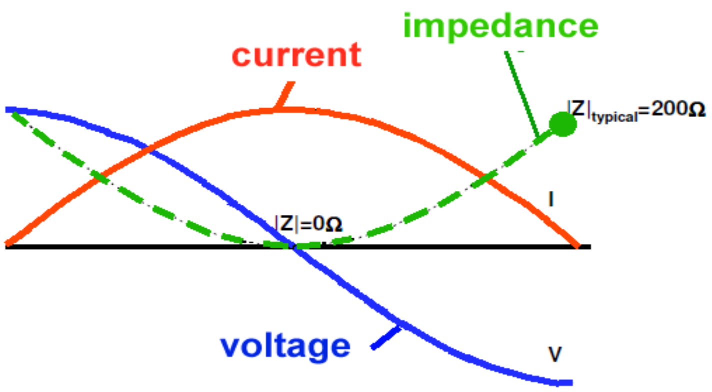



*Figure 2: Current distribution on the patch surface*




If you feed the antenna at the center of the left vertical patch edge (or near by) you'll get the current and voltage distribution as shown in the illustration.

At the right hand side of the patch the voltage has the same value but opposite phase (due to the line length of λ/2 = 180 degrees). **And exactly in the middle of the ptach you find U = Zero.**

But the current is zero at the start and the end of this transmission line and the maximum is found at the center.

At the start and the end of the line a radiation resistance represents the radiated energy. Each resistor has a value between 150 and 1000Ω (...depending of the antenna layout) and a λ/2 line does not change the impedance of a load resistor. Its value appears at the input and thus we find a parallel connection of both radiation resistors at the input = the connection of the supply voltage source.
Regarding this illustration we can say that for an input impedance of 200Ω at every side of the line a resistor of 400Ω can be thought.

Exactly at the center of the line we find U = 0 and thus the input impedance at this point is also Zero. That means that the input impedance starts with 200Ω at the left edge of the patch and decreases to Zero when walking to the center.
**So there must be a “feed point” with Z = 50 Ω along this way – but how to find him.....?**

**The patch width has no much influence to the resonant frequency. Starting with a square patch (= standard) the radiation resistance decreases and the bandwidth increases with increasing patch width.**
The design of patch antennas is usually done with an “**EM simulator for planar structures**” and the best choice for a private is the free **SONNET LITE** version. It is worldwide in use and well known.

**From my homepage / English section (www.gunthard-kraus.de) you can download a free Sonnet Lite Tutorial with the exact design description of a 5.8 GHz patch antenna.**

But one question must still be answered:
**Why and and how does such a patch radiate?**

So have a look at this illustration:

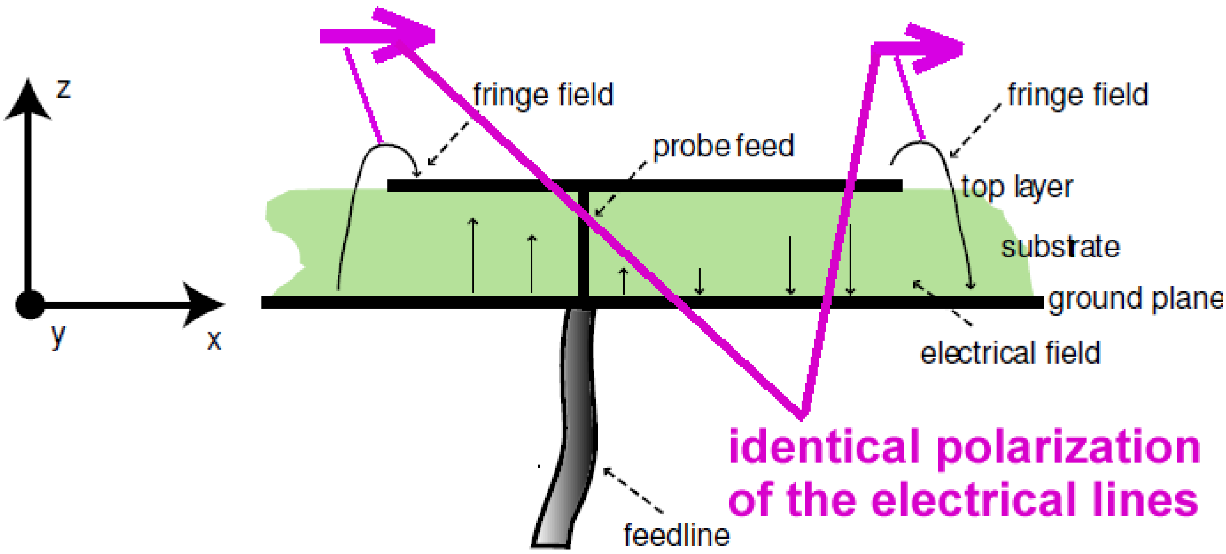



*Figure 1: Cross section of a patch antenna in its basic form*

Please examine the “fringe fields” at both edges of the patch! The voltages are in counter phase but the electrical field lines have the same polarization!
**Thus the left and right edge of the patch act as two “slot radiators” in parallel!**

(a slot radiator is the complementary version of a dipole. A dipole is a metal wire in an air environment. The slot radiator is an “air slot in a metal environment”. Thus the directions of electrical and magnetic field lines for the slot radiator are interchanged in comparison to the dipole).
The illustration gives also the answer to the polarization of the radiated electrical field in form of the two pink arrows.

If the ground plane is a bit greater than the patch, then you have a “shielding” and no radiation is possible in the downward direction (= in the illustration the patch will only radiate upwards).

The antenna diagram for the radiated power of a dipole is a “**8**”. But now one half of the “8” is missing and you get only a vertical circle for our patch antenna.

## 16.3. Measuring Antenna Properties

**a) Resonant Frequency, Input Impedance and Radiation Resistance**
This is best done with a Vector Network Analyzer connected to the antenna and a Smith chart for the presentation of the result. In this diagram the impedance shows normally “a loop around the center”. And the center means an impedance of exactly 50 Ω.
The distance of any curve point to the center equates the reflection factor “r” at this frequency which can be used to calculate the complex input impedance.

**Caution:**
Patch antennas show several “perfect matching points” when sweeping over a wider frequency range – but at only one frequency radiation is possible, if the patch length equates half a wavelength. But how can I check that?

Very simple: connect your antenna to the Network Analyzer and approach your hand to the antenna. A part of the radiated energy is reflected by your hand to the antenna and the Analyzer accepts this as additional reflected wave when measuring S11. So the S11 curve on the screen will jump like a rabbit with the the same rhythm of the hand movement.

**b) Cross Polarization**
Use two identical antennas. The first is connected to the output of a signal generator which is accorded to the antenna's resonant frequency. The second antenna feeds a Spectrum Analyzer as a receiver and is located in the far field (minimum distance = 10 wavelengths). Now align the frequency and the position (= direction) of both antennas for maximum input level at the receiver.
Afterwards loosen the connection between the second antenna and its jack a little bit and turn the antenna by 180 degrees. At 90 degrees there should no signal be found (= perpendicular polarization) but in practice you find a level with a reduction between 30 and 40 dB. This level difference to the maximum value is the cross polarization in dB.

**c) Horizontal Diagram**
Difficult to measure with the normal equipment as before (= demands an absorbing chamber and a turntable with exact degree calibration). Starts with the two antennas aligned for maximum receiver level, then the receiving antenna is turned around itself for 360 degrees and the level changes measured.
For the complete diagram (including the vertical part) some times a helicopter is used.

**d) Antenna Gain**
In this case the maximum level from the receiving antenna is **compared to the received level from an “isotropic radiator**”. The level difference is named “**antenna gain**” and the result is given in

**dBi**

Sometimes the simple dipole is used as reference, but this dipole has already a gain of 2.15 dBi.
Thus the “**gain referred to the dipole”** is measured in

**dBd**

and you get it if you reduce the dBi value by 2.15 dB.

But now the answer to the question: how can the antenna gain be measured?
A standard method uses the “Friis” formula as given in chapter 15.1. and the explaining text is the same as above for the “cross polarization:

**“Use two identical antennas. The first is connected to the output of a signal generator which is accorded to the antenna's resonant frequency. The second antenna feeds a Spectrum Analyzer as a receiver and is located in the far field (= a minimum distance of ca. 10 wavelengths) . Now align the frequency and the direction of both antennas for maximum input level at the receiver.”**

**The distance between the two antennas “d” must be known as exactly as possible – and also the transmitted power level.**
Then says the Friis formula (See chapter 15.1):

$$P_{received} = P_{transmitted} * G_{transmitting\_antenna} * G_{receiving\_antenna} * (\lambda / 4*\pi*d)^2$$

If you use two identical antennas then the gain values are also identical. And after changing to the calculation in dB and dBm you get this form of relationship:

**Antenna Gain in dBi =**

**[(Received Level in dBm) – (Transmitted Level in dBm) - 20*log( λ / 4*π*d)] / 2**

For your information:
**A typical gain value for Patch Antennas is ca. 6......7 dBi.**

## 16.4. Design of a WLAN Patch Antenna for linear Polarization

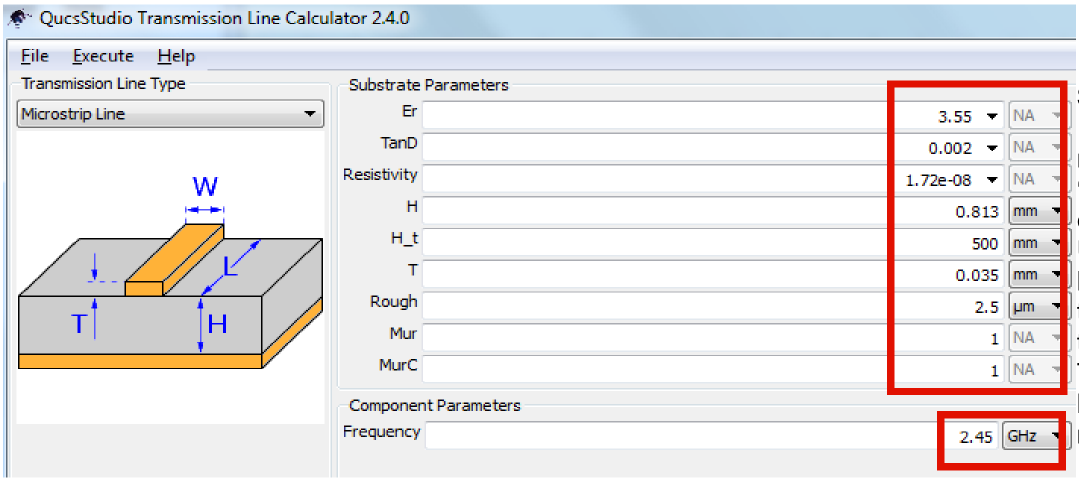
## 16.4.1. Patch Design using qucsstudio

We know:

The electrical length of the Microstrip Line used as Patch Antenna must be **λ/2 = 180 degrees.**
The patch width determines the radiation resistance, the quality factor Q and the bandwidth (increasing the width increases the bandwidth but decreases radiation resistance and Q).
So let us start with the typical standard version of a **square patch.**

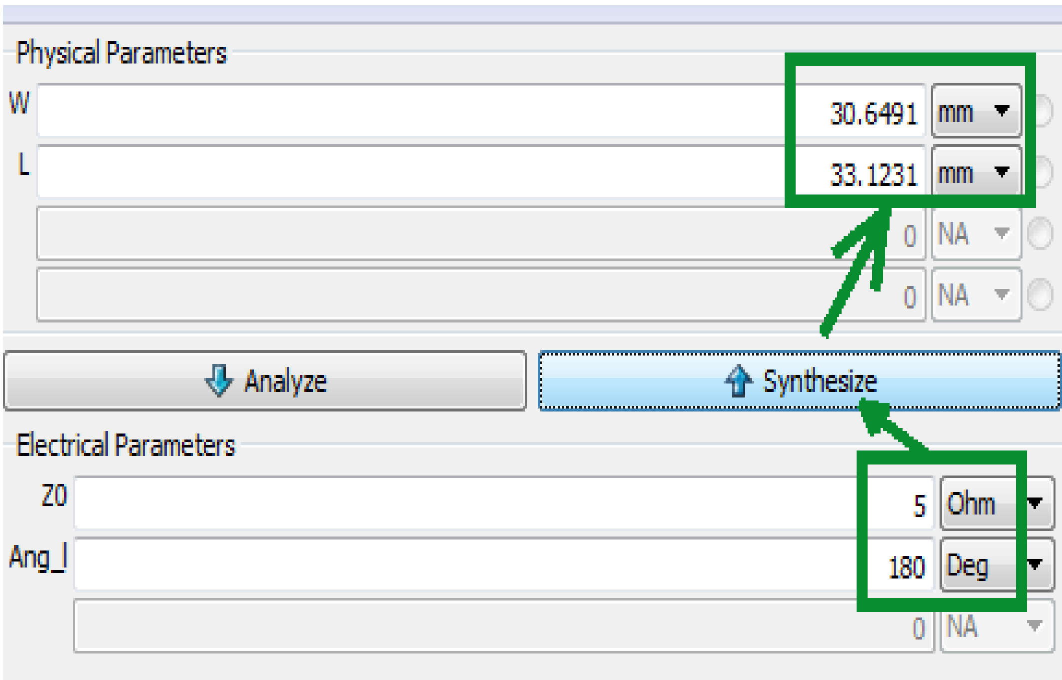
**The PCB material is Rogers RO4003 (height = 32 mil = 0.813 mm, $\varepsilon_r$ = 3.55). Transmitting frequency is 2450 MHz.)**

**Step 1:**
Start qucsstudio and open a new project named "**patch_antenna**". Save the opened schematic as "***patch_01***"



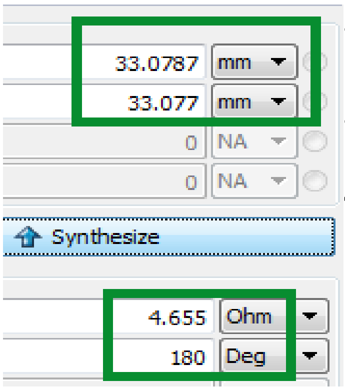

**Step 2:**
In the "**Tools**" menu you find **"line calculations".** Use "**microstrip line**" as transmission line type.
Then enter in the **left half of the menu:**

**er = 3.55 / TanD = 0.002 / H = substrate height = 32mil = 0.813 mm/ H_t = cover height = 500 mm / copper thickness = 35 µm / roughness = 2,5 µm / frequency = 2.45 GHz**




The design frequency is **2.45 GHz** and there we need an electrical length of **180 degrees.**

The patch shall be a **square (length = width)** and experience shows that therefore a characteristic line impedance of **ca. 5 Ω gives a good start for a width with nearly the same value as the length.**




**Step 3:**
The result is not perfect. So please vary the impedance of the line and synthesize as long as you get equal values for width and length.

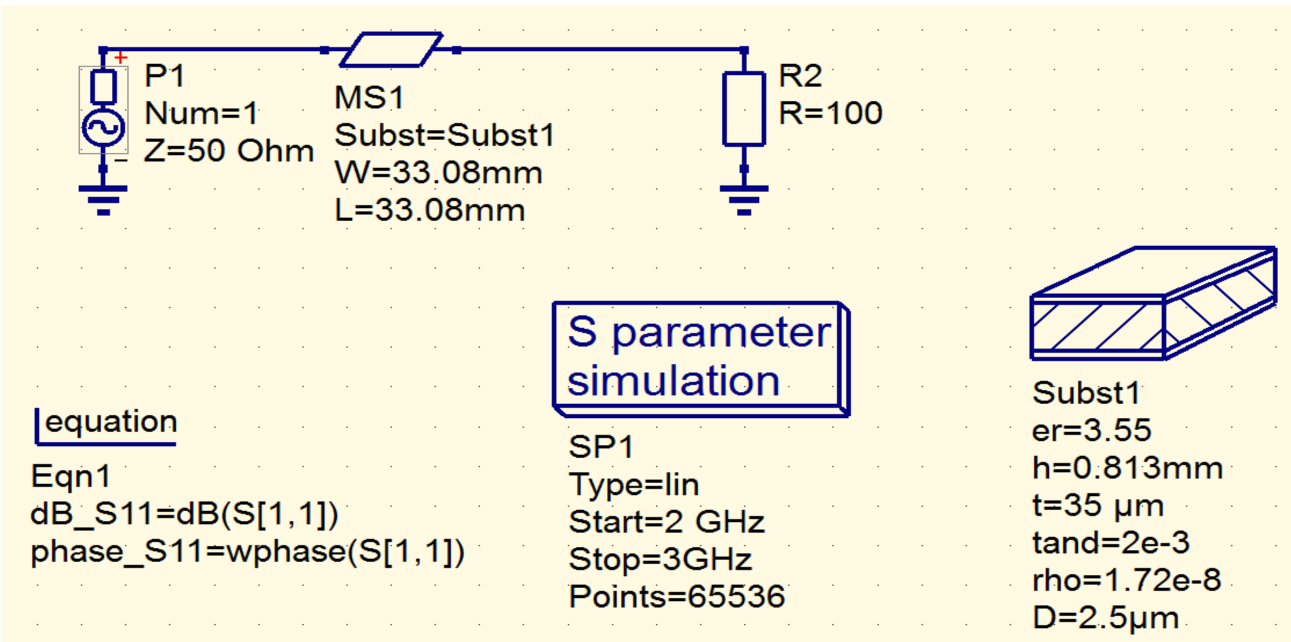
This is the solution for length = width = 33.08 mm and you need a value of

**Z = 4.655 Ω**

**Step 4:**
Now test the result with a simulation. Use a microstrip line with this impedance and a length of 33.08mm. Connect a resistor of 100 Ω to the end of the line.

**If you terminate a λ/2 line at the end with a pure resistor then you'll measure exact the value of this resistor at the input – without any capacitive or inductive admittances, but only at this frequency!**

Let us use a resistor value of 100 Ω:




**To do list:**

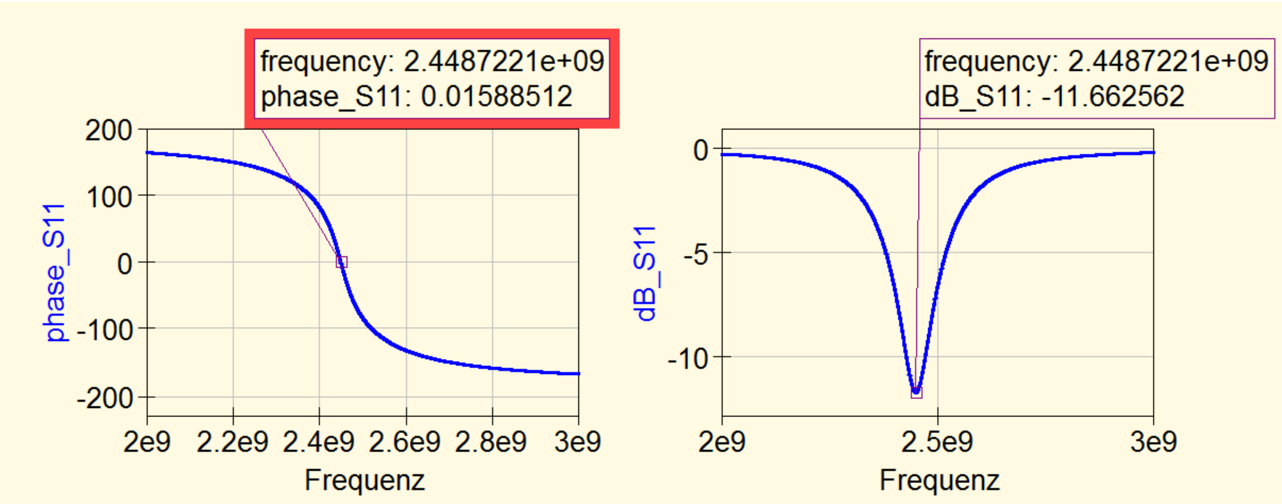
a) The microstrip line MS1 uses a length of 33.08 mm, a width of 33.08 mm and substrate “Subst1”. This information must be entered in the property menu of the line (MS1 can be found in „**Components / Transmission Lines / Microstrip Line“**).

b) The Substrate “Subst1” comes also from the „**Components / Transmission Lines**” menu and you have to enter the properties as given above in the illustration.

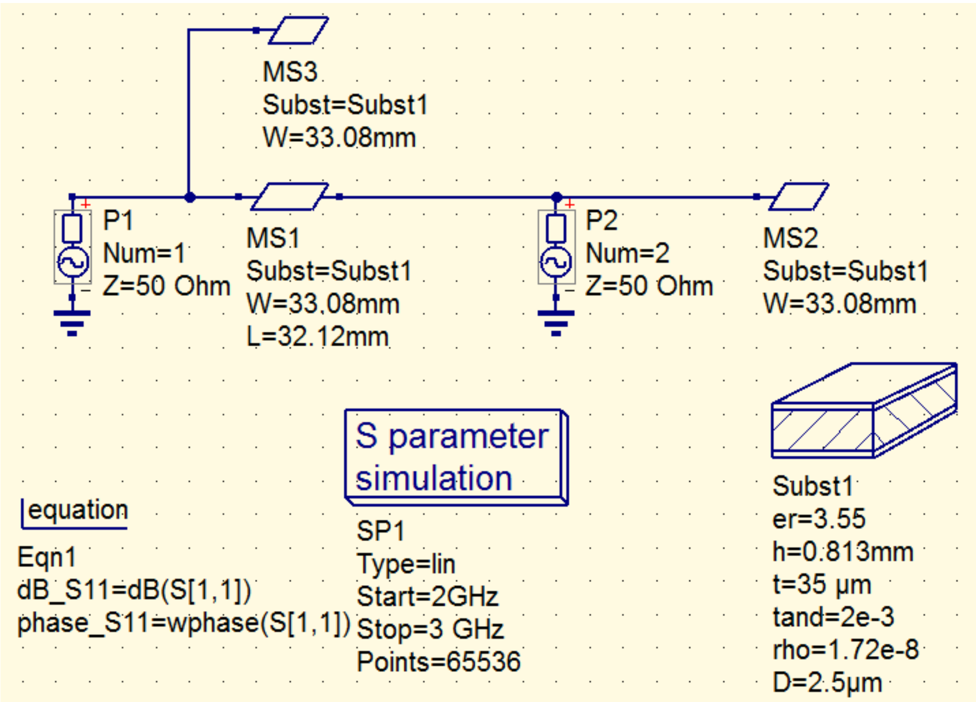
c) We need the equations for the S11 magnitude in dB and the S11 phase.

d) We simulate linear from 2....3 GHz using 65 536 points.

This is the result using markers with high resolution:




In the left diagram we find the “frequency for zero phase” at 2.44872 GHz – this is an error of 1.28 MHz = 0.05 %..... OK!




**Step 5:**
We must not ignore the fringing fields at every end of the line:

This can be done by the part named “microstrip open” from “components transmission lines”.
When connected to every edge and set to the correct line width you must shorten the line length.
You can stop the process of iteration when you have reached a **“frequency for zero phase” of 2.45 GHz.**

This is achieved by a **new line length of 32.12 mm and a line width of 33.08 mm.**

**But it is impossible to calculate the radiation resistance of this patch antenna using qucsstudio.** **The free EM simulation software Sonnet Lite will do this job for you without great efforts** if all losses (of the metal and the PCB substrate and the air environment) are set to zero. The only remaining resistor is then the radiation resistance!
(You can download a free tutorial for Sonnet Lite from my homepage www.gunthard-kraus.de with a patch antenne example project and do this job yourself).

**Here comes the Sonnet simulation result:**

At the patch input we find a total input resistor of **457 Ω** which is the parallel connection of a radiation resistor of **914 Ω at every patch edge. This is due to the two radiating slots connected by a "half lambda transmission line which does not transform the value of the termination resistance at the line end when regarding the line input**.

But how is it possible to match this high input resistance of **457 Ω to our 50 Ω system?**

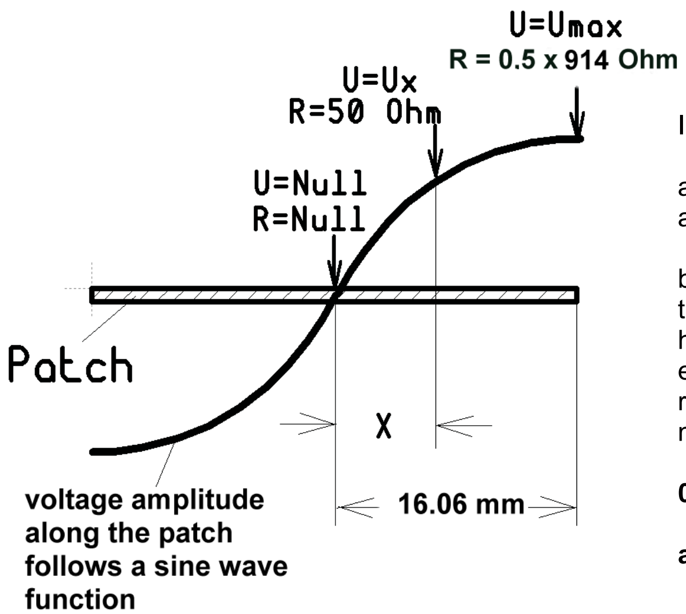
## 16.4.2. Matching the Antenna to 50 Ω

Two methods are in use:

a) Using a λ/4 match line or

b) Using a „**Coaxial Feed**" placed on the lower side of the PCB.

**Let us design method b) with the "underground feed".**




In this illustration you see that

a) that the voltage amplitude and the impedance at the patch center is zero.

b) At every patch edge we find the maximum of the voltage but only a small current. This gives a high input impedance at the patch edges and the exact value is the parallel connection of the two radiation resistances (= 914 Ω). Thus we measure

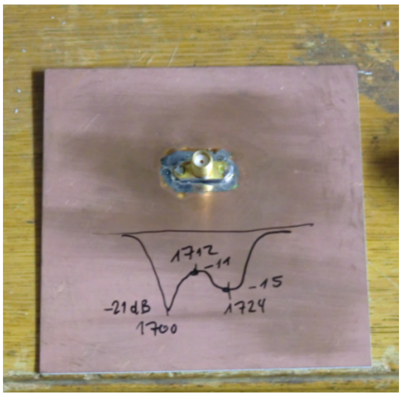
**0.5 x 914 Ω = 457 Ω**

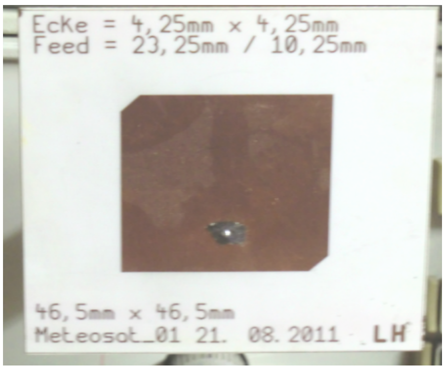
**as input impedance.**

**Now a point "x" can be found between patch center and patch edge where exactly 50 Ω are measured as input impedance!**

If you now drill a hole at this point of the PCB and solder this patch point to the center conductor of an SMA jack (located at the lower side of the PCB) then you have reached the goal.

Two illustrations of a Meteosat antenna shows this idea.



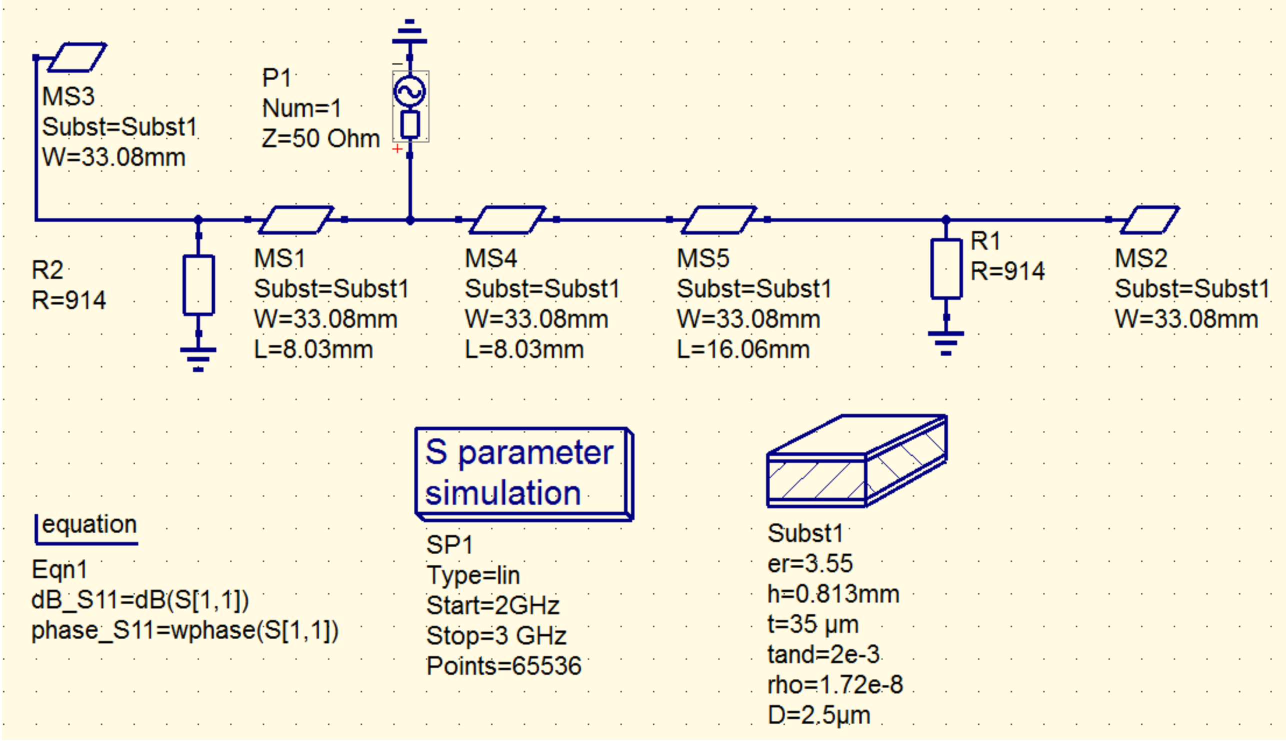
Now we try find this point “x” with a qucsstudio simulation.

**Therefore we divide the patch into two identical halves (at Z = zero Ohms). Thus you get two line pieces with a length of**

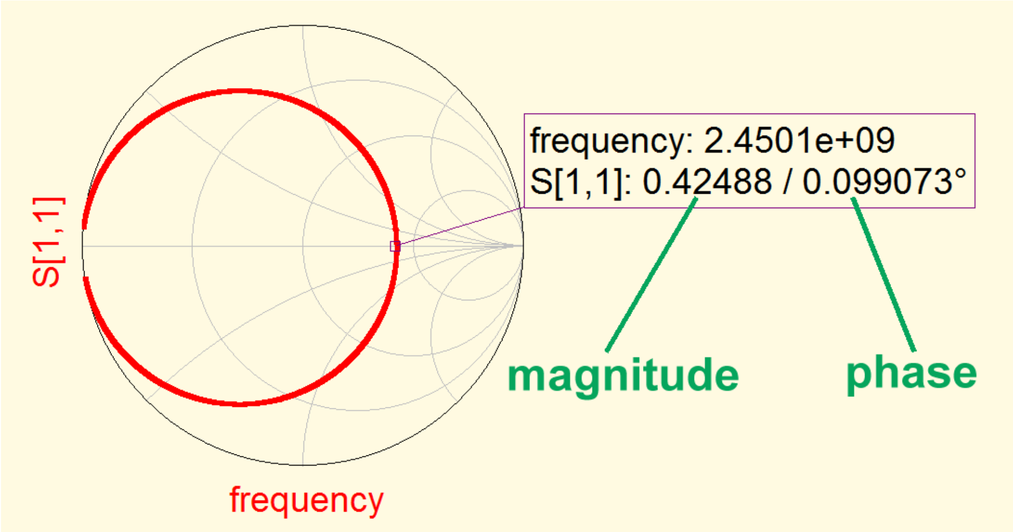
**32.12 mm / 2 = 16.06 mm.**

| |
|---|
| Now divide the left line piece again in two halves an connect Port 1 to this point.<br>So the patch is now divided in three pieces:<br><br>**= 8.03 mm + 8.03 mm +16.06 mm = 32.12 mm** |

**Then simulate this schematic:**




Present S11 in the Smith chart and mark the resonance frequency where the phase is zero:

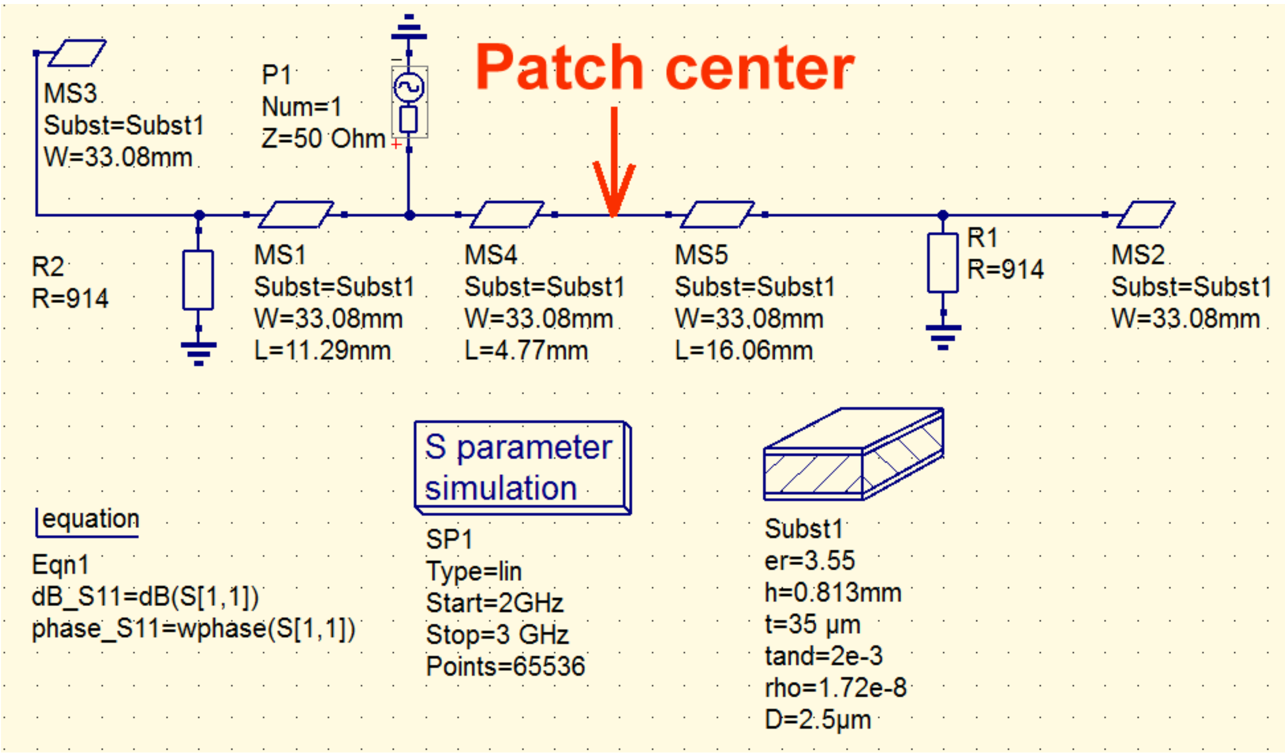


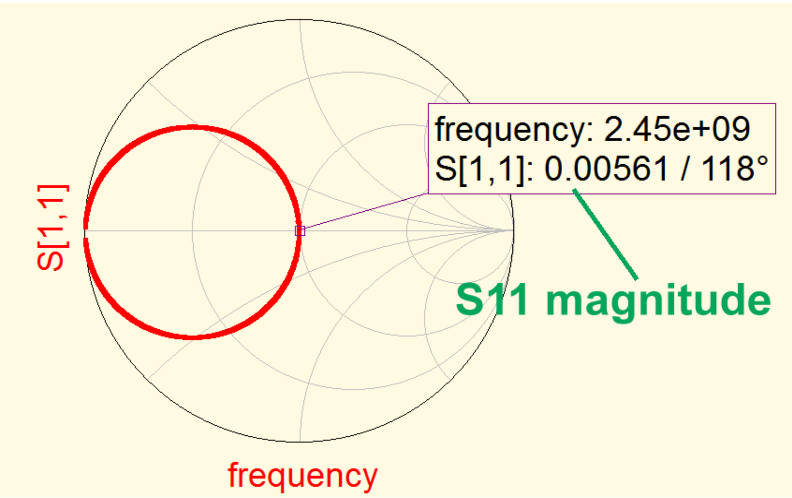

For perfect match the red S11 circle must run through the center of the Smith chart.

But:
**In our example the circles passes the center at the right hand side. Thus the input impedance is still too high at the chosen point and we have to move towards the patch center.**

(If the S11 curve would have passed at the left of the center then the input is too low and we would have to move in direction to the patch edge).

**Please pay attention:**
**If you “shift the feed point along the patch” the sum of the two line pieces must always have a value of 16.06 mm = half patch length!**

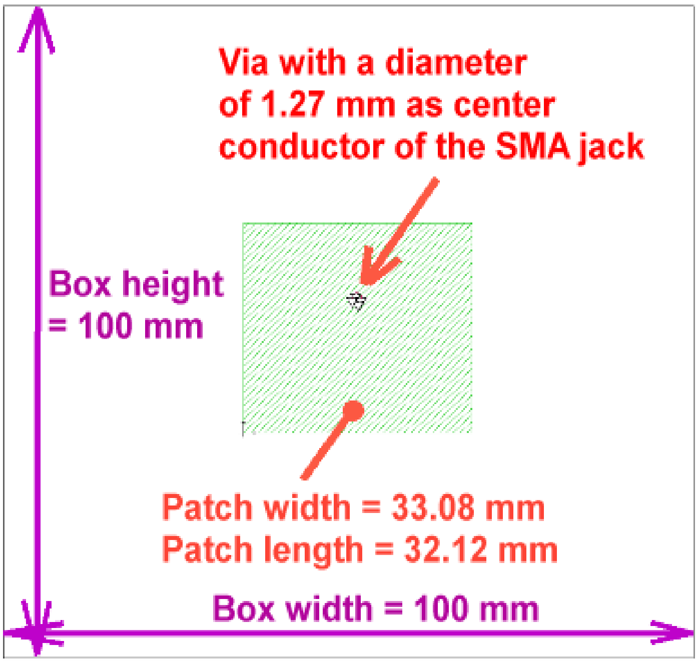



This is the end of the efforts: a feed point located at **4.77 mm** left from the patch center will do the job.

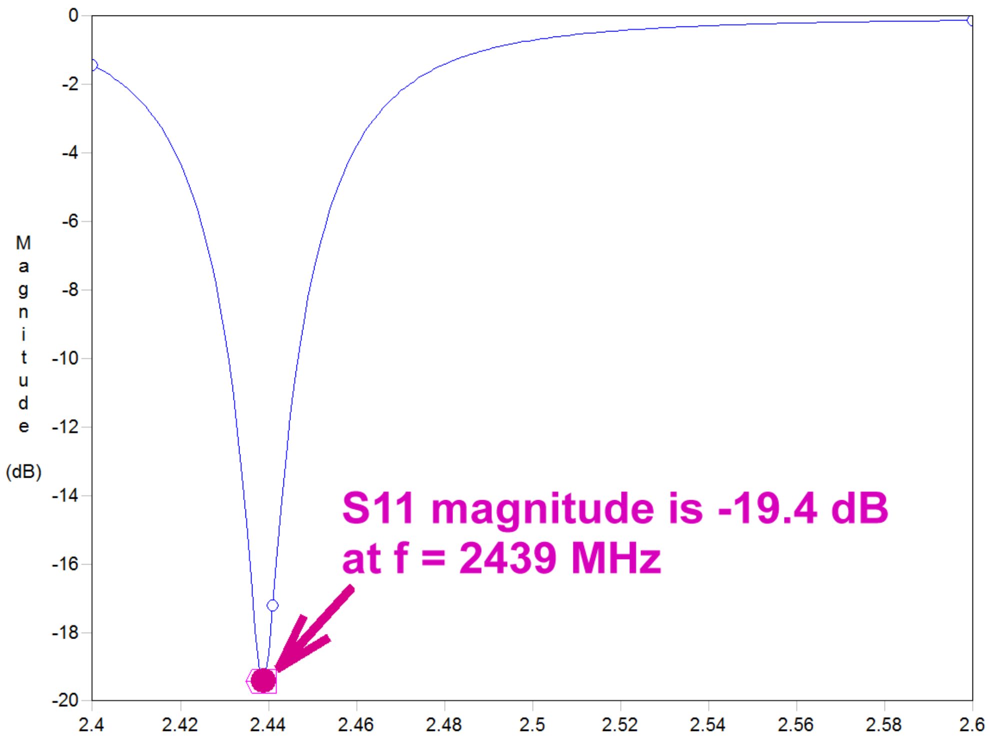
And here come the dimensions for the PCB design:

**Patch width =** **33.08mm**
**Patch length =** **32.12mm**
**Feedpoint =** **4,77mm at the left of the patch center = a distance of 20.83 mm from the right hand edge**




To check the design before manufacturing a prototype we use this SONNET LITE simulation.

The result is very fine:



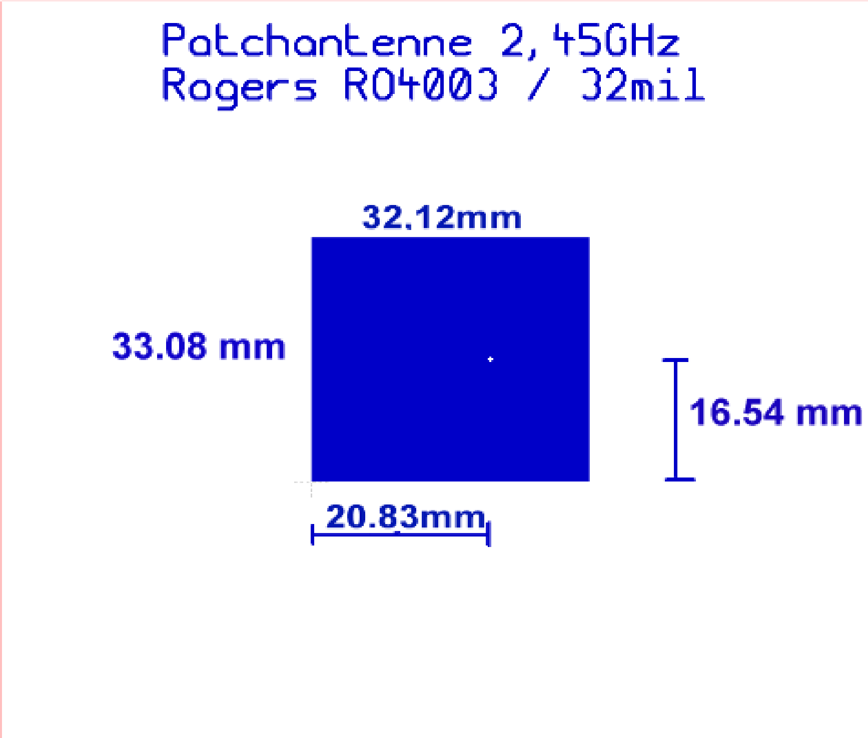

The via is an inductance and reduces the resonant frequency to 2439 MHz instead of 2450 MHz.

**The magnitude of S11 has nearly a value of -20dB.**

Very good....

But now we have reached the border of simulations and the prototype time starts...

On the next page you can see the PCB printout (sent as Gerber Plot to the manufacturer).




**This layout shows the fabricated and tested PCB**

**The outer dimensions are**

**100mm x 100mm**

**And now the final run with the network analyzer:**

**Resonance frequency = 2443 MHz**
**S11 better than -30dB.**

Well done...

## 17. Modulation

### 17.1. Introduction

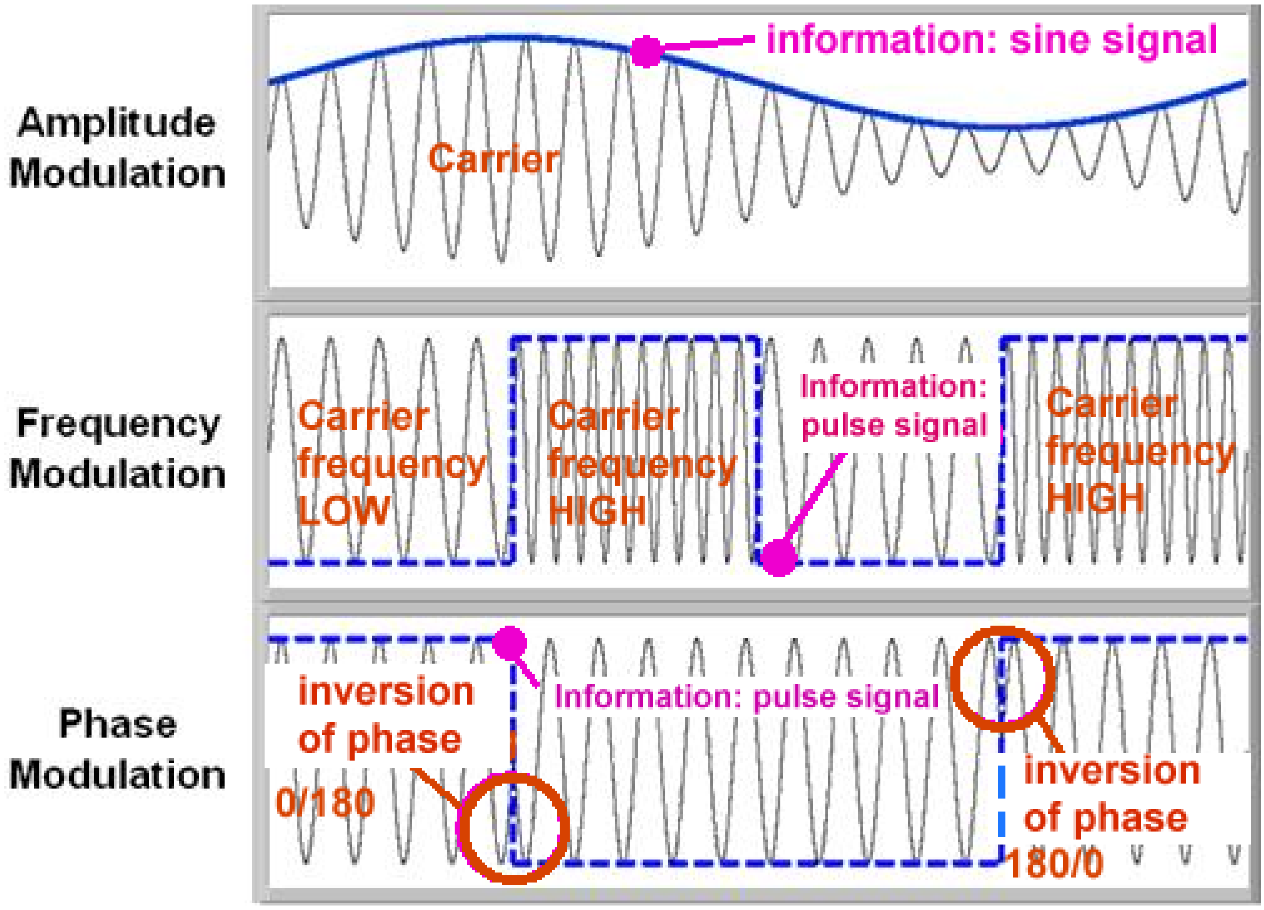
100 years ago wireless communication started with the Morse key, followed by the transmission of voice and music. But this problem had first to be solved:

For signals in the audio frequency range the wavelength is too high to radiate by an antenna with good efficiency, due to the law:

**Use an antenna length of a quarter wavelength for optimum efficiency. When the antenna length falls below 10% of λ, the radiation efficiency decrease rapidly and dramatically.**

Example:
A tone with f = 1 kHz has a wavelength of 300 km. So a good antenna should have a length between 30 km and 75 km...
**Thus another principle was realized:**
**Use a quick running donkey (= high frequency which can easily be radiated) and put the load (= your information) on his back. This process is called: "Modulation".**

But three different principles of modulation came more and more in use. This illustration shows the fundamentals (source: application note of National Instruments):




**Amplitude Modulation (AM) means:** the **carrier amplitude is varied due to the information signal.**

**Frequency Modulation (FM) means:** the **carrier frequency is varied due to information signal.**

**Phase Modulation (PM) means:** the **carrier phase is is varied due to information signal.**

But modern communication technique has produced a lot of sons and daughters of these simple methods. This shows a short overview found in Wikipedia:

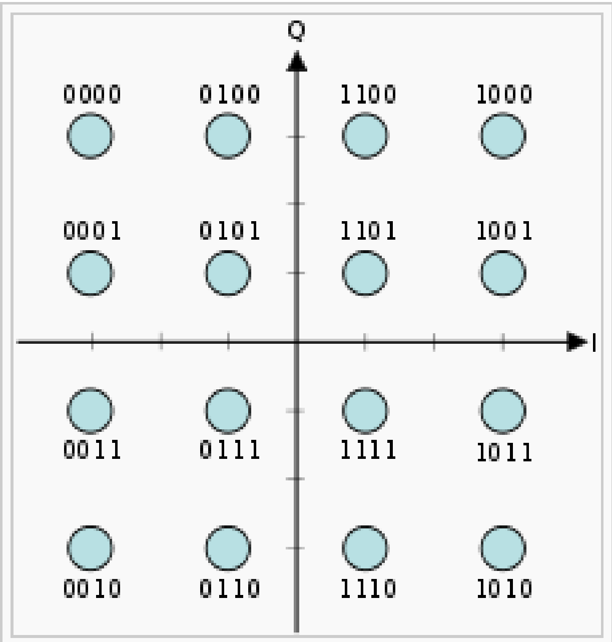
## See also [edit]

- Amplitude and phase-shift keying or Asymmetric phase-shift keying (APSK)
- Carrierless Amplitude Phase Modulation (CAP)
- In-phase and quadrature components
- Modulation for other examples of modulation techniques
- Phase-shift keying
- QAM tuner for HDTV
- Random modulation

## See also [edit]

- Amplitude-shift keying (ASK)
- Continuous-phase frequency-shift keying (CPFSK)
- Dual-tone multi-frequency (DTMF), another encoding technique representing data by pairs of audio frequencies
- Frequency-change signaling
- Multiple frequency-shift keying (MFSK)
- Orthogonal frequency division multiplexing (OFDM)
- Phase-shift keying (PSK)
- Federal Standard 1037C
- MIL-STD-188

# Very modern: QAM

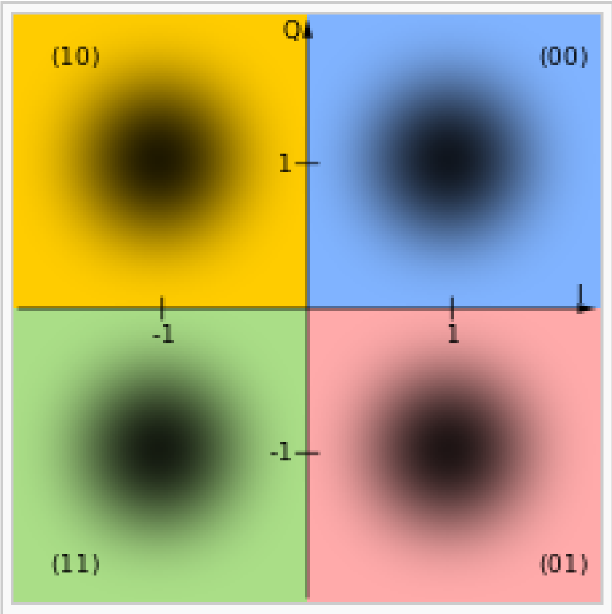



Today AM and FM are combined to "**QAM**". Hereby two orthogonal axes form a cross. The horizontal axis is named **"I" (for "in phase"),** the vertical axis is the **"Q" axis (for "quadrature phase")**. Now a point in this diagram is defined by the pointer length and the angle between pointer an "I" axis and this point represents a bit combination.

In this illustration you can find a 16 QAM with gray code.

**16 different bit combinations (= symbols) can be transmitted.**

**Every symbol differs from its neighbors by only one bit.**




**constellation diagram of a 4-QAM transmitting 2 bits.**

**In every quadrant an "allowed landing zone" must be defined in which the transmitted symbol can be recognized**

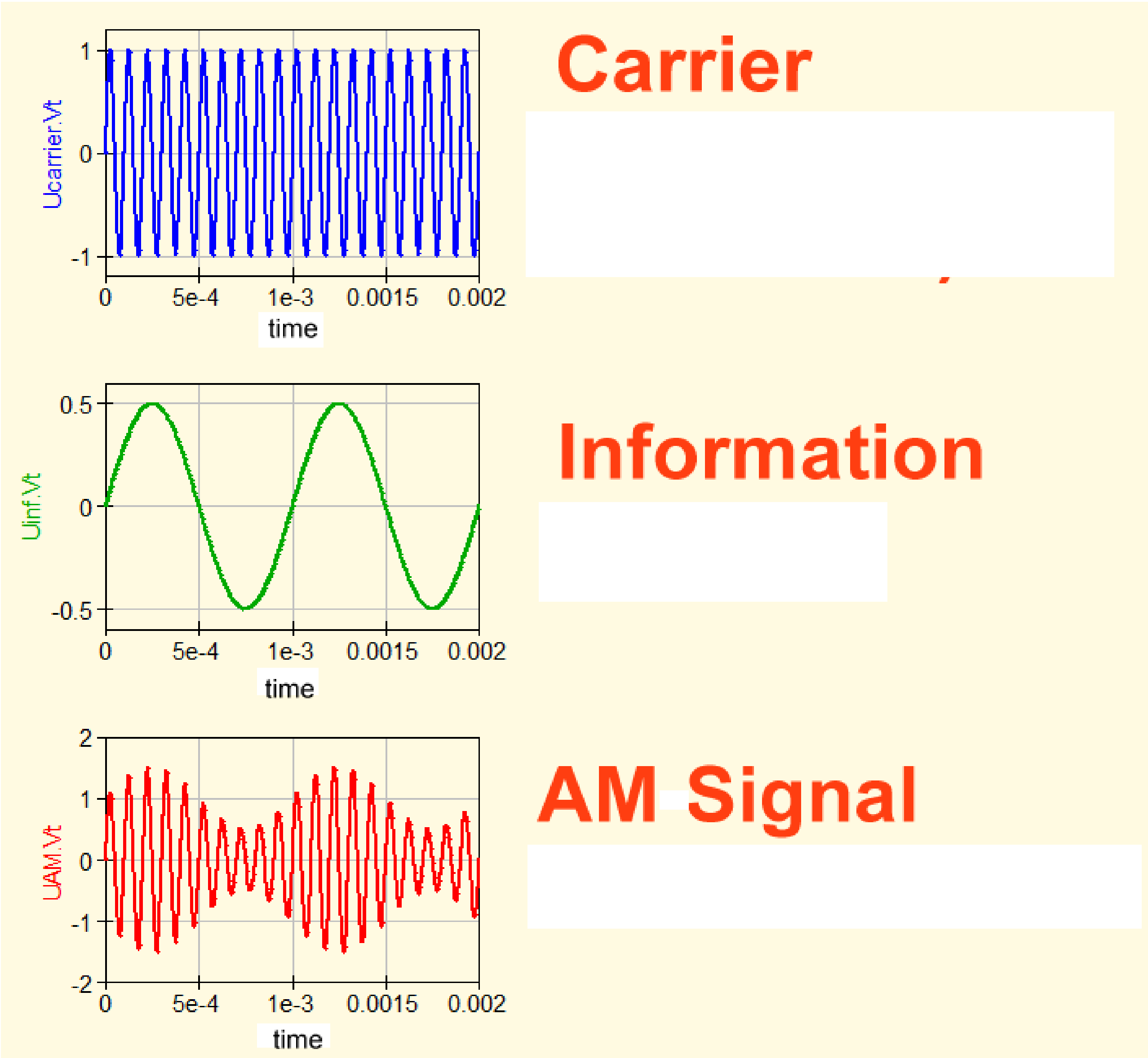
This illustration shows the definition of a 4 QAM transmitting four different symbols - each with 2 bits.

## 17.2. Amplitude Modulation

### 17.2.1. Background

**Principle:**
**The carrier amplitude is varied (= "modulated") by the information.**

Example:
Let us regard a sine wave carrier with f = 10 kHz which is modulated by an information signal (= sine wave with 1 kHz).



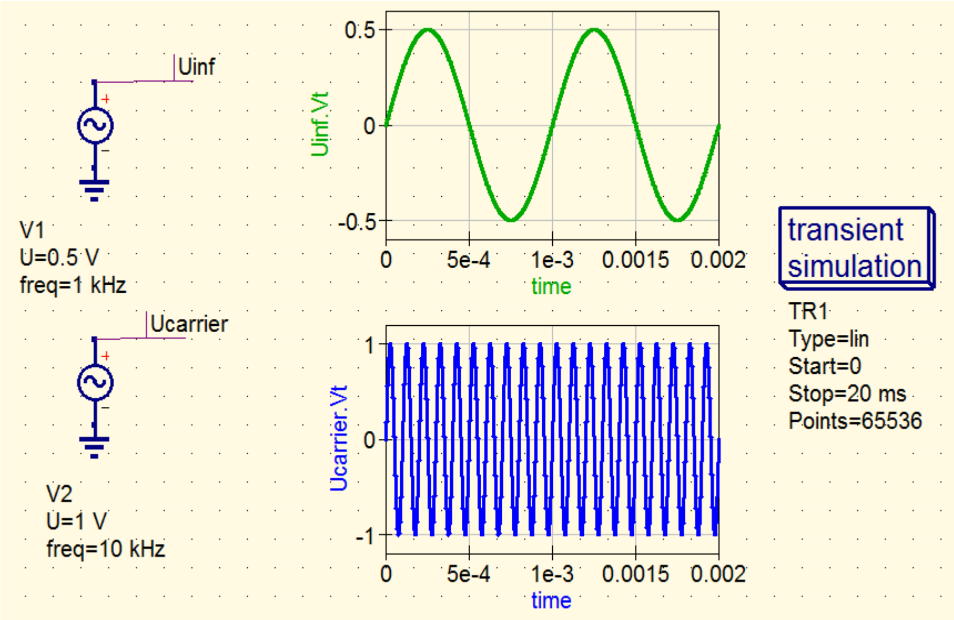

**If you analyze this process using mathematical laws, the you will find**

**a) a multiplication of two different signals, followed by**

**b) an addition.**

This shows a simulation.

We use two AC sources (they come form “components / sources”) for carrier and information and set the properties:



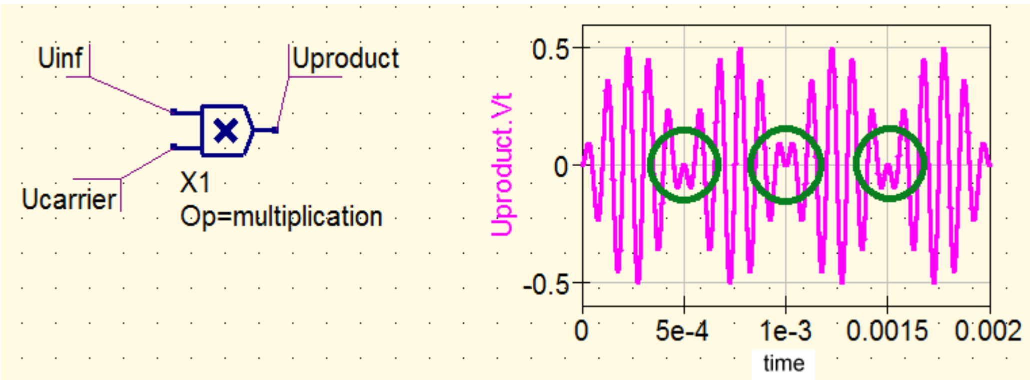

This is the information (= sine wave with 1 kHz and 0.5 V)....

….followed by the carrier with f = 10 kHz and 1V.

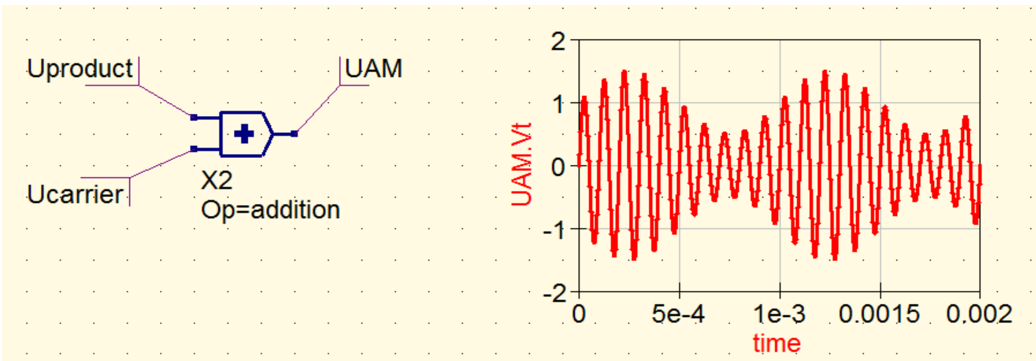
Now we have to **multiply** these two signals using a part named “**operation**” (which comes form “Components / system components” and is set to “multiplication”)
The result is the “**modulation product”** which has the following properties:

a) The **product amplitude follows the information**, but

b) at **every negative half of the information sine wave** the output phase must be **inverted.**




At first the **multiplication...**




...then an **addition of the modulation product to the unmodulated carrier signal,** done by an additional “operation” part – but now set to “**addition**”.

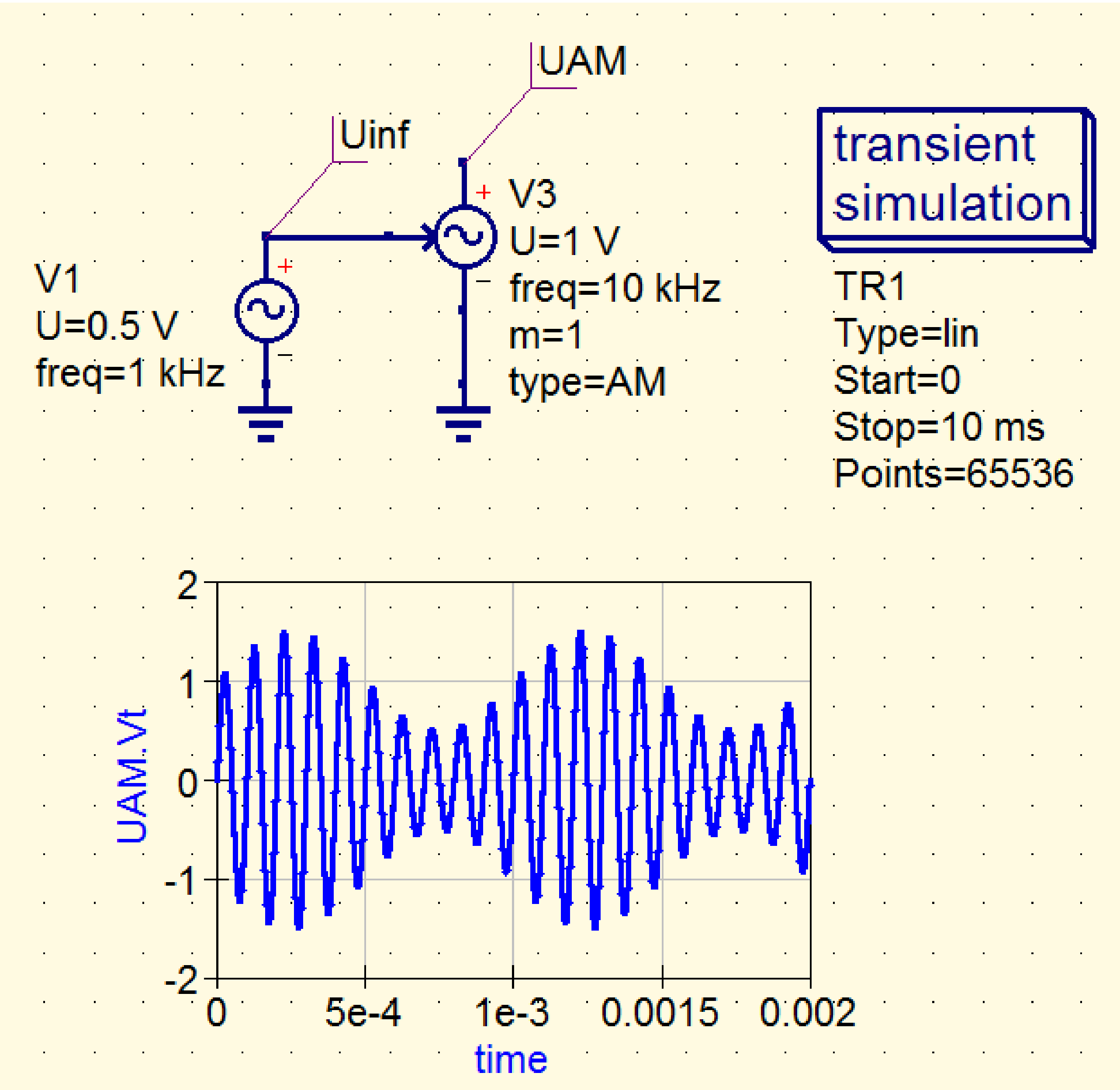
**qucsstudio offers for modulation purposes an own “modulated source” in “components / sources”, which must be set to “AM”.**

So please use this schematic, add a **voltage source for the information signal (1 kHz / 0.5 V), set the carrier to 10 kHz / 1 V and enter a modulation index m = 1.**

(Remark:
The **modulation index “m”** can be used to vary the “intensity of modulation” – it is the gain for the information signal before modulating).
**Thus a value of m =1 in our example gives an AM modulation degree of**

**Uinf / Ucarrier = 0.5 V / 1 V = 0.5 = 50 %.**

Increasing “m” to 2 will produce a 100 % modulation for the given voltage values)



### 17.2.2. AM Spectrum

An AM signal is not a “sine wave with constant amplitude and frequency from eternity to eternity”!
So let us analyze the complete frequency content and start with the modulation product.

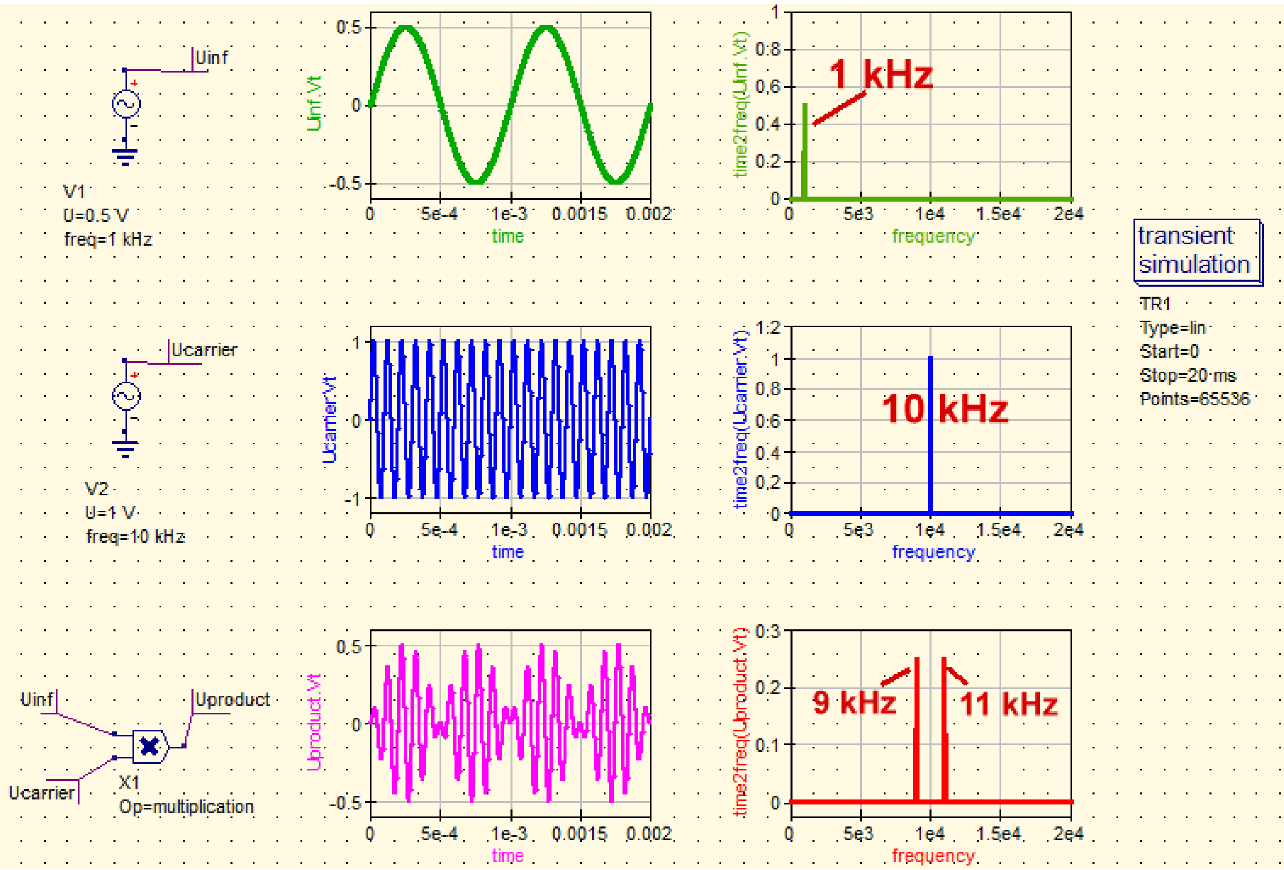
We got this product as result of the multiplication of two different sine waves. Therefore mathematic says:

$$\cos(\alpha)\bullet\cos(\beta)=\frac{1}{2}\left[\cos(\alpha+\beta)+\cos(\alpha-\beta)\right]$$

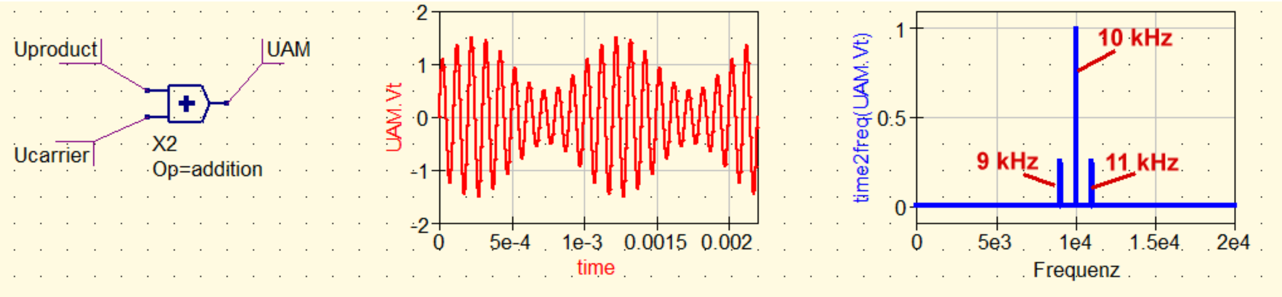
***The frequency content of the product result is the sum and the difference frequency of the two input signals!***

**Carrier and Information cannot longer be found at the output.**

This can be seen when simulating this multiplication in the Time Domain and in the Frequency Domain:



**If the original carrier (= not modulated) is added then you get a spectrum with 3 lines**":




If you now have a look at the amplitude of every line:

a) the **carrier has a peak value of 1 V**

b) every "**side band frequency**" must have an **amplitude of half the information voltage peak value** = 0.25 V = 0.5 V / 2 (...so says the formula and the simulation...)

**Remark:**
The information signal consists usually of a lot of different spectral lines (when transmitting speech or music). Thus we speak of a **lower side band (LSB)** and an **upper side band (USB)** beside the carrier in the spectrum.

**Attention:**
**This is identical to a frequency conversion of the information signal with the carrier frequency as "new zero point". Every side band has the half information voltage amplitude and contains the same information, but the frequency scaling of the lower side band (LSB) is mirrored and runs inverse...**

...and the carrier itself does not contain any information and could be reduced or suppressed....

## 17.2.3. AM Demodulation

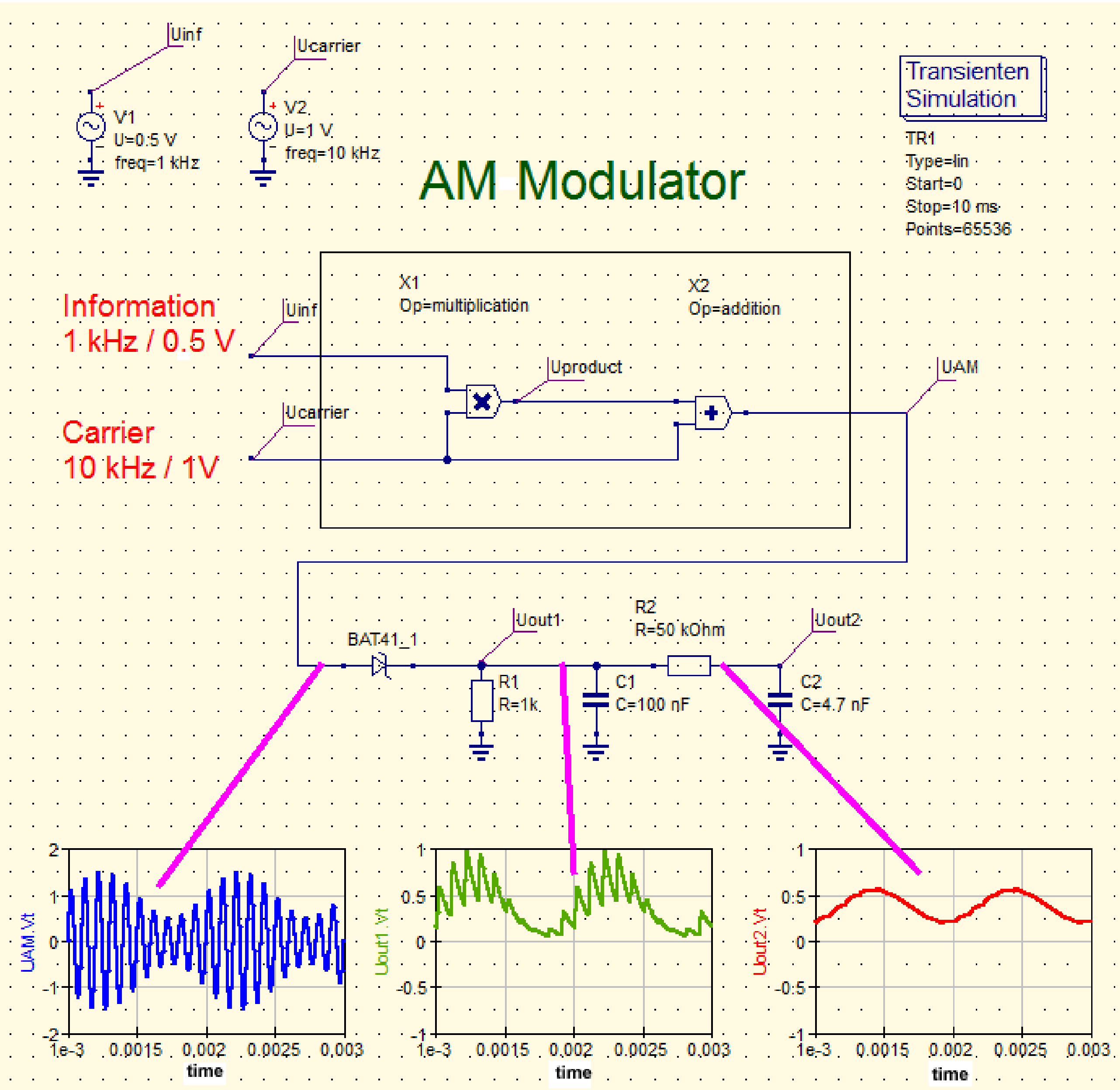
## 17.2.3.1. The classical AM Diode Demodulator

We use the detailed schematic to generate an AM signal and the same voltages as in the last chapter (information = sine wave with 1 kHz / 0.5 V, carrier = sine wave with 10 kHz / 1 V).

The generated AM signal feeds an one pulse rectifier using a Schottky diode BAT41, a load resistor with R = 1 kΩ and a capacitor with 100 nF in parallel connection.

At the load resistor you can already recognize the information, but a rest of the carrier is still worrying. Thus a low pass filter (R = 50 kΩ / C = 4,7 nF) follows to suppress this carrier rest.

Simulate from 0....10 ms with 65536 points. (...the Schottky is found as BAT41 in “**library / Schottky Diodes**“).



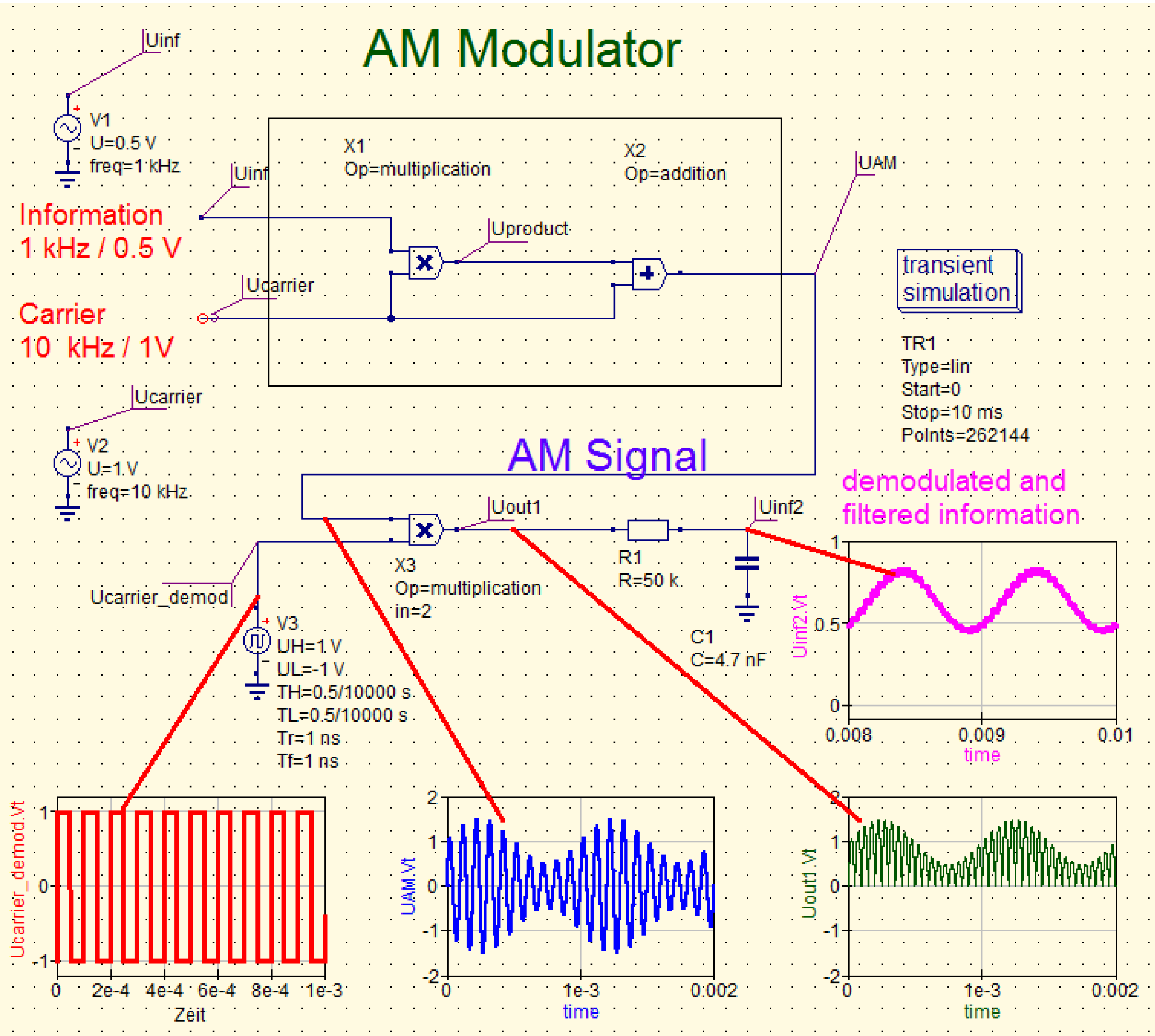
## 17.2.3.2. AM Demodulation using a Product Detector

This is state of the art, but needs more efforts than the diode demodulator.

**Import:**
**This demodulator type is able to demodulate AM signals with an amplitude of few micro volts without any distortion!**

**The principle is very simple: multiply the AM Signal by a square wave with exact the same frequency and phase – but with the amplitude values "+1" and "-1".**

Thus the circuit will work like a "**two pulse rectifier**". The square wave will invert the polarity of the AM signal during it's "-1" phase. A LPF must follow to suppress the carrier frequency and the harmonics and so to filter out the desired information.



## 17.3. SSB = Single Side Band

The “Double Side Band AM Signal” consists of 3 parts:

***the Lower Side Band LSB***

***the Upper Side Band USB***
**(Both side bands contain the same information).**

**the unmodulated carrier which does not contain any information. So this part could be reduced or totally suppressed to save transmitter energy.**

If the carrier is suppressed then we still have two identical side bands (...the LSB is a “mirror” of the USB) and an additional suppression of one side band (= **SSB = Single Side Band** operation). This offers a lot of advantages:

**You need only half the bandwidth for the transmission.**

**The complete transmitter power is now pushed into the remaining side band. This gives a greater maximum distance between transmitter and receiver and / or a better signal to noise ratio.**

If you have a break in your information (e .g. when somebody is speaking) no transmitter energy is needed for this break time. This saves energy and increases the efficiency.

**Disadvantage:**
a) Higher technical efforts for an SSB transmitter and an SSB receiver. Demodulation is only possible with a product detector.

**b) You must exactly know whether you receive an LSB or an USB – due to the inverted frequency scale of the LSB. And for a correct demodulation you must know the exact frequency of the suppressed carrier.**

Generating SSB signals is an own science. Three different methods are possible and in use:

a) the **filter method** which uses crystal filters to filter out the desired side band.

b) the **phasing method** which compensates the undesired side band by addition of this side band with inverted phase.

c) State of the art and winner of the game is today the usage of **IQ signals and a half complex mixer.**

Let us have a look at this modern technique.

## 17.4. Real, Imaginary and Complex Signals

Every electrical signal which you can see (on an oscilloscope) or hear (coming out from a loudspeaker) is a **Real Signal.**

Signals which you only can **guess due to the observed effects** are **Imaginary Signals.**

A mixture of these two types is called "**Complex Signal**".

We use a diagram with orthogonal axis and a **phasor** to demonstrate this. The phasor is a pointer, fixed at the center of the diagram and with the properties "magnitude" and "phase". He rotates with its given frequency, but

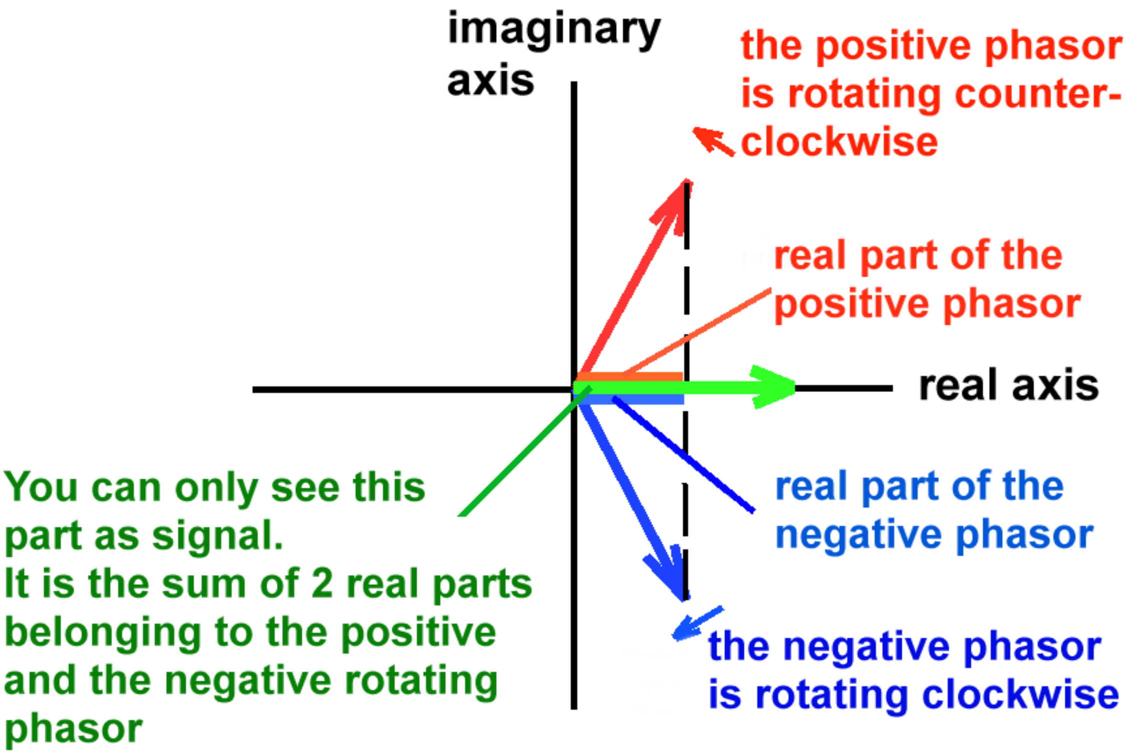
**If the phasor rotates counterclockwise then we say that its frequency is positive.**

**If the phasor rotates clockwise then its frequency is negative.**

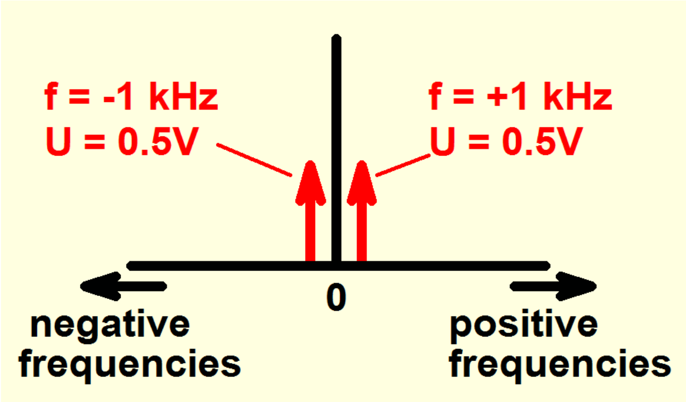
The **horizontal diagram axis is now the Real World**, the **vertical axis the Imaginary world.** Thus every voltage measured in a circuit and shown on an oscilloscope screen is the "**Real part of a complex signal**" and **can only be found on the horizontal axis of the diagram.**

**But here comes an important law:**
**"True" real signals consist of two parts = two phasors which rotate in opposite directions**. Each phasor has an amplitude value of 50% of the regarded real signal.

This illustration shall show that – the green pointer is the regarded real signal:



If you can't believe that: I'll prove the truth for you in the frequency domain.




a) At first we regard a **real information signal** (= 1 kHz tone with a peak value of 1 V) coming from a signal generator.

It consists of a **positive rotating phasor with f = +1 kHz / 0.5 V** and a **negative rotating phasor with f = -1 kHz / 0.5 V**

The mathematical description is – Thanks to Mister Euler! – not very complicated:

The positive phasor can be written as

$$U_{+1kHz} = U_{+1kHz\_max} * e^{j2\pi`*1kHz*t} = (0.5V)* [\cos(2\pi*1kHz*t) + j \sin(2\pi*1kHz*t)]$$

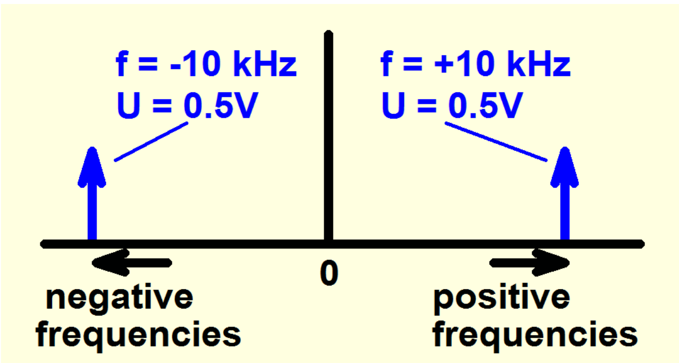
And this is the mathematical description of the negative phasor:

$$U_{-1kHz} = U_{-1kHz\_max} * e^{-j2\pi*1kHzt} = (0.5V)* [\cos(2\pi*1kHz*t) - j \sin(2\pi*1kHz*t)]$$

If you now add the two phasors then the imaginary parts will cancel and the rest is the real information signal with a peak value of

$$U_{1kHz\_max} = U_{+1kHz\_max} + U_{-1kHz\_max} = 2 * 0.5V = 1V$$

b) this real information signal will be multiplied by a **real carrier signal with f = 10 kHz and a peak value of 1V.**




This carrier signal consists of a **positive rotating phasor with f = +10 kHz / 0.5 V** and a **negative rotating phasor with f = -10 kHz / 0.5 V**

We use again Mister Euler and thus we write:

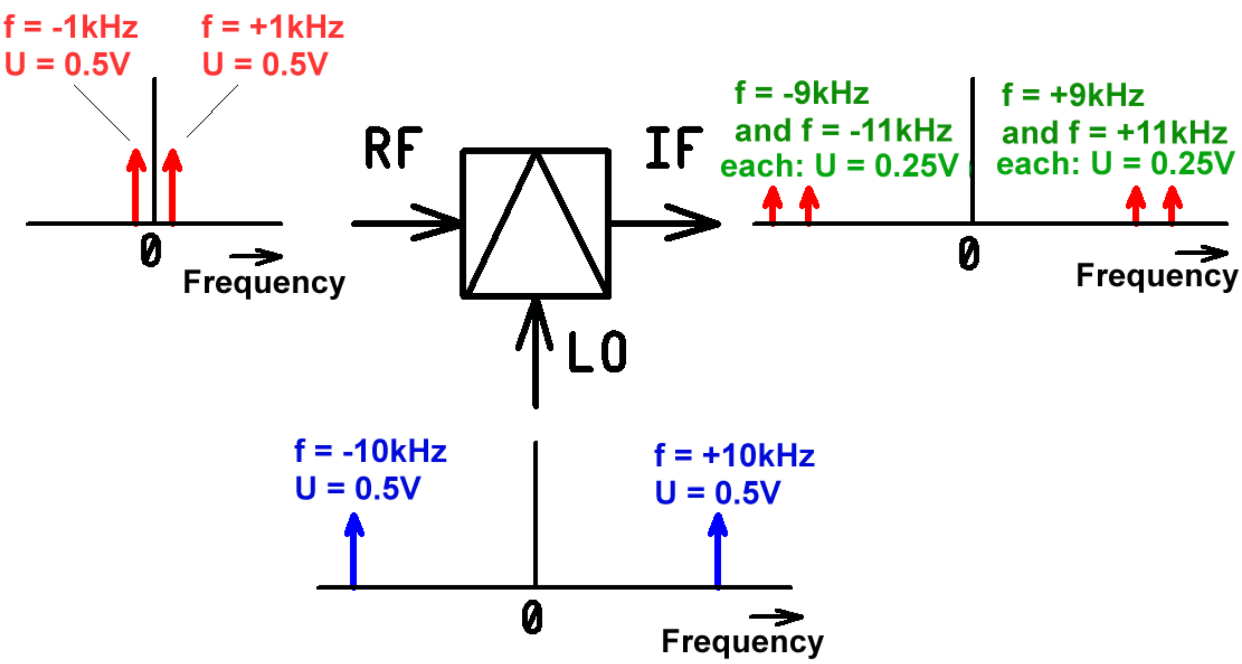
$$U_{+10kHz} = U_{+10kHz\_max} * e^{j2\pi`*10kHz*t} = (0.5V)* [\cos(2\pi*10kHz*t) + j \sin(2\pi*10kHz*t)]$$

$$U_{-10kHz} = U_{-10kHz\_max} * e^{-j2\pi*10kHz*t} = (0.5V)* [\cos(2\pi*10kHz*t) - j \sin(2\pi*10kHz*t)]$$

If you now add again the two phasors then the imaginary parts will cancel again and the peak value of the real carrier signal is

$$U_{10kHz\_max} = U_{+10kHz\_max} + U_{-10kHz\_max} = 2 * 0.5V = 1V$$

Now multiply all these Cosine signals and you get this result:




If you do not believe that: use again Mister Euler and you get:

$$(\cos\omega_1 t)\cdot(\cos\omega_2 t) = \left[\frac{\left(e^{j\omega_1 t}+e^{-j\omega_1 t}\right)}{2}\right]\cdot\left[\frac{e^{j\omega_2 t}+e^{-j\omega_2 t}}{2}\right] =$$

$$= \frac{\left[e^{j(\omega_1+\omega_2)t}+e^{-j(\omega_1+\omega_2)t}\right]+\left[e^{j(\omega_1-\omega_2)t}+e^{-j(\omega_1-\omega_2)t}\right]}{4} = \frac{1}{2}\cdot[\cos(\omega_1+\omega_2)t+\cos(\omega_1-\omega_2)t]$$

As result of the multiplication you get two real signals:

a) a real cosine wave with the sum frequency $f = f_{carrier} + f_{inf}$ and a value of 0.5 V

b) a real cosine wave with the difference frequency $f = f_{carrier} - f_{inf}$ and a value of 0.5 V

And if you have a sharp look at the output frequency plane in the above diagram, then you can see that the **information signal has been shifted in frequency!** But the new "zero Hertz point" lies now on $f = f_{carrier}$ and the fact of **negative frequencies is confirmed.**..

## 17.5. Analytic Pairs

Sounds like a secret but is not complicated!
Normally you don't transmit a single frequency - speech and music are a huge collection of different signals and their harmonics which continually vary frequency and amplitude.

If you want to halve the occupied bandwidth of the transmitted signal – simply suppress the positive or negative frequency part (= transmit only the LSB or the USB). This can be done by changing from a real signal to a complex signal and this is the necessary procedure:

**The real start signal is named „I“ signal (= In Phase Signal) and must not be touched or changed.**

**But this “I” signal feeds also a circuit named “Hilbert Transformer“**. At the output all spectral lines have the same amplitude as before, but the phase of every spectral line is shifted by 90 degrees!

**This artifical signal is named “Q signal” (= Quadrature Signal).**

If working with DSP then you have now two data streams “I” and “Q” which form a complex signal with either only positive frequencies (for a “Q” phase shift of +90 degrees) or only negative frequencies (for a “Q” phase shift of -90 degrees) and thus only half the bandwidth. This is the secret of the Analytic Pair....it is the same animal as an SSB signal....

## 17.6. Example: Half Complex Mixer to generate an SSB Signal

Therefore you need the following ingredients:

a) A **real** information signal **”$I_{inf}$“**, (speech or music), which must be **converted to a complex signal (= Analytic Pair)** by the aid of a **Hilbert Transformer**. The Analytic Pair can be described as:

$$U_{inf} = I_{inf} + j * Q_{inf}$$

b) a **complex** carrier $U_{carrier} = U_{carrier\_max} * e^{+j\omega_c t}$ **given as a sine and a cosine signal** and thus with only one positive frequency.

c) Two **Multipliers**

d) One **Subtracting Circuit**

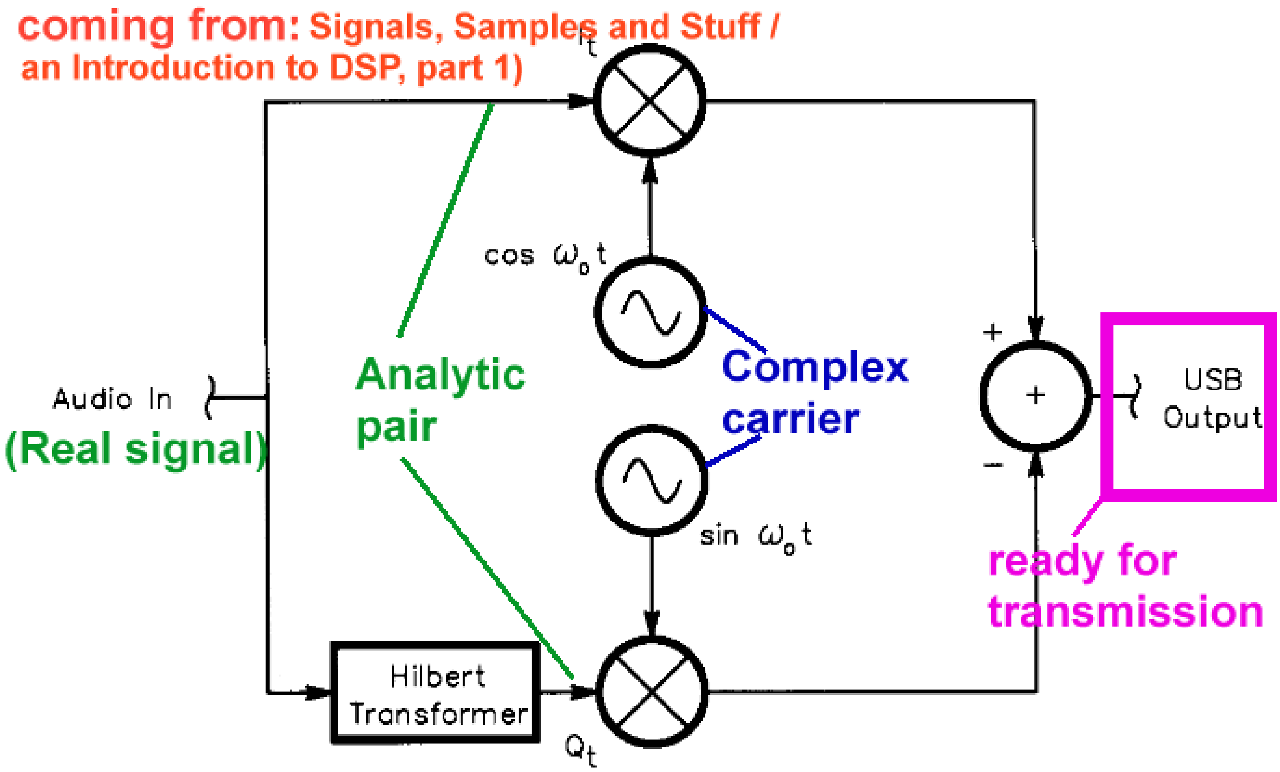
Now the information as Analytic Pair is multiplied by the complex carrier (with only one positive frequency):

$$U_{inf} * U_{carrier} = (I_{inf} + j * Q_{inf}) * [\cos(\omega_c t) + j * \sin(\omega c t)] =$$

$$[I_{inf} * \cos(\omega_c t) - Q_{inf} * \sin(\omega c t)] + j * [Qinf * \cos(\omega_c t) + I_{inf} * \sin(\omega c t)]$$

**The real part is marked in pink colour. This part must be realized by a practical circuit and can afterwards be transmitted (= converted to an electromagnetic wave by the antenna)**

(Remark:
A Hilbert transformer with a phase shift of 90 degrees is no problem for a Digital Signal Processor. But an analogue solution can only be realized in a small and limited frequency range...Sorry....)




This circuit produces an "**Upper Side Band = USB".**

**If you need the "Lower Side Band = LSB" then replace the subtracting circuit at the output by an adding circuit and change the lower sign from "-" to "+".**

## 17.7. Demodulation of Amplitude Modulated IQ Signals

This is a very easy task in the digital world:

Both parts "I" and "Q" of an **Analytic Pair have always a phase difference of 90 degrees.** Thus you can use Mister Pythagoras:

**(length of pointer)$^2$ = I$^2$ + Q$^2$**

The rest is simple for a Digital Signal Processor: square first "I", then "Q". Add the two squares and calculate the square root to get the pointer's amplitude = information signal.

## 17.8. Demodulation of SSB Signals

If you choose the analogue way then you need a signal with exact the same frequency and phase of the suppressed carrier during the transmission. Then a product detector as used in chapter 16.2.3.2. is necessary which is multiplied by the received SSB signal. The difference frequency at the output is the desired information.

**Using DSP and IQ signals and a Hilbert Transformer is more expensive and needs more efforts.**
Many information can be found in the Internet and the following article is a nice example:

### A DSP SSB DEMODULATOR

Figure 13 shows an example of a DSP SSB phasing method demodulator. Once the complex-valued BFO of $e^{-j(\omega_c n t_s)} = \cos(\omega_c n t_s) - j\sin(\omega_c n t_s)$ down-converts the RF SSB to zero Hz, it's sensible to decimate the multipliers' outputs to a lower fs sample rate to reduce the processing workload of the Hilbert transformer. We could have performed decimation by a factor greater than 10, but doing so would make the design of the post-D/A analog lowpass filter more complicated. The digital LPFs, whose positive-frequency cutoff frequency is slightly greater than 4 kHz, attenuate any unwanted out-of-baseband spectral energy in the down-converted signal and eliminate any spectral aliasing caused by decimation.

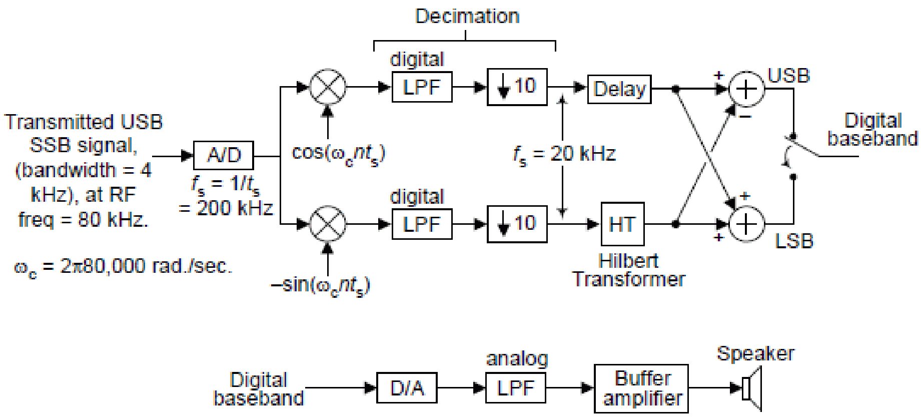



Figure 13

The Delay element in the upper path in Figure 13 is needed to maintain data synchronization with the time-delayed Hilbert transformer output sequence in the bottom path. For example, if a 21-tap digital Hilbert transformer is used, then the upper path's Delay element would be a 10-stage delay line [2].

**search result:**

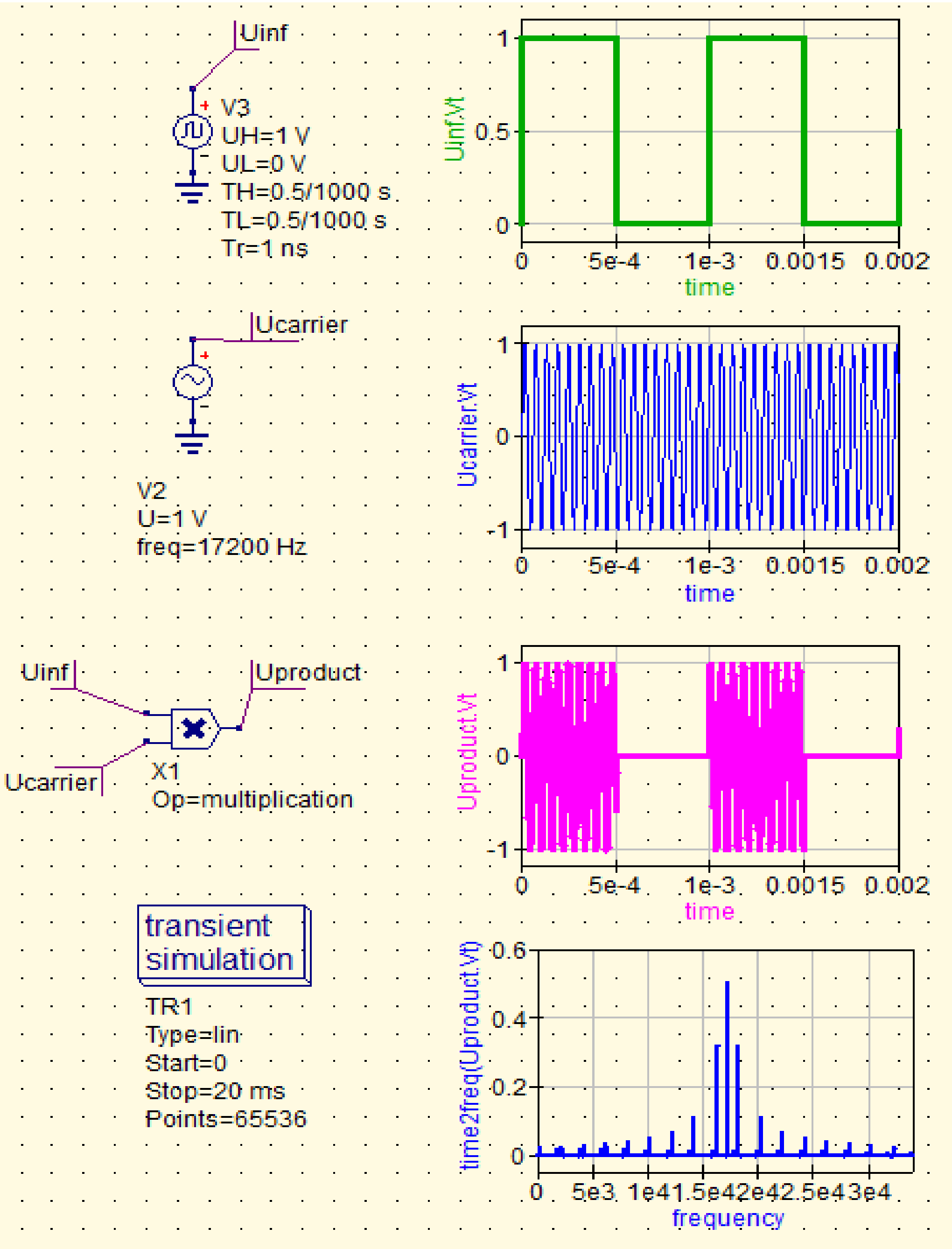
Understanding the 'Phasing Method' of Single Sideband Demodulation
**aus: http://www.dsprelated.com/showarticle/176.php**

## 17.9. Amplitude Shift Keying (ASK)

This is the good old Morse key switching “ON” and “OFF”. But this simple method is still today in use due to fact that we can assign two Bits (“0” and “1”) to this two-state and build up a digital communication. That runs very well and is less distorted by noise and intermodulation.

**To simulate such a Morse key we multiply a carrier by a rectangular pulse signal. Minimum value of the pulse is zero Volt (= key not closed), maximum value is 1 V (= key closed). The process is repeated with f = 1 kHz.**

The carrier frequency has a value of f = 17 200 Hz. This belongs to “SAQ” = a world heritage = a 200 kW transmitter from 1924 using an Alexanderson Generator). The carrier peak voltage is 1 V.




The frequency spectrum of the ASK signals shows the carrier at 17 200 and as LSB and USB the keying frequency of 1 kHz and its harmonics.

# 17.10. Phase Modulation

Another idea: the carrier amplitude is constant or not interesting, but the **carrier phase is modulated by the information signal**. Thus any amplitude variations or distortions can be eliminated by a limiter.

## 17.10.1. Making Phase Modulation

We have in stock (= components / sources) a **modulated source which can be switched between AM, FM and PM**. And the technical documentation delivers this information for PM:

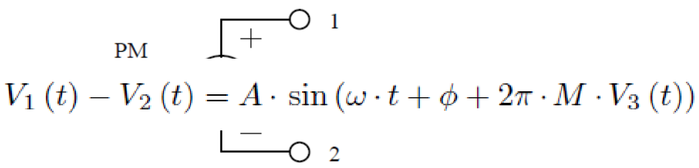


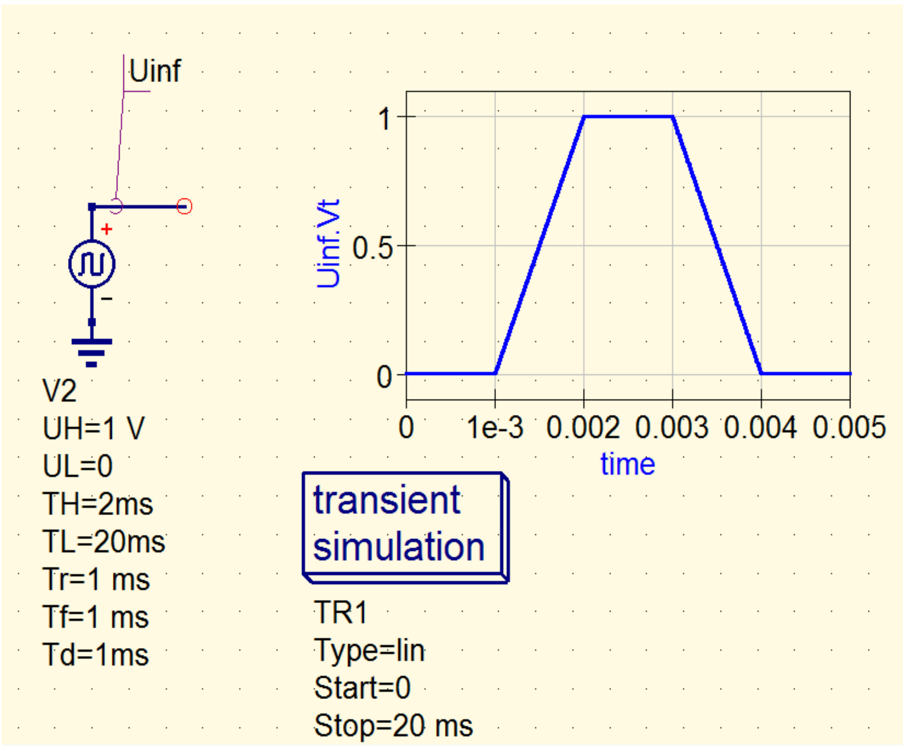

Figure 9.14: PM modulated AC source

We get a sine wave **[A*sin(ώ*t)] with a start phase „Φ“.**
The **Phase Modulation is done by the third expression in the brackets and is controlled by a voltage V3(t)**. Let's have a sharp look at this part:

**2π*M*V3(t) can be written as [2π*M] * V3(t)**

and you can see that the expression in the brackets is a “**phase shift in radians**”. **“M” is the Modulation Index** and for **M = 1 we get a phase shift of “2π” radians = 360 degrees.**

The bracket expression is now multiplied by **V3(t)** and thus the carrier phase will vary in the rhythm of the information signals amplitude and frequency!

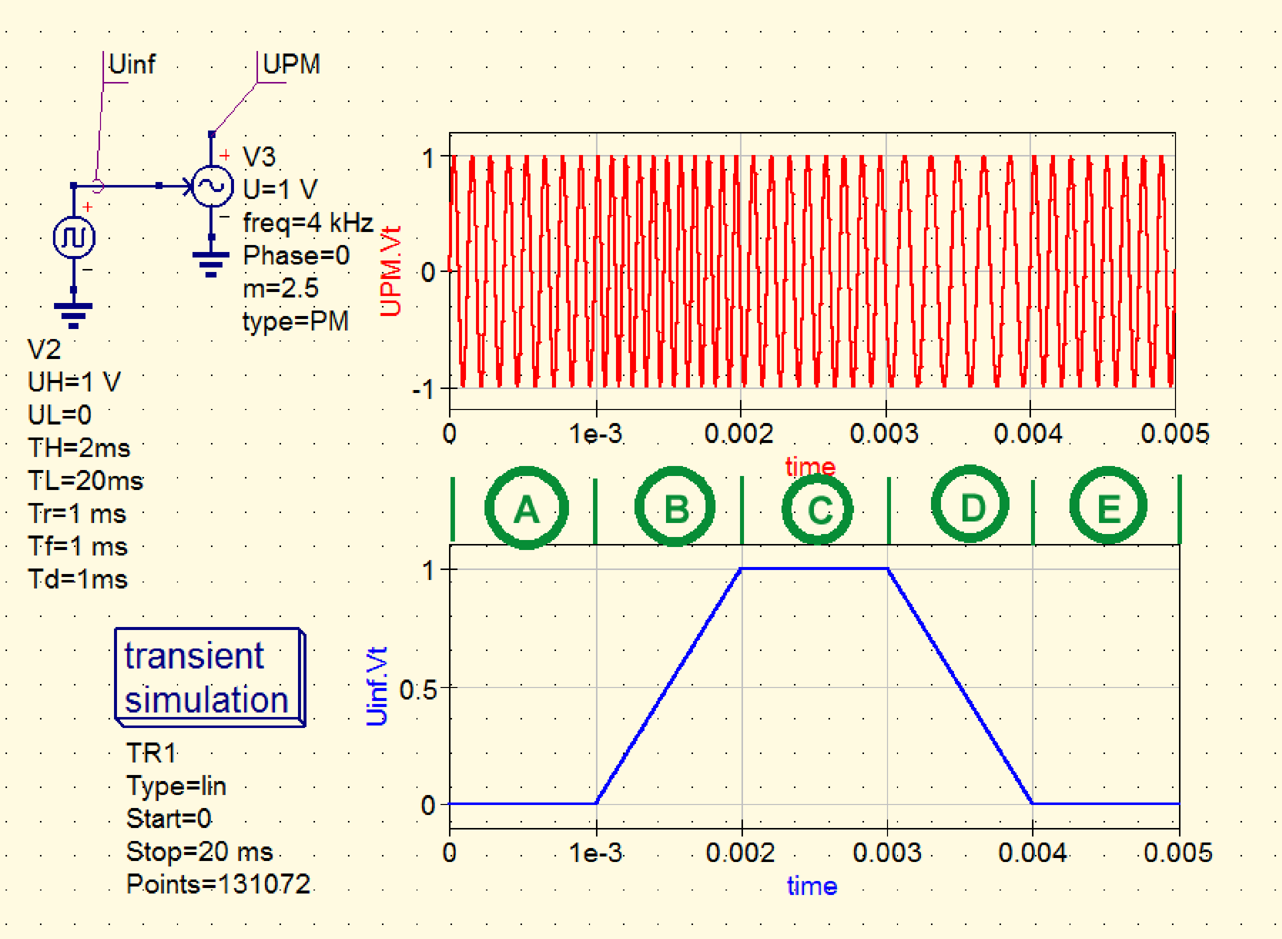



Let us begin with something simple.

The information is a single pulse. Starting after a delay of 1 millisecond the amplitude rises in 1 millisecond up to the peak value of 1 V = HIGH.
At t = 3 ms the amplitude decreases linearly to zero in a time of 1 millisecond. LOW time of the pulse is set to 20 ms.

Please draw this schematic and test it.

This circuit is now completed by the “**modulated source**” (found in “components / sources”) which is set to **PM, a carrier frequency of 4 kHz / 1 V, a start phase = 0 and a modulation index m = 2.5.**
**Let us analyze the 5 sections “A....E”.**




**Section „A":**
The information signal is zero. This gives a constant carrier sine wave with V3 = sin(ωt) and f = 4 kHz.

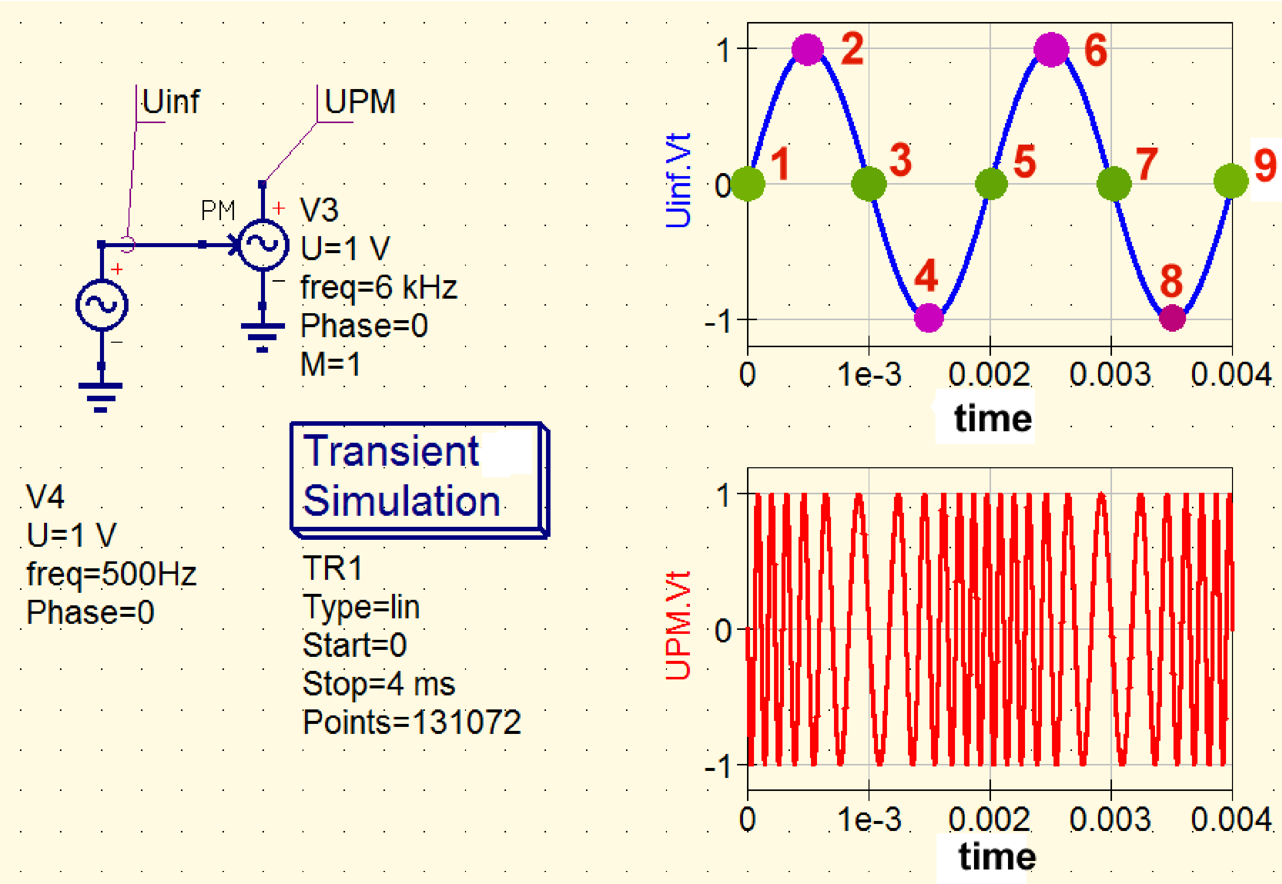
**Section „B":**
**The information signal rises linearly with time and forces an continuous advance of the carrier phase.** This is only possible with a **higher carrier frequency in this section.**

**Section „C":**
The information signal is **constant, thus the phase shift does not longer increase and must be constant**. Thus we observe a carrier signal with the **constant frequency** of 4 kHz (but the phase is shifted in comparison to the start phase).

**Section „D":**
The information signal is now **negative and the phase shift must decrease.** But this is only possible with a **lower carrier frequency in this section.**

**Section „E":**
The fall time is as long as the rise time of the pulse. Thus the total phase shift is reduced to the start value of section "A" and we observe again the constant carrier frequency of 4 kHz.

### 17.10.2. Result for a Sine Wave Information




**Section “1“ to “2“**
Due to the rising information voltage value the phase shift must increase and thus the carrier frequency is higher. But approaching to point “2” the information amplitude gets constant and he carrier frequency returns to the starting value.

**Section from “2“ over “3“ to “4“:**
The information voltage decreases and the phase shift follows (= lag in phase). The carrier frequency is lower and shows the lowest value at point “3” (= highest gradient of the information voltage).
At point “4” the information voltage is constant for a moment and this means a carrier voltage frequency of the start value.

**Section from “4“ over “5“ to”6“:**
The information voltage increases again and this causes again an advance in phase. Thus the carrier frequency must be higher. At point “6” the information voltage is constant, the phase shift advance has reached a maximum value and the carrier frequency shows its start value.

**Section from “6“ to “8“**
Have a look at section “2” to “4” – is is the same procedure. The information voltage decreases and thus the carrier frequency value is lower than the start value to reduce the phase shift.
At point 8 we find the start value of the carrier frequency but with a lag of phase.
And so on...

Examining the diagrams you find:
**The information voltage (which controls the phase shift) and the “carrier frequency of the moment” show a phase difference of 90 degrees!**

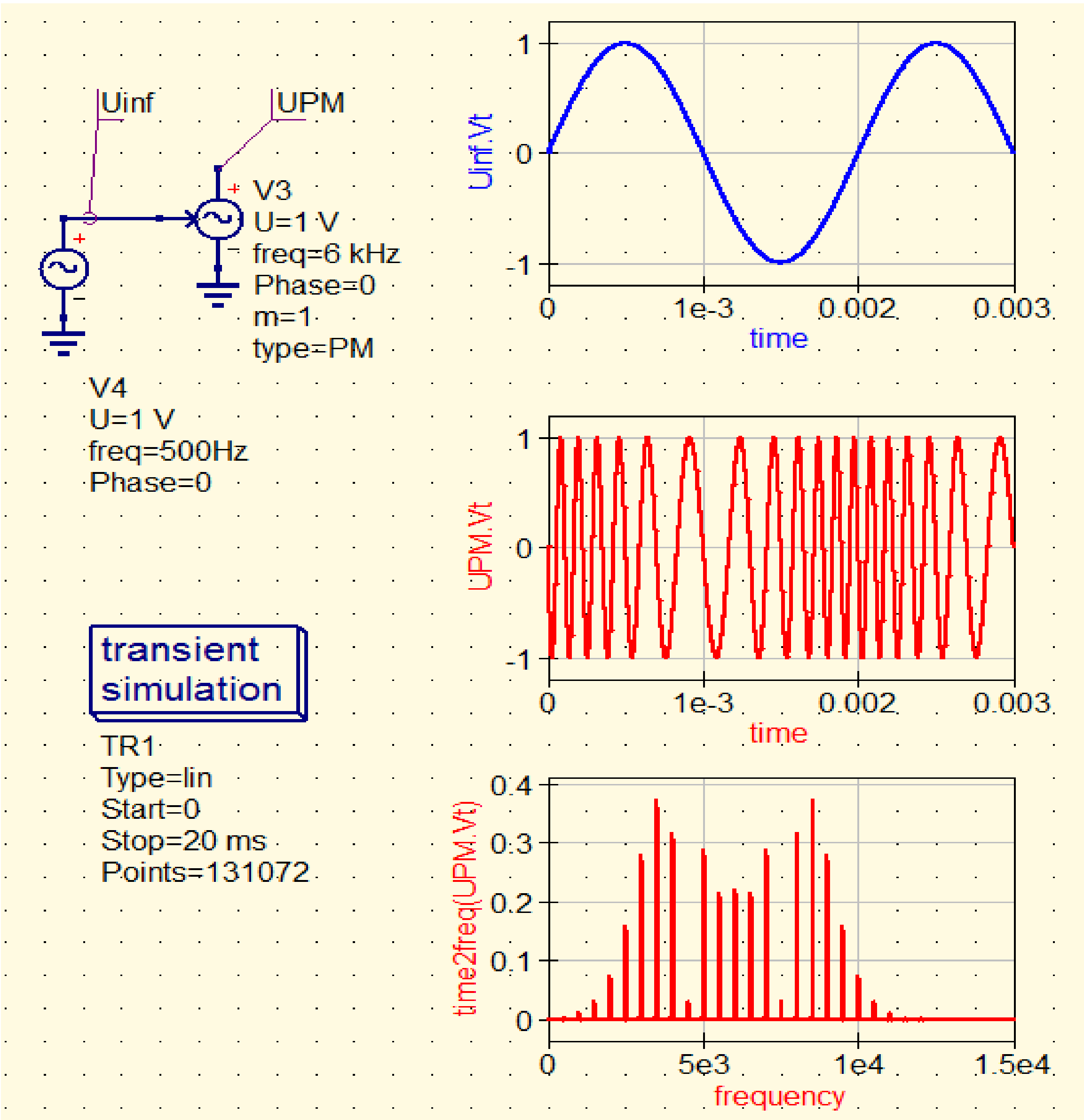
This is true because the mathematics say:
**The frequency is the derivative of the phase in the time domain.**

## 17.10.3. Phase Modulation demonstrated in the Frequency Domain

Let's have a look at the spectrum of a phase modulated signal. But to improve the resolution in the frequency domain please increase the simulation time to 20 ms
Thus the resolution will now be: **f = 1 / 20 ms = 50 Hz.**

**The information is a sine wave with f = 500 Hz / 1 V.**
**The unmodulated carrier has a frequency of f = 6 kHz and a peak value of 1 V.**
**The modulation index is m = 1.**

The simulated FFT spectrum is set to frequency range from **0....15000 Hz** and is calculated by the property
**time2freq(UPM.Vt)** of the diagram.

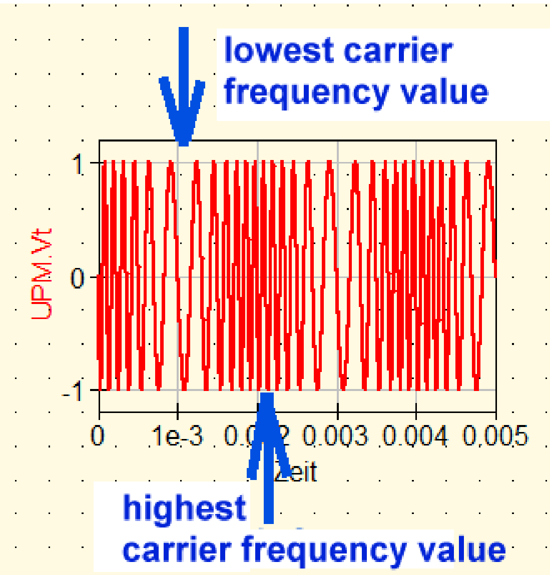



**The frequency spectrum consists of a lot of lines with a distance of the information frequency (500 Hz) to each other.**
The amplitudes of the lines vary due to a **Bessel function**.
The lines are centered around the unmodulated carrier frequency of 6 kHz.

**The number of lines (and thus the occupied bandwidth) increases with increasing modulation index “m”. But the amplitudes will at once vary due to the new Bessel function values for this new modulation index “m”.**

## 17.10.4. Some Information to Phase and Frequency Modulation

**Phase and Frequency Modulation cannot be distinguished if you only use one sine wave as information!**




But normally we have a collection of different information frequencies and thus we should know some facts and connections:d

**When FM - modulating by a sine wave we find a "minimum carrier frequency value" and a "maximum carrier frequency.**

Half the difference of these two frequencies is called

**„Frequency deviation = ΔF"**

Actual values for the frequency deviation are

**ΔF = +-75 kHz** for FM Radio Stations and

**+-2 kHz** for FM Walkie Talkies.

An FM Radio station transmits audio frequencies between 20 Hz and 15 kHz. But if you use a constant frequency deviation for the complete audio range you find an ugly effect:

**Due to the mathematical laws you will find that for constant frequency deviation the phase deviation will decrease with increasing audio frequency:**

**Phase Deviation =**
**(Frequency Deviation) / (Information Frequency)**

**But reducing the phase deviation increases the sensitivity to noise etc. and reduces the quality af the transmission! Thus the signal to noise ratio will decrease with increasing frequency.**

This must be compensated with a "**Pre Emphase**" in the FM transmitter: high audio signal frequencies will cause a higher frequency deviation to compensate this effect (= high pass filter used).
The FM receiver must do a correction by a De Emphase circuit (= a low pass filter).

And now the answer to the famous question:
**Which minimum bandwidth is necessary for a good FM transmission?**

This thumb rule will do the job:

**FM Channel Bandwidth = 2 * (Frequency Deviation + maximum Information frequency)**

Example for an FM Stereo Radio Transmitter using a maximum frequency deviation of 75 kHz and a maximum information frequency of 57 kHz (= RDS included):

**b = 2 * (75 kHz + 57 kHz) = 264 kHz.**

Please remember:
**Frequency modulation** (FM) works with **constant frequency deviation**. Thus the **phase deviation will decrease with increasing information frequency.** This is not good for the signal to noise ratio...

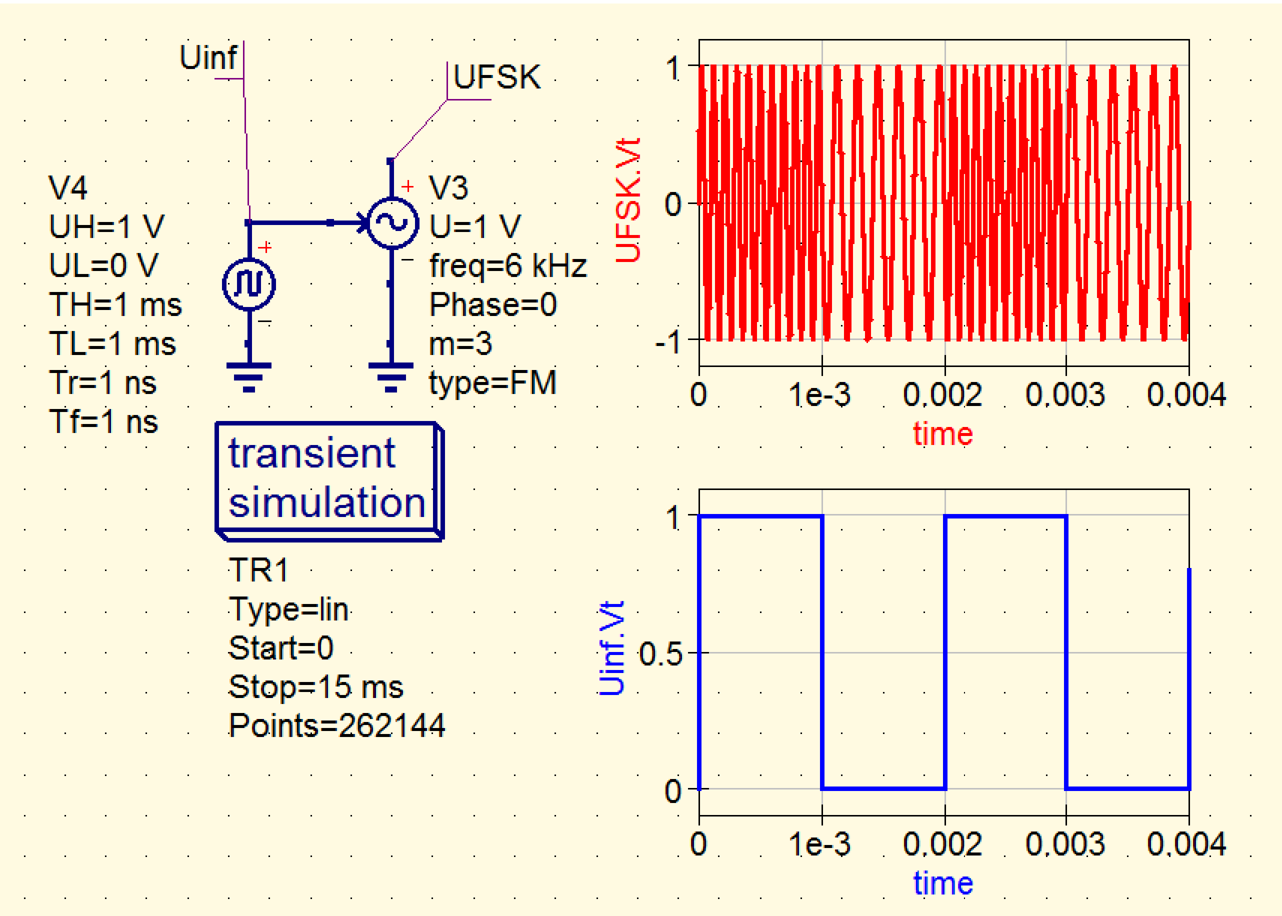
**Phase modulation** (PM) works with **constant phase deviation.**
Thus the frequency deviation will increase with increasing information frequency and thus you have always the same quality and signal to noise ratio. But the **occupied transmitter bandwidth will increase together with the information frequency....**
**==================================================================**

## 17.11. FSK = Frequency Shift Keying = Digital Frequency Modulation

This means that the two digits "0" and "1" are coded by two different carrier frequencies. The lower carrier frequency is named "**Space**" and the higher "**Mark**".

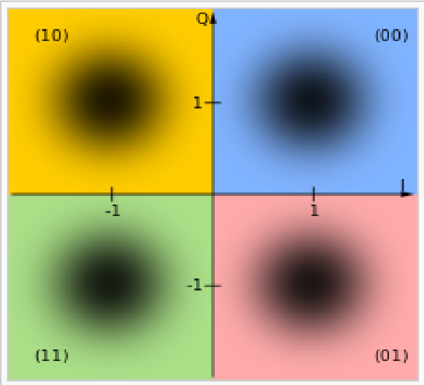
In qucsstudio we use the "**modulated source" and switch the property to "FM".**

This example shows the principle.



## 17.12. QAM in examples

### 17.12.1. 4 QAM (= 4 PSK)




**constellation diagram of a 4-QAM transmitting 2 bits.**

**In every quadrant an "allowed landing zone" must be defined in which the transmitted symbol can be recognized**

QAM means **"Quadrature Amplitude Modulation"**.
For a demonstration we use the modulated source in the PM mode.

The description of the PM mode in the "technical handbook for qucs" (coming from the qucsstudio homepage) tells us this expression which describes the working principle:

$$V_1\,(t) - V_2\,(t) = A \cdot \sin\left(\omega \cdot t + \phi + 2\pi \cdot M \cdot V_3\,(t)\right)$$

**"A" means the pointer length = carrier voltage amplitude** which must be constant in our example.

The rest of the brackets content is very simple:

**"ώt" defines the value of the carrier frequency we want to work with. Let us use f = 6 kHz.**

The "starting value of the phase shift" **ϕ** is set to zero.

In the last expression we use **M =1. The phase shift for every constellation is now controlled by a small file controlling the phase via V3(t):**

**For "45 degrees": V3(t) = 0.125**

**For "135 degrees": V3(t) = 0.375**

**For "225 degrees": V3(t) = 0.625**

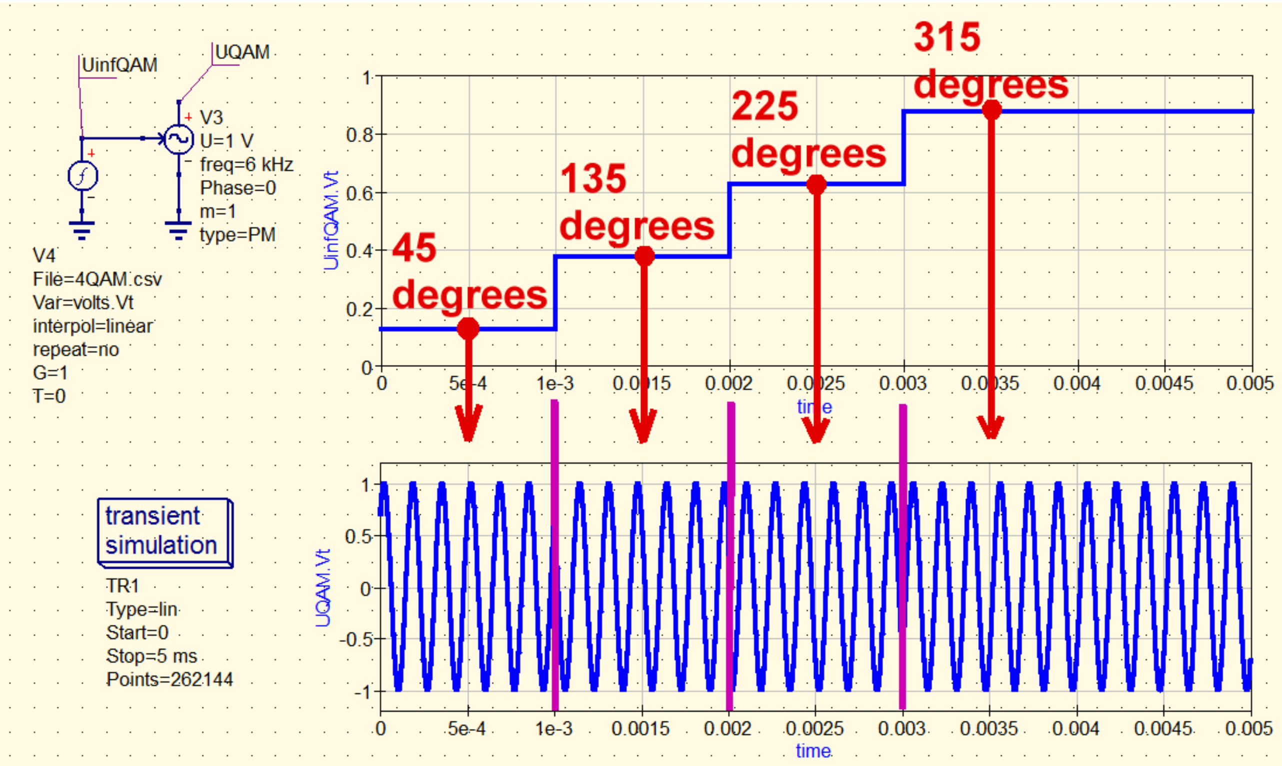
**For "315 degrees": V3(t) = 0.875**

**(For V3(t) = 1 you would get "2π*1" radians = 360 degrees....)**

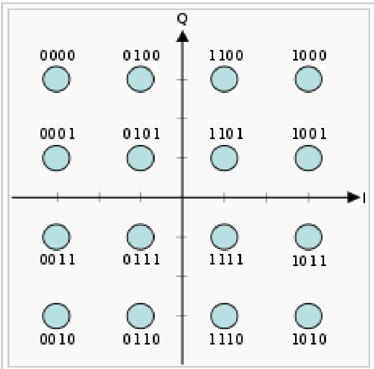
So please use a text editor to write the following "**4QAM.csv**" **file for the control of the PM source:**

```
0           0.125
1e-3        0.125
1.001e-3    0.375
2e-3        0.375
2.001e-3    0.625
3e-3        0.625
3.001e-3    0.875
4e-3        0.875
```

You find the nice simulation result on the next page.




## 17.12.2. The 16 QAM

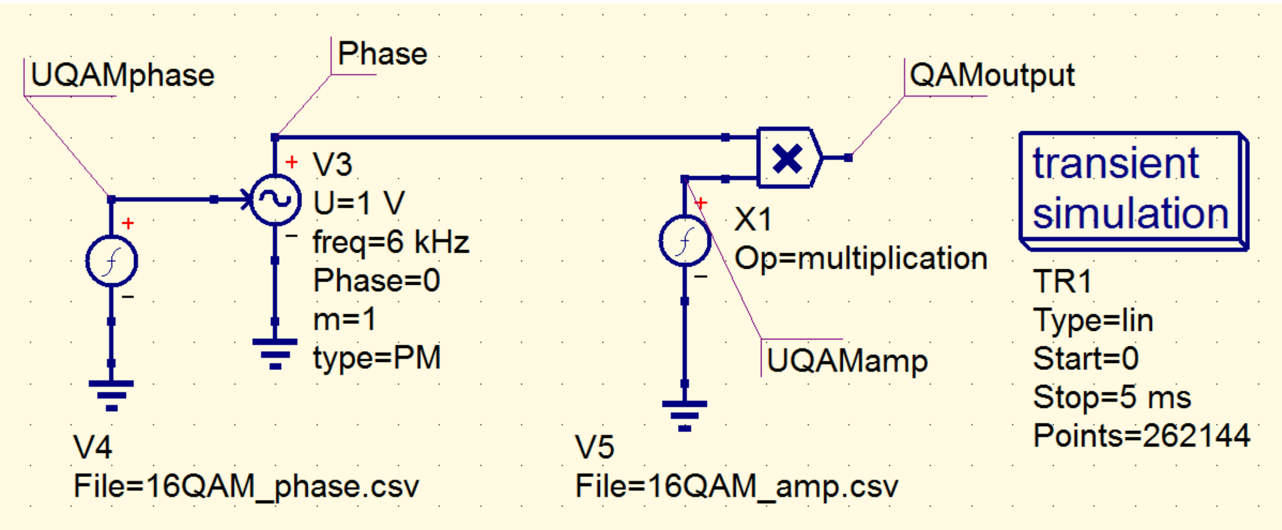



This is the 16 QAM constellation diagram. Every point stands for a symbol with four digits.

$$V_1\,(t) - V_2\,(t) = A \cdot \sin\left(\omega \cdot t + \phi + 2\pi \cdot M \cdot V_3\,(t)\right)$$

Regarding this formula for the Phase Modulator we have to set the phase of every symbol by V3(t) in the bracket, but the amplitude by the factor “A” at the front of the equation.

**So a simulation schematic would look like:**




Thus we need two *csv files to control the amplitude and the phase of the carrier due to the chosen symbol.

Then let us test the result by transmitting the symbols

**0000 / 0011 / 1110 / 1010**

with 16 QAM.
Use again a carrier with f = 6 kHz and a peak value of 1V.
Find the length and phase values of the four symbols by analyzing the above illustration.

Result:
**0000 = 100% of maximum pointer length and an angle of 135 degrees**

**0011 = 74% of maximum pointer length and an angle of 200 degrees**

**1110 = 74% of maximum pointer length and an angle of 290 degrees**

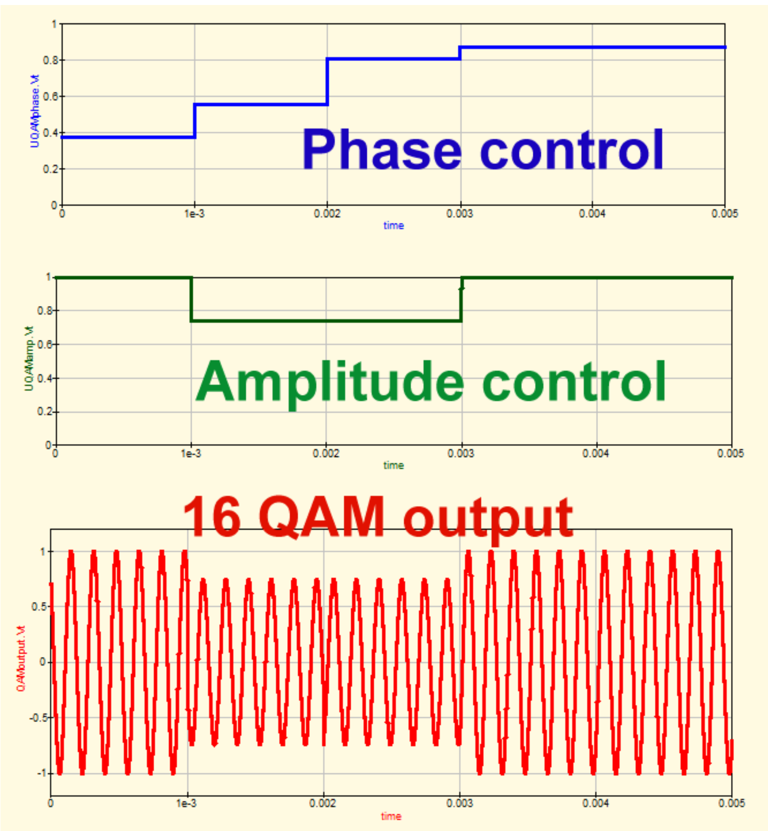
**1010 = 100% of maximum pointer length and an angle eo 315 degrees**

**Solution:**
Thais is the file “**16QAM_amp.csv**“...

| | |
|---|---|
| 0 | 1 |
| 1e-3 | 1 |
| 1.001e-3 | 0.74 |
| 2e-3 | 0.74 |
| 2.001e-3 | 0.74 |
| 3e-3 | 0.74 |
| 3.001e-3 | 1 |
| 4e-3 | 1 |

...and this the file **“16QAM_phase.csv**“

| | |
|---|---|
| 0 | 0.375 |
| 1e-3 | 0.375 |
| 1.001e-3 | 0.5555 |
| 2e-3 | 0.5555 |
| 2.001e-3 | 0.8055 |
| 3e-3 | 0.8055 |
| 3.001e-3 | 0.875 |
| 4e-3 | 0.875 |




The result.....

### 17.12.3. Demodulation of QAM Signals

No problem:

Use two product detectors in parallel and feed the inputs with the QAM signal.

Every product detector needs additionally a carrier signal with exact the same frequency as used for the QAM signal during transmission.

**But these two carriers must have a phase difference of 90 degrees.**

So you get the two axis "I" and "Q" of the coordinate system and you need only to multiply, followed by a good filtering to suppress the rests of the carrier signal.

# 18. Noise

## 18.1. Fundamentals

If you receive a message without any information than this sounds like a waterfall. Such a signal is named “noise” and is well known as the receiver sound between two stations. In that case the noise is produced by the atmosphere or the deep space. But every resistor or conductor generates noise due to the “movement of the electrons with increasing heat”, if the temperature exceeds zero degrees Kelvin”
But a “crypted digital message” or “pseudo random noise” sounds like the “original Johnson Noise”.

**Ideal noise = white noise = thermal noise = Johnson Noise has a special property: The Spectral Power Density (= power in every Hertz of bandwidth) is constant all over the regarded frequency range and that is typical for in an Ohmic resistor generated noise voltage.**
(But if you vary this property by the usage of filters you get “pink noise”....)

## 18.2. Noise -- where does it come from?

There is a quick answer: current in a resistors or conductor means moving electrons. But if heat plays a role (at increasing temperatures above Zero degree Kelvin) electrons start to bounde: to the left, the right, frontwards, backwards...there are collisions....and that is not the straight way from Minus to Plus. This takes also place without an applied voltage and so at the connections of the conductor or resistor a small “noise voltage” can be measured. This voltage can be calculated as

$$U_{NOISE} = \sqrt{\frac{4 \cdot h \cdot f \cdot B \cdot R}{e^{\frac{hf}{kT}} - 1}}$$

mit
h = Planck’s constant
k = Boltzmann constant = 1.38 x $10^{-23}$ J / Kelvin
T = absolute temparature in Kelvin
B = regarded bandwidth in Hz
f = center frequency in Hz
R = resistance value in Ω

This formula seems to be very complicated but up to 100 GHz and down to a temperature of 100 K you can use this smplified version:

$$U_{NOISE} = \sqrt{4 \bullet k \bullet T \bullet B \bullet R}$$

Square the formula and you get this formula version:

$$\frac{\left(\frac{U_{NOISE}}{2}\right)^2}{R} = k \bullet T \bullet B$$

**This is ist a simple power formula and means that in every resistor (independent of the resistor's value) the same noise power is generated** by the existing heat **and an open loop internal voltage $U_{noise}$ can be measured.**

Now we regard the resistor R as a voltage source with the open loop voltage $U_{noise}$ and the (noise free!) characteristic resistor R. If then the same (noise free) resistor R is connected to the source as a load you get perfect power match. In this case you will measure **half the open loop voltage Unoise over the load and the delivered power to the load is “kTB”.**
The intrinsic noise power of the resistor is rising according to the part's temperature and the regarded

bandwidth (..the noise voltage is then the square root of the noise power). There is no variation in the power density due to the regarded frequency and this kind of noise is called “**White Noise**”.

Important:
In the communication technique “**Levels**” are preferred instead of voltage values and this replaces (due to the logarithmic principle) the multiplication when connecting modules in series by an addition.
The new unit to work with is now “**dBm**” and always powers (and not voltages) are used for the calculations.
A reference power

**$P_0$ = 1 Milliwatt at the characteristic impedance of the system**

is used and every regarded power in the system is referred to this “reference value”. So you get the power level to

$$10 \bullet \log\left(\frac{\text{measured power}}{1\text{mW}}\right) \quad \textit{in dBm}$$

If you now have a sharp look at the noise power “**kTB“ you can also say:**

**“kT“ is the noise power in every “Hertz of bandwidth“ (= spectral noise density) and you get the total noise power when you multiply “kT” with the valid bandwidth.**

Changing to “thinking in levels” you should know:
Every resistor generates at room temperature ($T_0$ = 300 K) the same **internal noise level of**

**-167.8 dBm per Hz of bandwidth**

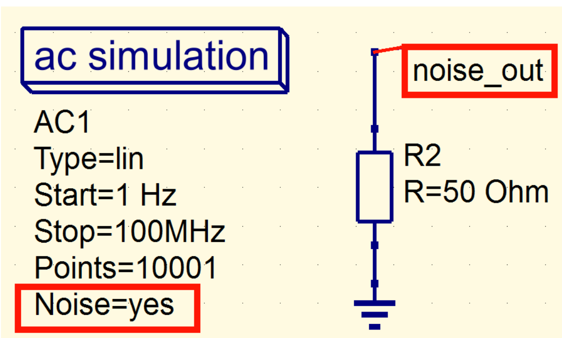
If you connect this resistor in parallel to another but noise free resistor with the same value then you get “power matching” and the available power level at the parallel connection decreases by 6 dB. So this measured level at the parallel connection is

**-173.8 dBm per Hertz**

And if the bandwidth is greater than 1 Hz:

**Measured maximum available power level in dBm is:**
**-173.8 dBm + 10log(bandwidth in Hz)**

**Task 1:**
**What is the “open loop noise level” and the measured open loop noise voltage for a resistor with 50Ω at t = 300 Kelvin for a bandwidth of 1Hz? Confirm this result by a qucsstudio simulation.**




**Solution:**
The internal noise level for every resistor is **-167.8 dBm** for T = 300 Kelvin and b = 1 Hz.

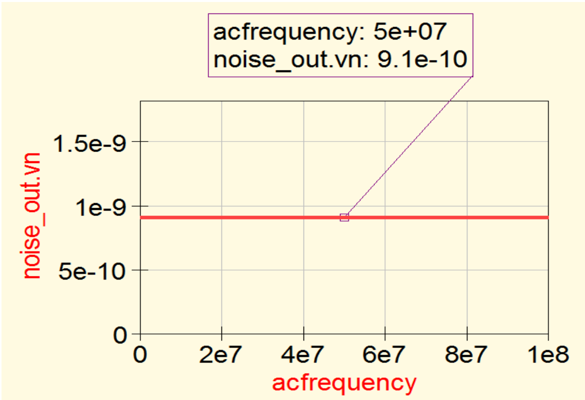
This equates to a power of **P = (1mW) * 10exp(level / 10) = 1.66 *10exp(-20) Watt**

Using a resistance value of 50 Ohms you get the internal open loop voltage to

**U = sqrt(P * 50Ohm) = 0.91 nV**
Now follows the qucsstudio simulation:

Pick a 50Ω – resistor and ground the lower end. On the upper end set a label named„**noise_out**“.

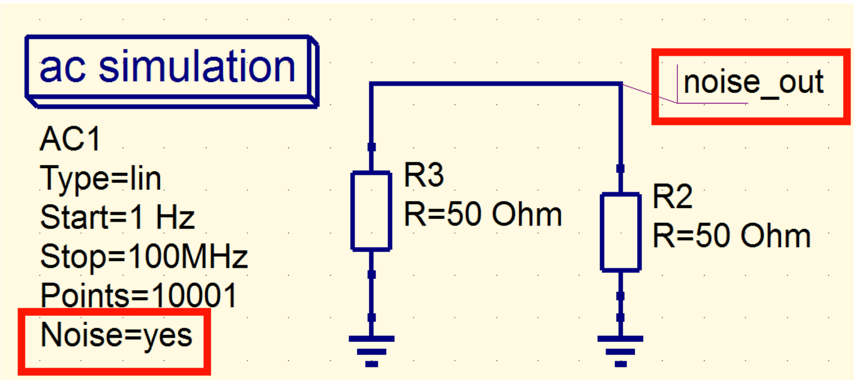
Now use an AC simulation for the range from 1 Hz up to 100 MHz with 10001 points. But (after editing the resistor's properties) change there to the menu „**Properties**“ and **activate the noise simulation.**




This is the correct result of the expected 0.91 NanoVolts shown by a frequency marker at 50 MHz.

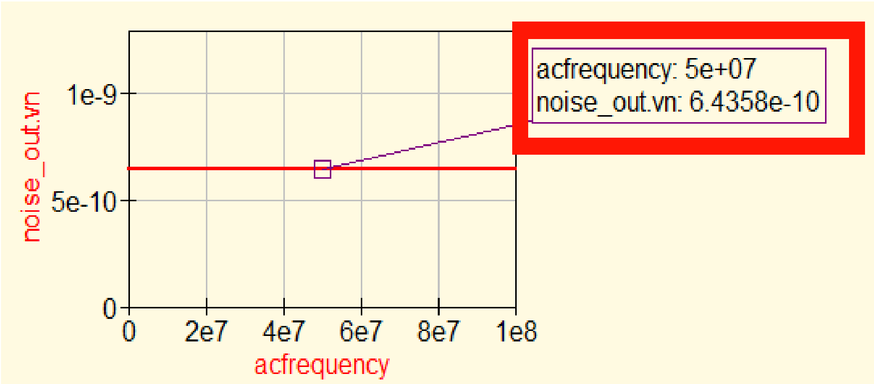
==========================================================================

**Task 2:**
Connect an additional resistor with a value of 50Ω in parallel. Now calculate again “noise_out”. Analyze and explain the result.




Qucsstudio works as follows:
At first the left resistor is regarded as a noise source with the internal characteristic resistor R3 of 50Ω and the internal open loop voltage of 910 PicoVolts. This source is connected to a noise free load with R2 = 50Ω and this is the case of “perfect power matching”. Thus we find a voltage of 455 PicoVolts per square root of 1Hz at the connection point named “noise_out”.
Now the procedure is repeated, but with R2 as noise source and a noise free load R3. This gives also 455 PicoVolts per square root of 1Hz and afterwards the two calculated voltages must be added.
**But:**
**These two signals are “not correlated” and thus only the “main square values” may be added:**

$$U_{noise\_total} = \mathrm{sqrt}[(0.455nV)^2 + (0.455)^2] = 0.635nV$$




This is exactly the simulation result.

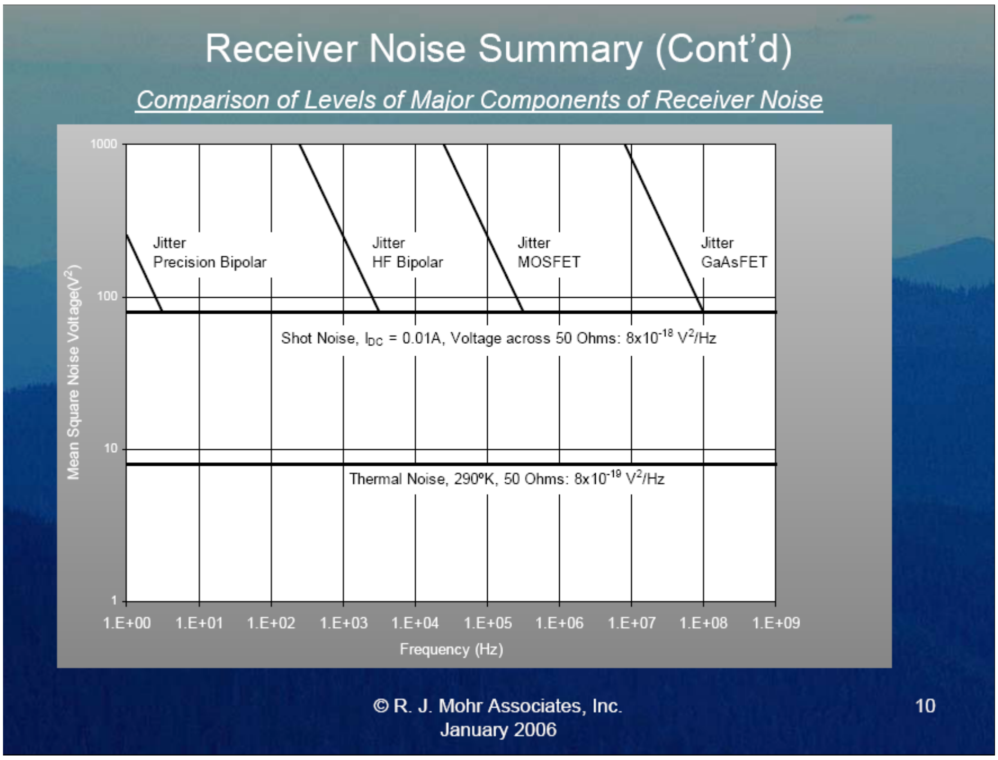
===================================================================================

## 18.3. Other Noise Sources

In every active part (like tube, bipolar transistor, junction FET, MOSFET, HEMT...) also additional kinds of noise are existing:

a) ***Shot-Noise*** is a broadband noise, generated in tubes and PN junctions due to non-uniform electron movement when traveling through potential differences.

b) ***Flicker-Noise* = *Jitter Noise* =„1 / f – Noise**“ is caused by contamination of the crystal structure in a semiconductor. This generates short changes in the constant current flow with the well known 1 / f spectrum of such “steps” including a “corner frequency”.
Please have a look at this informative diagram, published in an application note by Mohr Associates.




===================================================================================

# 18.4. Using White Noise to measure a Transfer Function

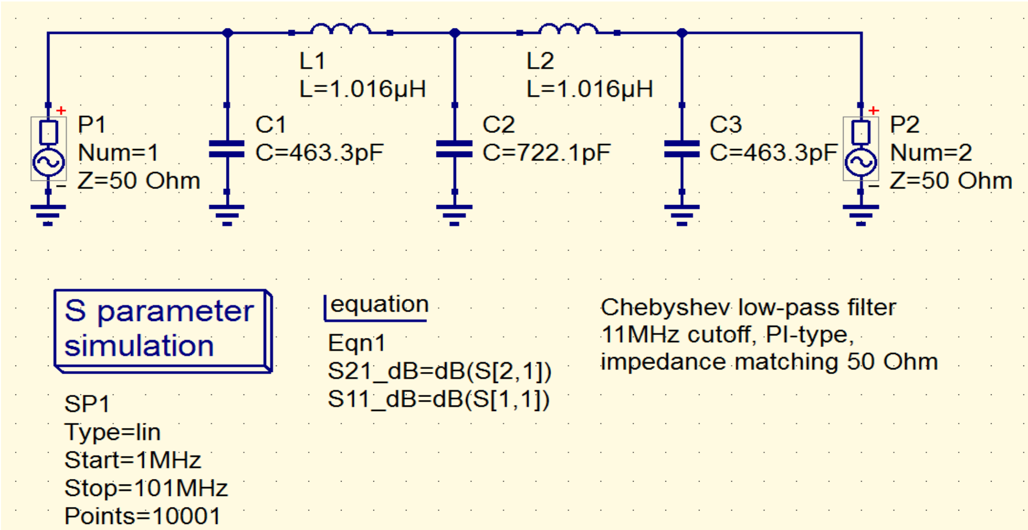
## 18.4.1. Remembering S Parameters

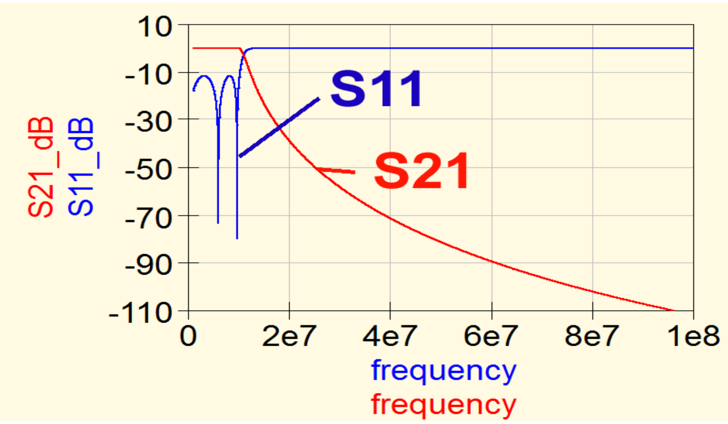
The Communication Technique needs permanently filters, amplifiers, couplers, cables.....and thus many kinds of these components are in permanent development and production. So conventions were necessary to ensure best transmission quality independently of the manufacturers and thus the S parameters were created.
Let us look at a Low Pass Filter with a pass band corner frequency of 11 MHz (as used to suppress everything above the RF receiver intermediate frequency with 10.7 MHz).
This LPF uses – like every module in the communication chain – the same "characteristic impedance" = input impedance = output impedance of 50Ω to ensure perfect power matching. If you now feed the LPF input with a short pulse over a 50Ω cable you find an incident wave traveling from the generator to the LPF's input. The amplitude of the incident wave is 50% of the generators's internal voltage (= open loop voltage) and when arriving at the LPF input two reactions are caused:

a) If the input impedance differs from 50 Ω then you don't have "perfect match" and a part of the arriving energy is not accepted and will be reflected as an "echo". This fact is expressed by the **Input Reflection S11**.

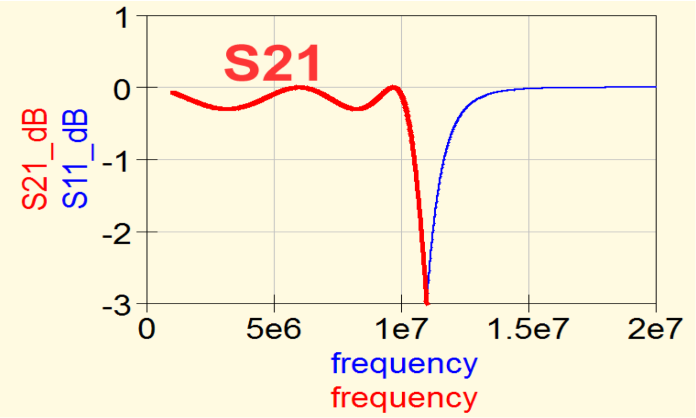
b) The rest of the energy enters the module (= the LPF) and causes an output signal with a greater or smaller amplitude. This reaction is described by the **„Forward Transmission S21" and measuring S21 is the usual method to find the transfer function.**

The procedure is now repeated at the output side and gives the two paramters "S22" and "S12" for a twoport.
This is now demonstrated by a qucsstudio S parameter simulation for this Chebychev LPF:




This is the simulation result for S11 and S21 in the frequency range from 1 kHz up to 100 MHz.




But to see the “Chebychev” ripple you have to zoom S21.

An important question:
**How can I measure the Forward Transmission S21?**

You have the choice:

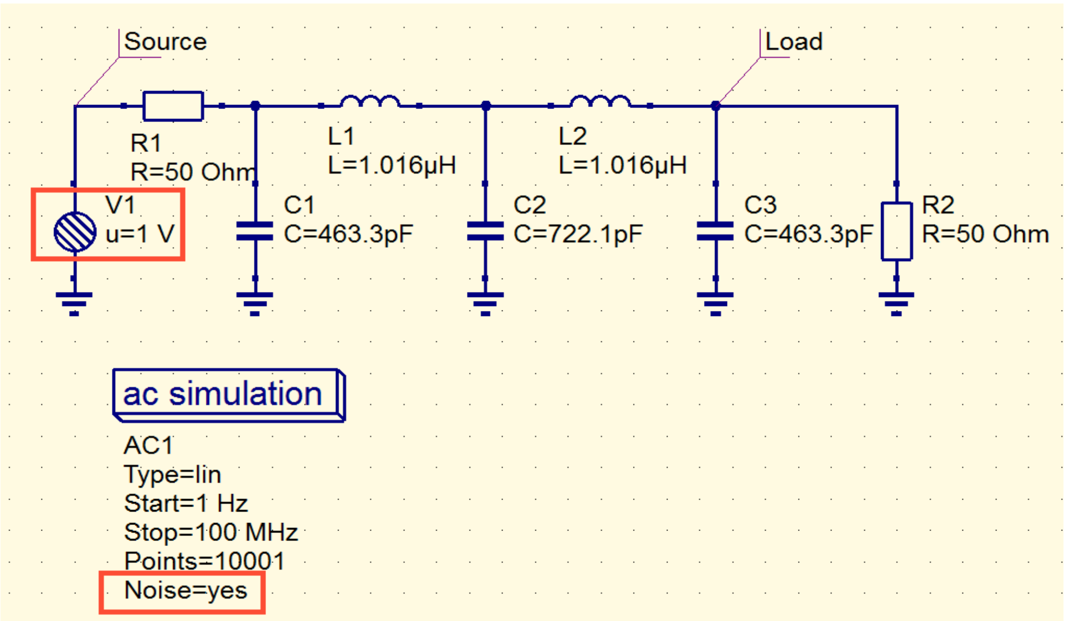
a) Use the S parameter simulation or a Vectorial Network Analyzer to sweep from the lowest up to the highest frequency. This is what we did in our example.

b) Use a „**Dirac Impulse**“at the input which represents a complete and constant energy spectrum up to infinity without any gaps. But afterwards you have to convert the impulse response at the output to a frequency response by the FFT.

c) Use white noise at the input and measure the spectral noise density at the output with a spectrum analyzer. White noise has (like a Dirac Impulse) a constant power density over the complete frequency range and so you can at once see the Transfer Function of the module as “envelope” of the output signal.

## 18.4.2. Simulating S21 using White Noise

The last method is a good choice for simulation and practice. So let us use the last LPF example.

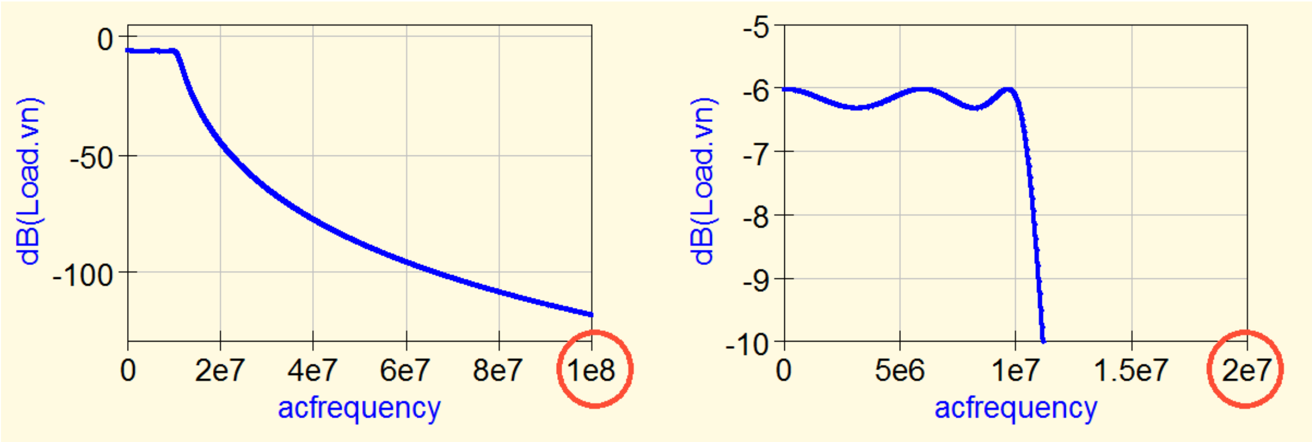


a) Please add the **label "Load" at the output and at the noise source the label „Source".**

b) Use the "**noise voltage source" coming from the "Components / Sources" menu**. Set the internal source voltage to „**1V / sqrt(1Hz)**". This high noise level is necessary to "override" the small intrinsic noise voltages of the resistors in the circuit and increases the simulation accuracy. **Activate "noise calculation = yes" in the properties of the source.**

c) Start an AC simulation from 1 Hz up to 100 MHz using 10001 points.

Now simulate and use a Cartesian diagram with indication of „**dB(Load.Vt)**" at the vertical axis.




In the left diagram you find S21 from Zero up to 100 MHz (= wide band simulation).
The right hand illustration shows the "Chebychev ripple" after zooing with an upper frequency limit of 20 MHz.

But to get the correct parameter S21 you have to use the formula:

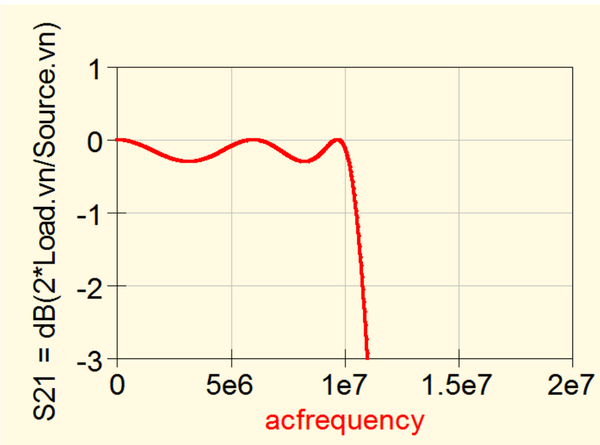
**S21 = (voltage at resistor R2 = point "Load") / (Incident Wave)**

**= $U_{Load}$ / (half internal source voltage**

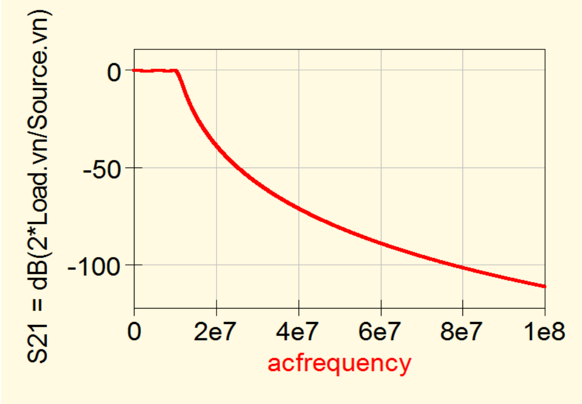
**...calculated in dB....**

Thus you have to change the "Graph Properties" of the left diagram to

## dB(2*Load.vn / Source.vn)




And this is the correct result for the frequency range from 0....20 MHz and an amplitude value range from +1....-3 dB. Nice, isn't it?........

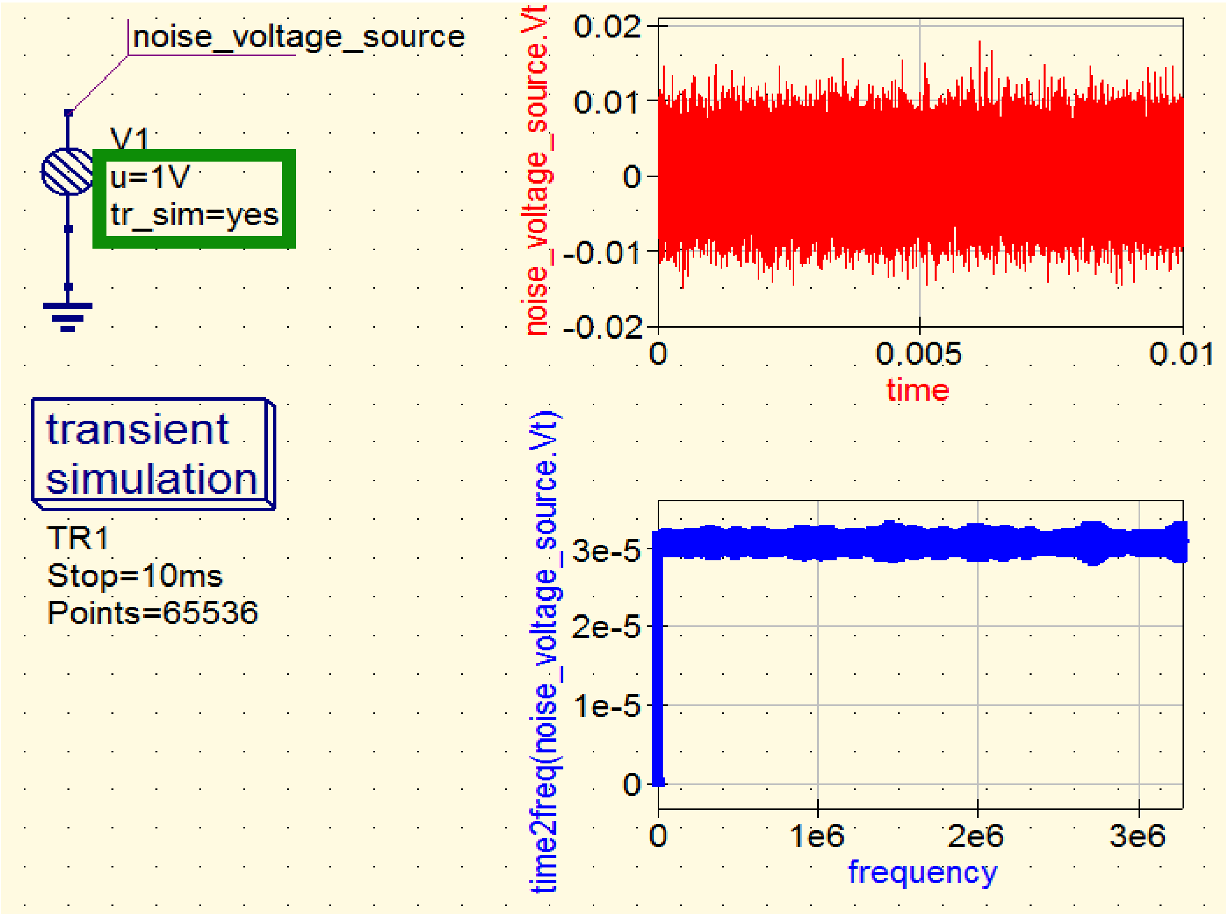



…. but the wide band presentation is also a pleasure.

## 18.5. Generating Noise Signals in the Time Domain with qucsstudio

This is a useful tool for the circuit developer, because you can now simulate irregular variations of a supply voltage, check the “Bit Error Rate” of a Digital Communication System, improve the linearity of AD converters...
In the folder “Components / Sources” you find the noise voltage source for this purpose to generate white noise. (…and if you want: also “pink noise” by setting coefficients in the property menu).

Please open the properties of the noise source, enter a voltage density of 1V / sqrt(Hz), activate “create noise during time simulation” by “yes”. Afterwards use the FFT to show the spectrum of white noise.




Who is interested in the power density vs frequency: simply use a formula for the vertical axis to square the voltage density!

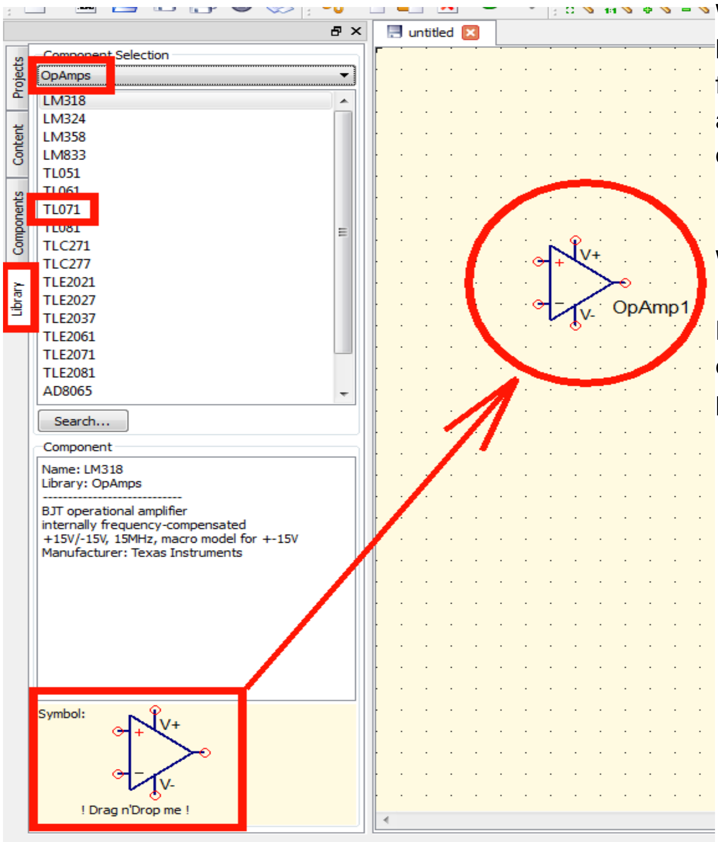
| time | noise_voltage_source.Vt |
|---|---|
| 0 | -0.00728 |
| 1e-6 | 0.0157 |
| 2e-6 | 0.00441 |
| 3e-6 | 0.0144 |
| 4e-6 | 0.000403 |
| 5e-6 | -0.00793 |
| 6e-6 | -0.0108 |
| 7e-6 | -0.0108 |
| 8e-6 | 0.00597 |
| 9e-6 | -0.0183 |
| 1e-5 | -0.00326 |
| 1.1e-5 | -0.00997 |
| 1.2e-5 | -0.000999 |
| 1.3e-5 | 0.0152 |
| 1.4e-5 | 0.00676 |
| 1.5e-5 | 0.0213 |

If anybody needs (for other purposes) complete series of noise amplitude values he can do this as follows:
Simulate from 0...10ms with 10001 points. This gives a time step of 1 microsecond.

When the simulation is finished: use "**Tabular**" presentation of the result.

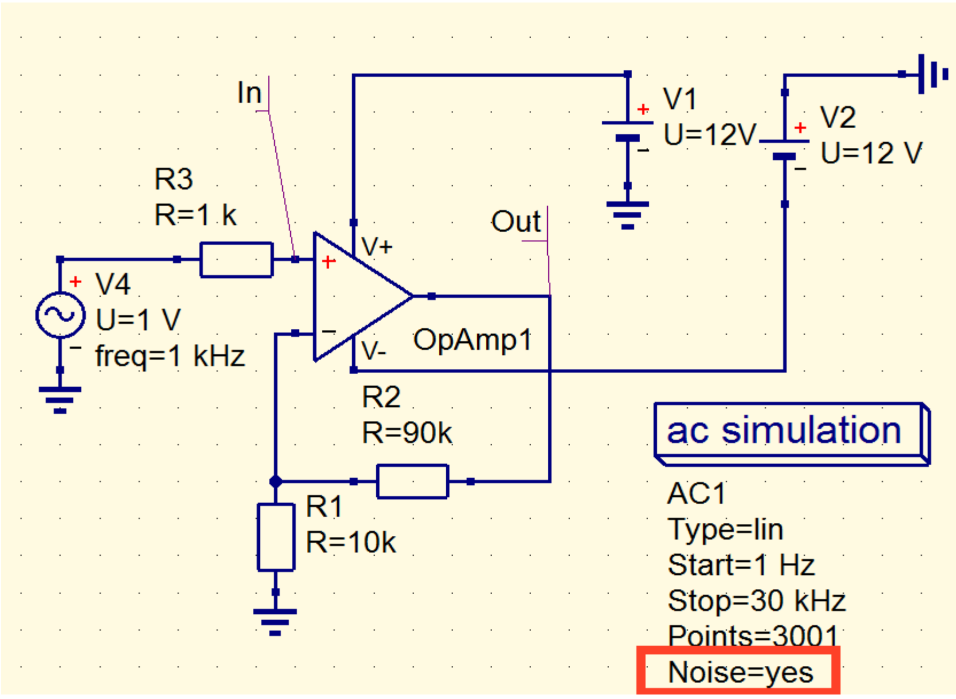
If you want to save this value series in a file: mark the curve "noise_voltage_source.Vt" in the Cartesian diagram. Then go to "Project" in the Windows task bar to click on **"Export to CSV". The program asks then for the CSV file name and the memory location.**

## 18.6. Simulation of an OPA circuit including Noise Figure NF

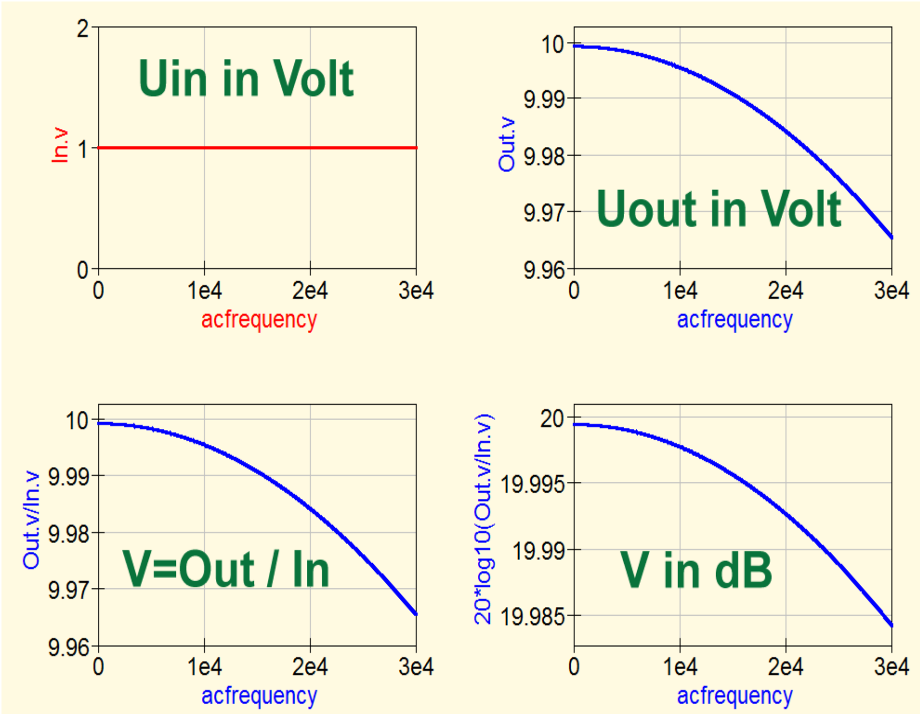



We use the well known OPA type "TL071" built by Texas Instruments. The model can be be found in the qucsstudio **Library** under **OpAmps** and the symbol can be dragged and dropped to draw the schematic.

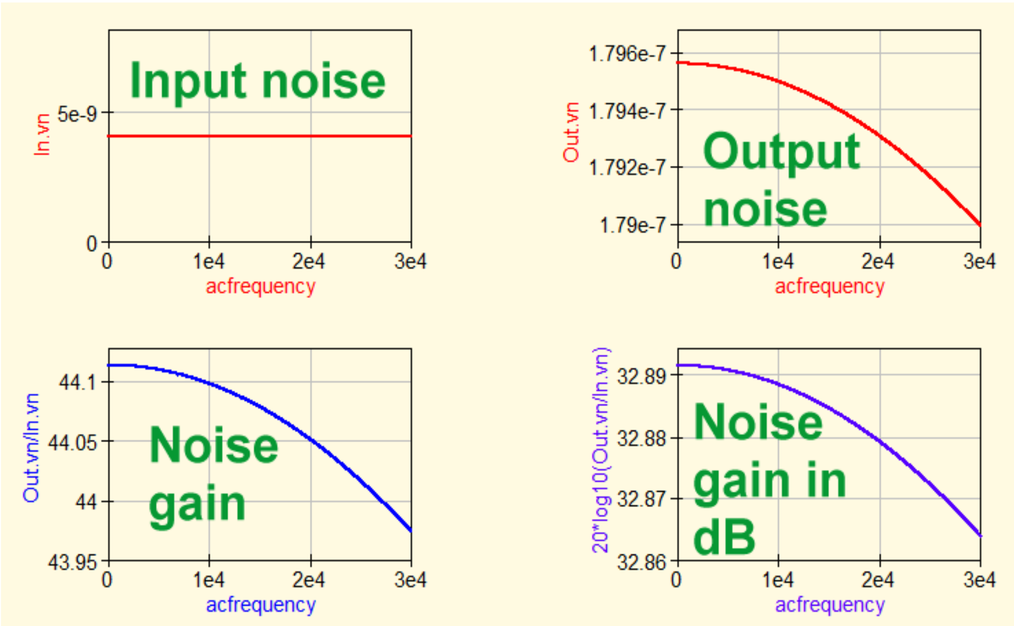
We use a non inverting amplifier with a gain of 10 in the frequency range from 1 Hz to 30 kHz. Draw the following circuit but do not forget to enable the noise simulation in the **AC sweep** properties...




**After the simulation show the input voltage, the output voltage, the linear gain and the gain in dB versus frequency in separated Cartesian diagrams.**




---




**Now repeat the procedure for the input noise voltage, the output noise voltage, the linear noise gain and the noise gain in dB.**

**Because the OPA is noisy itself, the output noise voltage is greater than the value**

**output = input x gain**

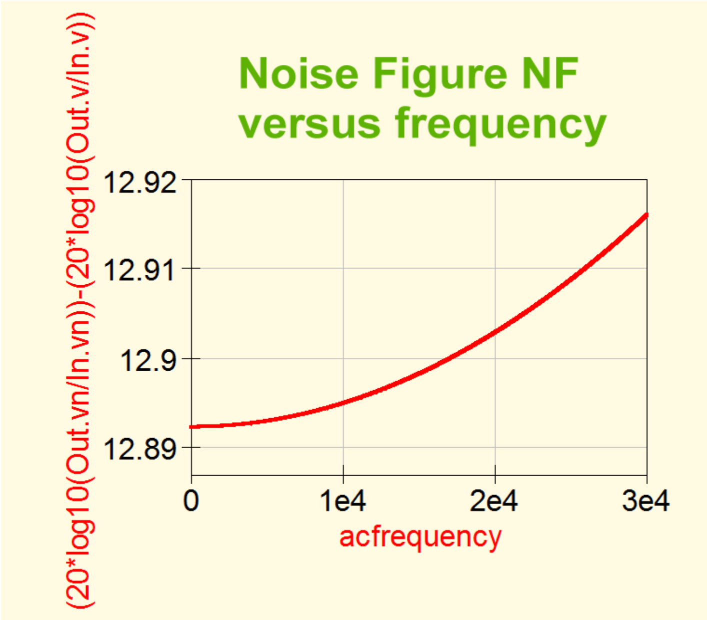
**Therefore if you subtract the “stage gain in dB” from the “Noise gain in dB” you will get the**

## Noise Figure NF of the stage in dB

**The regarded signal to noise ratio of the input will be reduced by the value of the noise figure NF when measuring at the output!**

The necessary formula for the Graph properties is:

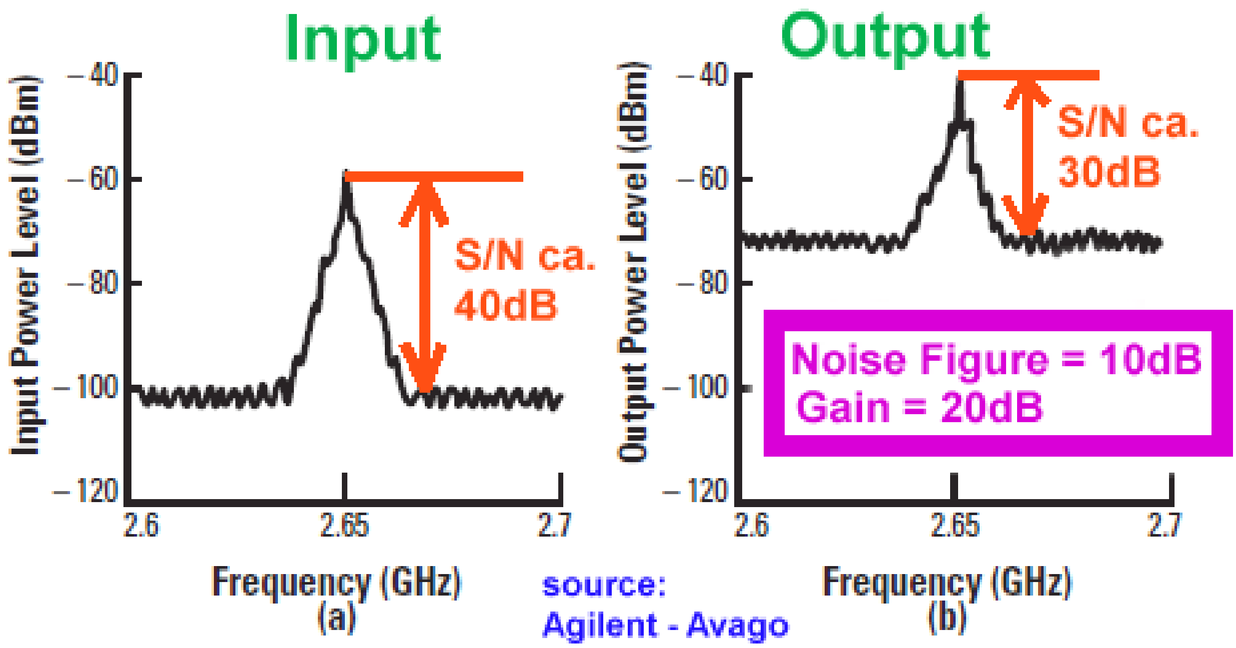
## (20*log10(Out.vn/In.vn))-(20*log10(Out.v/In.v))




...and this is the simulation result.

For better understanding comes here a fine example from an Agilent Avago application note using an amplifier for 2.65 GHz:

**gain is20 dB, the input level is -60dBm and the signal to noise ratio at the input is 40 dB.**



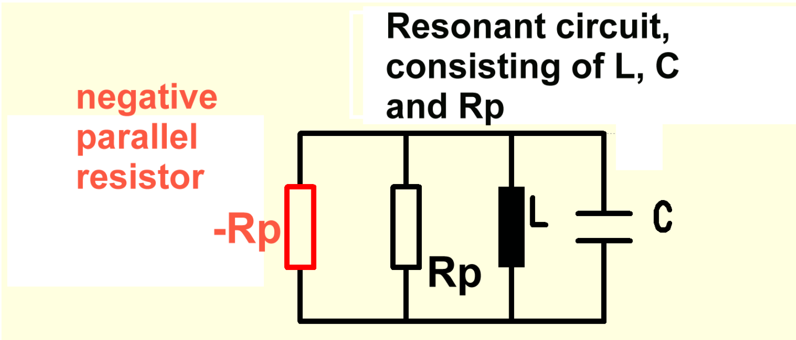

The input signal level appears increased by the gain of 20 dB at the output. But the noise level has increased by 30 dB and thus due to the S / N ratio reduction the **noise figure NF has a value of 10 dB (t**he input voltage to noise ratio of 40 dB is now reduced to 30 dB at the output).

**Remember:**
RF circuits are designed using S parameter files, published by the semiconductor manufacturers. Most of these S parameter files contain additional information for the simulation software to calculate the **"actual noise figure NF" and the "minimum noise fiigure Nfmin".**
**In chapter 12 of this tutorial you find a complete design example for a "LOW NOISE AMPLIFIER = LNA". Please have a look at this chapter to learn how to handle the noise matching procedure.**

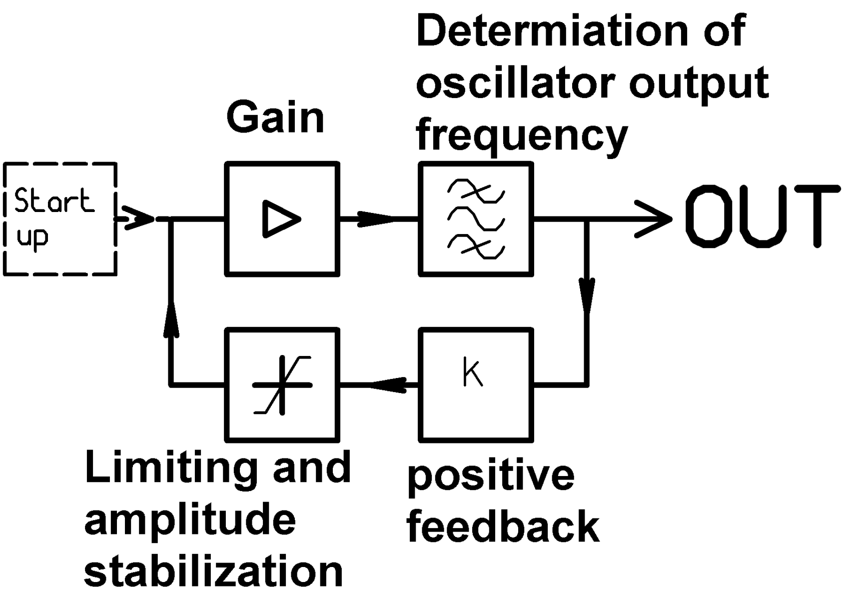
## 19. Sine Oscillator Simulation

### 19.1. Sine Oszillator Principle

Such an oscillator generates – due to its name – a sine wave with an exactly defined frequency and constant amplitude.
A lot of oscillator circuits are known but at first we deal with a principle which is very often used in the Microwave region.




If a resonant circuit is fed for one time with a "bucket of energy", an oscillation will occure. Caused by the circuit losses this will be a "damped oscillation" and soon be terminated.
But if a "negative resistance" is connected in parallel you can compensate the losses and now an oscillation with constant amplitude can be observed. The amplitude will even increase for "over compensation" and a limiter circuit is then necessary to stop this rise.




But for frequencies below 1000 MHz this is the mostly used principle.

The "Startup Unit" feeds the amplifier ininput with a signal of the desired output frequency. This signal is amplified and filtered out by a band pass filter. Then it can be used at pin "OUT" for other purposes.

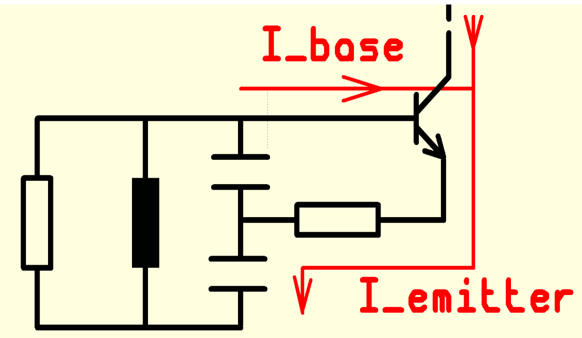
**Important:**
**The "OUT" signal is "in phase" with the signal at the amplifier input.** So a "**positive feedback network**" always takes a part of the OUT signal to feed the amplifier input and to make the startup signal expendable. If more than the "necessary minimum" is permanentlly fed back you get an increasing OUT signal amplitude – and thus the limiting circuit must then start its action.

The positive feedback signal can be expressed as **„k*Uout" and so you find the well known condition for oscillating:**

### k*Gain > 1 (including "in phase")

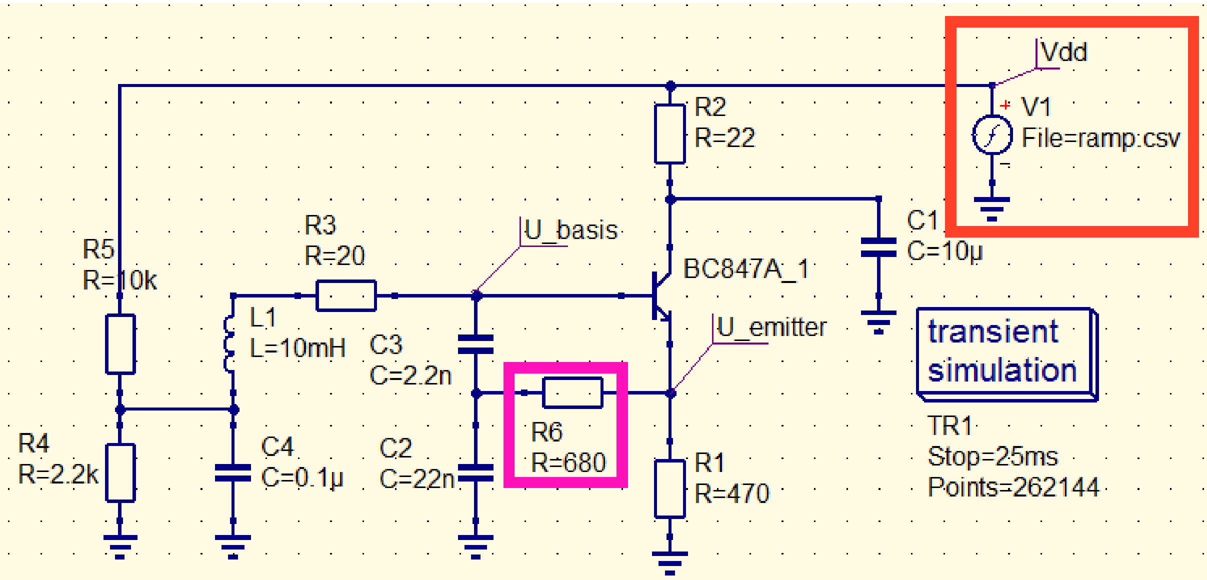
The startup procedure" is not complicated. The noise generated in the amplifier itself contains also very small parts with the output frequency. These parts are filtered out, sent back to the input and amplified. So a feedback signal with increasing amplitude is effective and generates (after amplification) a rising OUT signal which must be limited and stabilized.

## 19.2. Simulated Oscillator Circuit




We use a Colpitts Oscillator Circuit with an npn type transistor BC847C working in collector grounded operation.

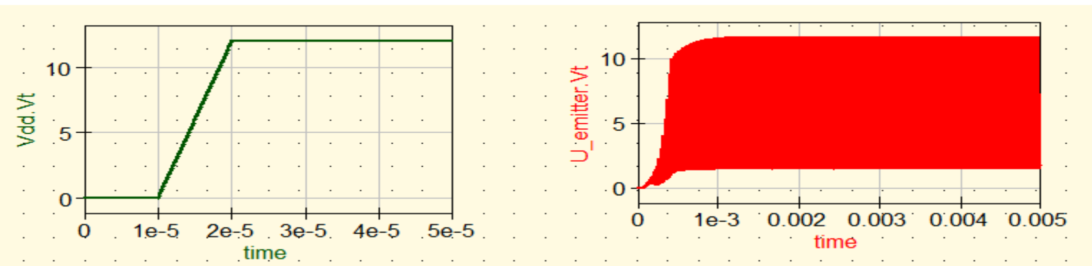
**Principle**:
The voltage of the resonant circuit feeds the base of the transistor which consumes a base current "**I_base**".
The transistor itself is a "**current amplifier**" and so the resonant circuit is fed with the (amplified and in phase!) emitter current to compensate the circuit losses.
The **positive feedback factor "k"** is defined by the value of the resistor between the emitter and the resonant circuit: increasing the resistor's value decreases the positive feedback.
Now see the circuit in practice, expanded by the operating point setting network:

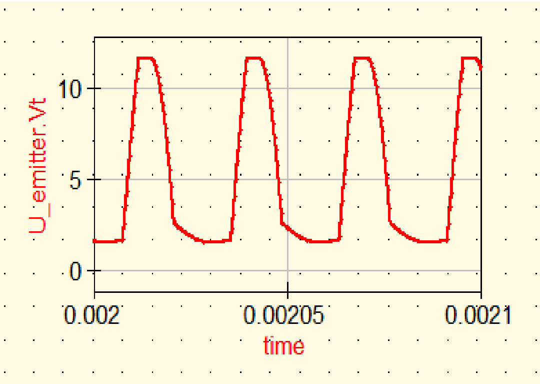



An important detail is the (red marked) **power supply**:
An oscillator simulation needs a "**startup push**" and this is done by programing **the supply voltage as a ramp signal.**
Vdd starts at time t = zero with a value of zero Volts and stays so up to 10 µs.From t = 10µs to t = 20µs the amplitude rises linearly up to +12 V and stays afterwards on this value.
Therefore a "**ramp.csv**" file (= comma separated value file) must be written to use a "**data based voltage source" (= PWL source = piecewise linear voltage source)**. This file must be saved in your actual oscillator project folder:

| | |
|---|---|
| **0** | **0** |
| **10e-6** | **0** |
| **20e-6** | **12V** |
| **40e-6** | **12V** |

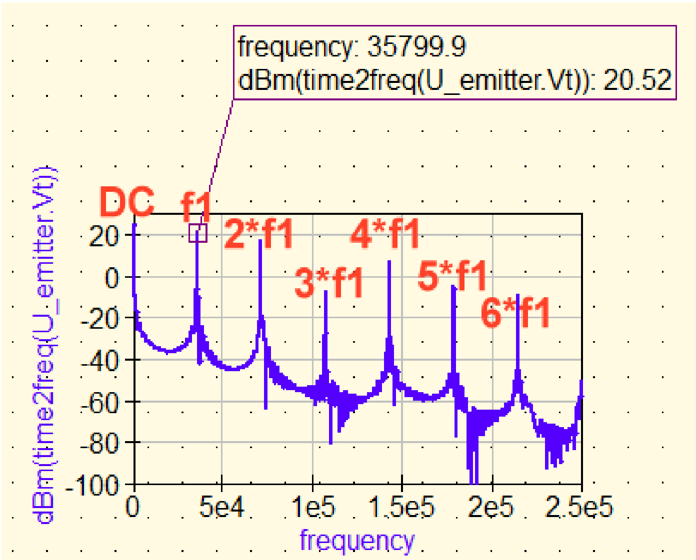
Then simulate and show Vdd resp. U_emitter:






The “expontential rise” of the emitter voltage (followed by limiting) can very well be seen in the right diagram. But there is no symmetry in the signal and the reason is at once found when using the Zoom function:

**The positive feedback is too great and thus the transistor is “strongly driven into saturation”. This causes an extremly distorted signal with a lot of even and odd harmonics.**




To proof this a FFT of “U_emitter” is called by the Graph Formula

**dBm(time2freq(U_emitter.Vt))**

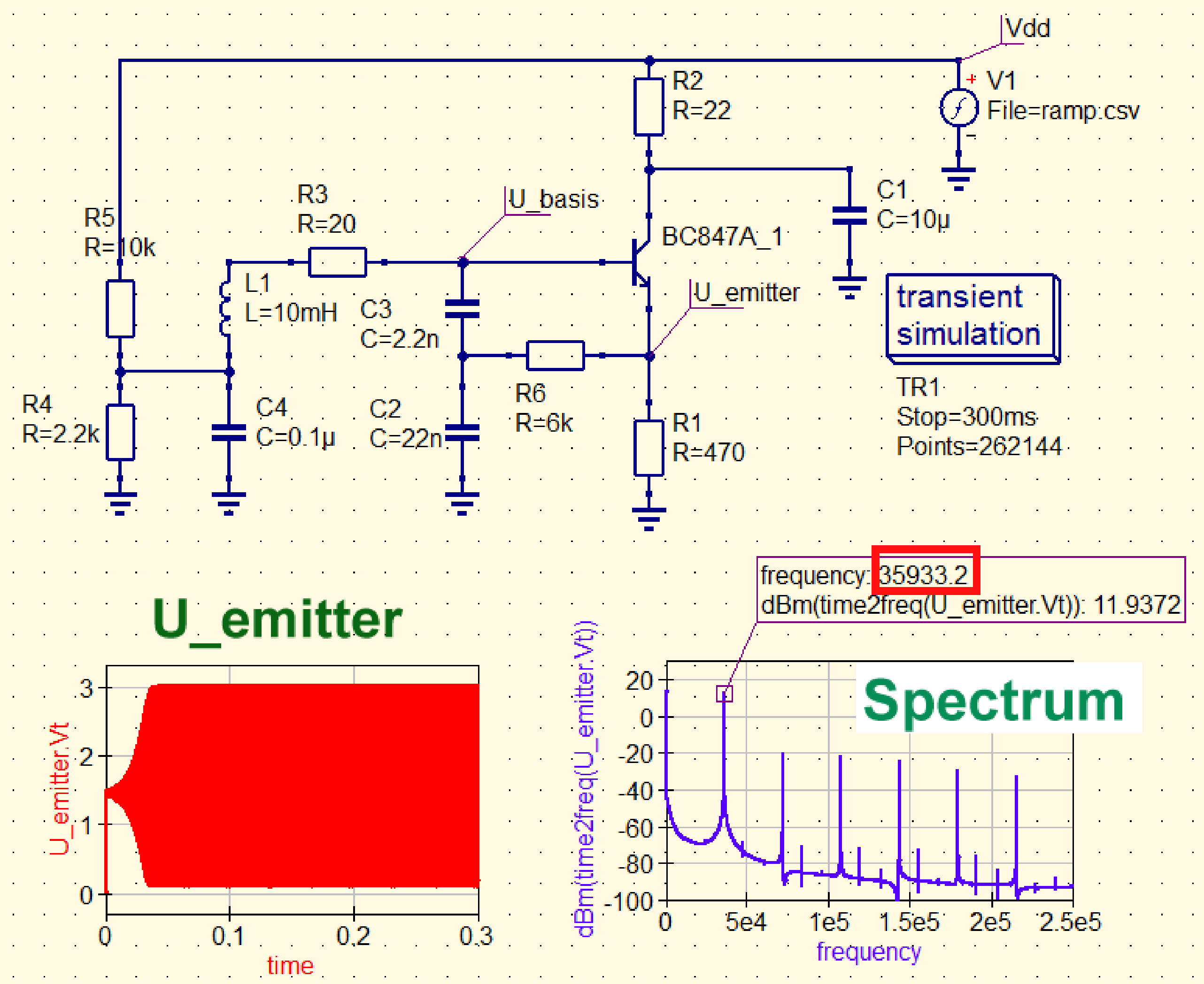
and this is the confirmation.

In the above circuit diagram the resistor **R6 / 680Ω** is marked in red, because this is the “positive feedback channel”. Increasing the resistance value reduces the part of the emitter current which feeds the resonant circuit to compensate its losses. This improves the waveform but the starting process is delayed more and more.
And if you continue with increasing the resistors value.....the oscillation will suddenly stop!

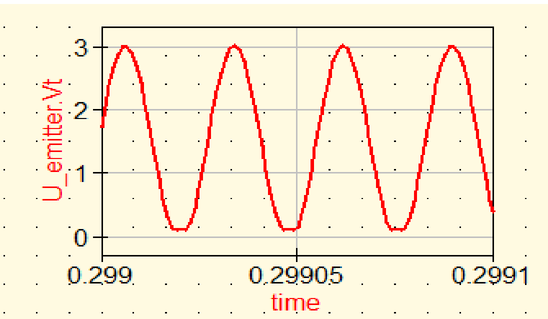
**Task:**
**Try to find out the resistor value which stops the oscillation. Have also a look at the “oscillation start delay” and the improved curve form including the spectrum of the harmonics.**

**To observe these effects use a simulation time from 0 to 300ms and increase the resistor's value from 680Ω up to 6 kΩ.**

Solution for **R6 = 6 kΩ:**




Start delay has increased to **more but 25 ms for R6 = 6 kΩ**, but the **amplitudes of the harmonics are now significantly reduced.** And the oscillator's output signal freqency can exactly be determined in the spectrum using a marker (f = 35933 Hz)



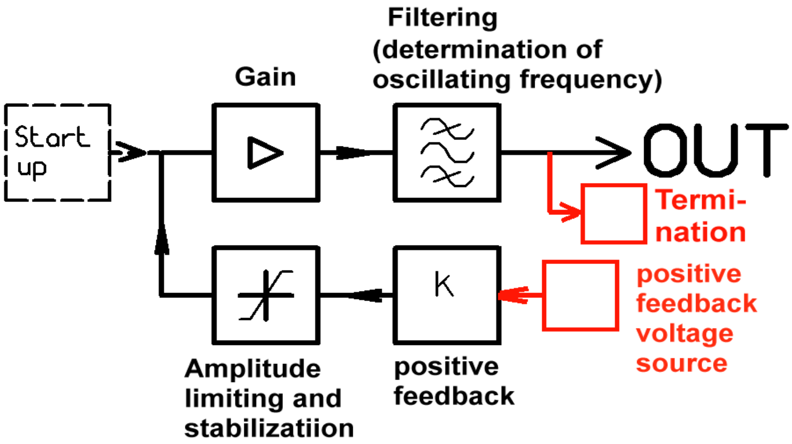

But improvements seem to be possible when regarding the emitter voltage in the Time Domain: the lower half of the curve is still damaged – we'll have a look how to cure this problem in chapter 19.4 (= amplitude limiting and stabilization)

## 19.3. Simulation in the Frequency Domain

Simulation in the Time Domain provides a lot of information about the curve form, the frequency, the harmonic content etc. But is difficult to know how far (and without any risk) the positive feedback can be reduced to improve the quality of curve form and so to reduce "overdrive effects" and harmonic distortion.

**A Simulation in der Frequency Domain provides these additional information** including the "phase lag", which is responsible for the side band noise. But how to simulate this?
We use the circuit diagram of chapter 19.1. But we cut the connection between the output and the positive

feedback channel.

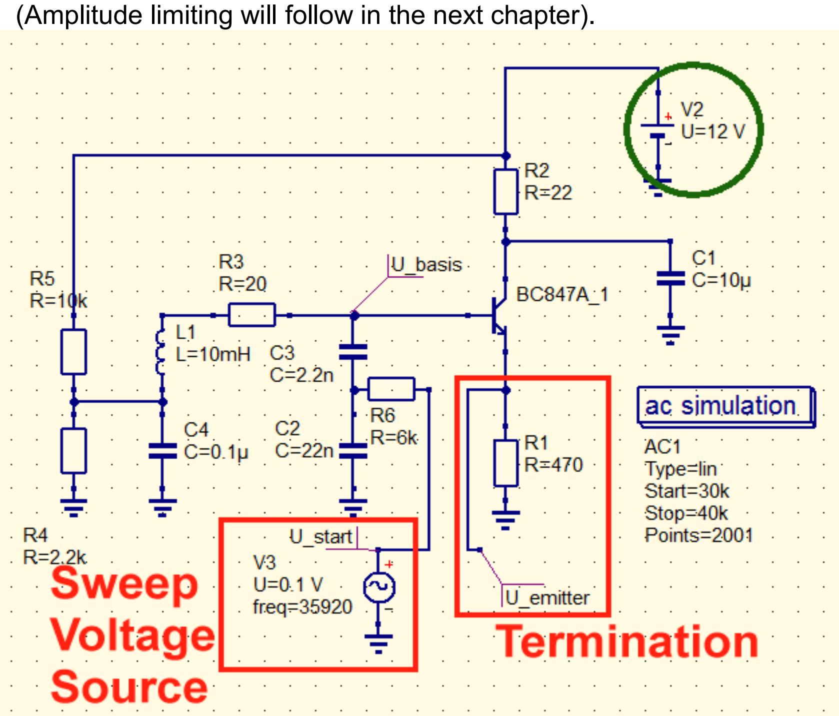



A separate voltage source serves now as "positive feedback generator" and we observe the voltage at the OUT termination (...the "termination" represents the input impedance of the positive feedback channel).

And if now the internal resistor of the voltage source complies with the impedance "seen from the OUT point in direction to the flter output", then you"ll get the correct loop gain as a result.

**The loop Gain" must be greater than 1 (= great than zero dB) and the OUT signal must be in phase with the source voltage. Then an exponentially increasing output voltage is generated.**

(Amplitude limiting will follow in the next chapter).




This is the simulation schematic for the AC-Sweep.

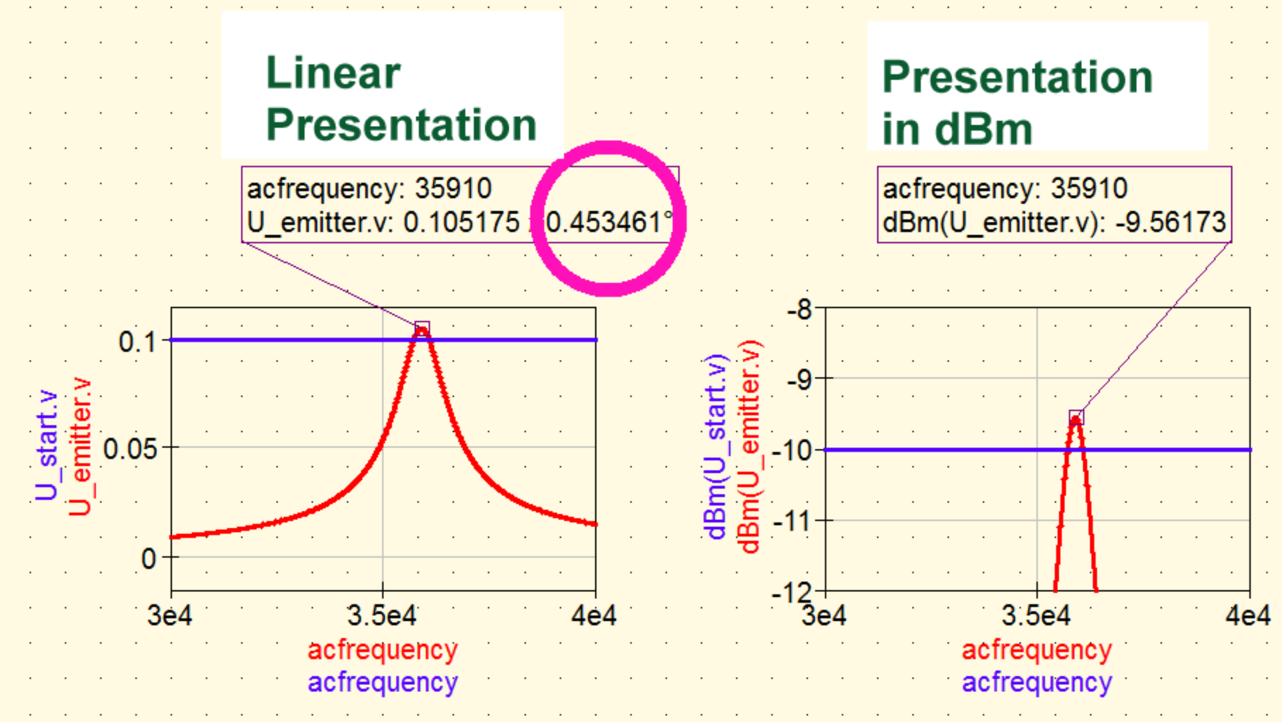
a) **The power supply voltage must be constant for an AC sweep. So we use a DC voltage of +12 V.**

b) The **termination** is a simple affair, because the internal resistor of the transistor at the emitter pin is 8 **Ω for the quiescent DC current of 3 mA. Thus R6 can be ignored in comparison to R1 = 470 Ω.**

c) The internal resistor of the transistor at its emitter (8 Ω) is – as discussed! – extremely small in comparison to R6 = 6 kΩ. Thus the internal resistor of the positive feedback voltage source can be set to zero.

A sweep from 30 kHz to 40 kHz is used. The swept amplitude of the internal voltage of the feedback voltage source is set to a value of U = 100 mV..

And this is the result:




**The Loop Gain is higher than 1 (and greater than zero dB).** This means that the **circuit will oscillate.**

Remark:

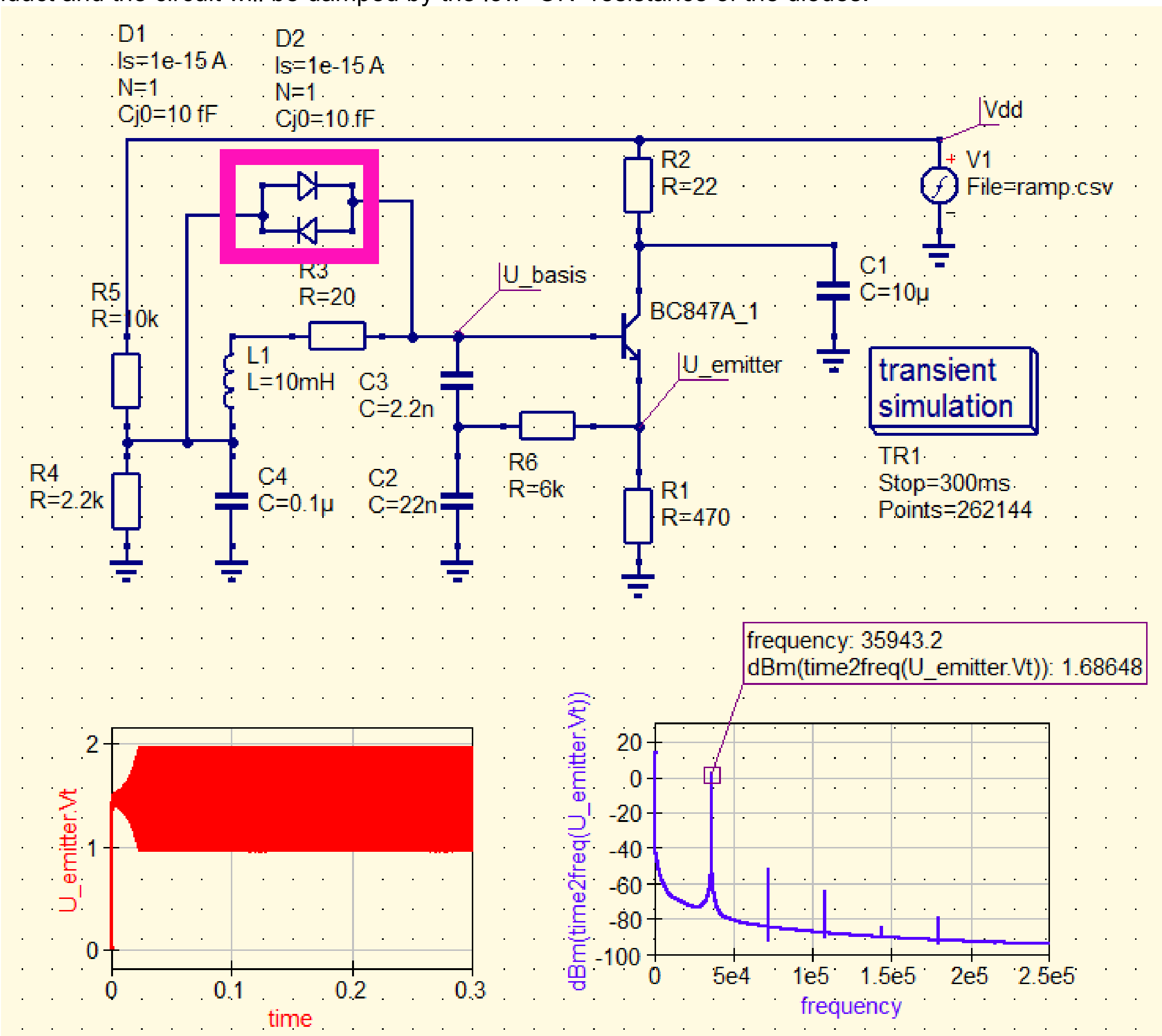
a) In the left diagram (= within the pink circle) we find the evidence that also the **necessary phase condition (feedback signal in phase with termination signal) for a oscillation is fulfilled.**

b) In the right diagram we find a **gain reserve of 0.5 dB.** This is sufficient for a sure start but does not drive the transistor into a heavy saturation.

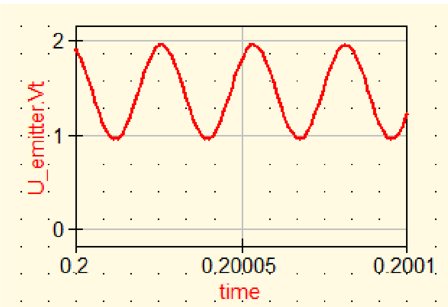
## 19.4. Amplitude Limiting and Stabilisation

Smooth and soft and symmetrical – these are the most important goals for this task. So the distortion of the sine wave is reduced AND the even harmonics are nearly suppressed. This means that saturation or an overdrive of the amplifier should be avoided.

One simple solution are two diodes in anti parallel arrangement connected to the resonant circuit. (remark: R3 must be included because this is the representation of the coil losses).
By this symmetrical manner the peak resonant circuit voltage is reduced to 0.7V, because the diodes start to conduct and the circuit will be damped by the low "ON" resistance of the diodes.




**The distortion of the sine wave shows now a massive reduction.** The "double frequency" is reduced by nearly 60 dB and the "triple frequency" even by nearly 70 dB.
The oscillating frequency has raised a little bit because the transistors capacitances are less driven by the RF voltage (...transistor capacitances are voltage dependent and increase with reduced collector to emitter voltage).




And if we now have a look at the output voltage – there is no reason for any worrying...

## 20. Developement of a Microstrip Low Pass Filter for 1700 MHz

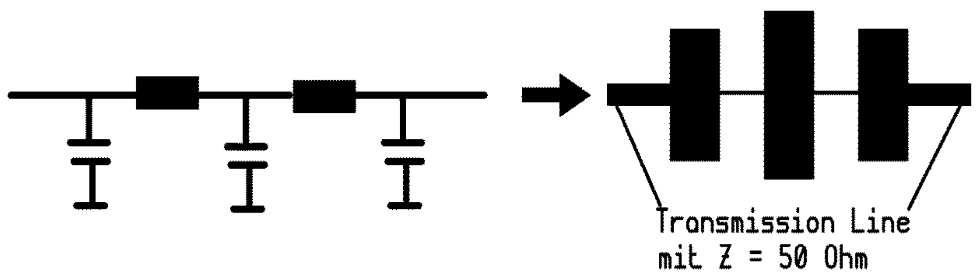
### 20.1. Remarks

If filters for frequencies above 1000 MHz are needed then microstrip line solutions are often preferred. In this case you have to use a double sided copper plated low loss PCB. On the upper side you find the filter structure, the lower side can be regarded as an infinite and perfectly conducting ground plane.
Advantages are the the high accuracy in a series production, no necessarity of align, no mounting of discrete parts etc. Disadvantages are the increasing structure dimensions when decreasing the design frequency.

Low loss material is an absolute demand. Of proven quality are the Rogers product, but do Teflon dielectric only use at very high frequencies or for its extremely low loss property. The handling of Teflon is very difficult due to it's bad mechanical properties ( = like chewing gum) – and the price is high.
So please prefer material of the „RO4000“ serie – it is much cheaper, the loss tangent is only 10...30% higher but the mechanical handling is perfect (high mechanical stabilibility like the glass woven and epoxy reinforced and well known material named FR4).
But pay attention: FR4 is once more cheaper and in common use, but the losses are higher (the quality factor has only a value of 0.02 in comparison to 0.002 for Rogers RO4003) and for frequencies higher than 1500 MHz everything is getting worse: the loss tangent factor increases rapidly, the dielectric constant decreases....not good for high quality circuits!

### 20.2. Design Procedure for a Microstrip Low Pass Filter

A lot of proposals and design procedure are existing. But most of them use complicated mathematical transformations which can often not be understood easily.




**Thus we use another and simple way:**
At first the low pass filter is designed as an LC circuit using an Online Filter Calculator (and as „Pi“ - circuit). Afterwards we replace the cpacitors and inductors by microstrip line pieces.

(Short line pieces with a great width represent capacitors, very small line pieces serve as inductors).

**But:**
**for a correct filter function the electrical length of the line pieces must be between 10 and 30 degrees referred to the wavelength at the cutoff frequency for the filter (….starting at 45 degrees such line pieces start to transform impedances and thus the filter properties differ more and more from the desired values).**

To avoid wave guide properties of the PCB (= higher order waves must not be possible which cause unexpected attenuations in the passband) reduce the PCB thickness as much as possible. Normally in a microstrip line only TEM waves should exist but if you ignore this recommendation you can get waves with electric field parts in direction of the energy propagation. These wave parts would be short circuited by the center conductor of an SMA socket at the output and so energy would be missing. Then you get the „holes“ in the passband.
In practice a thickness of 32 MIL (0.813mm) is good up to 2 ...3 GHz and at 10 GHz a thickness of 10 MIL (= 0.25 mm) is in general use.

## 20.3. Filter Specifications

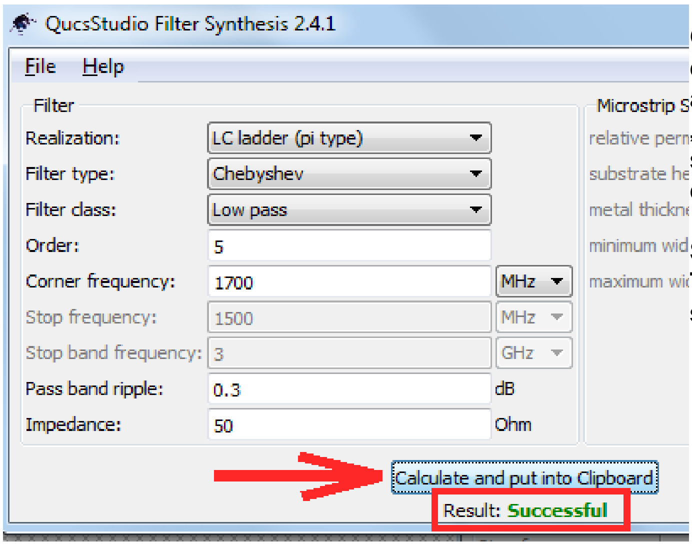
A frequency of **1500 MHz must not be attenuated, but then the attenuation should increase rapidly.** So we define these specifications:

| | |
|---|---|
| **Filter order:** | **n = 5** |
| **Filter type:** | **Chebychev version, Pi type** |
| **Maximum Pass Band Ripple:** | **0.3 dB** |
| **3dB Corner Frequency:** | **ca. 1.7 GHz** |
| **Characteristic System Impedance:** | **Z = 50 Ω (symmetrical filter version)** |

PCB material: **Rogers RO4003, coated with copper on both sides** with the following properties

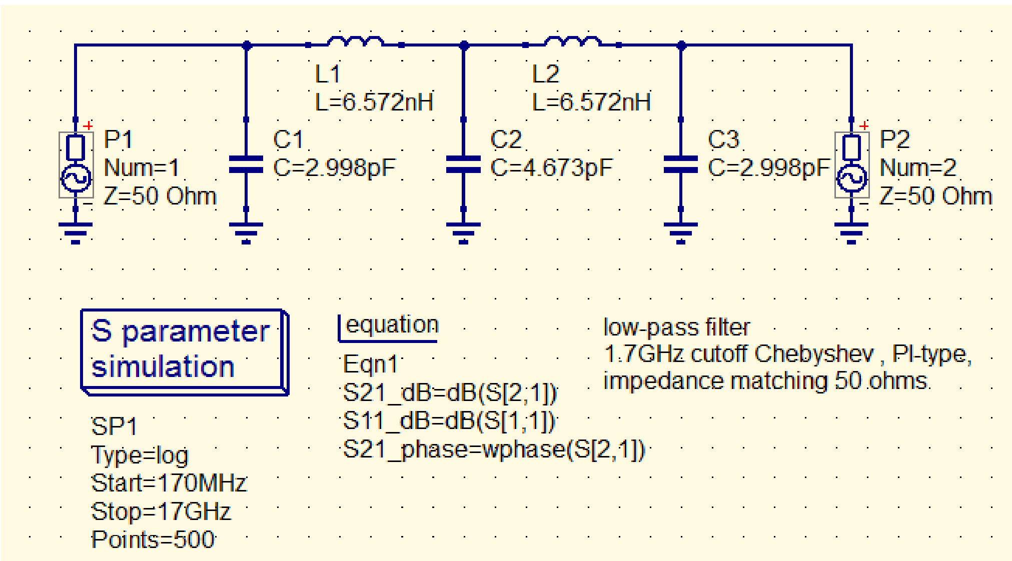
| | |
|---|---|
| **Substrate Name:** | **= RO4003** |
| **PCB Height „H“** | **= 32 mil (= 0.813 mm)** |
| **Dielectric Constant εr** | **= 3.55** |
| **Loss Tangent TAND** | **= 0.002 at f = 2 GHz** |
| **Cover Height HU** | **= 13 mm** |
| **Copper Plating Thickness (top resp. bottom cover)** | **= 1.3 mil = „1 oz“ (= 35µm)** |
| **Surface Roughness** | **= 2.5 µm** |

## 20.4. qucsstudio Filter Calculator Work

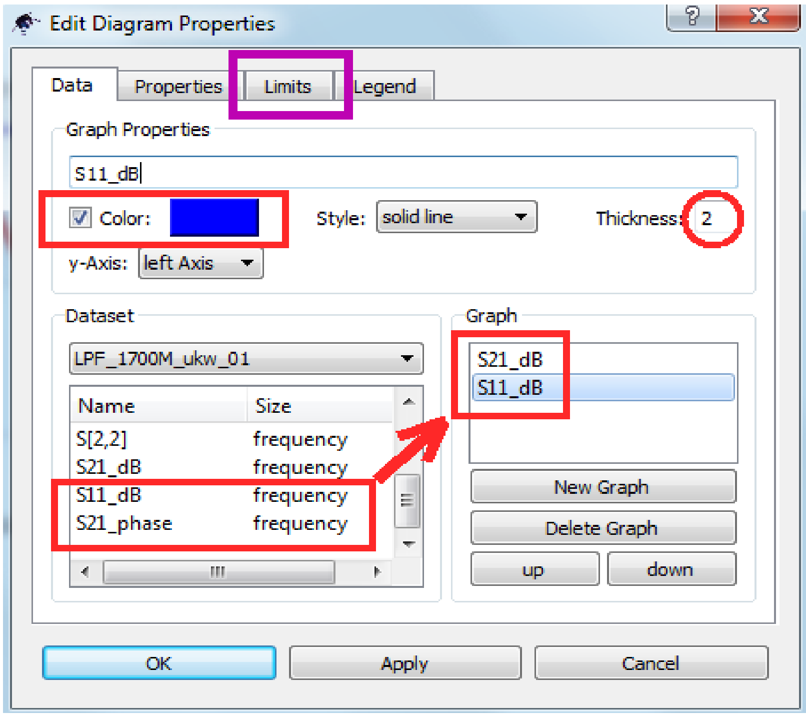



Open „**Tools**“ and „**Filter Synthesis**“, then choose „**LC Ladder (pi type)**“. Enter the filter data and you should get this illustration. After pressing „**Calculate and put into Clipboard**“ you'll see a short message „**Successful**“ and now you can close this menu.

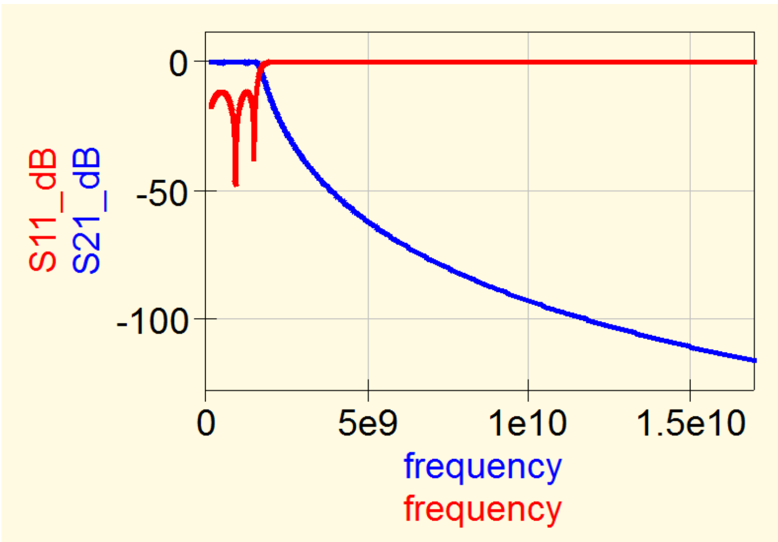
**Short information:**
**This message will already disappear after 3 seconds....**




**Use <Control> + <v>** to insert this schematic, then simulate by pressing the „gear wheel symbol“ in the upper right corner of the screen.

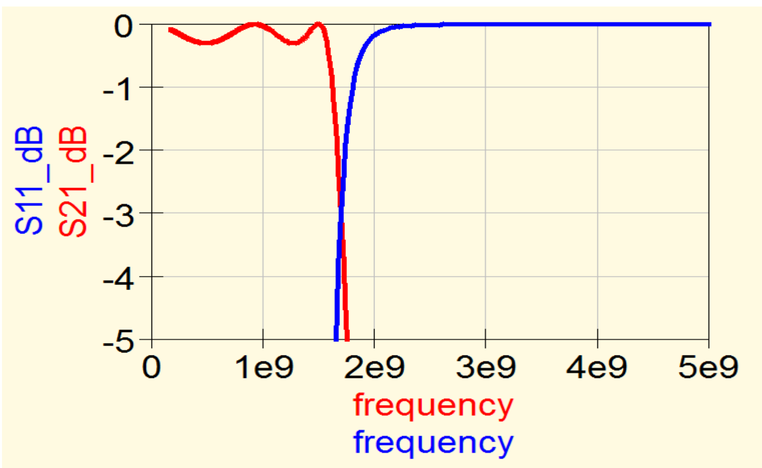



Use a cartesian diagram to present the results. The diagram is fixed after a left mouse click at the cursor and when placing it on the screen the property menu opens automatically.
There add **„S11_dB“ and „S21_dB**“ to the Graph list (using „New Graph”) and set curve colour and line width. Use „Apply“ to prove your choice and terminate with „OK“.




You get this result for a frequency range up to 15 GHz.

**This is a very coarse solution if you would like to see the „Chebychev Ripple“.**
**Thus right click on the diagram, followed by „Edit Properties“ and „Limits“. Now set the horizontal scaling to “Zero up to 5 GHz with a tick of 1 GHz” and the vertical scaling to “-5 dB up to zero dB with a tick of 1 dB”.**

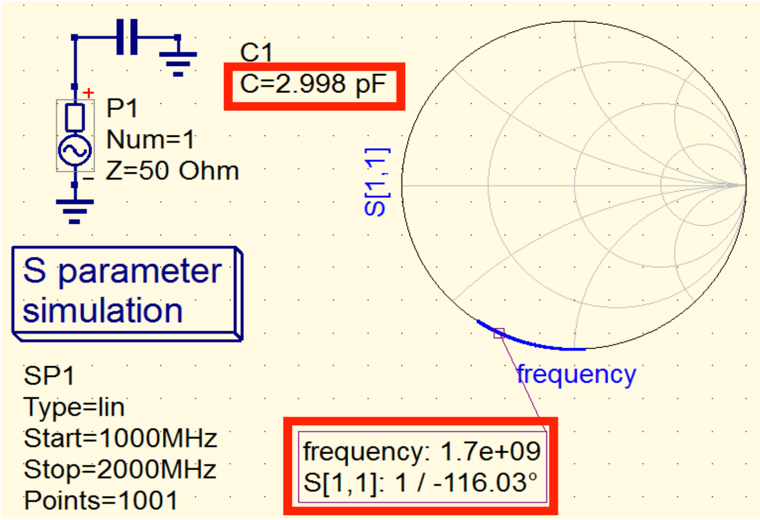



Very nice, isn't it?

## 20.5. Microstrip Lines as Capacitors or Inductors

### 20.5.1. Replacing Capacitors

Due to the schematic in the last chapter we need **two capacitors with 2.998pF and one capacitor with 4.673 pF.**




**At first we use qucsstudio and a smith chart to calculate S11 at f = 1700 MHz for every capacitor.**
We draw this schematic, start with C = 2.998 pF, simulate and apply a frequency marker to determine the correct value of S11 at the desired frequency (1700 MHz).

But please change the marker's properties to „**5 digits**“ and „**magnitude / angle (degree)**“. Now you can change the frequency in steps of 1 MHz using the horizontal arrow keys. The result ist
**S11 = 1 / -116.03 Grad for 2.998 pF**
and
**S11 = 1 / -136,33 Grad for 4.673 pF**

Now we need a similar schematic with an open ended microstrip line replacing the capacitor. But be aware of the following developer's experience:

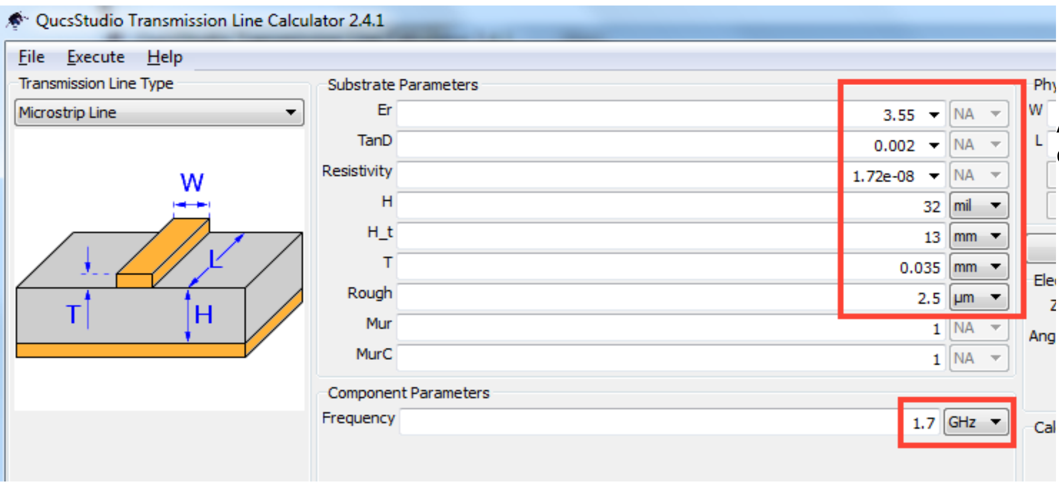
**The electric length of the microstrip line piece must be between 10 and 30 degrees (at the corner frequency).**

**Its chacteristic impedance should be 4....5 times smaller than the characteristic impedance of the feed line with 50Ω. This will increase the line width.**

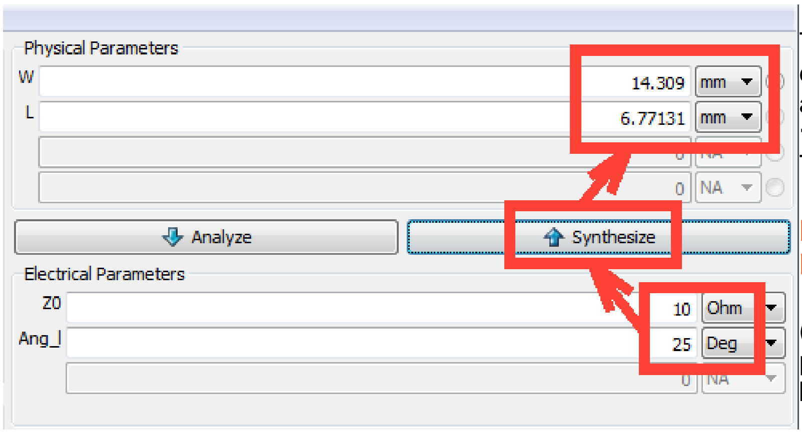
**The ratio of line width to line length must be between 1:1 and 5:1.**

We start the Line Calculator (found in „Tools“) and enter the following PCB data in the left half of the menu:

| | |
|---|---|
| **Dielectric Constant Er** | **= 3,55** |
| **PCB Thickness„H“** | **= 32 mil (= 0.813 mm)** |
| **Loss Tangent TAND** | **= 0.002 at 2 GHz** |
| **Cover Height “HU”** | **= 13 mm** |
| **Copper Plating on both sides:** | **= 1.3mil = „1oz“ (= thickness of 35µm)** |
| **Surface Roughness** | **= 2.5µm** |



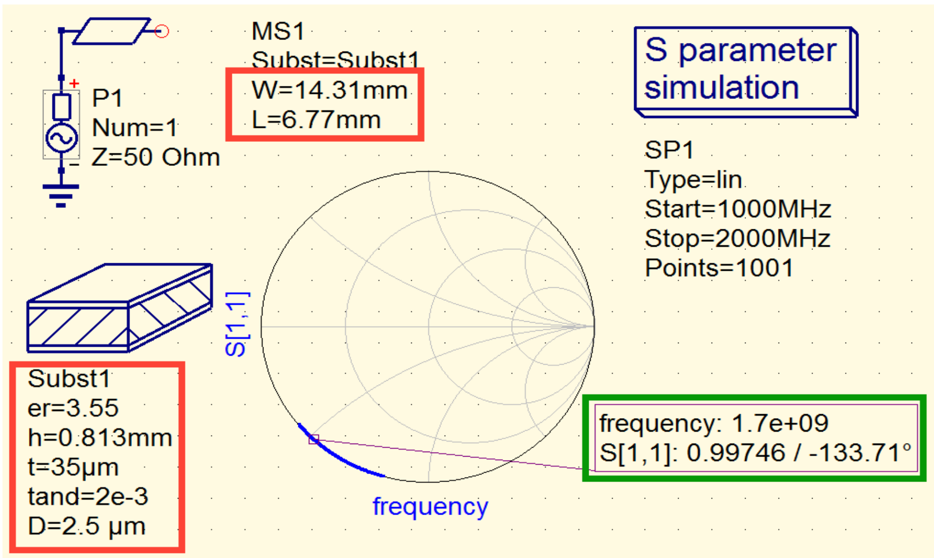

A design frequency of **1.7 GHz** is used.




Then we enter (as discussed) a characteristic impedance **Z = 10 Ω and an electric length of 25 degreees at 1700 MHz.**
The simulation result is a

**Line Width of 14.31mm and a Line Length of 6.77 mm.**

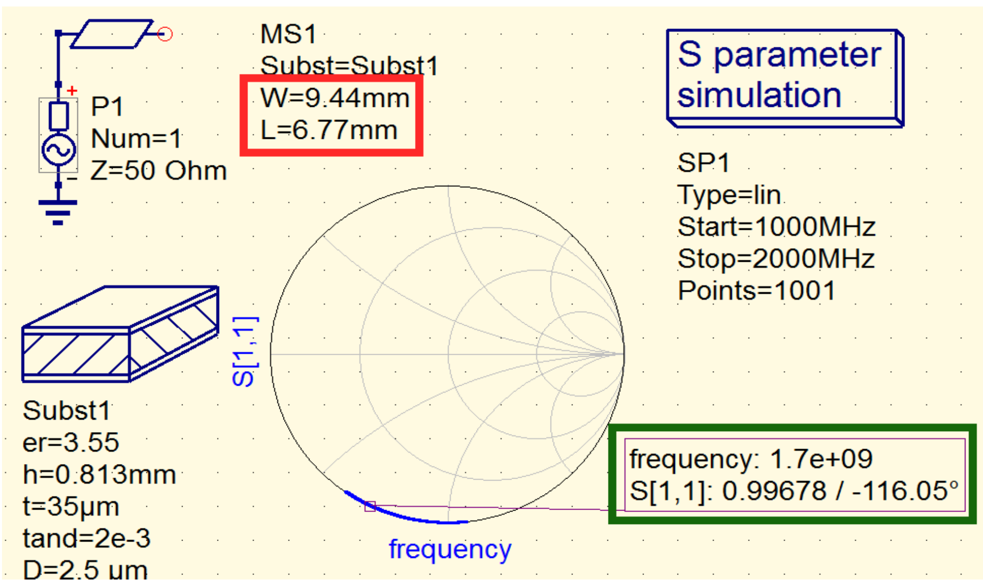
(...do not use more than 2 post comma positions...PCB production accuracy is limited...).




This is the new simulation schematic using an open ended mcirostip line instead of a true capacitor.
The sweep runs from 1...2 GHz with a tick of 1 MHz.

The microstrip line can be found in „**components / transmission lines**“, but we need also the „**Substrate**“ - Block to enter the PCB properties as given in the specifications.
The S11 simulation result is marked in green and meets better the conditions for the capacitor with **4.73 pF.**

Thus we let the length of 6.77 mm unchanged and vary only the width to realize an S11 value of “**1 / 136.33 degrees**”.
With a **Width of 15.35 mm and a length of 6.77 mm for C = 4.673 pF we hit the target**.




Now we repeat the procedure, let the length of 6.77 mm unchanged and vary only the width to replace the capacitor with **2.998 pF**.
For the demanded S11 value of

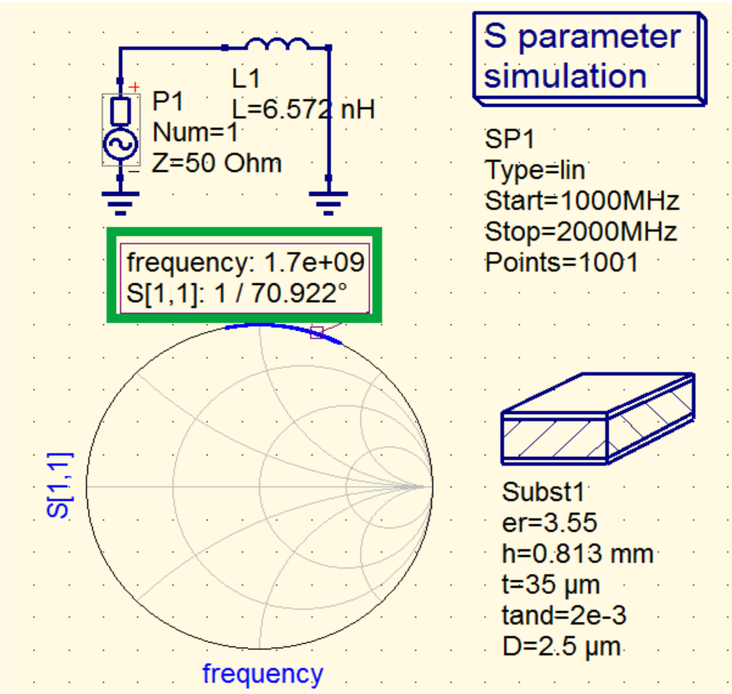
**1 / 116.03 degrees at 1700 MHz**
we find a line piece with the dimensions

**width = 9.44 mm**

**length = 6.77 mm**

as solution.

## 20.5.2. Replacing the Inductors




We start an S11 simulation for an inductor with **L = 6.572 nH.** The result is

**S11 = „1 / +70.922 Grad" at 1700 MHz.**

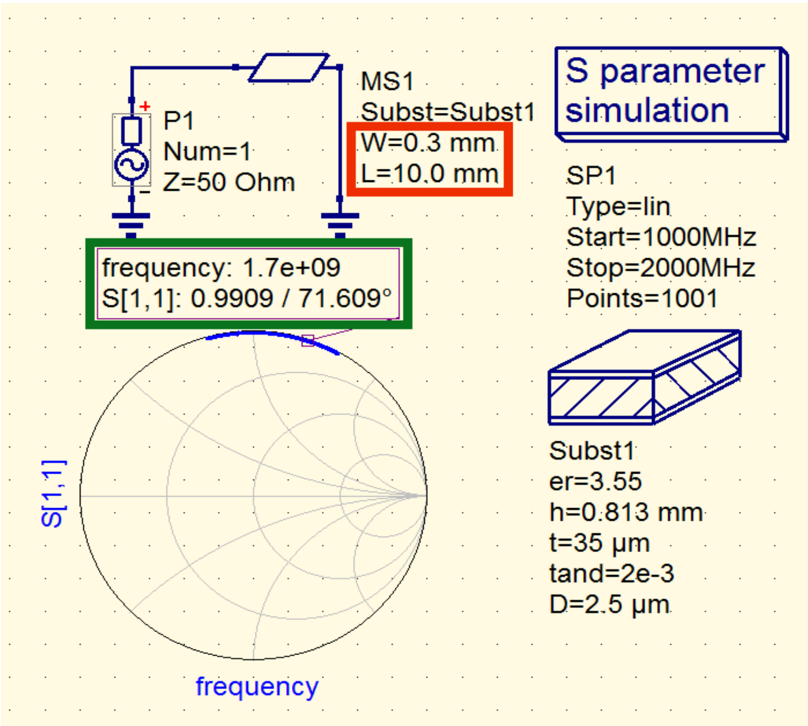
To replace this inductor we use a microstrip line piece which is

**short circuited at its end and**

**its width must be much smaller than the width of the feed line with Z = 50Ω and**

**the electrical length must be between 10...30 degrees at the corner frequency**

**Advice:**
**Do not use a line width below 0.25...0.3mm. Below this width value the losses increase and the PCB manufacturer worries about heavy manufacturing difficulties..**




The first simulation trial runs with a **line width of 0.3 mm and a length of 10mm.**

The simulation says that S11 has a value of
**S11 = 1 / +71,609 degrees**
and it seems that the line is too short (the correct S11 phase value would be 70.922 degrees).

**Thus we need a line length of 10.1 mm and a width of 0.3mm**

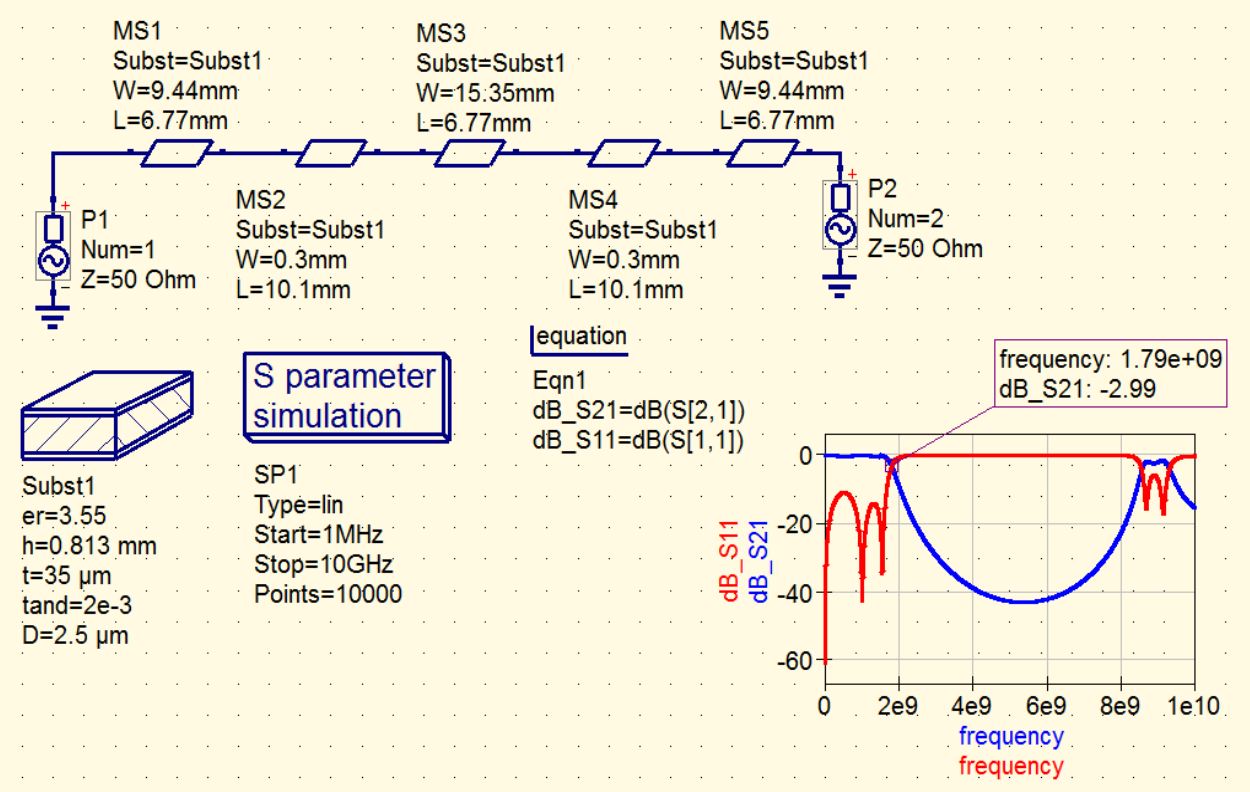
**This gives**
**S11=1 / 70.932 degrees**
as demanded...

Now we can simulate the complete microstrip line filter structure.

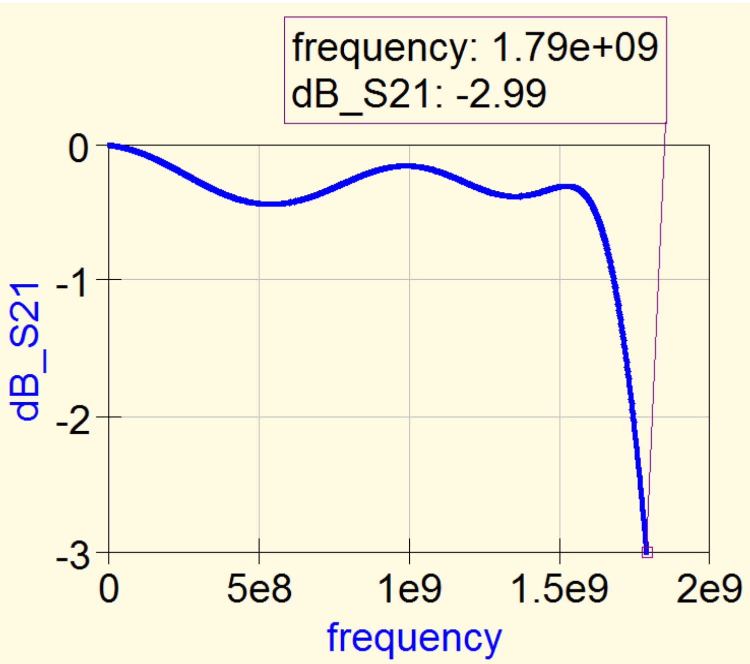
## 20.6. The complete Microstrip Filter

### 20.6.1. Optimization of the Design

That is an easy task because we have simply to connect all line pieces in the correct order and add two ports. Then we simulate from 0 up to 10 GHz.



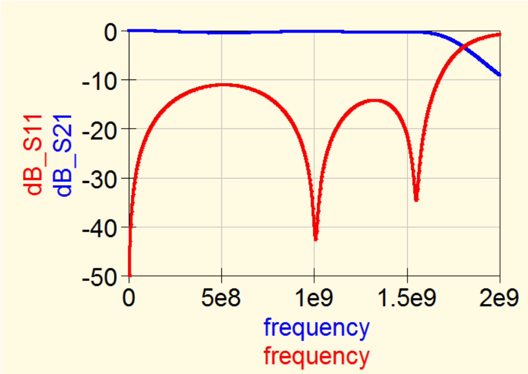

**But the Stop Band Attenuation does not increase "monotone up to the infinite" like filters consisting of capacitors and coils do. Lines are lines and with increasing frequency they start to transform impedances.**
Thus the stop band attenuation shows a maximum value (= S21 shows an minimum value of -42 dB) in the frequency response and afterwards the attenuation decreases to values below 10 dB...
Developer's experience says that typical minimum S21 values of -40 dB give "good microstriop filters...




If we use the Zoom Function for the frequency range from 0.....2 GHz and S21 values from 0....-3,dB we can admire the Chebychev ripple.

But:
**The ripple shows different amplitude values and the corner frequency has raised up to 1.79 GHz!**

The different ripple values are caused by different S11 (= reflection) values in the passband.
Thus we change the diagram scaling and add the S11 curve.




Now we can see the reason: the height of the two S11 “hills” is different but the correction is easy:

**Reduce the width of the “center capacitor” from 15.35 mm to 14.3mm**

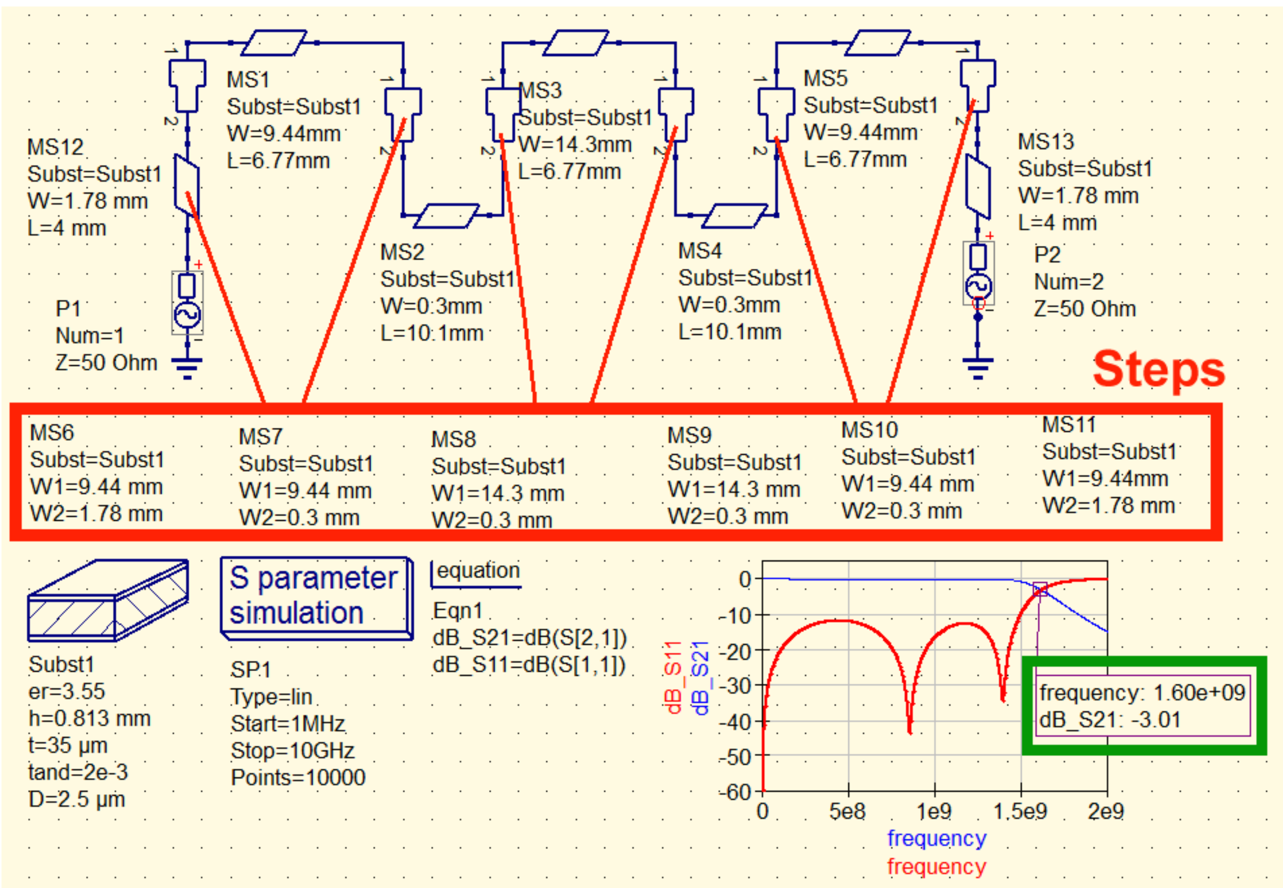
But if you believe that a simple reduction of the corner frequency from 1.79 GHz to 1.7 GHz will finish the job and this is the end.....wait still a moment...

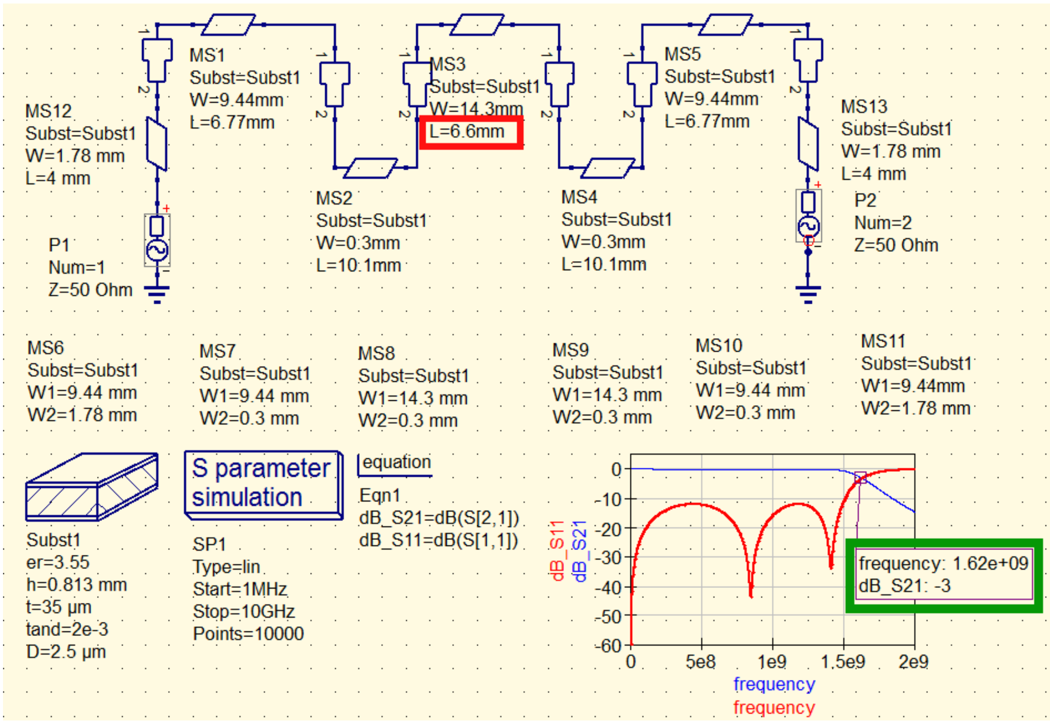
## 20.6.2. Realistic Circuit with Steps

If two microstrip lines with different widthes are connected together you get an effect named “**Step**”. The current which flows in the “broad line” is pressed together when entering the small line. This effects equates to an additional inductor. An when leaving the small line and entering the broad line it takes some time to expand again the current pathes to fulfill the complete width of the broad line. This gives “empty edges which hang beside the current way” – and this means an additonal capacitor!

**In the qucsstudio line library you find the necessary correction part (= step model) and you have to enter the PCB properties and the widthes of the two “lines in collision”**

But do not forget that the filter structure needs also a **50 Ω feed line at every side as connection to the used SMA sockets – and this gives additional steps....**
Use the filter calculator to simulate the correct width of the 50 Ω feed line (**result is 1.78 mm**) and use a length of 4 mm in the simulation. Be surprised by the simulation result...



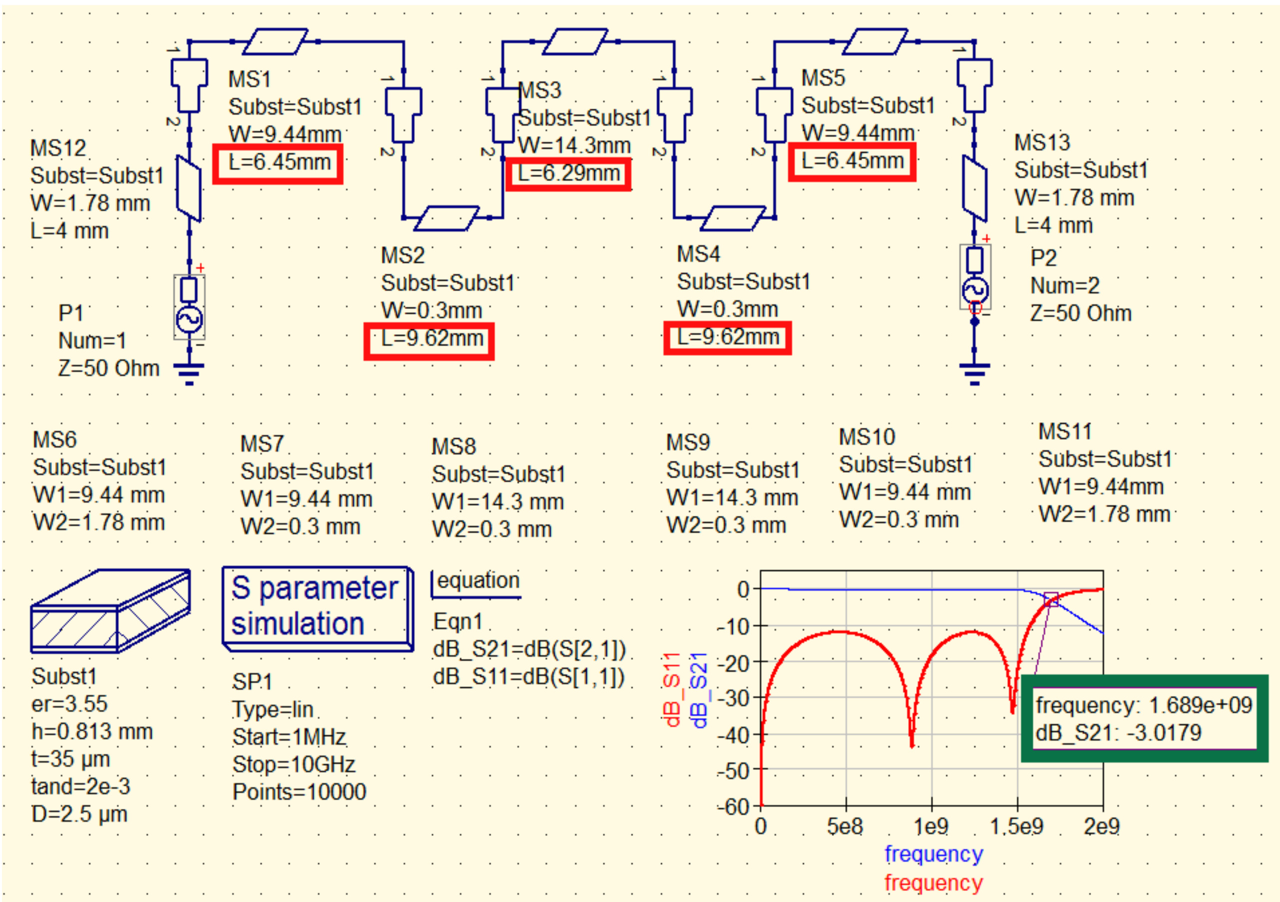



The corner frequency dropped to 1600 MHz and the S11 “hills” show again different heights.

The S11 curve can be corrected by a length reduction of the center microstrip line (...do NOT vary its width, otherwise you must also correct the adjacent steps...).

**Reduce the length from 6.77 mm to 6.6 mm and the S11 problem is eliminated.**

Now we must correct the simulated corner frequency and this is simple: **shorten all length values by the factor 1620 / 1700 = 0.953**




This is a fine result. Here you should stop every variation and simulation and continue with a PCB prototype.

## 20.7. The Filter Prototype

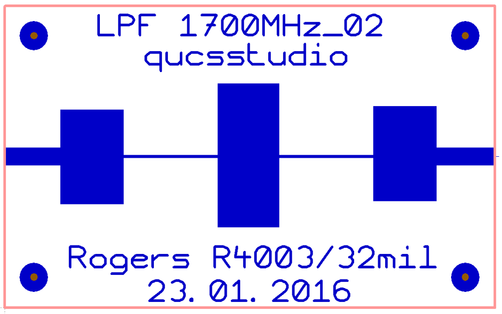
**Information:**

a) The circuit uses two **„50Ω -Feed Lines"** with a width of **1.78 mm**.

b) The LPF starts and ends with capacitors of **2.998 pF**. These are replaced by microstrip line pieces (**width = 9.44 mm / length = 6.45 mm**)

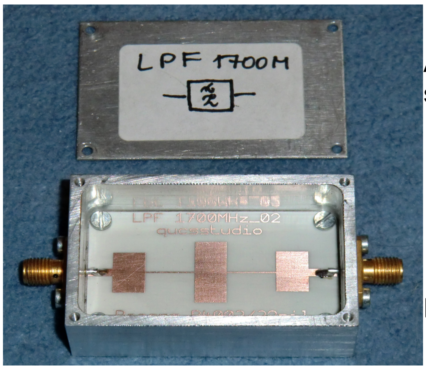
c) The inductors are also realized by line pieces (**width = 0.3 mm / length = 9.62 mm**)

d) The "central capacitor with **4.673 pF"** is a line piece with a width of **14.3 mm and a length of 6.29 mm**



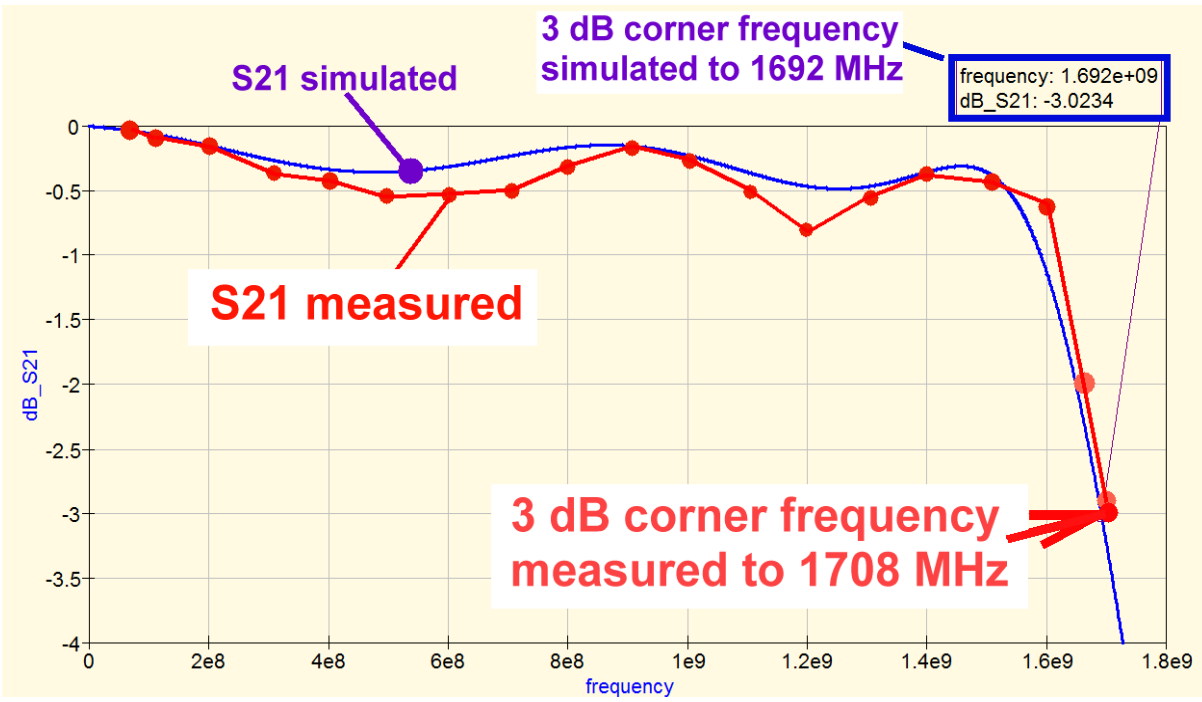

This is the PCB.

**(Dimensions = 30 mm x 50 mm. Dielectric is Rogers RO4003, thickness = 32 mil = 0.813 mm dick, copper cladded on both sides with a thickness of 35 µm).**

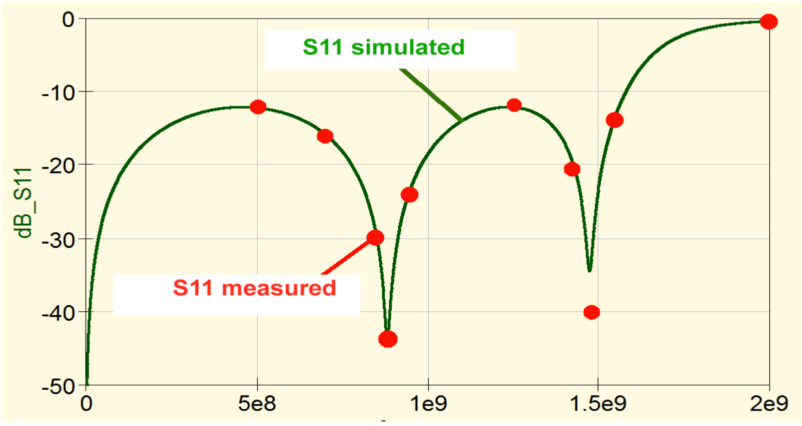



And this is the final product using a milled alumina case and SMA sockets

Now the test results. At first the S21 curve in the pass band:

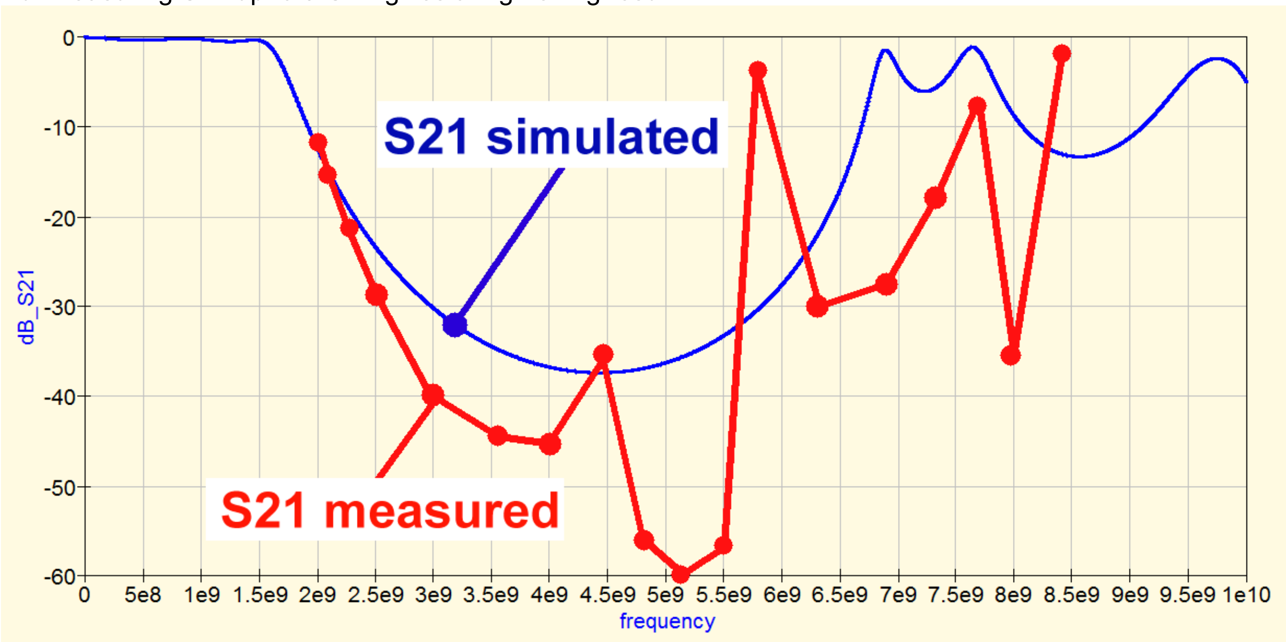


**The corner frequency is OK but the passband attenuation due to the losses is a little bit higher.**




The S11 reflection curve is as desired

But measuring S21 up to 8 GHz gives a frightening result:

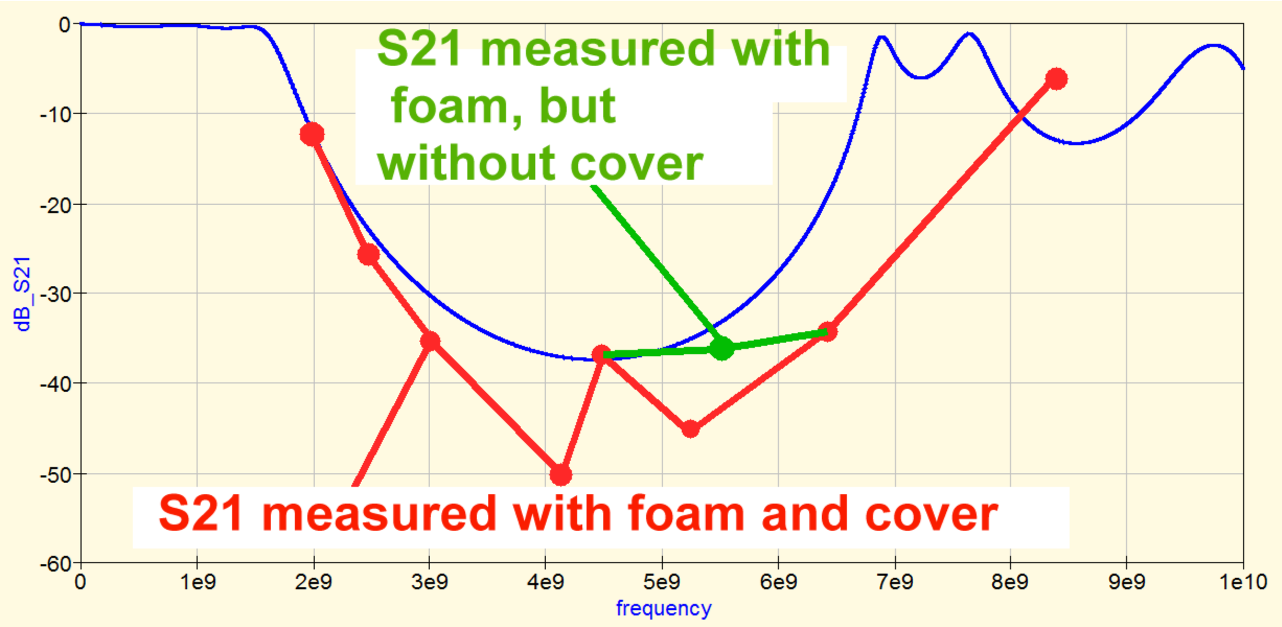



**Several years of experience in the microwave range deliver the solution of the effect: .The alumina case with cover is a CAVITY !**

To eliminate this effect use "**conducting foam = damping material for high frequencies". Fill the room above the PCB with this material but leave some millimeters of free space over the PCB surface!**

In this case and for a quick test the cheap foam material for IC packages was used.

Now repeating the S21 measurement will tell us the success.




The result is convenient but not perfect at all. It seems that **the alumina cover is still "looking through the foam"** and the application of special microwave damping material would cure the problem.

If this is successful we'll find a stop band attenuation of nearly 40 dB up to 6 GHz. Not so bad...

# 21. A Microstrip Band Pass Filter for GPS (f = 1575 MHz)

## 21.1. Information and Specifications

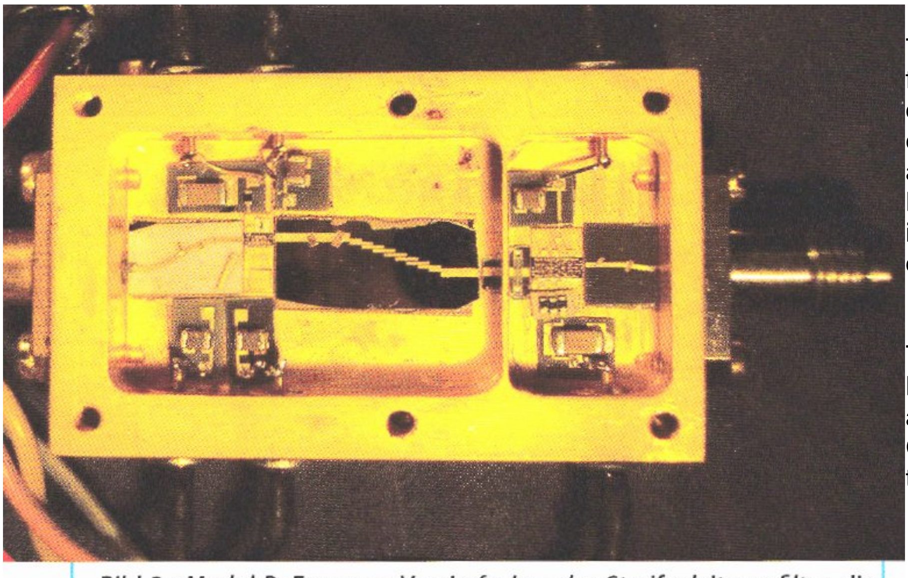


*Bild 2 : Modul B: Frequenz-Vervierfacher, das Streifenleitungsfilter, die Durchführung in die kleine Gehäusekammer, das 3 dB-Dämpfungsglied und schließlich das Verstärker-MMIC*

This filter type is in use for freqencies above 1000 MHz due to the simple an cheap construction and the high accuracy also in greater series. Dimensions decrease with increasing frequency and dielectric constant.

The illustration shows such a BPF in a frequency multiplier for an output frequency of 61 GHz. Compare the filter dimensions to the SMA sockets..

### Specifications for f = 1575 MHz:

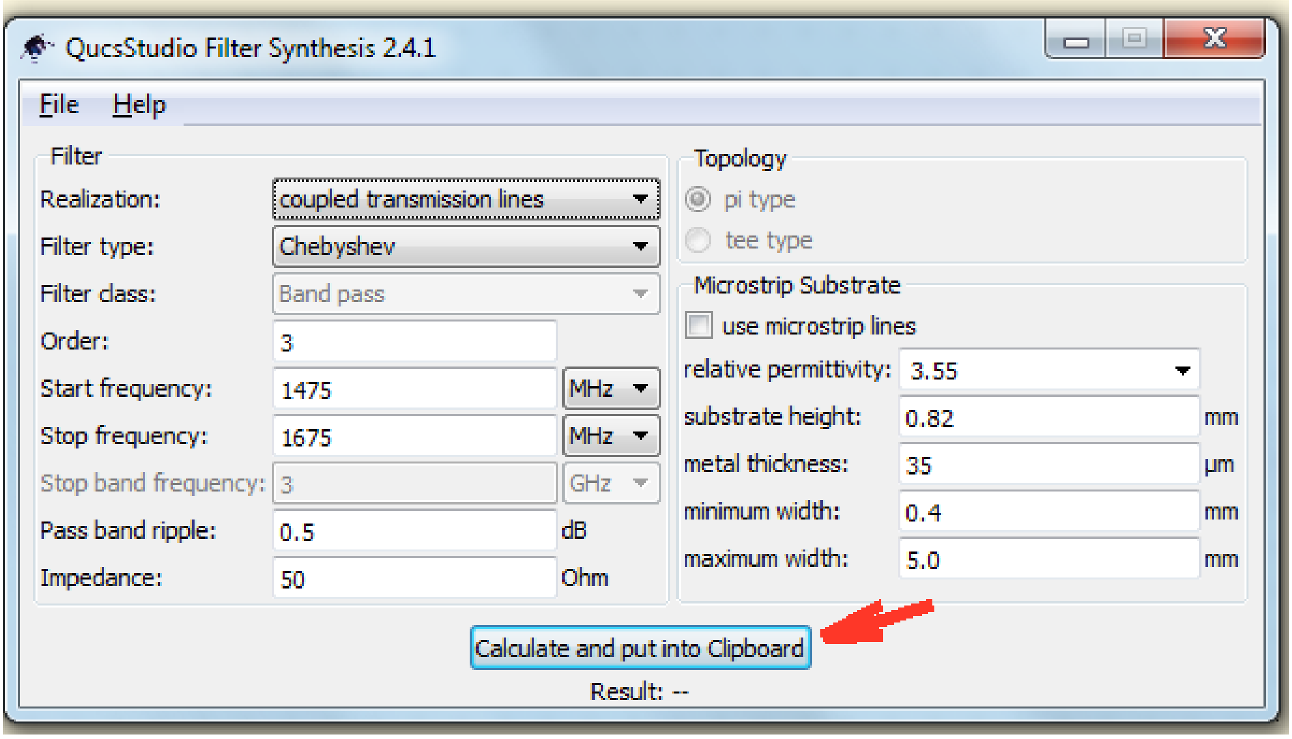
| | |
|---|---|
| **Filter Type:** | **Chebychev** |
| **Characteristic Impedance:** | **Z = 50 Ohm** |
| **Pass Band Ripple :** | **0.5 dB** |
| **Filter Order:** | **n = 3** |
| **Center Frequency:** | **1575 MHz** |
| **Bandwidth:** | **200 MHz** |

### Printed Circuit Board:

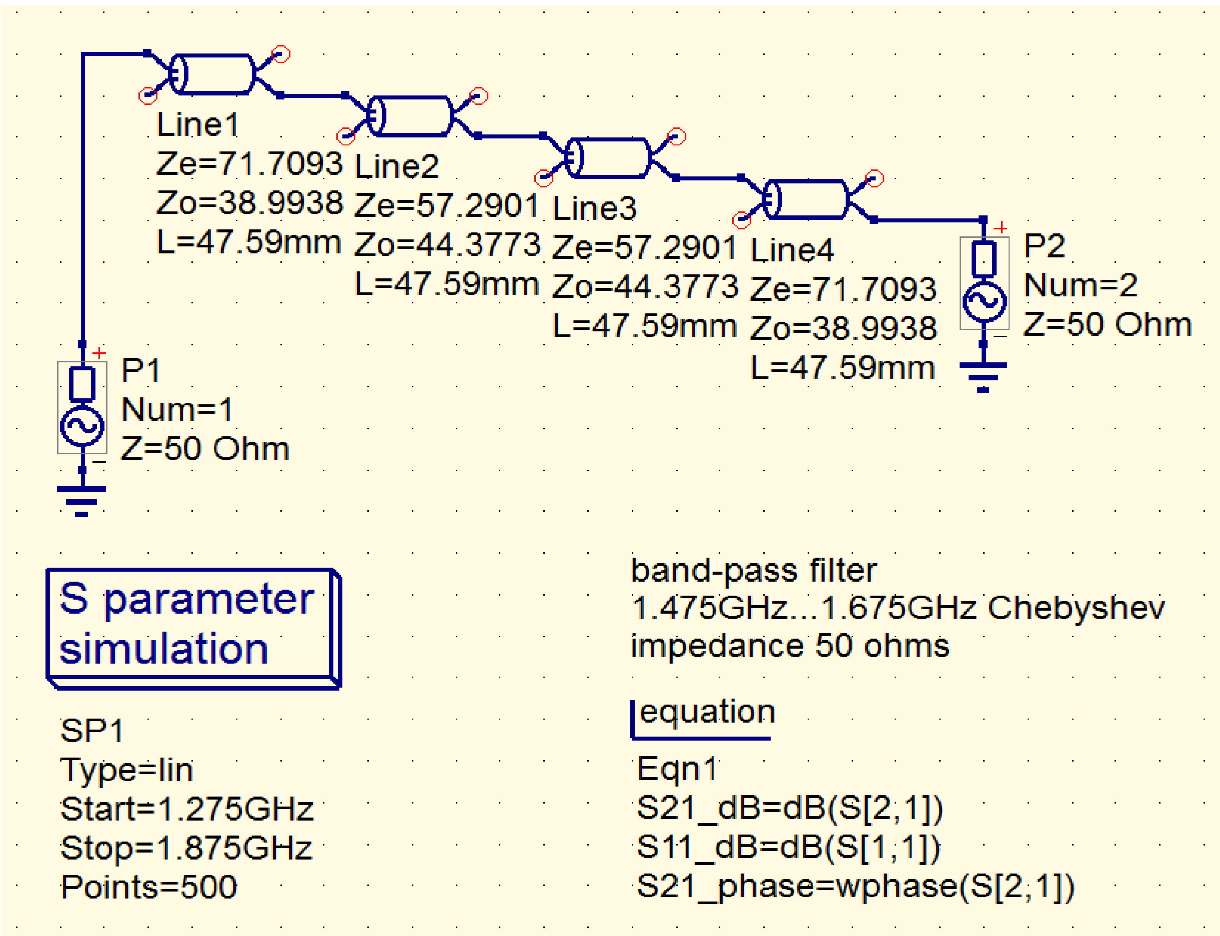
| | |
|---|---|
| **Material:** | **Rogers R04003** |
| **Thickness:** | **32 mil (= 0.813 mm)** |
| **Dielectric Constant:** | **3.55** |
| **Loss Tangent tand:** | **0.002 at f = 2 GHz** |
| **Copper Cladding on both sides** | **35 µm (= 1.35 mil = 1oz)** |
| **Surface Roughness:** | **1 µm** |
| **Cover Height:** | **13 mm** |

## 21.2 Filter Design using the qucsstudio Filter Calculator

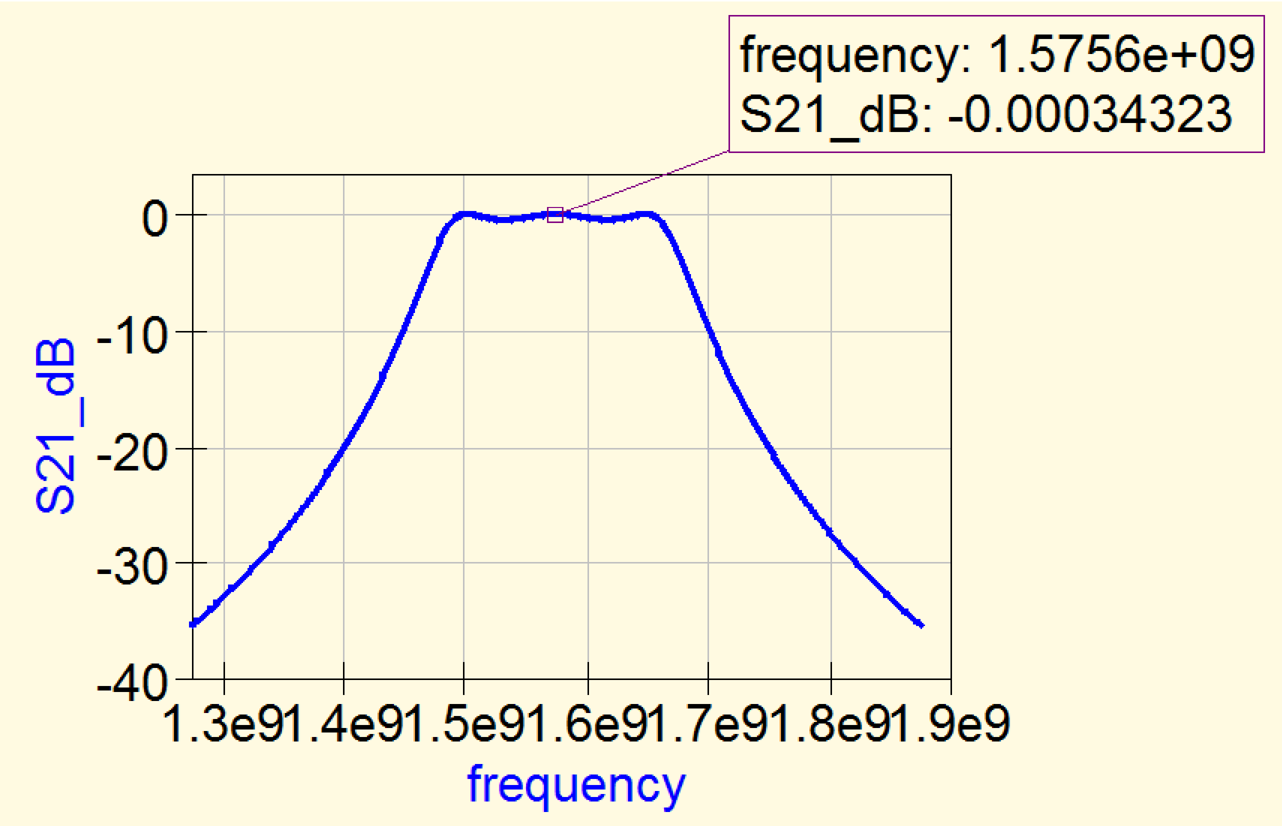
Press „**Tools**“ , then „**Filter Synthesis**“ for the following entries:




After pressing „**calculate**“ you get at once the message „**successful**“ and the result is copied to the clipboard. Close the menu and past the already prepared simulation schematic using **<CTRL> + <V>.**



This should be the simulation result:

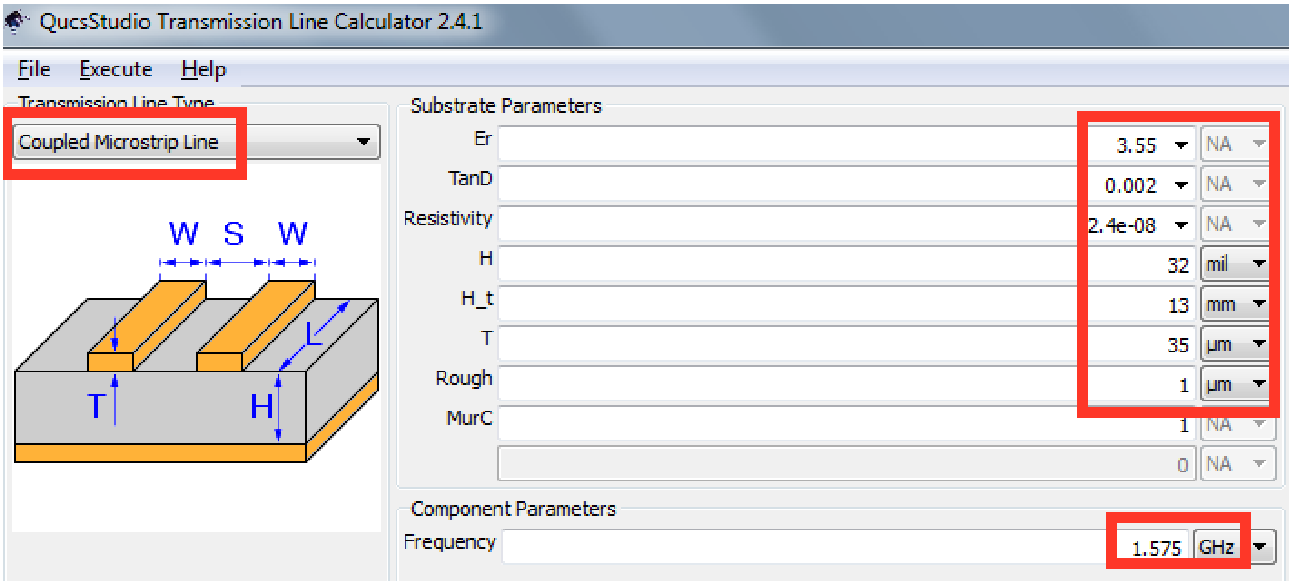


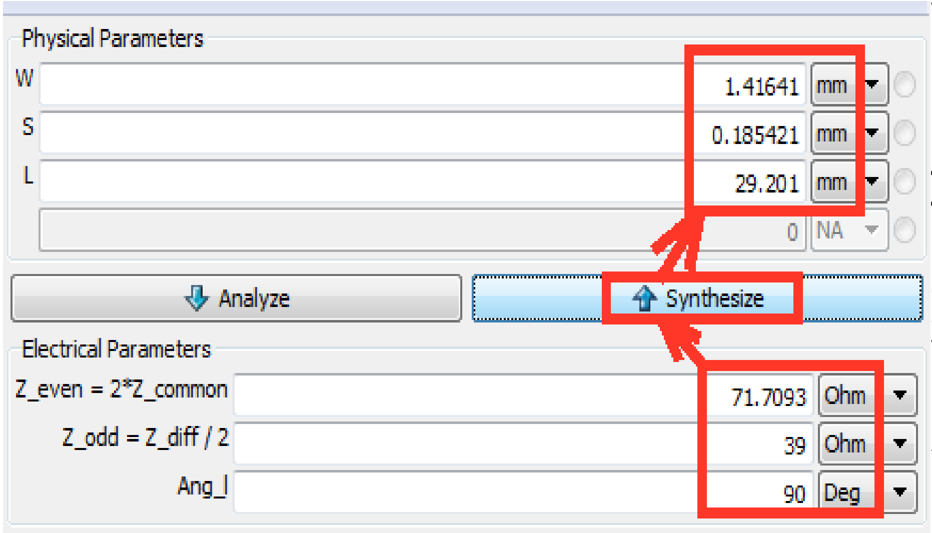

This looks very fine....so the schematic can now be transformed into a coupled microstrip line version.

## 21.3. Calculating the Line Properties using the Line Calculator

This starts always with the entries in the left half of the menu:

**Line Type / PCB Properties / Center Frequency:**






We enter the characteristic impedances of the first and last coupled line in the right half of the menu:

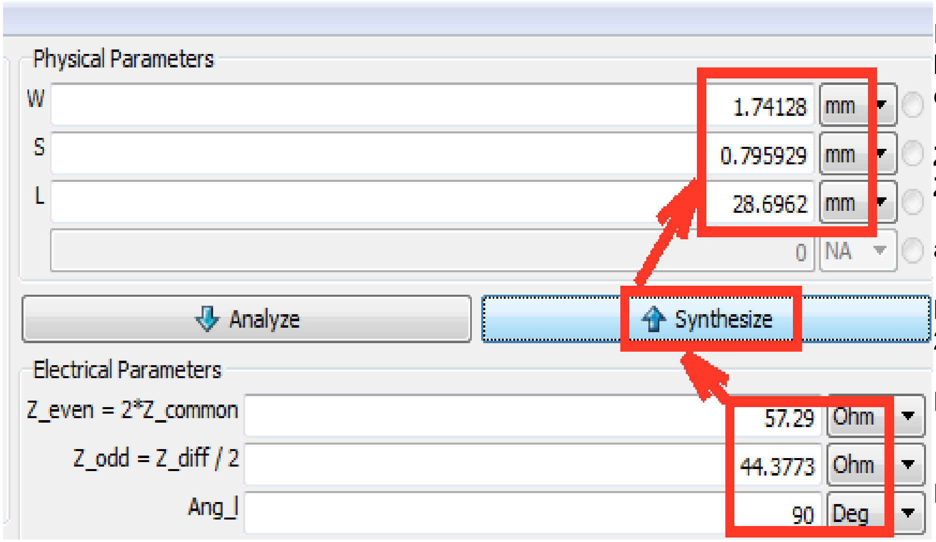
**Zeven = 71.7093 Ohm**
**Zodd = 39 Ohm**

including an **electrical length of 90 degrees at a frequency of 1575 MHz.**
This gives a

**Mechanical Line Length = 29.2 mm**

**Line Width = 1.42 mm**

**Line Separation = 0,19 mm**




For the two identical coupled line pairs at the center we enter

**Zeven = 57.29 Ohm**
**Zodd = 44.3773 Ohm**

and find a

**Mechanical Line Length = 28.7mm**

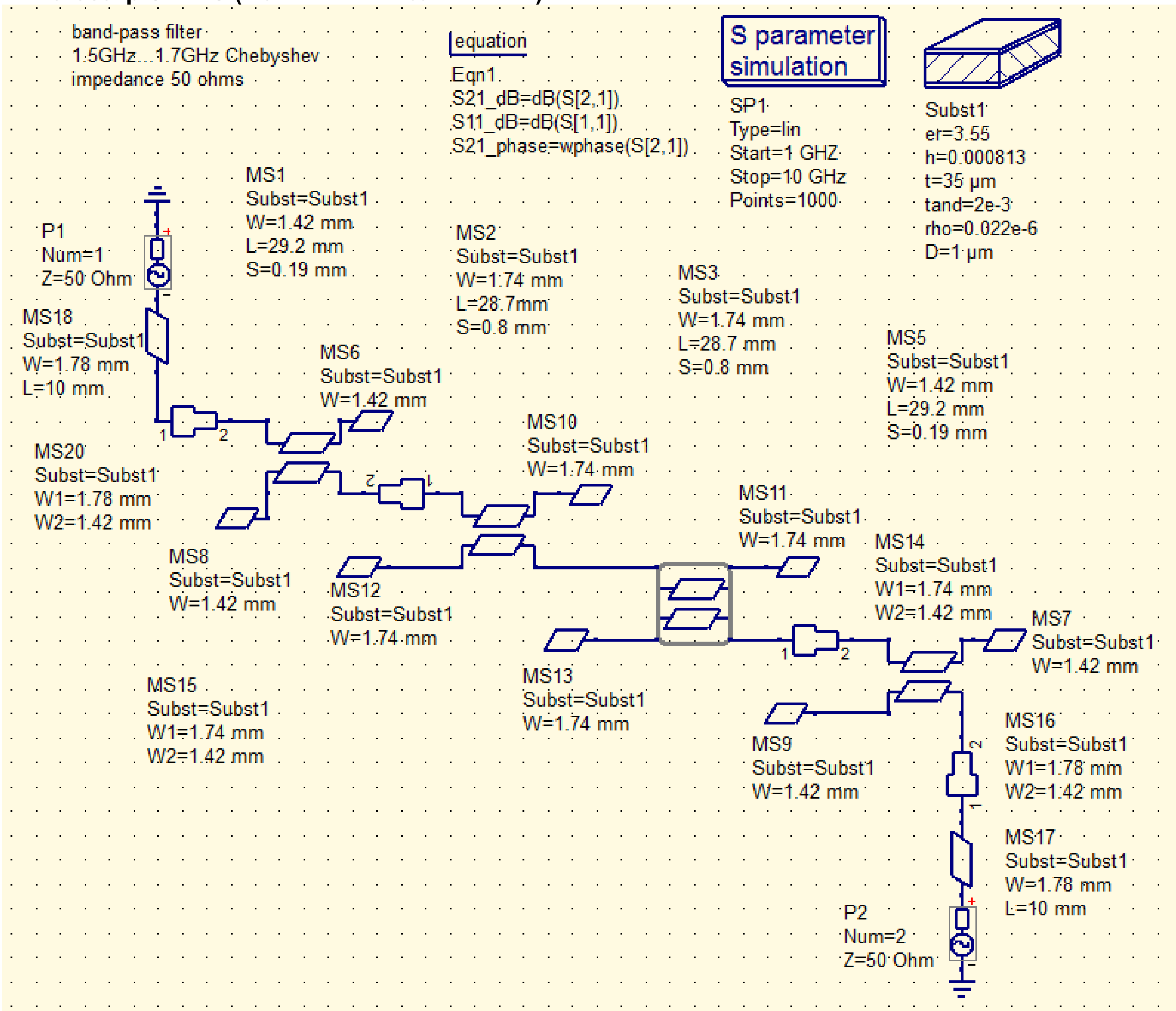
**Line Width = 1.74 mm**

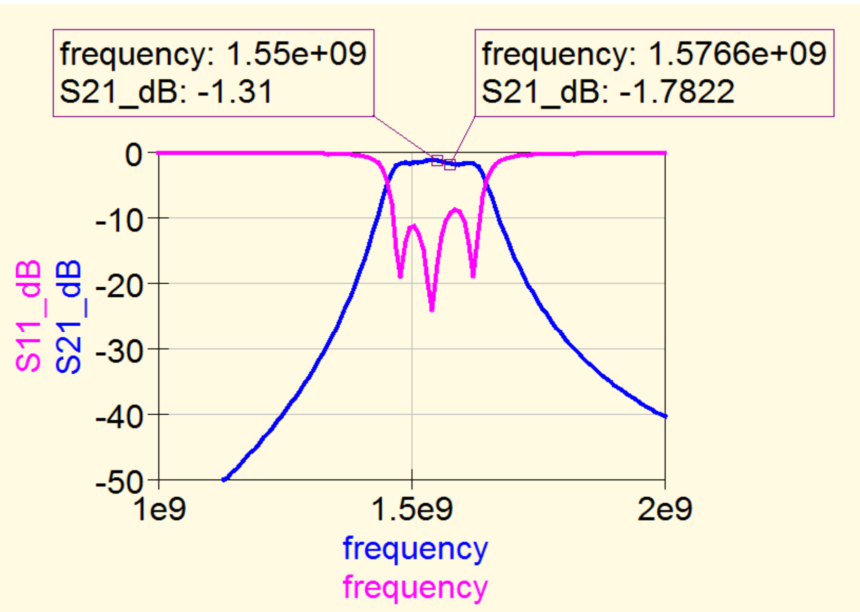
**Line Separation = 0.8 mm**

## 21.4. Filter Design and Optimization

Now we create the complete microstrip filter schematic. But we must not forget to terminate every „open end" of a line with a part named **Microstrip open"** (coming from „**Components / Transmission lines**"). But do not forget to enter the correct line width...
In the „**Substrate**" block the correct properties of the PCB material must be entered. „**Substrate**" can also be found in **„Components / Transmission Lines".**
In this schematic microstrip lines with different widths are connected together and so additional errors will occure. This must be corrected by **„Steps"** and this correction model can also be found in „**Components / Transmission Lines**".
At last the complete filter must be connected to the SMA sockets at each side of the case. This is done by 50Ω microstrip feed lines. If you use the filter calculator you'll get a **line width of 1.78 mm** for this line type (...please test it out). Do not forget: an additional step must be added at each side between feed line and the filter.....
This is now our „part collection":
**2 coupled microstrip line pairs, width = 1.74 mm / length = 28.7 mm / line separation = 0.8 mm**
**2 coupled microstrip line pairs, width = 1.42 mm / length = 29.2 mm / line separation = 0.19 mm**
**4 Microstrip OPEN, width = 1.74 mm**

**4 Microstrip OPEN, width = 1.42 mm**
**2 Feedlines (microstrip), width = 1,78 mm. Length = 10 mm**
**2 Microstrip STEPS (from 1.78 mm to 1.42 mm)**
**2 Microstrip STEPS (from 1.74 mm to 1.42 mm)**




The simulation result is not so bad, but

**the center frequency has decreased. If you have a look at the two markers you find a reduction from 1.575 GHz to 1.55 GHz**

**And the right „hump of S11“ is too high. This means an increased reflection coefficient |S11| which is higher than -10 dB. Thus S21 will also be reduced in this region.**

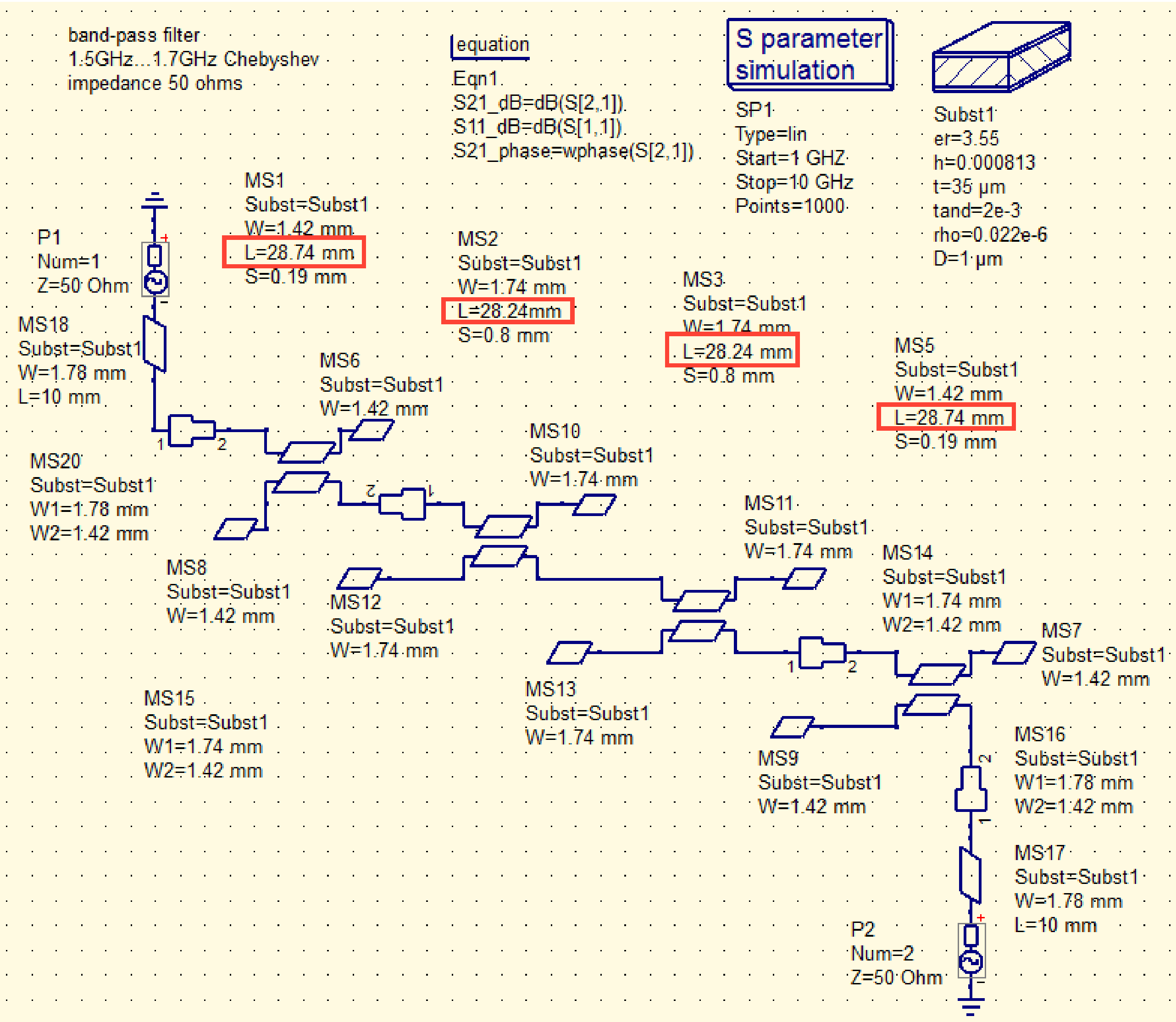
This needs some work...

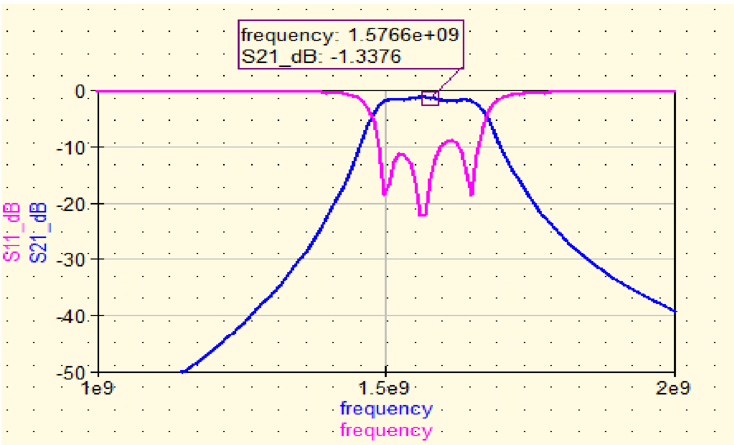
...but not too heavy. At first we shorten all line length values by the factor

**1.55 GHz / 1.575 GHz = 0.984**

to achieve again a correct center frequency of 1575 MHz. The new line length values for the coupled pairs are now

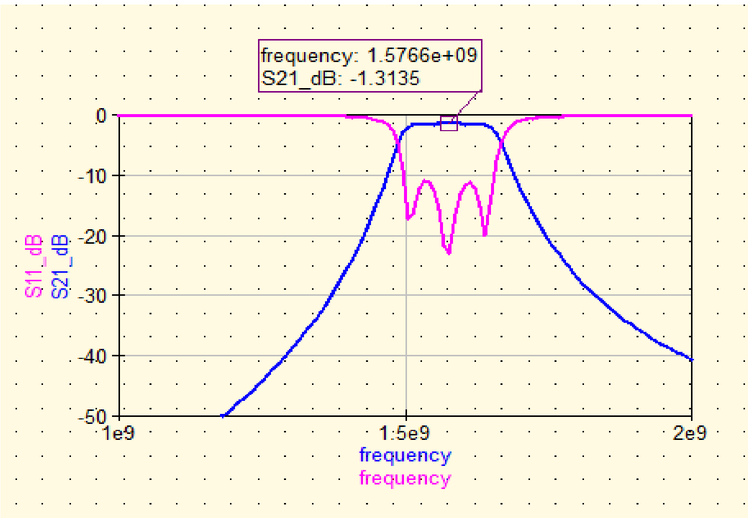
**28.74 mm resp. 28.24 mm**




**The result is a correct center frequency.**

The next steps needs patience. By trial and error two identical „humps of S11“ must be achieved and the correct way is **to modify „line separation“ and „line length value“** of the couples line pairs.



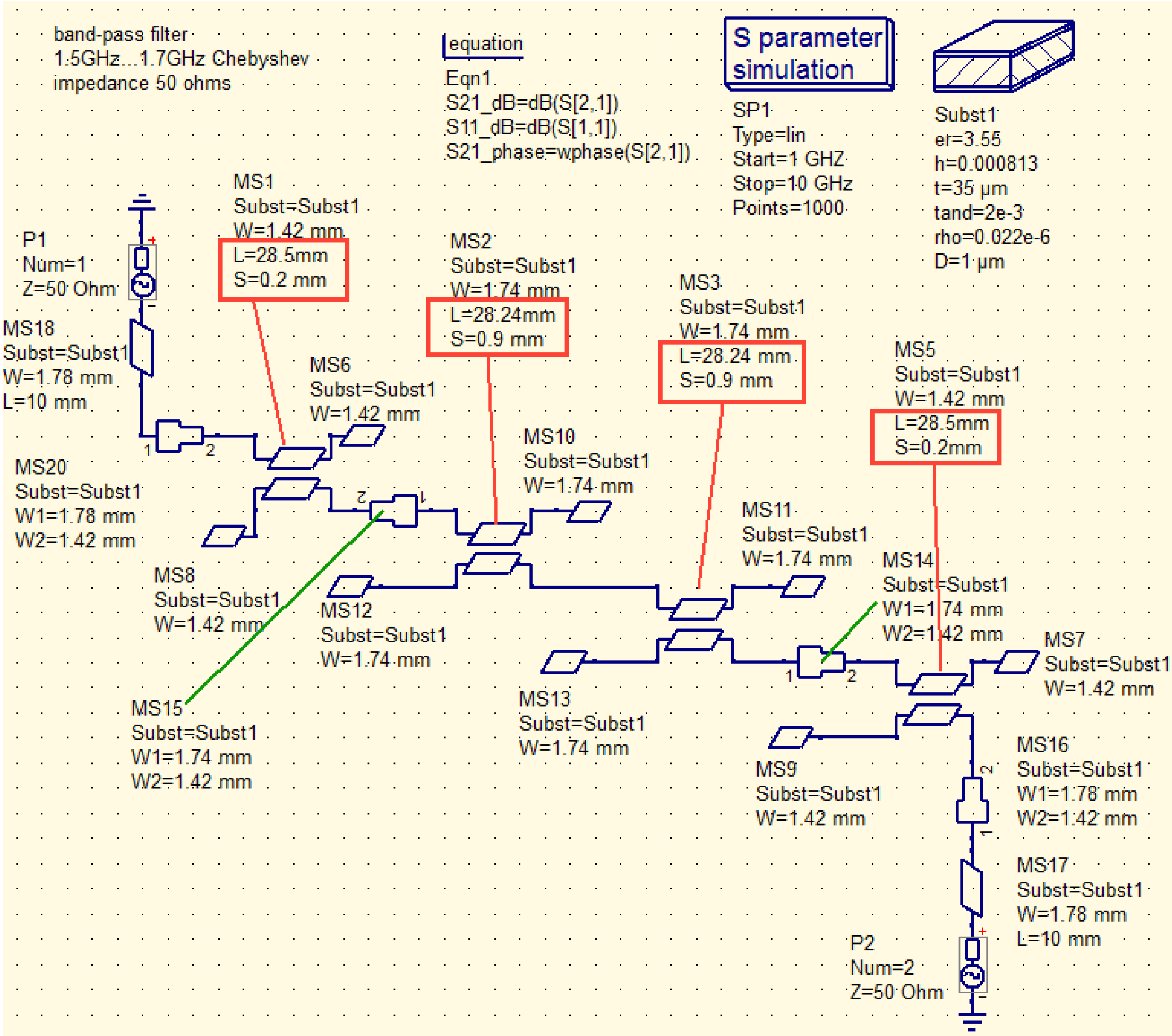

This is the fine final result....and the start point for a prototype design.

**Line 1 and line 4:**

**line width = 1.42 mm**
**line length = 28.5 mm**
**line separation = 0.2 mm**

**Line 2 and line 3:**

**line width = 1.74 mm**
**line length = 28.24 mm**
**line separation = 0.9 mm**

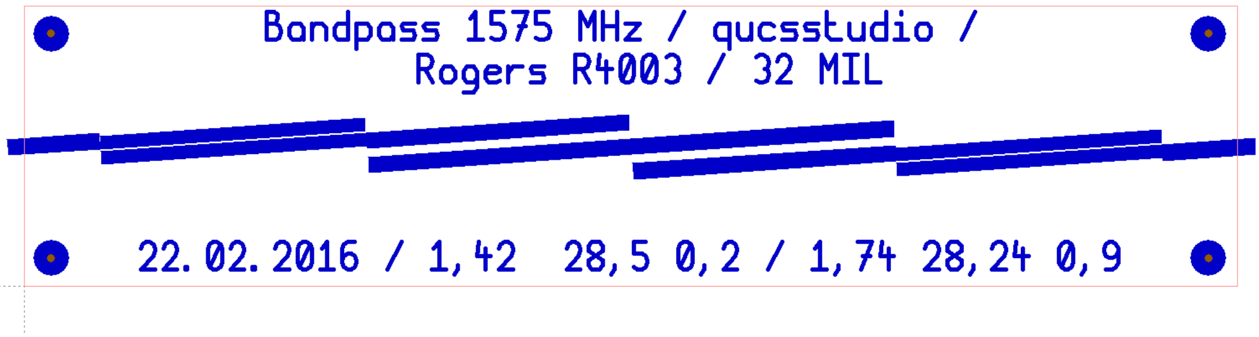


## 21.5. The Printed Circuit Board

Dimensions are 30 mm x 130 mm, thickness is 32 mil = 0.813 mm, material is Rogers R4003.

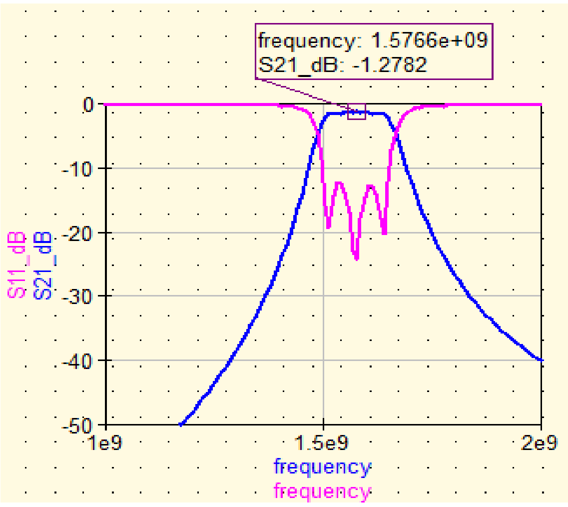
| | |
|---|---|
| **Line 1 and 4:** | **width = 1,42 mm / length = 28.5 mm / line separation = 0.2 mm** |
| **Line 2 and 3:** | **width = 1.74 mm / length = 28.24 mm / line separation = 0.9 mm** |
| **Feed line width** | **= 1.78 mm** |

Please note these data on the PCB (including the versions number). This could be very useful for your overview if a complete serie of different PCB trials is manufactured....

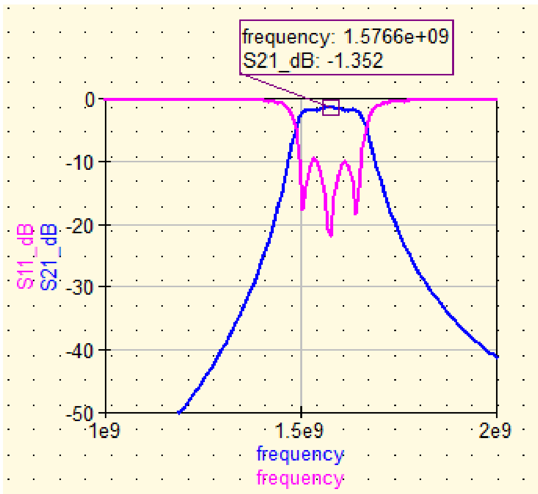


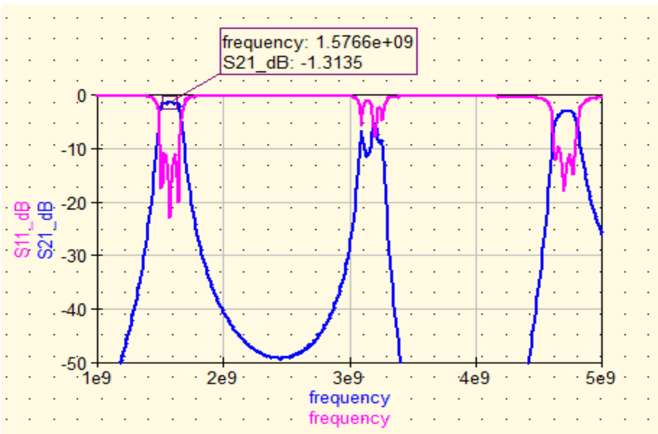

**Remark:**
**The efforts for the PCB maker and the accuracy demands are high....please have a look at the following simulations. The line separation of line 1 and line 4 was varied between 0.18 mm and 0.22 mm in the simulation. This has great influence to the S11 curve:**

**Line separation = 0,18 mm**



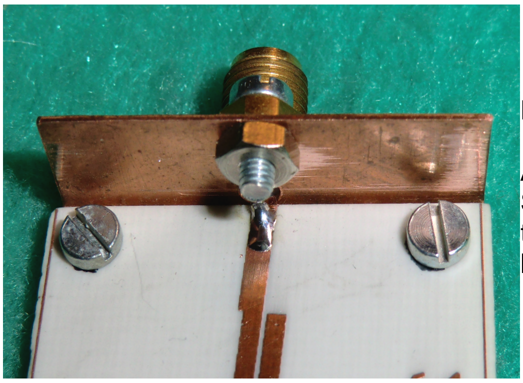

**Line separaton = 0,22 mm**




**But**
**Do never forget: as soon as you use transmission lines for filter constructions, you'll always be stressed by the line's property of „transforming impedances". Thus you get the well known S21 stop band curve with limited attenuation (resp. with S21 values higher than -10dB at „harmonic line length values" like λ/2, 3λ/4, λ......).**
**The left illustration proves this: the filter is only usable up to 2.8 GHz for S21 values smaller than -40 dB.**

## 21.6. Measured Results

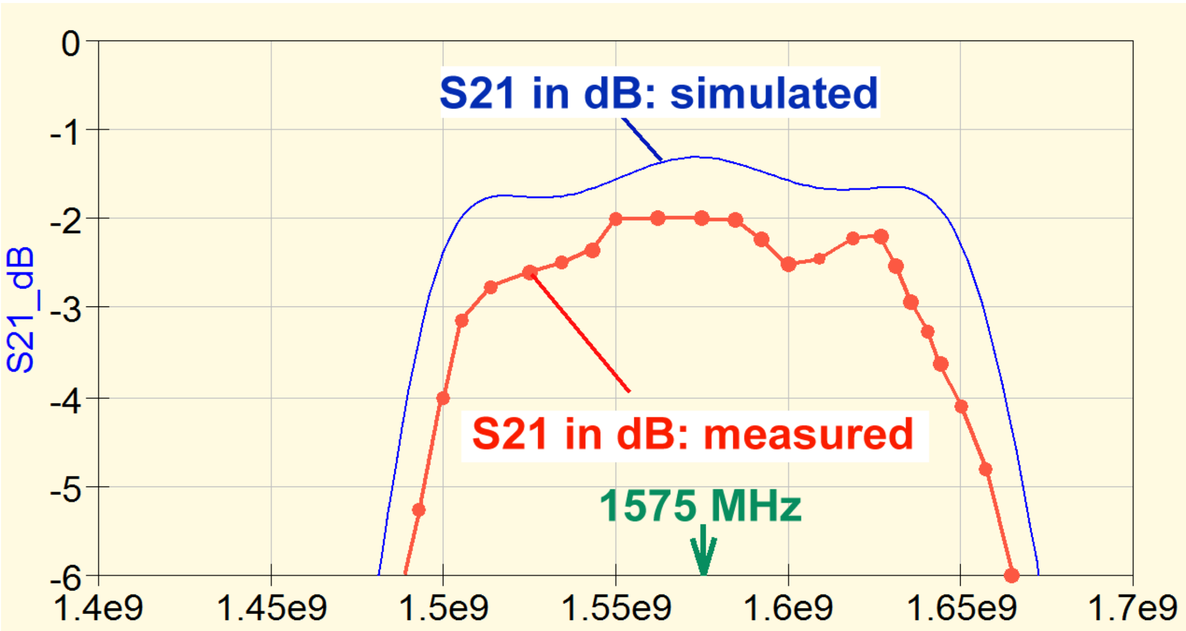


Let us start with the PCB and the test fixture:

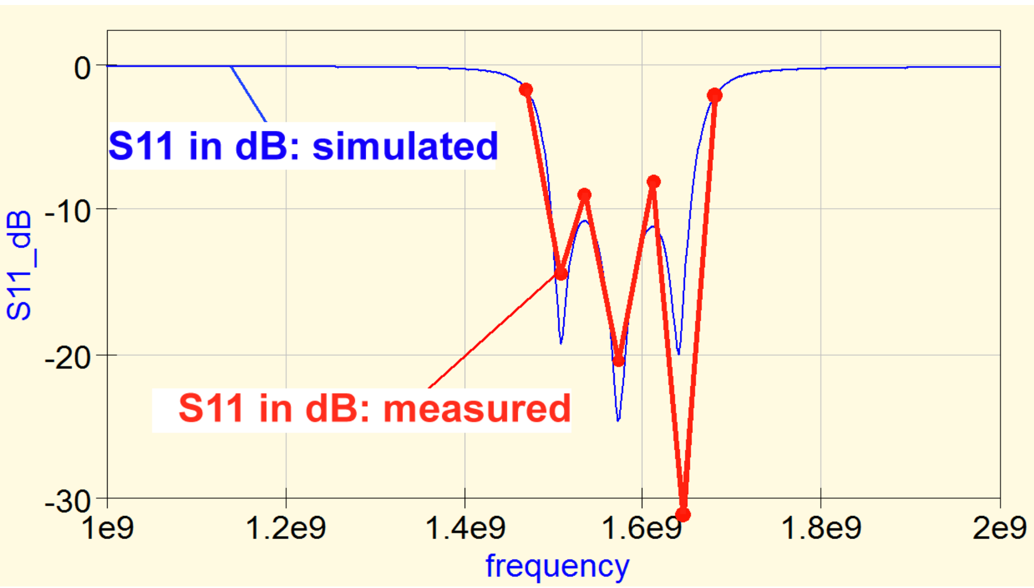
At every edge of the PCB a folded sheet copper is used to fix the SMA socket. The center conductor of the socket is soldered to the microstrip feed line of the filter. This gives a connection with low reflection.

Now the S21 curve was measured using a Vectorial Network Analyzer. But the result was a fine surprise...
The measured passband attenuation is only ca. 1 dB higher than simulated!

The

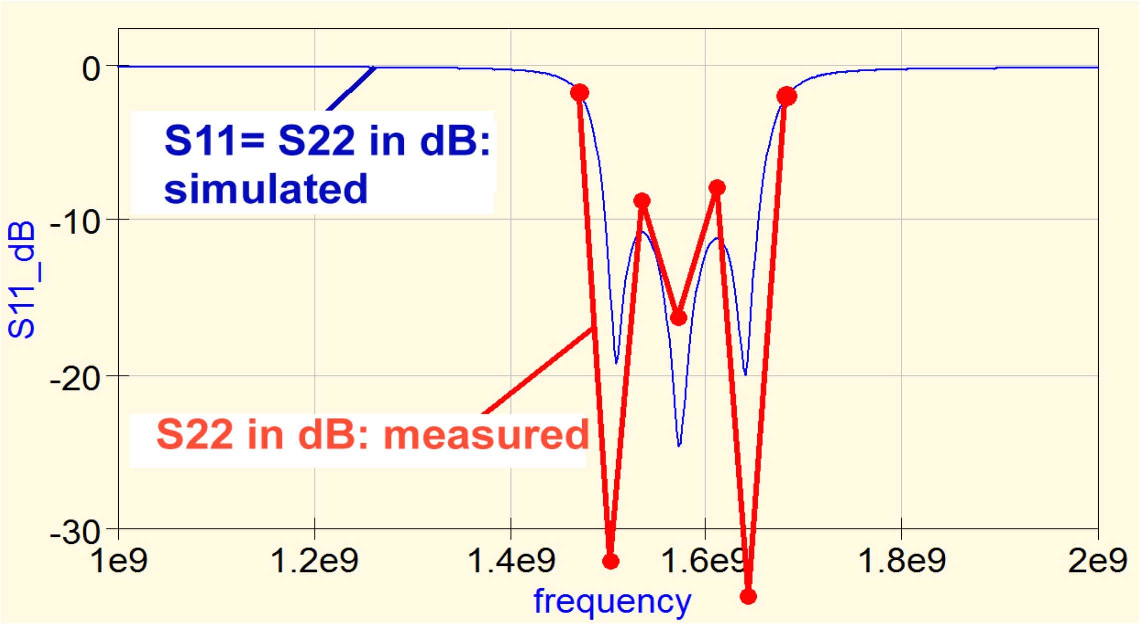



measured bandwidth is only a little bit smaller than desired but the passband ripple has increased – due to an increase of the reflection coefficients S11 and S22:




This is S11....but the two humps show different heights...

The reverse measurement to find out S22 is surprising: the right S22 hump stays higher. But due to mechanical tolerances (= scattering of the coupled line pair dimensions) on the PCB the S22 curve differs from the S11 curve.



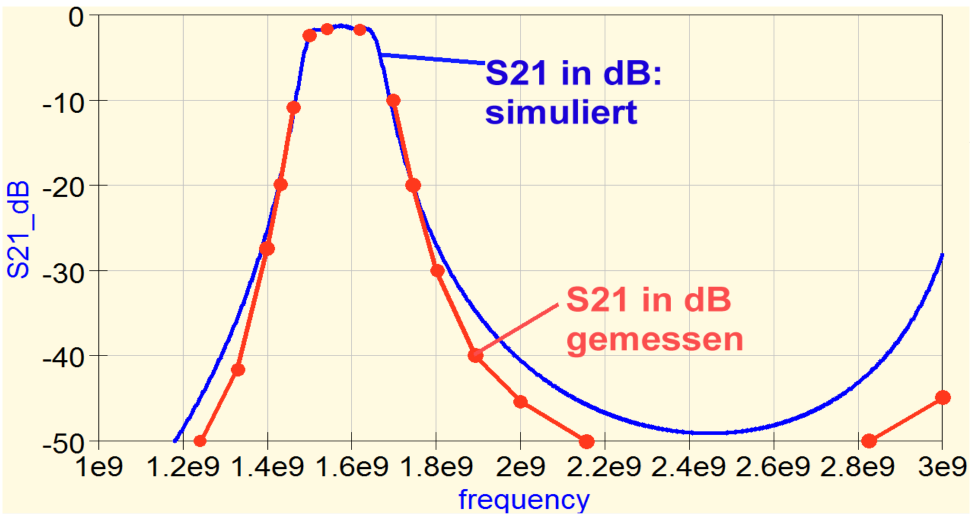

Summary:

**a) The right reflection hump is higher than the left one**

**b) The reflection amplitude is too high (S11 and S22 amplitudes = -8 dB and -9 dB, demanded = -13 dB).**

**This causes additional development work and PCB design of the next test PCB (...see chapters 21.4 and 21.5 for the way).**

But for a first prototype...not so bad...




At last a simulation to show the stop band attenuation up to 3 GHz.

**Maximum attenuation is nearly 50 dB (due to |S21| = -49 dB at 2.5 GHz).**

Thus: a lot of thanks to qucsstudio and the program's author Michael Margraf.