QucsStudio-Tutorial

Part 2:

Harmonic Balance Simulations

Version 1.0

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

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

January 1st, 2016

Table of Contents Page

1. Harmonic Balance: Why?

Simulations in the **Time Domain** (e.g. with SPICE) give a lot of information concerning the curves of currents and voltages and an FFT provides all information to the new frequency content caused by the nonliearities of the parts in the circuit. But this is always only a „snapshot“ because all input signal frequencies are held constant during the simulation.

An „**ACSweep**“ **ignores every nonlinearity and shows only the Transfer Functions for sine wave input signals with constant amplitude**.

So you find a „Simulation gap“ if you want an AC sweep and afterwards present the „nonlinearities over frequency“ for different amplitudes or frequencies of the input signals.
This gap is now closed by the „Harmonic Balance Simulation“ which is a free part of qucsstudio.
But remember: Harmonic Balance is always a Frequency Domain Simulation!

2. Hamonic Balance: How?

This is an ingenious trick and thus patented.
If a circuit combines linear parts (like resistors, capacitors, coils..) and nonlinear parts (like diodes, transistors, FETs) the program acts as follows:

a) **Linear Parts (including connections and nodes) are collected in a „linear subcircuit“**.

b) All the **nonlinear parts (including connections and nodes) are collected in a „nonlinear subcircuit“**.

c) Now the **two subcircuits (or „blocks“) are connected together by exactly named connection lines.**
Now we have again our correct circuit and the input signals can be applied.

See this nice illustration found in [1]:

Let us define the following symbols:

M = number of (independent) voltage sources
N = number of connections between linear and non-linear subcircuit
K = number of calculated harmonics
L = number of nodes in linear subcircuit

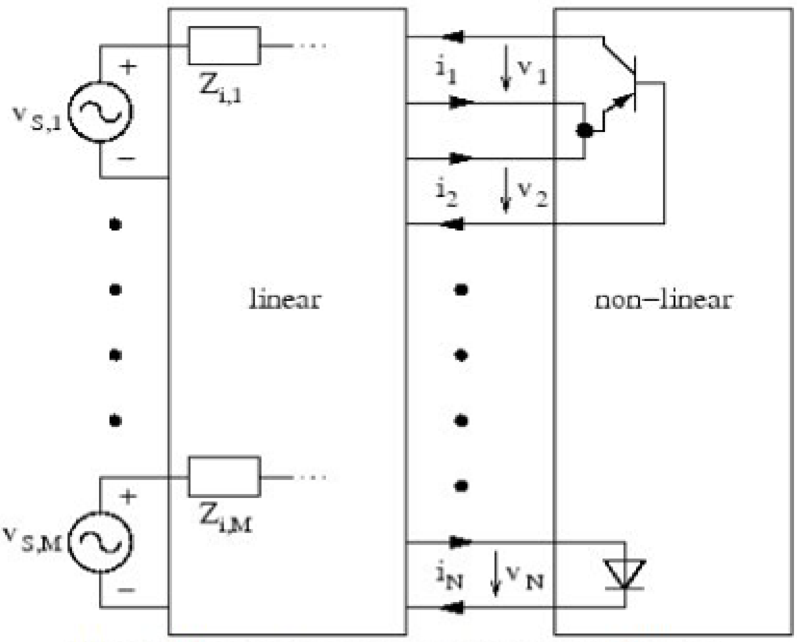


Figure 7.1: circuit partitioning in harmonic balance

The answer to the question „why all these efforts?“ is very simple:

Everything in the **linear subcircuit** can at once be calculated in the **Frequency Domain without any difficulties or problems**.

All **calculations and simulations in the nonlinear subcircuit can (after a FFT) be done in the Time Domain.** So you get as **a result every „current curve distortion“ which can be FFT re-transformed into a frequency spectrum for every connection between linear and nonlinear subcircuit. These results are sent back to the linear subcircuit.**

There is a lot to calculate....because the user must at first tell the „**maximum order of the regarded harmonics“** in the nonlinear subcircuit. **Then the program starts to calculate and compares the „energy entry from the linear subciruit into the nonlinear subcircuit (for every connection!) to the energy „which ist given back from the nonlinear subcircuit to the linear subcircuit in form of a frequency spectrum“. If no identy is reached then a new trial with altered values will be started.**
This needs a lot of iterations up to the point of a correct „Harmonic Balance“.....and thus the user must also enter (before starting HB simulation) the maximum number of trials and the maximum allowable error at the end of the simulation.

Theory and mathematical background are complicated. If somebody is interested in this: please read [1]..

But the possibilities and results of a successful Harmonic Balance Simulation are fascinating. Additionally working with a „Parameter Sweep“ you have a huge stock of data and results. You can calculate the relationship between the input voltage and the distortions over the frequency, the 1 dB compression point, the IP3 point, the spectrum of generated harmonics.....and so on...and so on....

Let uns start with an example which can be downloaded from the qucsstudio homepage, named

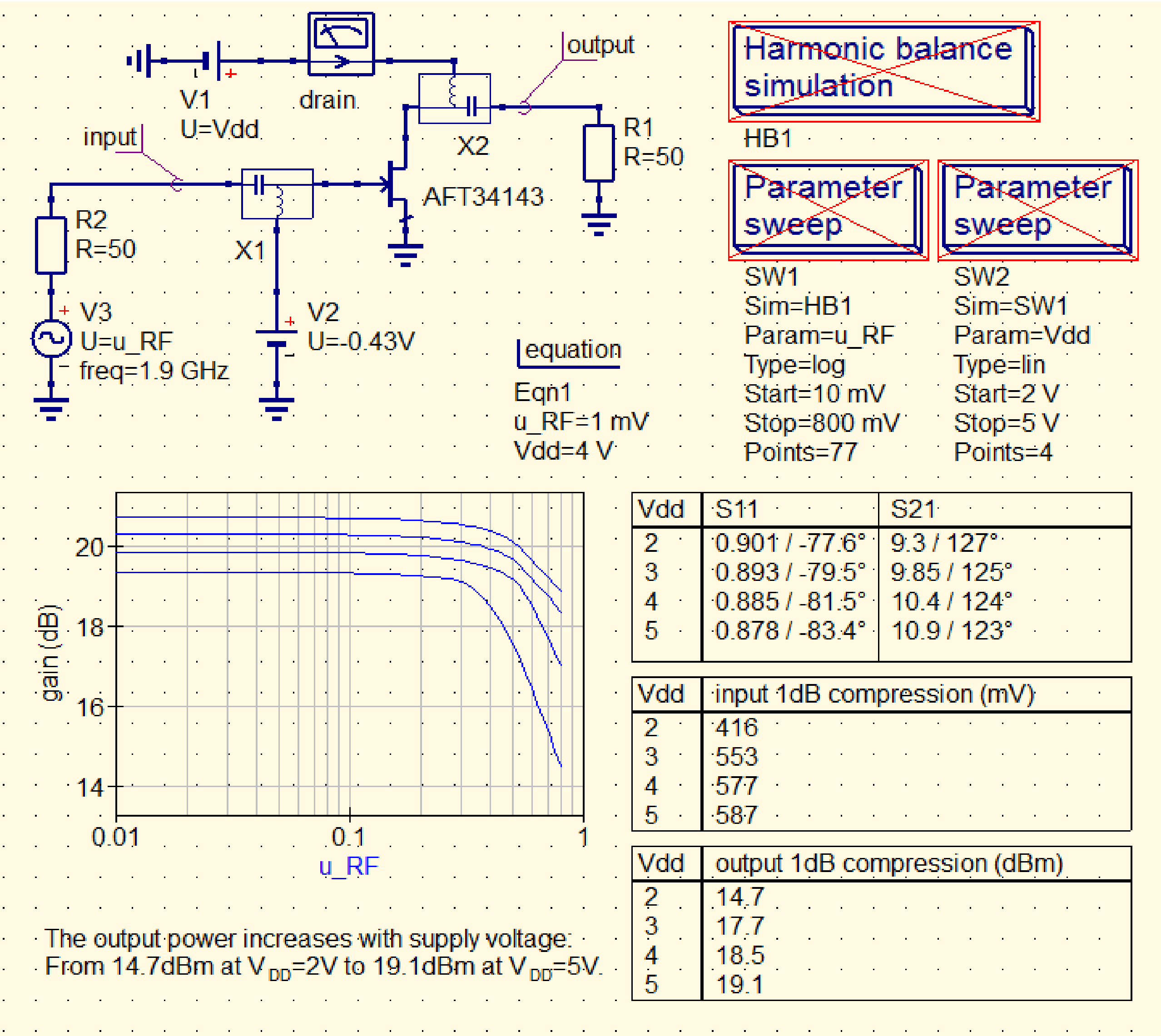
„FET_P1dB.sch“

There you find a collecton of „**examples**“, followed by „**HB-Analysis**“. Download and extract the packjage using the path „**Project / Extract package“.**

3. First Example: a FET Amplifier Stage explored by a Parameter Sweep

3.1. Preparations

At first check the identity of your downloaded example and this schematic.



Vdd	S11	S21
2	0.901 / -77.6°	9.3 / 127°
3	0.893 / -79.5°	9.85 / 125°
4	0.885 / -81.5°	10.4 / 124°
5	0.878 / -83.4°	10.9 / 123°

Vdd	input 1dB compression (mV)
2	416
3	553
4	577
5	587

Vdd	output 1dB compression (dBm)
2	14.7
3	17.7
4	18.5
5	19.1

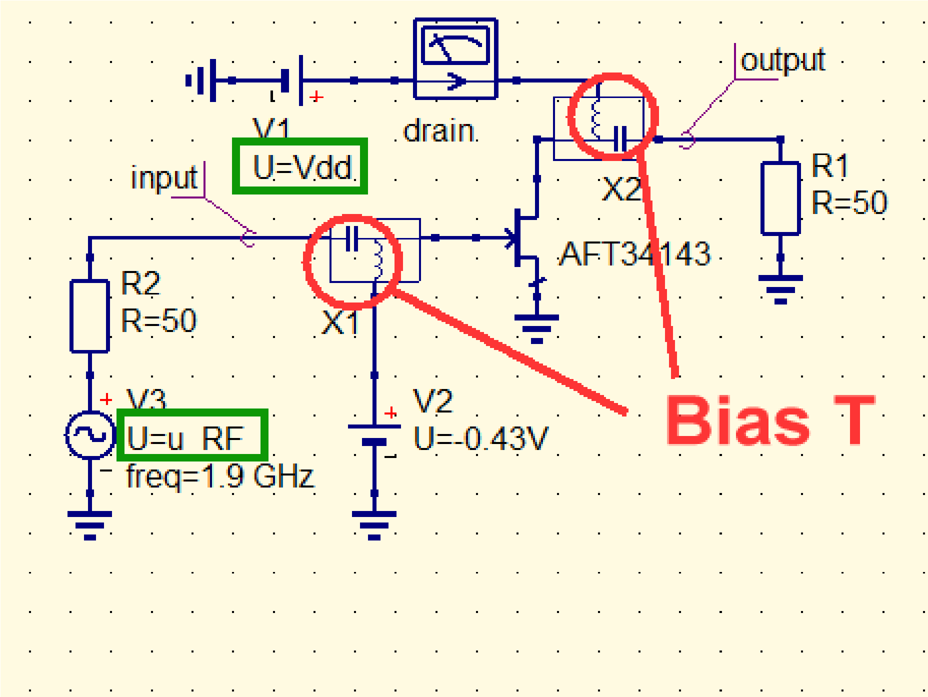
The next steps can be found in the above schematic. Please open „edit“ in the Menu taskbar and click on

Deactivate / Activate

Now you find a small rectangle hanging on your cursor and if you now click on both „Parameter Sweep“ and on „Harmonic Balance“, these three functions are deactivated.

(At every moment you can roll back this action by repeating the procedure)

Now impose order on your screen and shift these deactivated directives away from the schematic – we want to examine and to explain the circuit itself!



The Gate Bias Voltage is fed to the gate by a „**Bias T**“ and the voltage value is **-0,43V.**

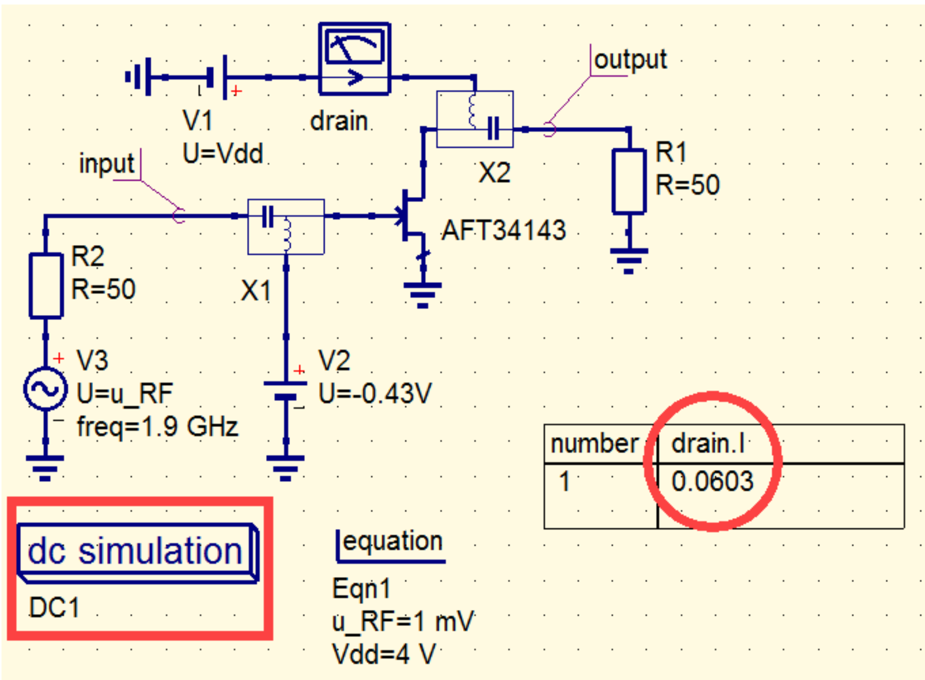
The supply voltage ist named „**Vdd**“ as preparation for a Parameter Sweep

The Input RF Signal comes from the voltage source V3. Its frequency is f =1.9 GHz and the amplitude is also given as a variable named „**u_RF**“ (for another Parameter Sweep).

At the Drain pin another „**Bias T**“ is used to separate the power supply voltage Vdd and the RF output voltage at R1.

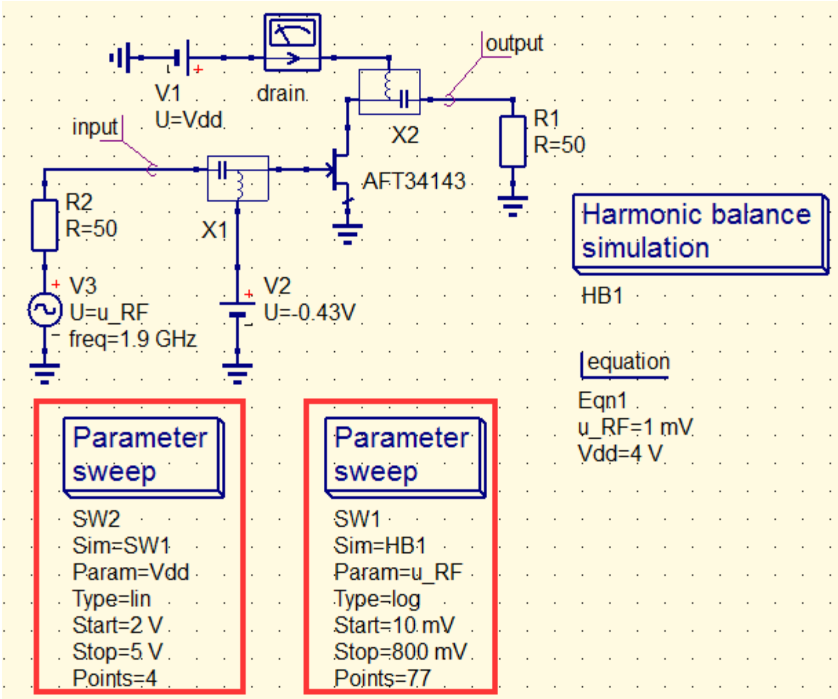
Task:
Find the DC quiescent current of the FET.

Solution:
Pick „DC Simulation“ (path: „**Components / Simulations**“) and simulate. The results are given in a **table.**



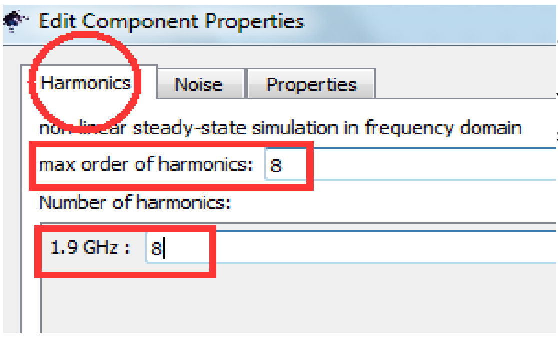
This means:
You get a Drain current of **60 mA** **for a Drain voltage Vdd of +4 V and a Gate voltage of -0,43V**

Now we continue with the Parameter Sweep

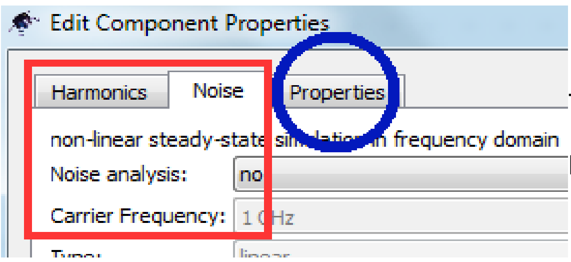


At first we activate the three inaccessible directives (= Harmonic Balance Simulation and two Parameter Sweeps).

Now right click on „**Harmonic balance simulation**“ to open the properties.



On the first card you find the confirmation of chapter 2 („How?“):
You have to enter the highest order of harmonics for the steady state simulation, default is „8“.



The next card sets the noise simulation properties.

Please mark „**no noise simulation**“ for this example.

On the last card you find the simulation name „HB1“ and

a) the **settings for the relative and absolute tolerance when convergence is reached**

b) the **maximum number of trials (= iterations)** before an error message is edited.

Still a sharp look at the two Parameter Sweeps beside the schematic:

a) „SW1“ sweeps the input RF voltage „u_RF“ in 76 logarithmic steps from 10 mV up to 800 mV

b) „SW2“ sweeps the supply voltage „Vdd“ using the values „2V / 3V / 4V / 5V“.

With equation equ1 the start values (u_RF = 1 mV and Vdd = +4V) are given for a non-parametric simulation.

Remember:
For a **result presentation of a non-parametric simulation in the Frequency Domain** you have only to enter the **name of the variable in the Graph Property equation.**

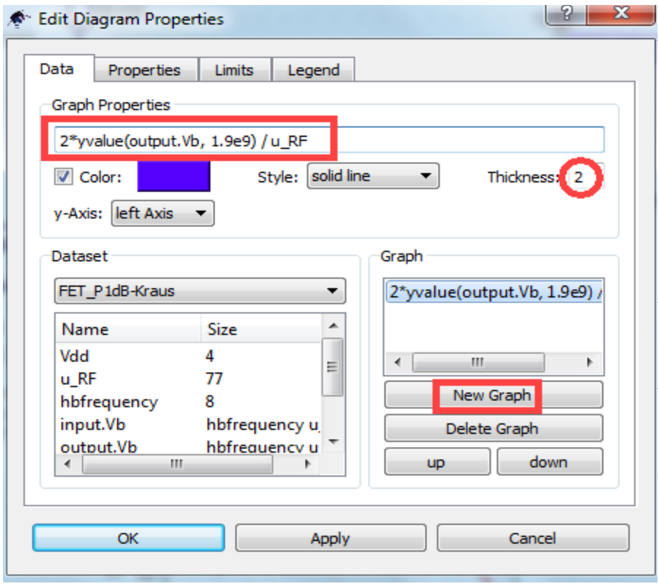
For a result presentation of a **Parameter Sweep the Graph Property equation must start with „yvalue...“**

But now....simulate!

3.2. Simulation of the Relation between Forward Transmission S21 and Supply Voltage

you need a cartesian diagram for the presentation. Enter under „Graph Properties“ the following equation for S21:

2*yvalue(output.Vb, 1.9e9) / u_RF



This can easily be understood, because this is the well known definition of S21:

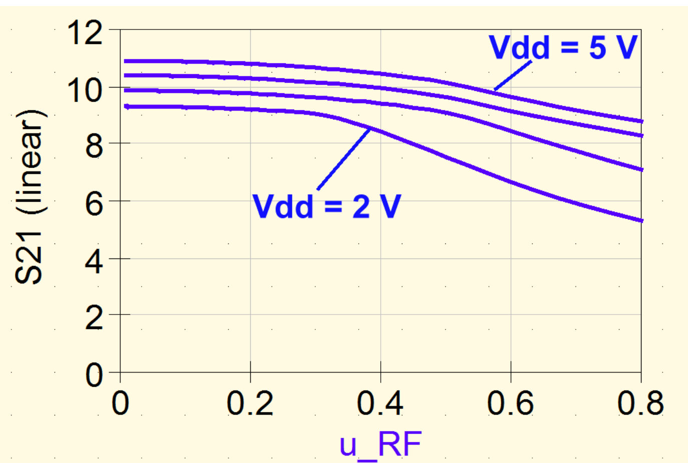
S21 = output signal at R1 divided by the incident wave (u-RF / 2)

But have a sharp look at the output voltage expression in the equation:

You must use **„output.Vb,1e9“**

which is the result of the Harmonic Balance Simulation for the used frequency f = 1.9 GHz.

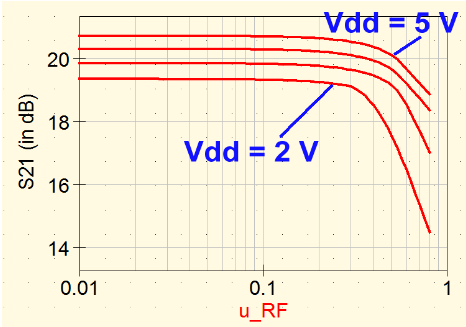
A line thickness of „2“ is used in the diagram. The vertical axis is scaled linearly from 0.....12.
The horzontal axis is scaled linearly from 0....800 mV for the RF input voltage „u_RF“.



This is the result.

Task:
Present **S21 in dB** and mark the vertical axis as „**S21 (in dB)**". Choose a red line with thickness „2".

Solution:
Write a new Graph property equation:

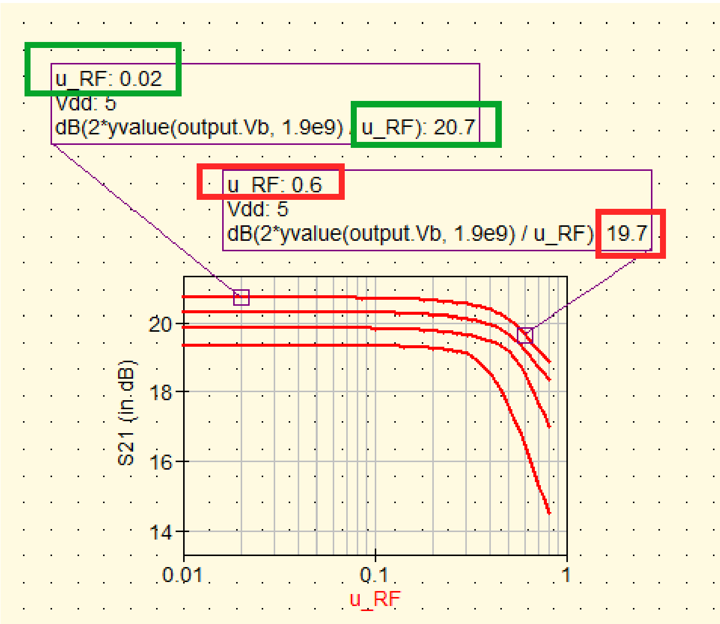


dB(2*yvalue(output.Vb, 1.9e9) / u_RF)

What a nice look....

3.3. The „1 dB Compression Point"

If the input voltage „u_RF" is increased the output voltage will follow up to the „saturation" and reaches a constant value. Thus a „1 dB compression point" is defined and this point marks a gain reduction of 1 dB.



Task::
Find the 1 dB compression point for a supply voltage **Vdd = +5V**

Solution:
Use the last diagram „S21 versus u_RF" and set two markers:

The first marker is placed at a low value of „u_RF" where the gain is still constant (here: u_RF = 20 mV). Then check S21 (here: S21= **20.7 dB**).

The other marker is shifted upwards to the point where the gain is reduced by 1 dB (now: S21 = **19.7 dB**). At this point the input voltage has a peak value of 0.6 V.

Remark:
0.6 V is the **peak value of the input signal „u_RF"**. The **RMS value** would be

u_RF = 0.6 V / sqrt2 = 0.424 V.

And if you are interested in the **„level at 50 Ω expressed in dBm**" another calculation is necessary:

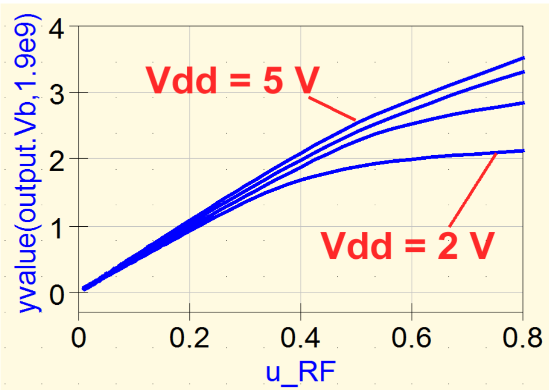
„Input 1dB compression point"= 20*log10(0.424V / 0.223V) = +5,6 dBm

(0.223 V is the RMS reference voltage for P = 1mW at 50Ω)

You get the „**Output 1dB compression point**" by adding the gain (S21 = 19.7 dB) at this point. This gives a value of
+5,6dBm + 19,7dBm = +25,3 dBm

Task:
Simulate the 1 dB Compression Points for Vdd = +2V / +3V / +4V

3.4. Linear Presentation of the Transfer Characteristic for different Values of the Power Supply Voltage Vdd



Open a new cartesian diagram and enter the equation:

yvalue(output.Vb,1.9e9)

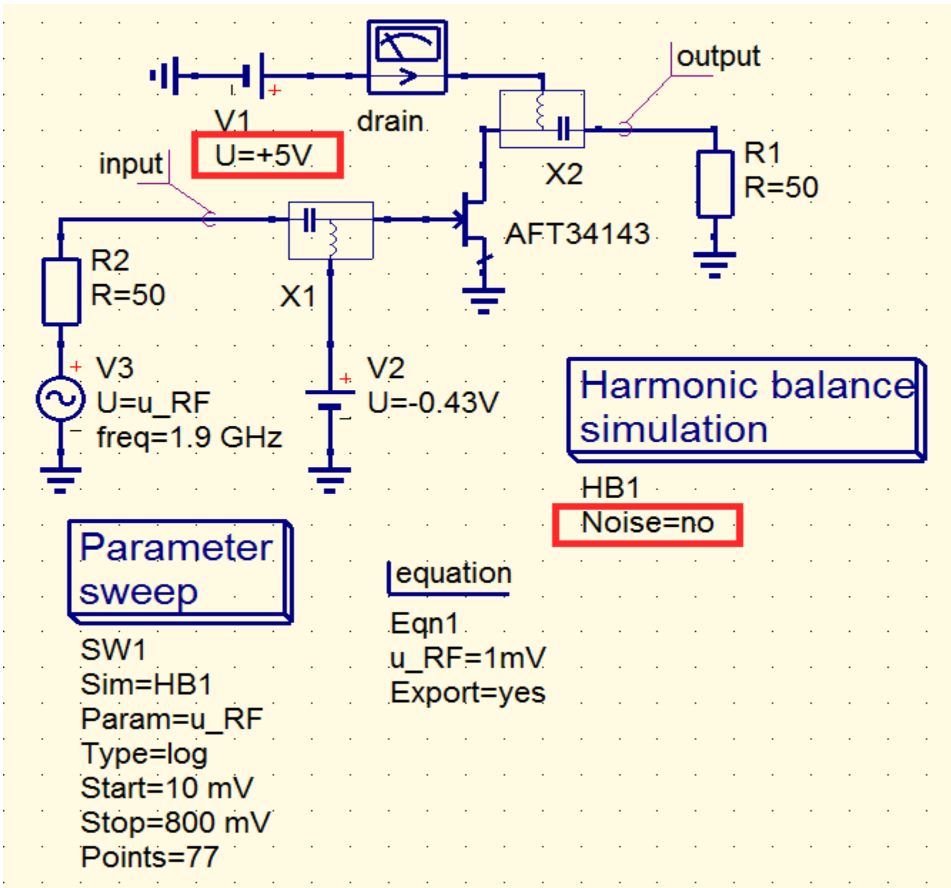
This is not complicated and can be read as follows:

„Show the simulated output voltage „output.Vb“ at f = 1.9 GHz versus the input voltage „u_RF“ for the four different power supply voltage values, using a parameter sweep“.

„---

4. Second Example: Examining the FET Amplifier Stage for a fixed Operating Point

4.1. Simulation Schematic



Let us use a **fixed supply voltage (Vdd = +5 V) and deactivate the parameter sweep SW2** used in the last chapter.used in the last chapter.

Please open the properties of the „Harmonic balanced simulation“ directive and make the entry „**noise = no**“ visible.
Then simulate.

4.2. Ideal and real Transfer Characteristic

We use a cartesian diagram after the simulation and write the following Graph property equation for the presentation of the real Transfer characteristic at f = 1.9 GHz:

yvalue(output.Vb,1.9e9)

(Use blue colour and line width = 2 for the curve).

To get the „ideal transfer characteristic curve“ is a little complicated:

1) **Calculate the maximum value of the Input to Output ratio at f = 1.9 GHz:**

plot(**u_RF,(max(yvalue(output.Vb, 1.9e9) / u_RF))***u_RF)

----

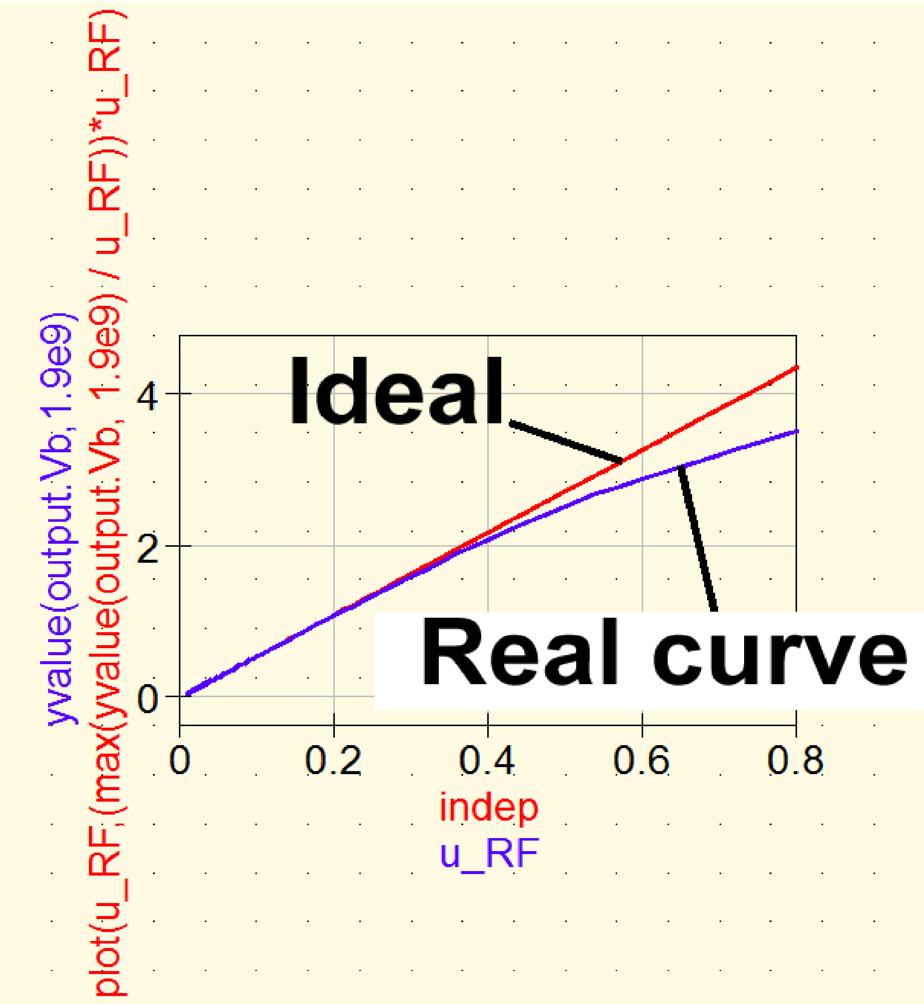
2) **Then multiply this maximum value by the input voltage „u_RF“ :**

plot(u_RF,(max(yvalue(output.Vb, 1.9e9) / u_RF))***u_RF**)

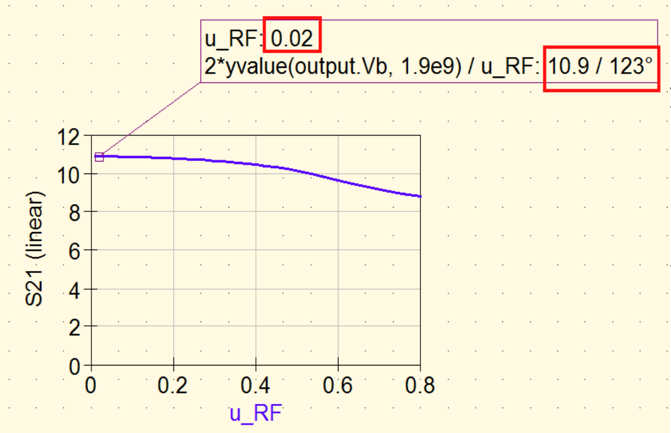
----

3) At last plot this result also in the same diagram. Use red colour and a line width of „2“:

plot(u_RF,(max(yvalue(output.Vb, 1.9e9) / u_RF))*u_RF)



4.3. Linear Presentation of S21 for different Input Voltage Amplitudes



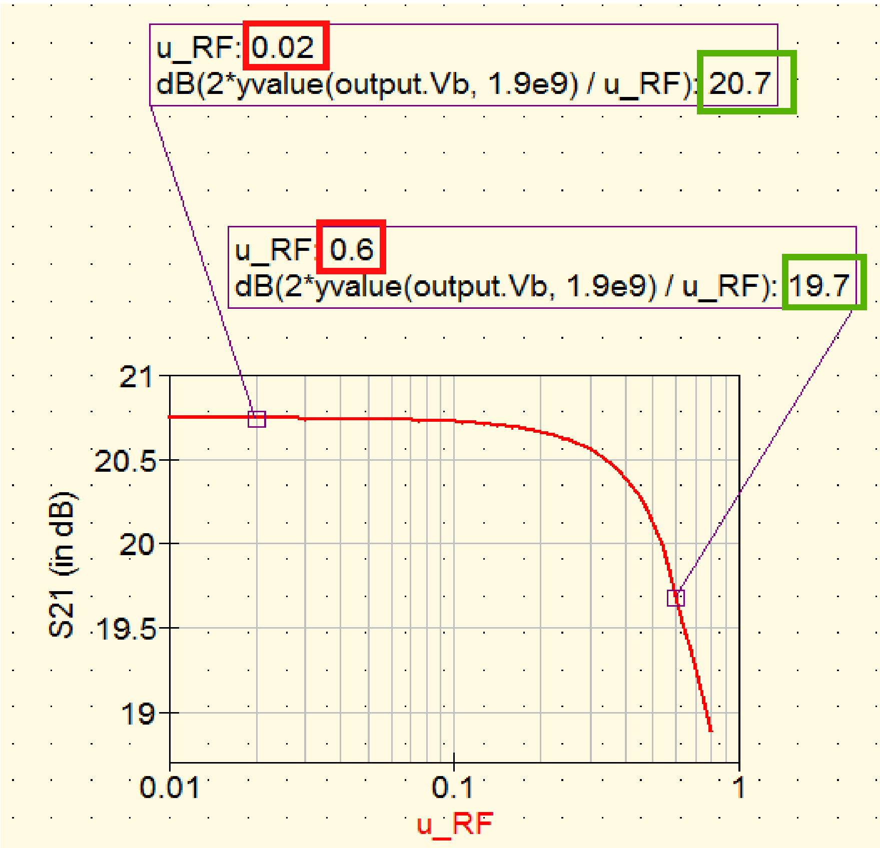
As in the last project we use the well known formula to calculate „S21“ at 1.9 GHz:

2*yvalue(output.Vb,1.9e9) / u_RF

Using a marker we find without any problems the value for S21 at f = 1.9 GHz for an input voltage u_RF = 20 mV (= peak value) to

10.9 / 123°

4.4. Presentation of S21 in dB (including 1 dB compression point) for different Input voltage Amplitudes



Also a well known procedure:

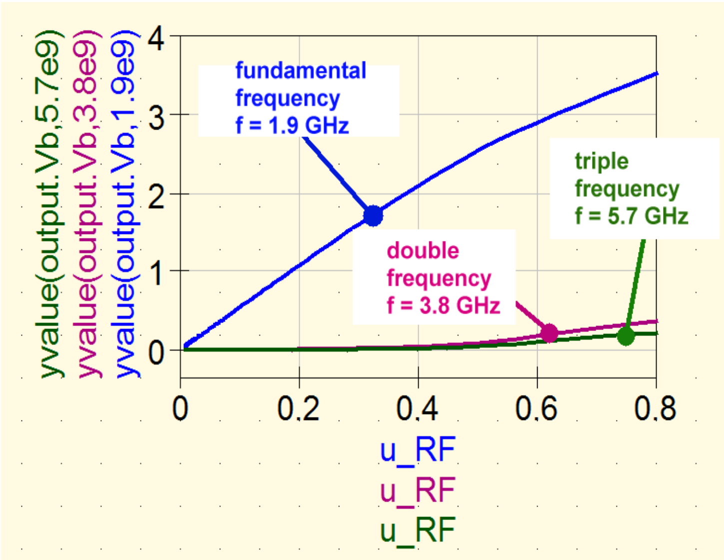
We find (by usage of a marker) a start value of **S21 = 20,7 dB at an input voltage u_RF = 20 mV for f =1.9 GHz.**

And shifting a second marker to the point where S21 is now 1 dB lower (S21 = 19.7 dB) we find that this correlates with a peak value of 600 mV of the input voltage u_RF.

4.5. Rise of the Harmonics for increasing Input Voltage Amplitude (linear Presentation)4

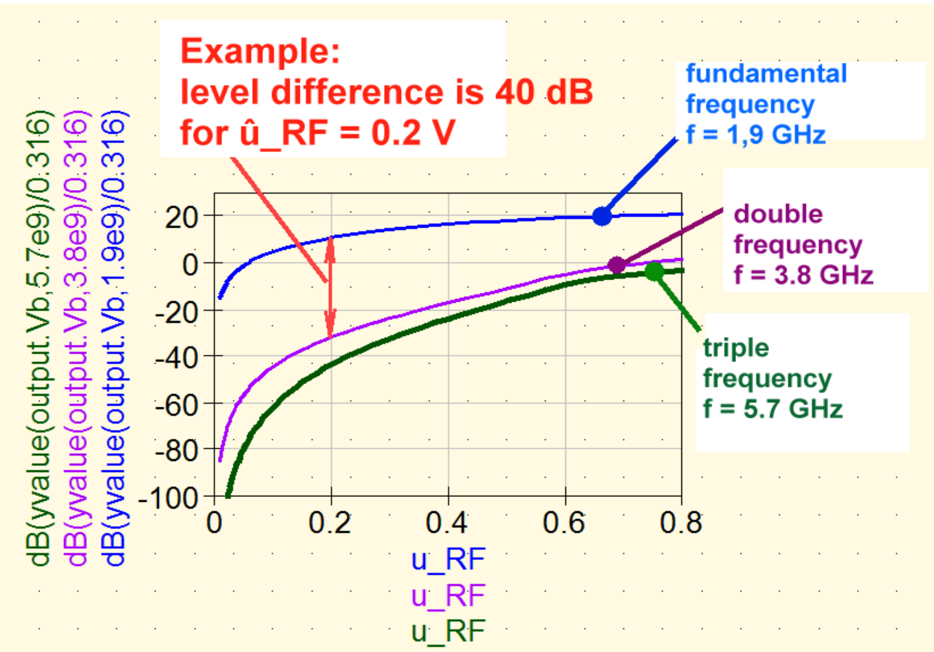
This is a quite simple affair because the following Graph Properties equation will do the job:

yvalue(output.Vb,[frequency])



This is the diagram showing the fundamental frequency, the double and the triple frequency (peak values)

4.6. Rise of the Harmonics for increasing Input Voltage Amplitude (presentation in dBm at 50Ω)



This diagram is needed very often to find the level difference between the Fundamental and a Harmonic Frequency.

But never forget that qucsstudio always calculates peak amplitude values (...also valid for the Harmonics).
Thus when looking for levels scaled in „dBm“ you have at first to change from peak values to RMS values.

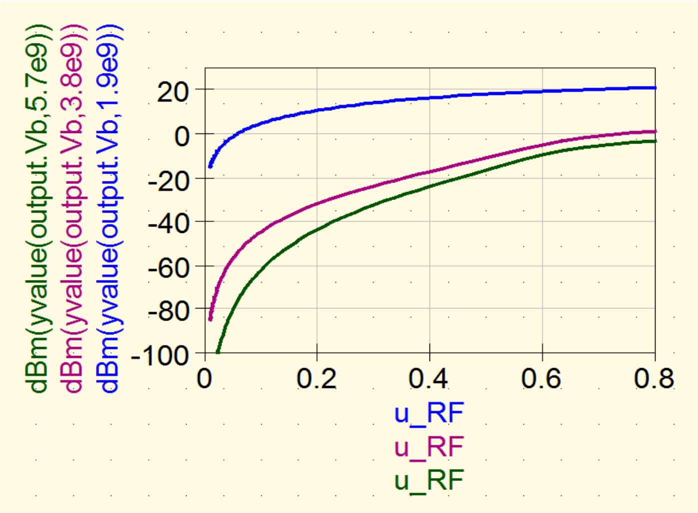
Remember:
P = 0 dBm = 1 mW at the characteric system resistance.

Example for Z = 50 Ohms:
P = 1 mW at 50 Ohms gives an RMS voltage of 0.223 V and a peak voltage of 0.223*sqrt(2) = 0.316V

So to change from „dB“ to „dBm at 50Ω“ for this diagram the following Graph property equation was used: **(yvalue(output.Vb,[frequency]) / peak reference voltage)**

Attention:

This equation is necessary if the charactistic system impedance differs from the default setting of „Z = 50Ω“



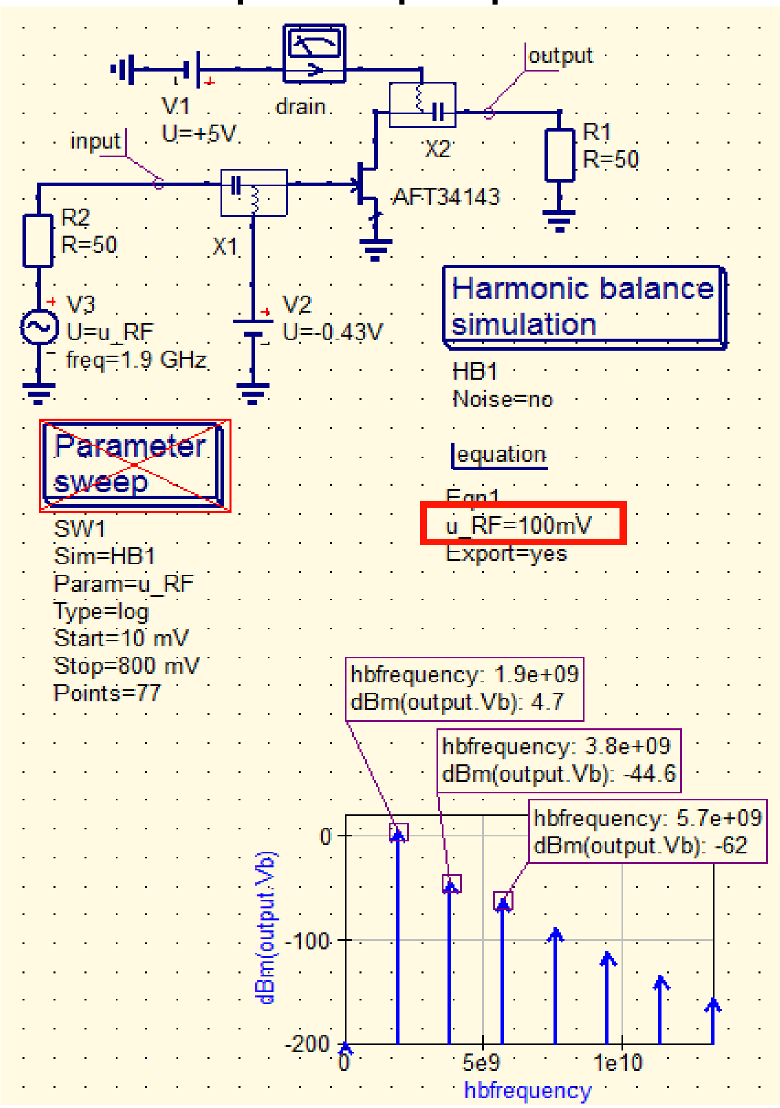
For Z = 50Ω this calculation is much simpler, because this case is already prepared for you. Enter the equation

dBm(value(output.Vb, [frequency]))

to get the same „dBm at 50Ω“ curves as before!

Please compare....

4.7. The complete Output Spectrum



If you want to see the **complete frequency spectrum at the output, deactivate all parameter sweeps and avoid „yvalue...“ in the graph property equation.**

Now if you are looking for all harmonics for an input peak voltage value of u_RF = 100 mV:
Enter this value in the equation“Eqn1“ and simulate. Then write the graph property equation as

dBm(output.Vb)

for a cartesian diagram and use a marker to indicate every harmonic frequency and amplitude.

4.8. The IP3 Point

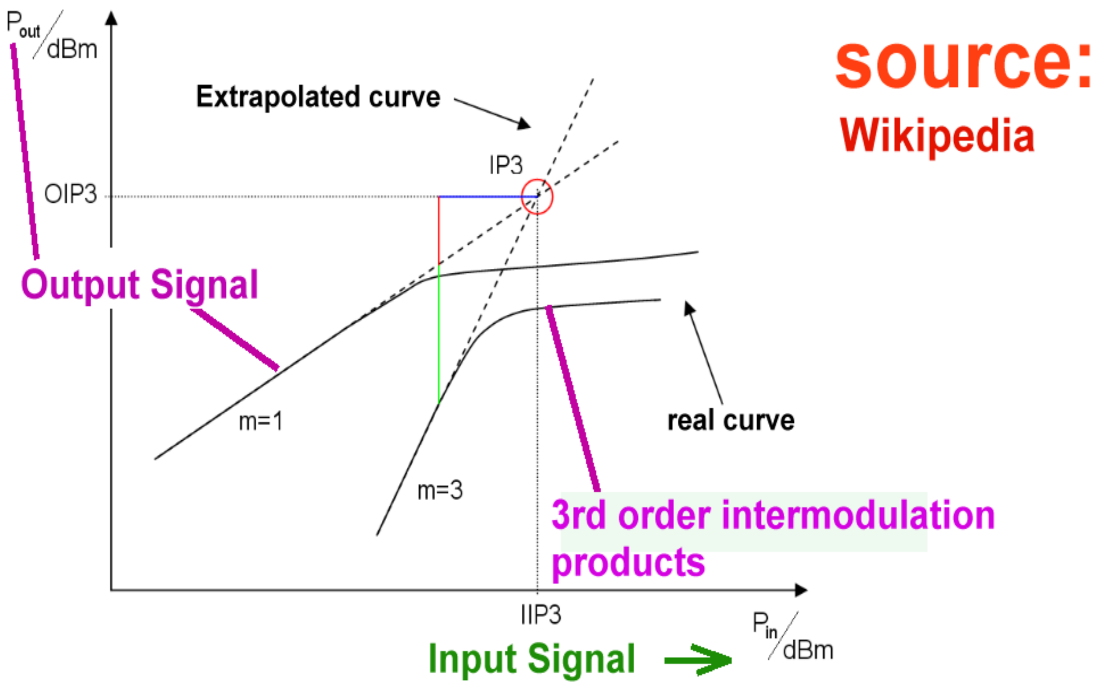
4.8.1. Fundamentals

In chapter 14.4. (of part 1) this text can be found:

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:



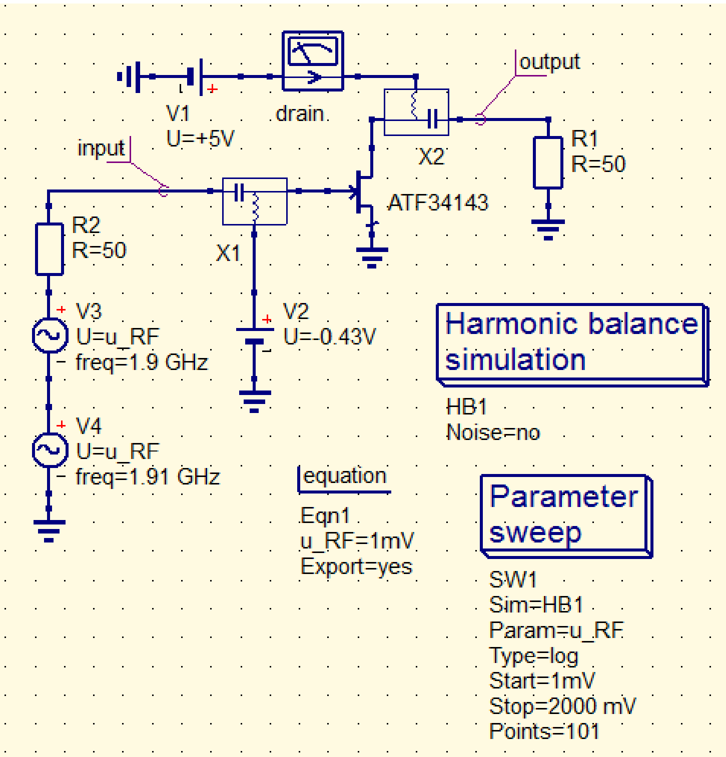
But if you exactly know this IP3 point, then you can for every input or output level calculate the level difference to the undesired 3rd order intermodulation products! **(You have simply to solve two equations for straight lines....)** **And so you can for this case define an intermodulation free dynamic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

4.8.2. Simulation of IP3 using the Transfer Characteristic

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

The Levels are 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....)

Example:



At the input two voltage sources (with f1 = 1.9 GHz and f2 = 1.91 GHz) are connected in series. This gives a frequency difference of 10 MHz. In this case you'll find the IP3 products at

(1.9 GHz - 10 MHz) = 1.89 GHz

and

(1.91 GHz + 10 MHz) = 1.92 GHz

The voltage values of both sources must always be indentical. So we use the **variable u_RF** and a **value range from 1 mV to 2 V in the parameter sweep.**

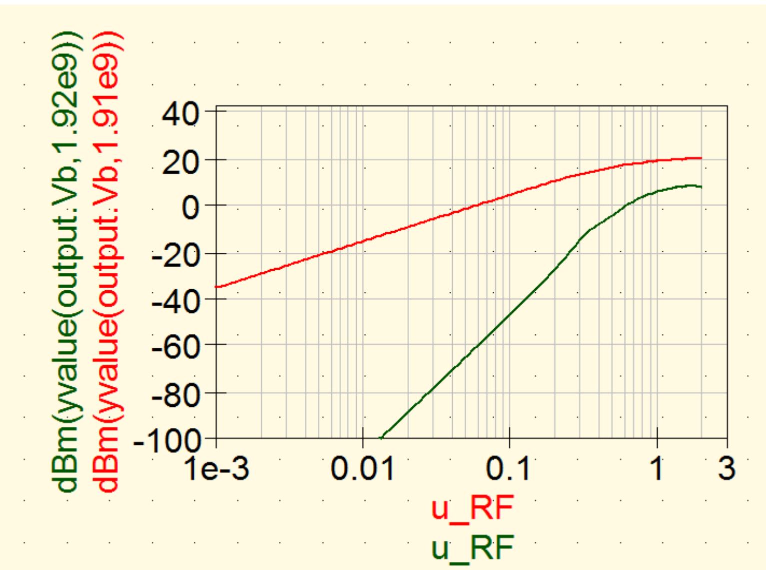
To present the simulation results in a cartesian diagram use the two equations:

dBm(yvalue(output.Vb,1.91e9))

and

dBm(yvalue(output.Vb.1.92e9))

Then you'll see the output power levels in dBm for the „fundamental frequency f2 = 1.91 GHz" in red and the „upper IP3 product at f = 1.92 GHz" in green



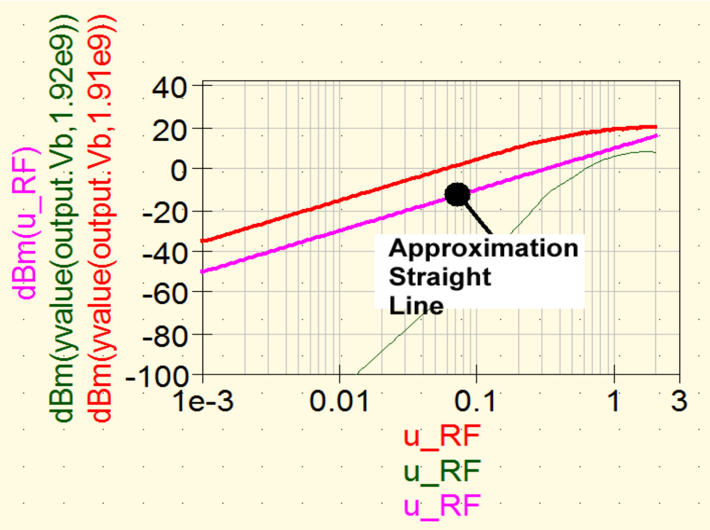
Remark:
If you want to calculate the frequencies of the third order distortion products then use the formula:

$$f_{IP3} = 2 \cdot f_1 - f_2$$

and you'll get f=1.89 GHz and f=1.92 GHz. The amplitudes and relationships are the same for both products.

But you need a logithmic horizontal x axis to see the linear part of both curves!

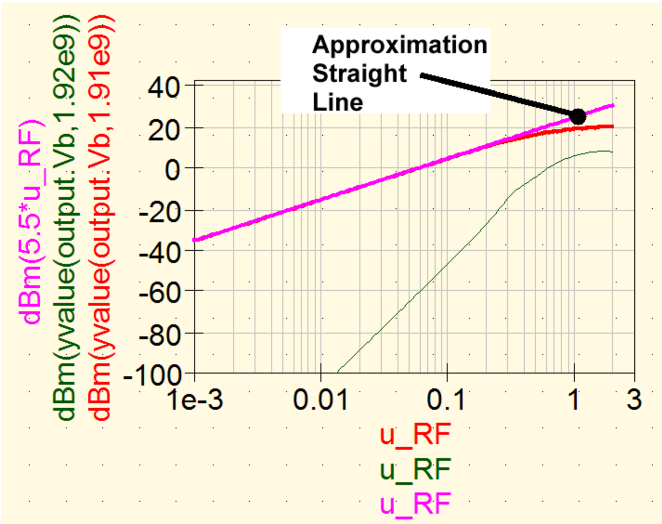
Now we must approximate every curve by a straight line, but that isn't very difficult.



Let us start with the fundamental frequency of f = 1.91 GHz and write a simple equation for the next graph:

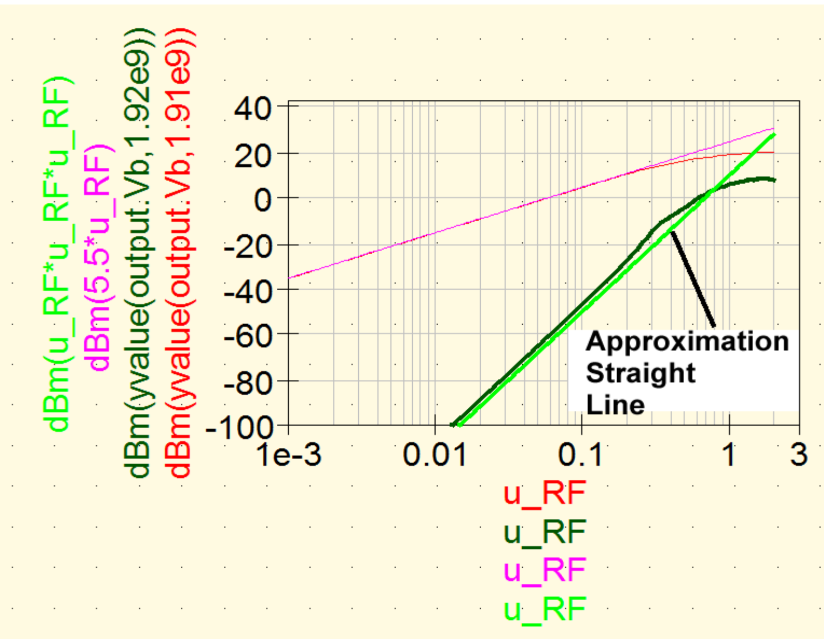
dBm(u_RF)

As you can see: the pink line must be a little bit shifted upwards and for „dBm“ scaling you have simply to multiply u_RF by a constant factor to do this.



The final solution is

dBm(5.5*u_RF)



For the IP3 product use the same procedure. But remember that this is a „third order product“ and „Third order“ means that you have to calculate

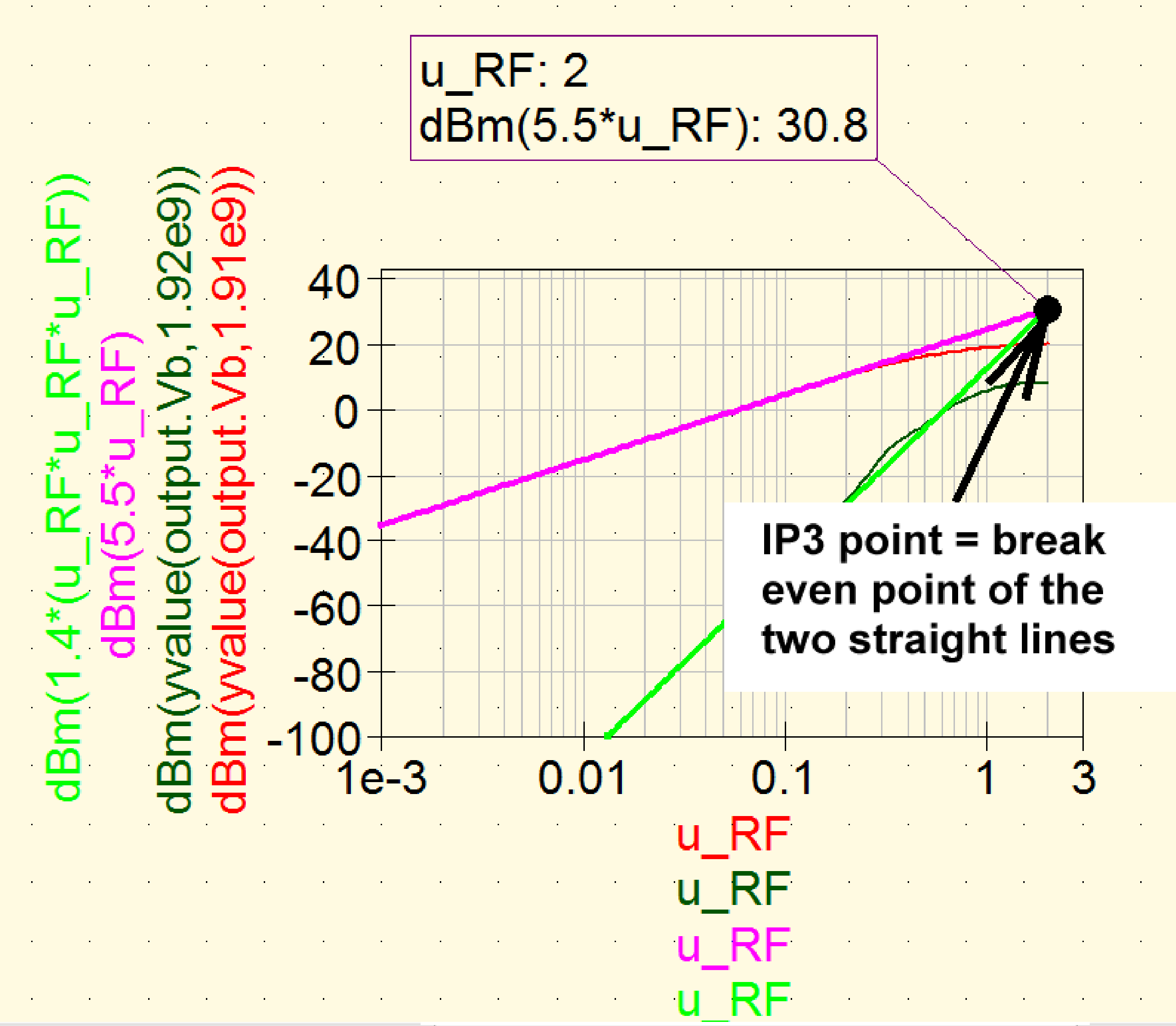
u_RF*u_RF*u_RF

by the equation. Afterwards shift the (green) line upwards by multiplying by a constant factor to coincide with the simulated IP3 product.

This is the final equation form for the green line:

dBm(1.4*(u_RF*u_RF*u_RF))

Then extrapolate the two lines (green and pink) up to the (theroretical) break even point and set a marker to this point!



Now we have found what we wanted to see:

The „Output Intercept Point OIP3“ has a value of +30.8 dBm (...calculated as peak value...) and (+30.8 dBm – 3 dB =) +27.8 dBm calculated as RMS value.

This is achieved at an input peak voltage value of **u_Rf = 2V** (...see the horizontal diagram axis).

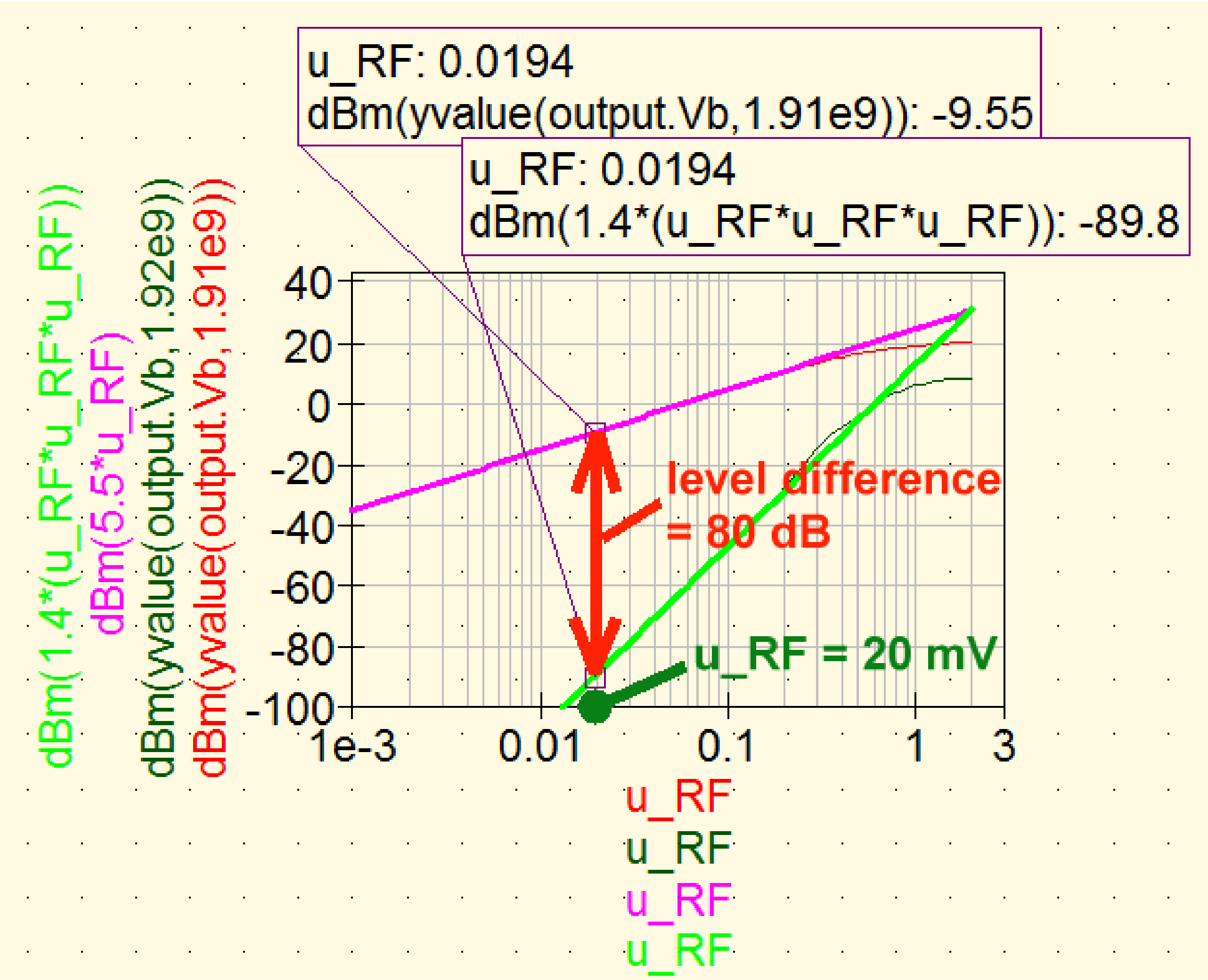
This would be an RMS value of **2V / sqrt(2) = 1.41V**

And this coincides with an RMS level of **20*log10(1.41 V/ 0.224V) dBm = +16,8 dBm**

Now we have found the „Input Intercept Point IIP3“ with +16.8 dBm.

Task:
An input voltage with a peak value of 0.02 V is applied to the input. Try to find the attenuation of the third order intermodulation produkt.

Solution:
Use the above diagram and draw (in your brain!!) a vertical line at the point for u_RF = 0.02 V starting at the horizontal axis.
Mark now the break even points of the vertical line with the two approximation straight lines in the diagram using two markers.
Calculate the level difference in dB.

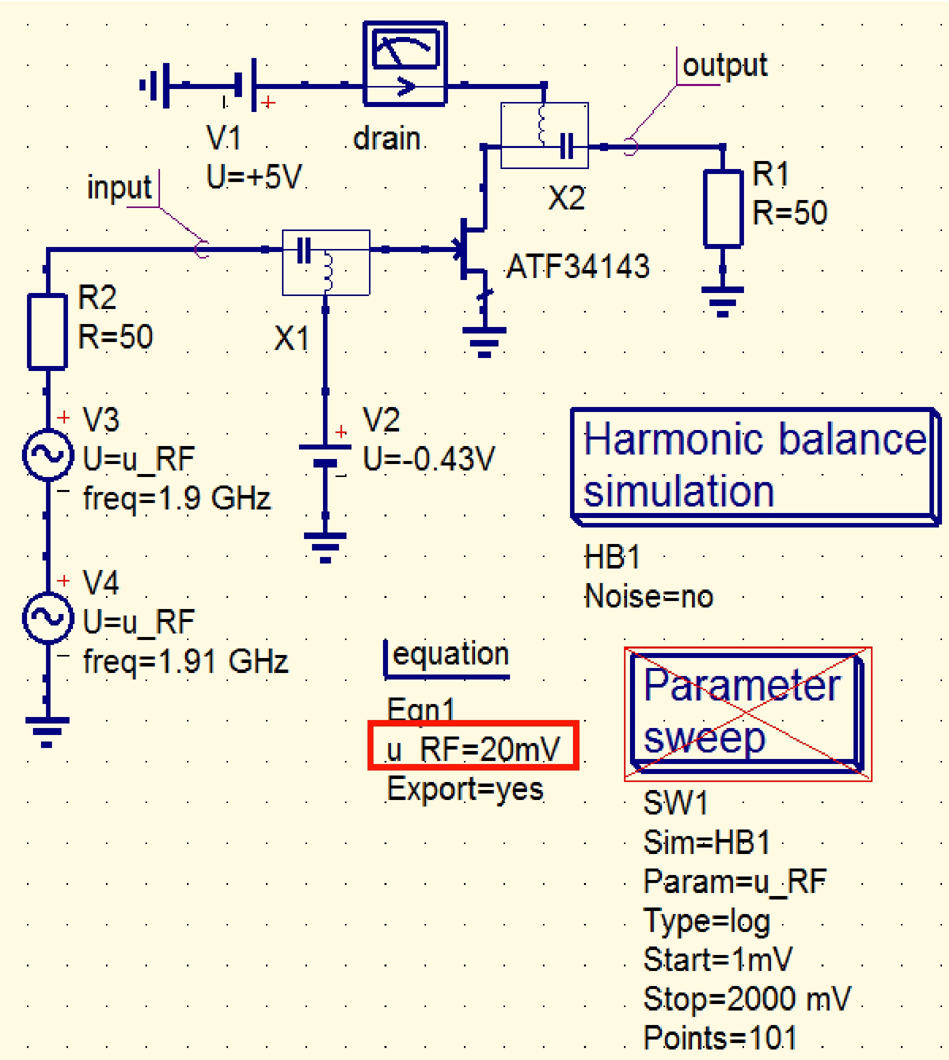


The level difference (= attenuation of third order product) is 80 dB.

Task:
Theory says that an increase of 20 dB of the input voltage u_RF causes an increase of 60 dB of the third order intermodulation product.
Thus the level difference (= attenuation of the third order product in relation to the input level) will be reduced by 40 dB in this case.
Please check this using the above diagram.

4.8.3. Simulation of OIP3 using the Output Spectrum

As already mentioned: for a spectrum presentation you have either to de-activate all parameter sweeps or to avoid the expression „yvalue“ in a Graph property equation!

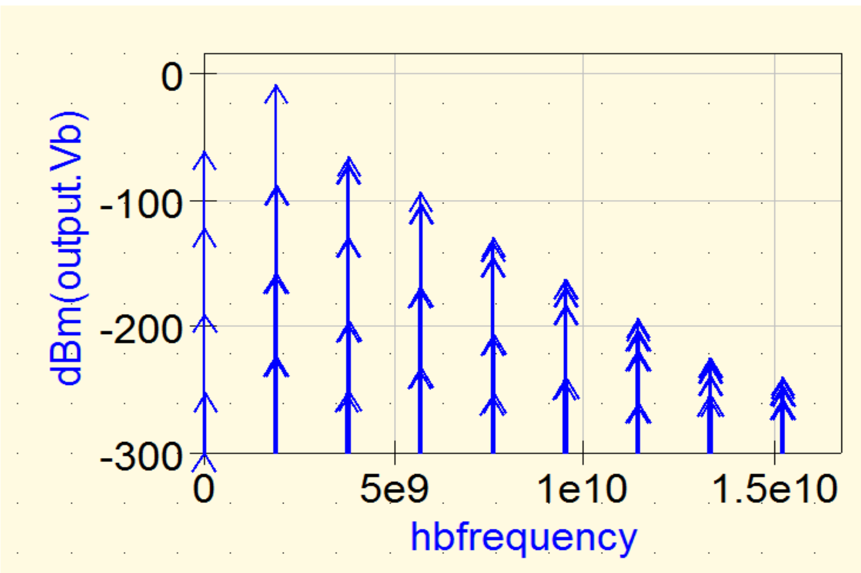


Let us use the same schematic as in the last chapter 4.8.2. (to simulate the IP3 point).

But now we write

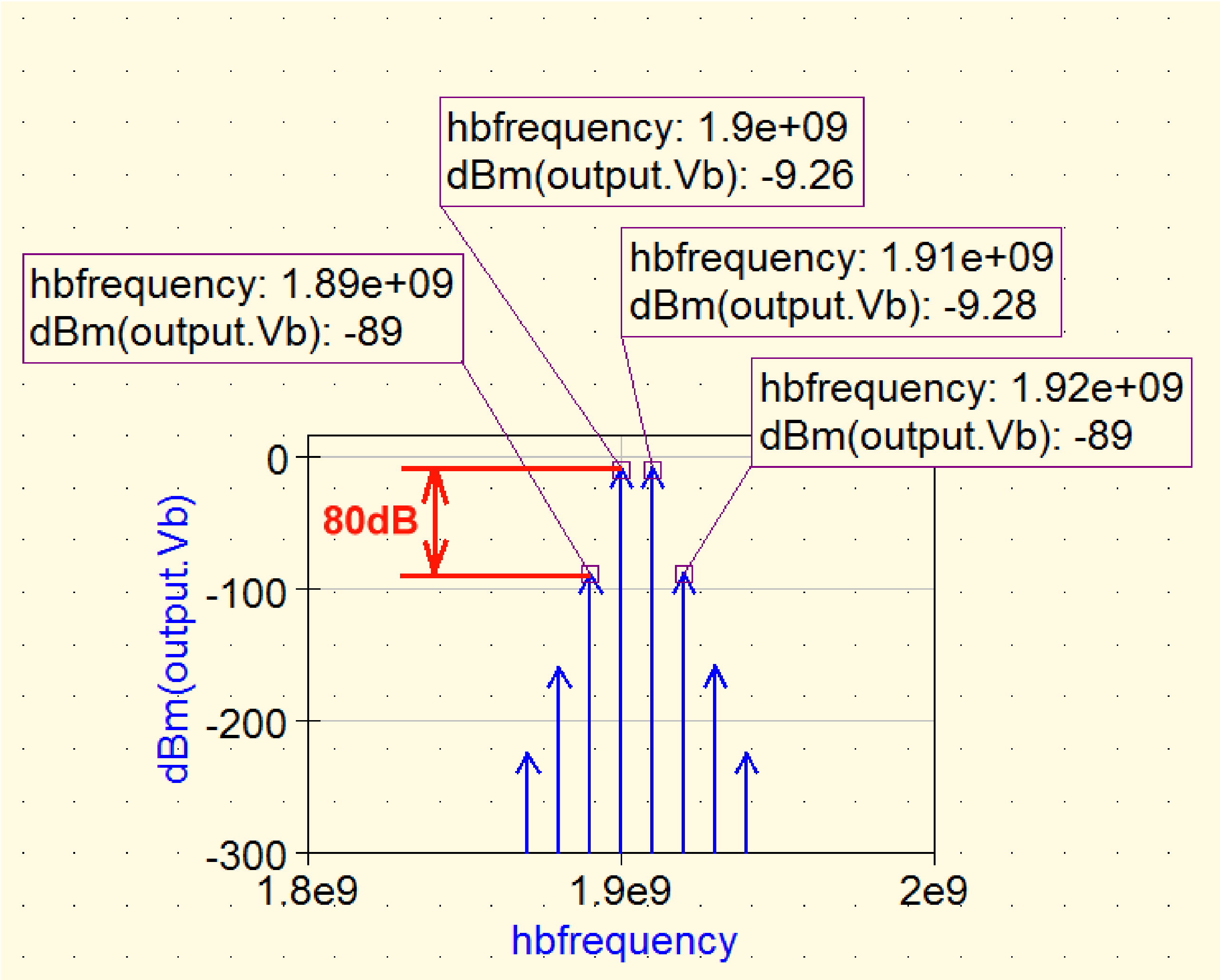
dBm(output.Vb)

as equation for the Graph properties.



This is the result and an additional analysis of the diagram is necessary...because if you have a sharp look at the lines you see them presented as arrows....they look a little strange...

Let us use the zoom function to present only the frequency range from 1.8....2.0 GHz!



This result is already well known!
We see the two amplified input signals with 1.9 GHz and 1.91 GHz at the output. Additionally we find the IP3 products on each side of them (frequencies: 1,89 GHz und 1,92 GHz). The level difference is exactly 80 dB – just like in the last chapter!

But now the vertical axis is scaled from -300 dB up to 0 dB. Thus we find additional products of higher order –.. and the slope of such a signal increases with the order of the intermodulation product.

So an „overload“ of an amplifier should strictly be avoided...

5. A Half Complex Mixer“ to generate an SSB Signal

This is a modern version of the good old „phase method of SSB generation“. But at first we have to discuss some new terms.

5.1. Analytic Pairs

Sounds mysterious, but isn't.
If you listen to music or a talk, a lot of signals are in action continually changing frequency and amplitude. Thus we use the expression „frequency band“ for such kind of signals. But this has an interesting property: you find a positive and a (mirrored!) negative band symmetrically arranged beside Zero Hertz!
If you now want to halve the bandwith: simply change to a **complex signal** = use either the positive OR the negative part. Is is done as follows:

a) The signal with the double sided spectrum is the **real signal** (..as coming from a generator, a microphone, a loudspeaker....) and is named **„I“ Signal** (= in phase signal). This signal must not be changed.

b) additionally you use a circuit named „**Hilbert Transformer**“. If you connect the real signal to its input, then you get at the output a **constant phase shift of exactly 90 degrees for all input frequencies**! This artificial output signal is named **„Q“ signal = „Quadrature Signal“**.

Using digital signal processing you have now two data streams named „I“ and „Q“ which are the two components of a complex signal! This is the famous analyctic pair. It can be defined as

I + j*Q

and you find only positive frequencies for „I + j*Q“ resp. only negative frequencies for „I – j*Q“

5.2. Example: a Half Complex Mixer to generate an SSB Signal

You need the following ingredients:

a) A **real information** **U = „I_{inf}“** (= speech, music..) which is converted to an Analytic Pair by the aid of a **Hilbert Transformer. Its form is now:**

$$U_{inf} = I_{inf} + j * Q_{inf}$$

b) A **already complex carrier signal** which can (due to Eulers law) also be written as

$$U_{carrier} = U_{carrier\_max} * e^{+j\omega_c t} = U_{carrier\_max} * [\cos(\omega_c t) + j * \sin(\omega c t)]$$

Attention: in practice you use the amplitude value „1“ for the carriers's amplitude and a Sine and a Cosine voltage source with the same frequency. This represents a signal with only one positive frequency.

.

c) Two **multiplying circuits.**

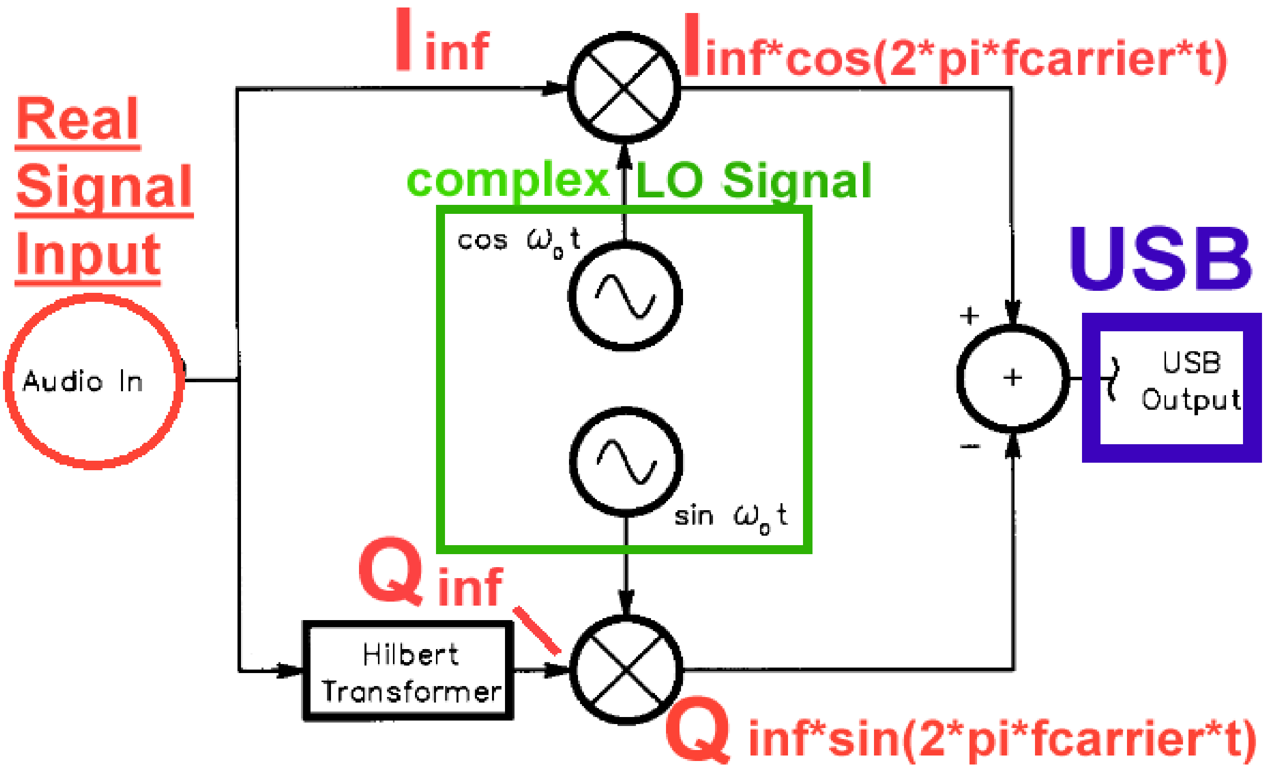
d) one **subtracting circuit**

Now multiply the Information's Analytic Pair by the **complex carrier with an amplitude value = 1 and only one positive frequency:**

$$U_{inf} * U_{carrier} = (I_{inf} + j * Q_{inf}) * [\cos(\omega_c t) + j * \sin(\omega c t)] =$$

$$[I_{inf} * \cos(\omega_c t) - Q_{inf} * \sin(\omega c t)] + j * [Q_{inf} * \cos(\omega_c t) + I_{inf} * \sin(\omega c t)]$$

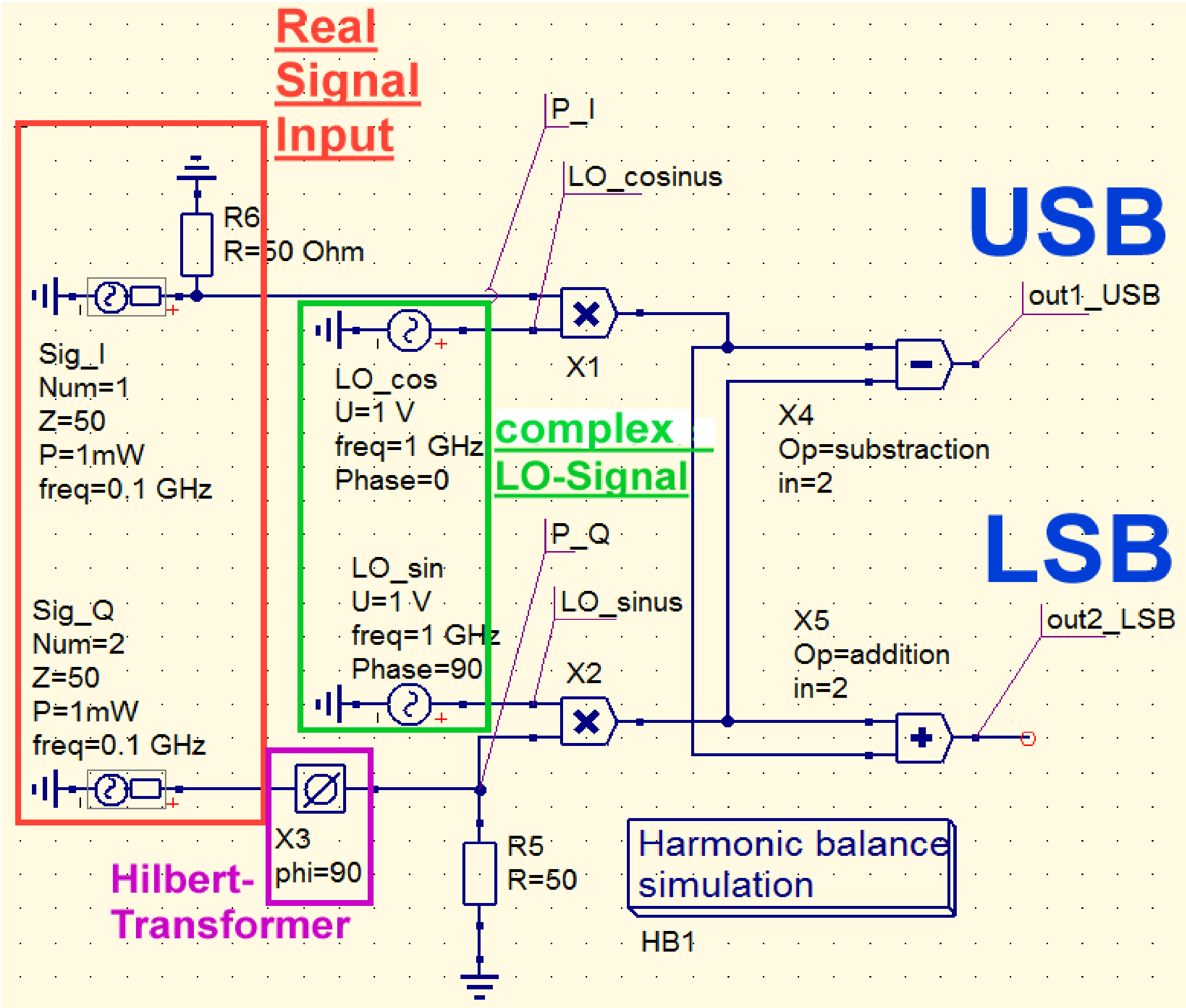
The **real part** of the product is **marked in red. Only this part (= USB = upper side band) can be transmitted into the air and this part must be realized by the following circuit.**



Remark:
The Hilbert Transformer with 90 degrees of constant phase shift is not complicated when realized in digital signal processing. But an analog version with a greater bandwidth.....can be a life task.....Sorry....

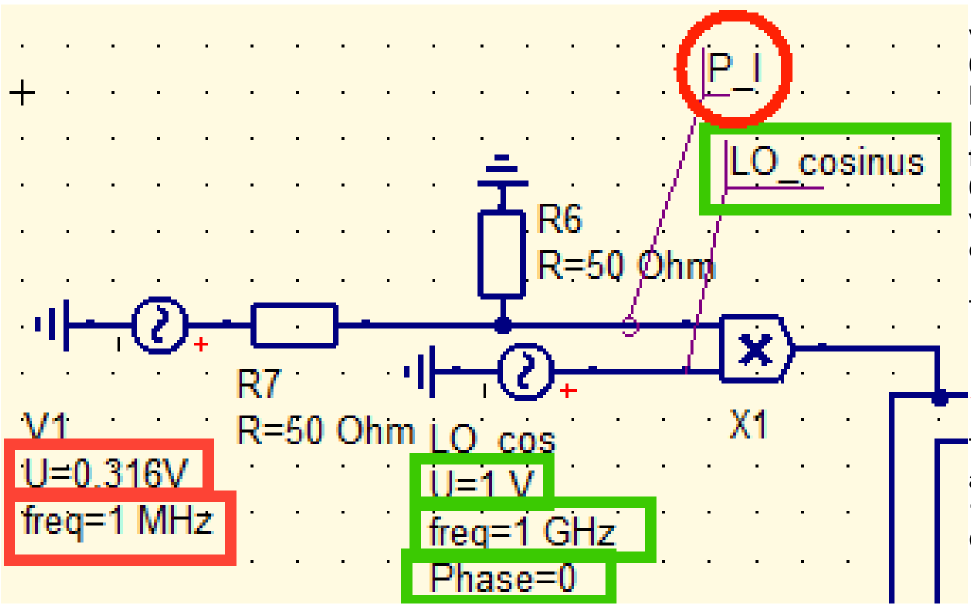
This circuit generates an „Upper Side Band = USB“
If you need the „Lower Side Band = LSB“, then change the sign of the signal in the lower part of the circuit. This can be done by replacing the subtracting circuit by an adding circuit.

Now realize this principle in qucsstudio (see next page...).



This is the correct realization and by using Harmonic Balance we'll find some interesting details.
The **real RF input signal is presented by two identical voltage sources**. The first source is the **„I“ Channel,** the second source **feeds the Hilbert transformer to generate the constant phase shift of 90 degrees for the Q Channel.**

This is the „I“ Channel including the multiplying circuit as regarded example:



The peak internal voltage value of the RF source is **0.316 V.**
But at the input of the multiplying circuit we find only the half value of 0.158 V due to the voltage divider consisting of R7 and R6.

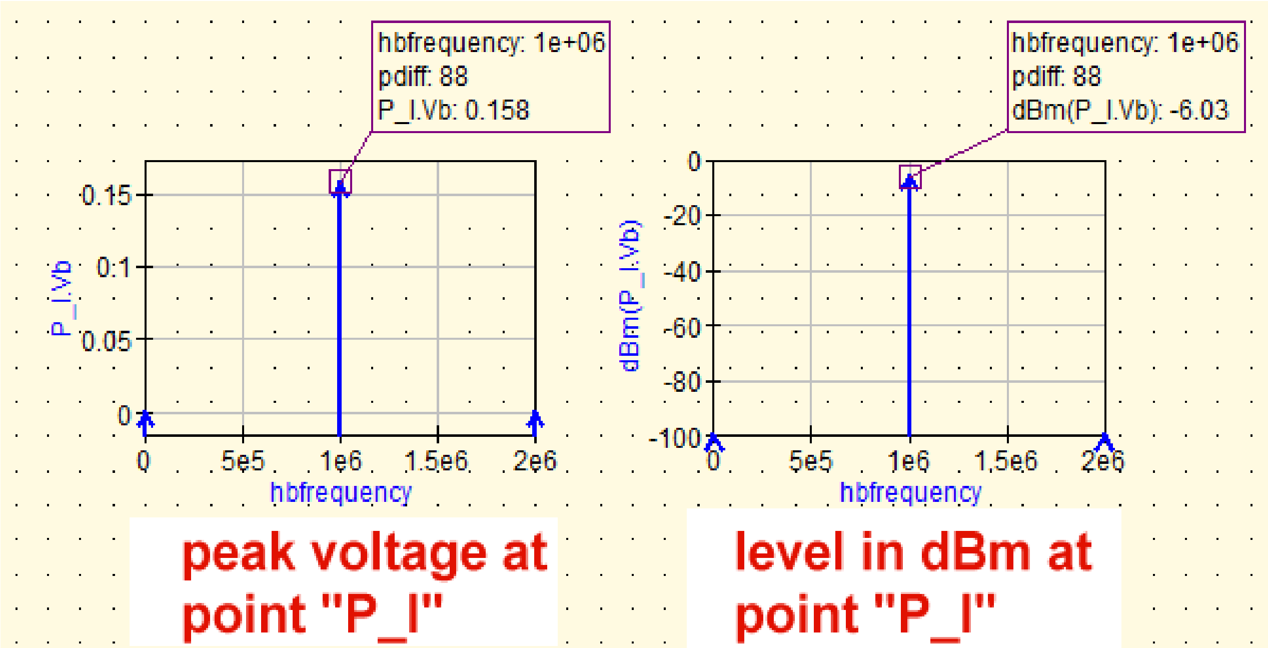
This input signal is named

„P_I“

The **LO signal has an amplitude peak value of 1 V and a frequency of 1 GHz.**

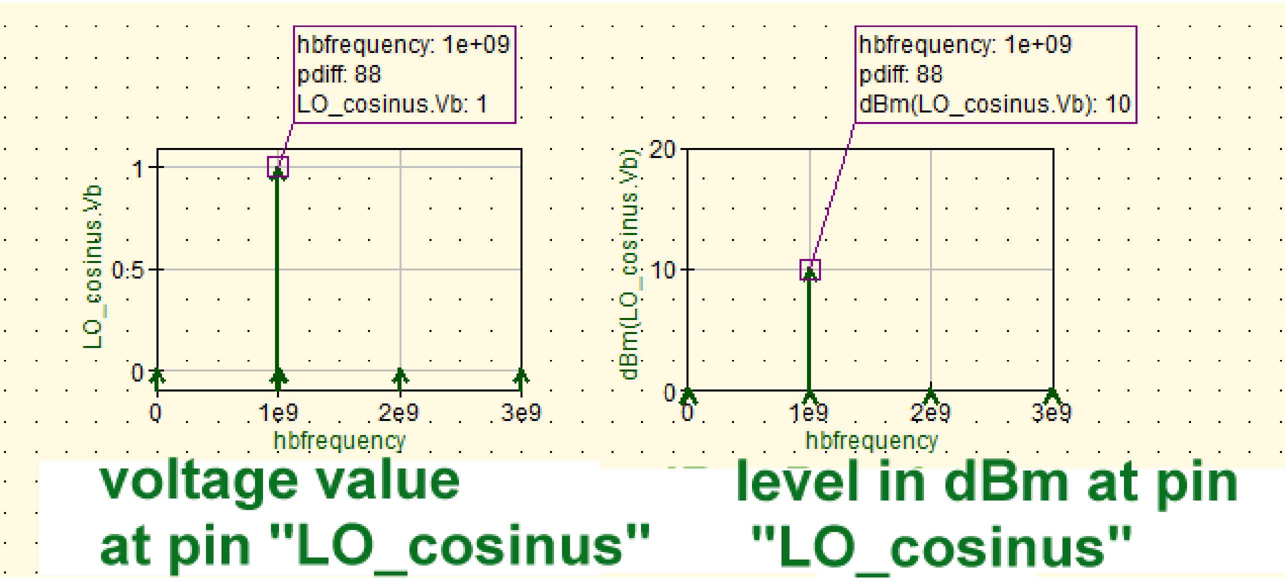
That is the evidence:

at point „P_I“ we observe a peak voltage of 0.158 V and this equates to a level of -6 dBm and the RF signal frequency is1 MHz.



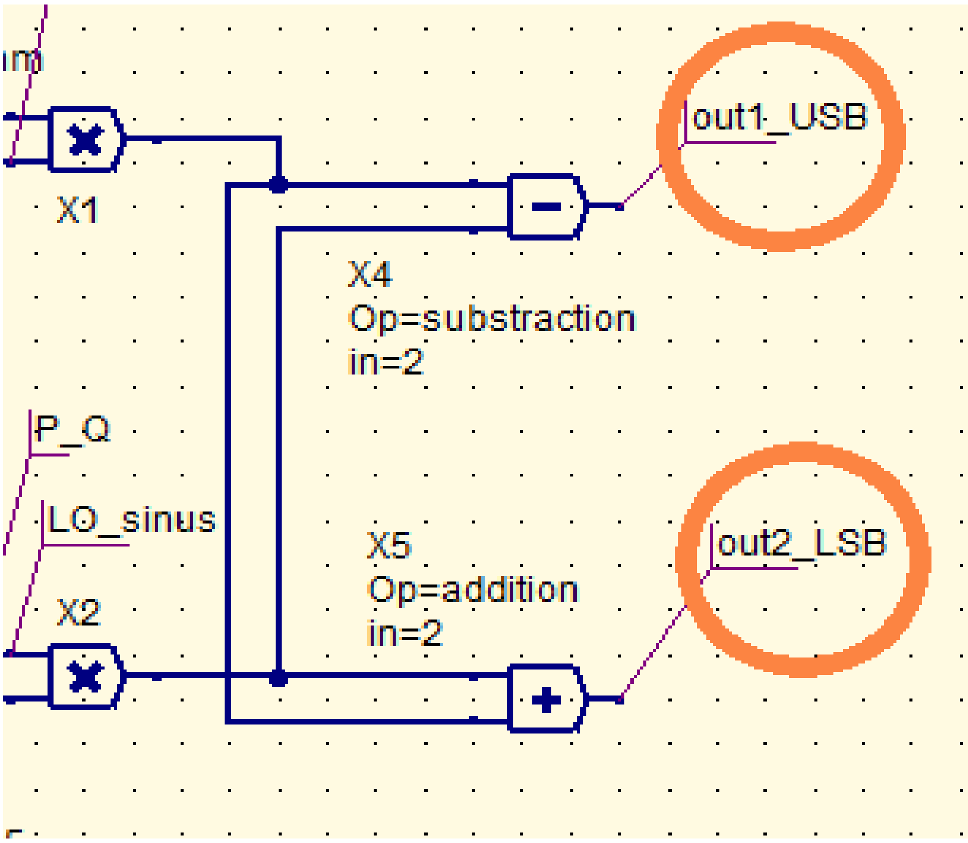
*

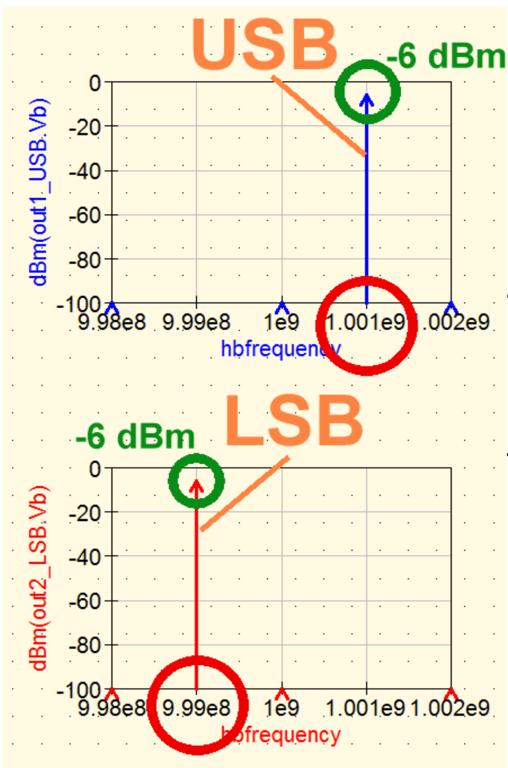
At the **„LO_cosinus“ pin we find a peak voltage of 1 V = a level of +10 dBm (**LO frequency is 1 GHz9



But the output is interesting:
We have to **add** the output signals of the two multipliers to get the **LSB signal.**

And a **subtraction** of the output signals gives the **USB** signal:





The levels of both sidebands equate to the RF input signal level (= -6dBm).

...and we get a sum frequency of 1001 MHz and a difference frequency of 999 MHz.

5.3. Parameter Simulation of the „undesired Side Band's Rejection“ for different Phase Errors

The rejection of an undesired side band depends on the exactness of the 90 degree phase shift of the Hilbert transformer. So we start a parameter sweep to show this rejection for phase shifts between 99 and 92 degrees.

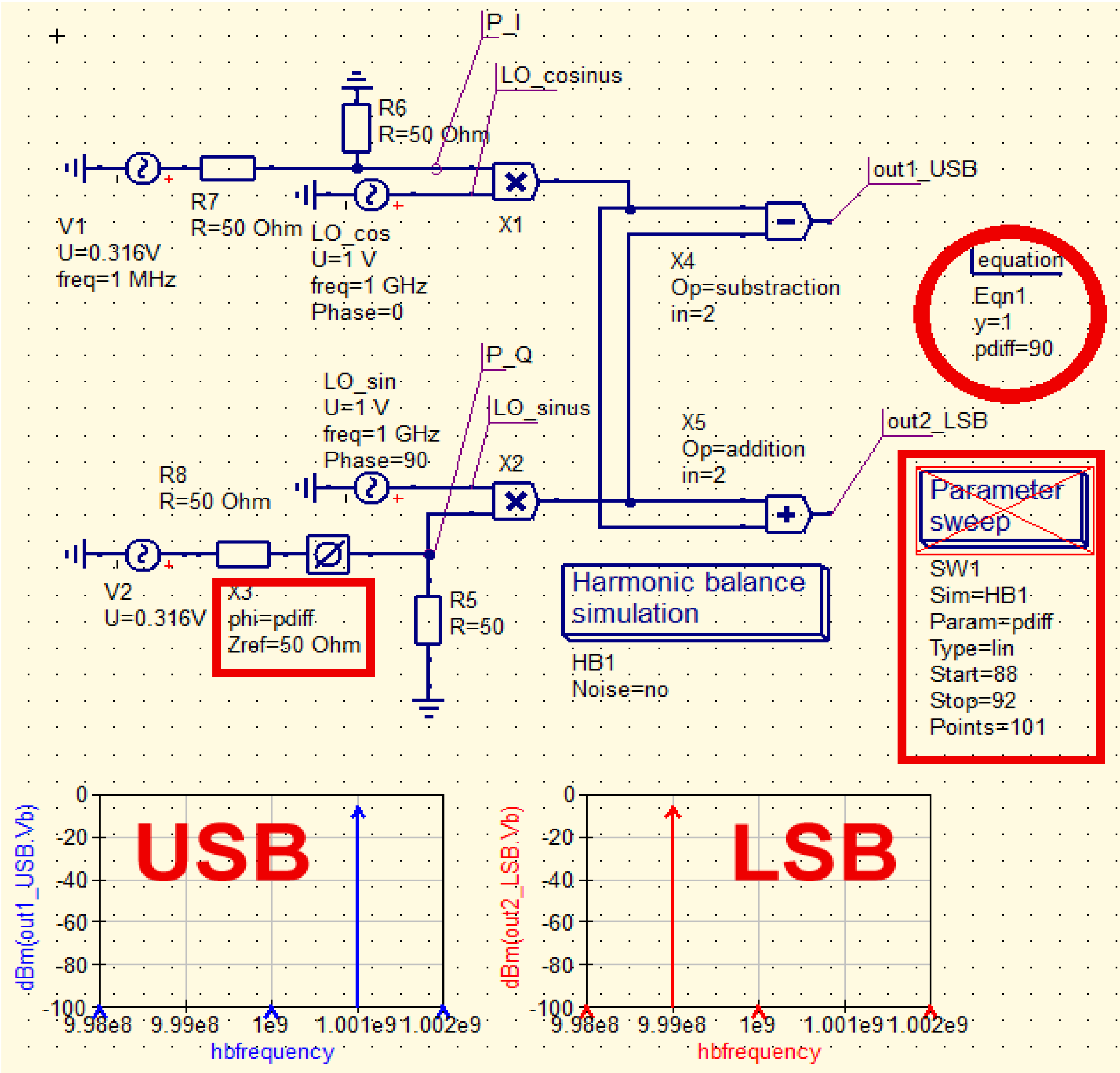
Please modify the schematic as follows:

a) In the property list of the Hilbert Transformer a **variable „pdiff“** is used for the phase shift.

b) We need an equation to set the **start value of the phase shift to 90 degreees.**

c) The parameter sweep **varies the angle „pdiff“ from 88 to 92 degrees using 101 steps**. durchgespielt.

Then de-activate the parameter sweep and simulate. Compare the result to the above simulation result – it should be identic.

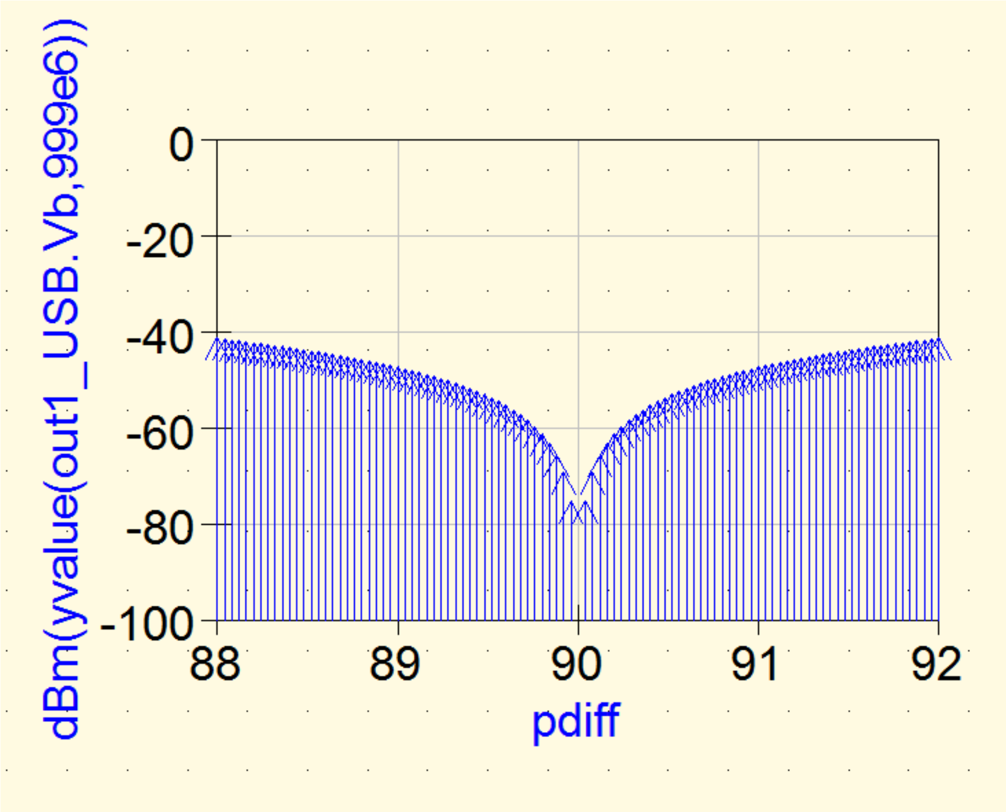


But now.....activate the parameter sweep and simulate.

We present in a cartesian diagram the USB signal but show only the **frequency range of the undesired LSB with f = 999MHz**. So we observe the level of this signal in relation of the phase error.

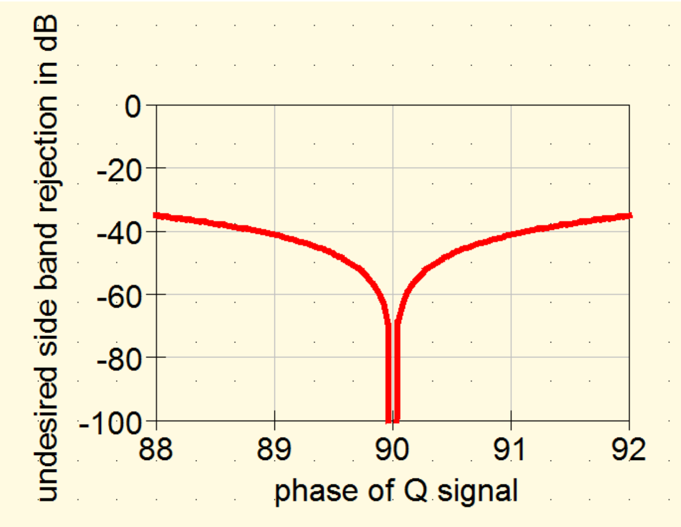
This is the necessary Graph property equation:

dBm(yvalue(out1_USB.Vb,999e6))



Remember that the level of the desired USB was „-6 dBm“. With this information you could now easily calculate the „undesired side band's attenuation“ – in your brain or by the modified formula

dBm(yvalue(out1_USB.Vb,999e6))+6



This would be rthe presentation example.

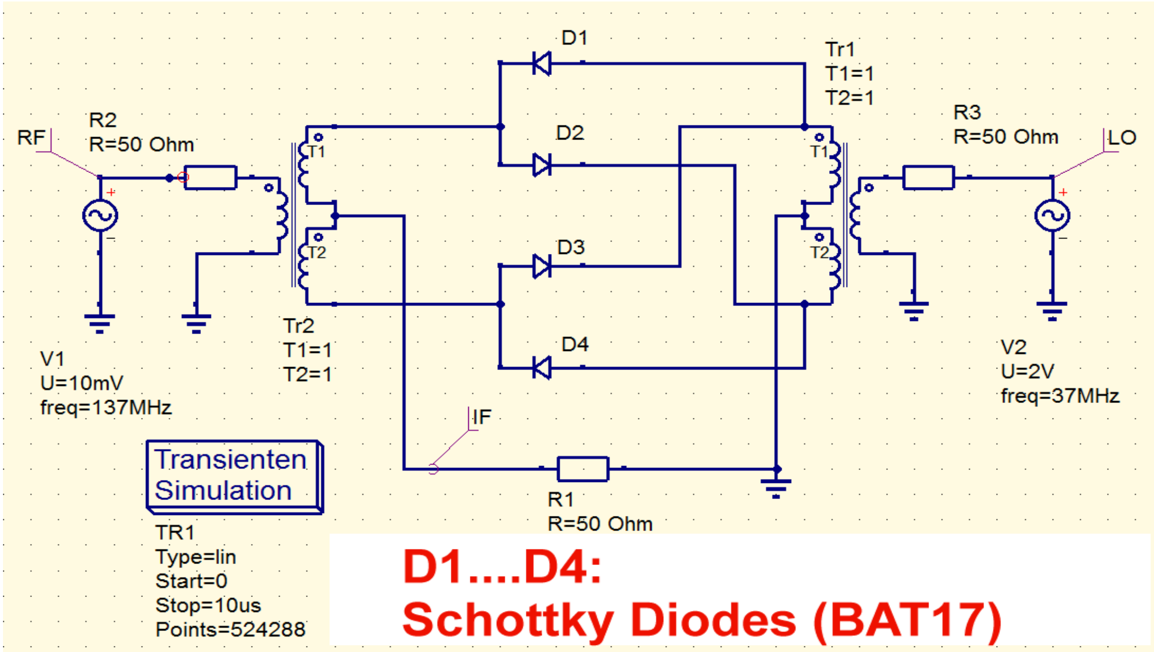
6. A Double Balanced Mixer (DBM)

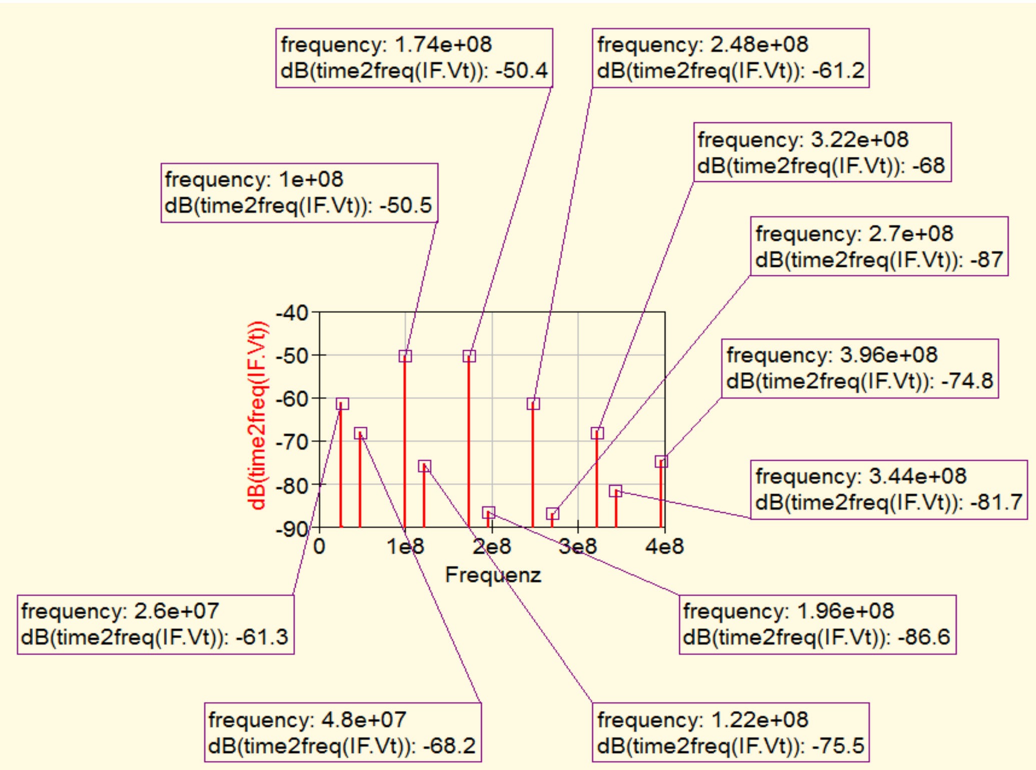
6.1. The Schematic (coming from chapter 14.3. in part 1 of the tutorial)

This circuit was investigated in the mentioned chapter and we have simulated

- **the output spectrum**
-
- **the conversion loss**
-
- **the IP3 point**

What can HB add to this information?





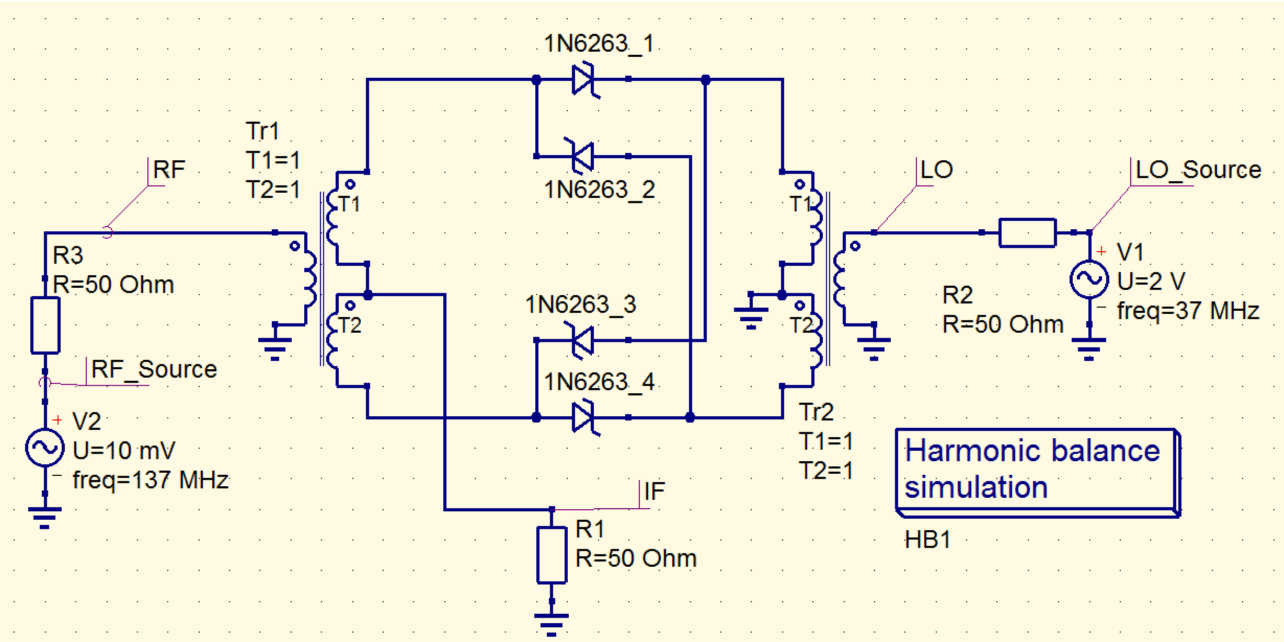
This was the simulated output spectrum in chapter 14.3 of the tutorial's part 1.

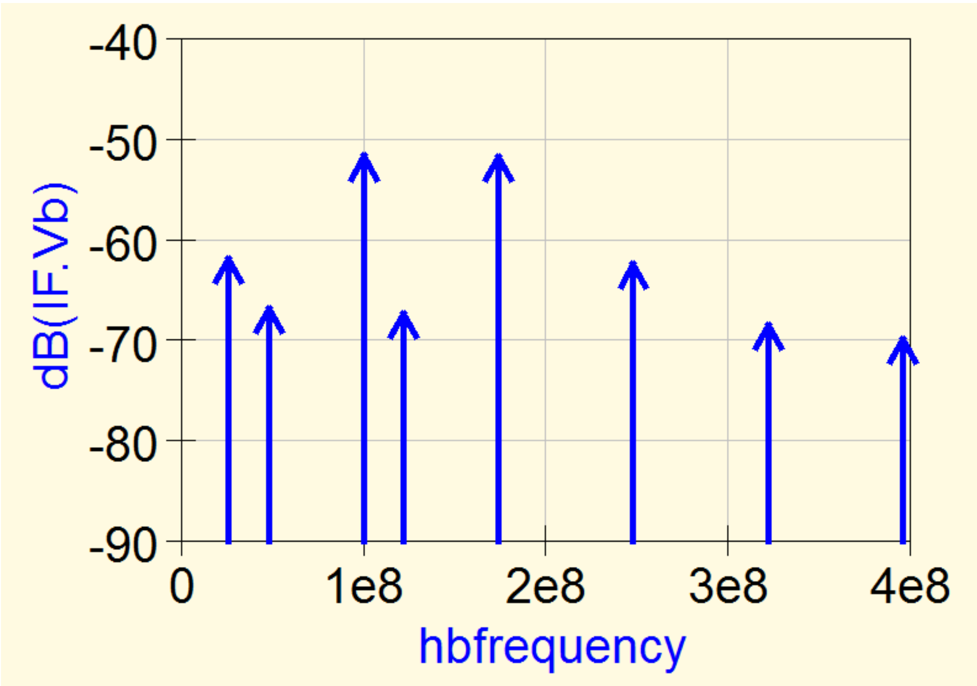
(calibration still in „dBV“)

6.2. Used Schematic for the Harmonic Balance Simulation

There is only one modifaction: now **Schottky diodes „1N6263“ are used because they switch faster and show lower capacitance values.** They can be found under „Schottky diodes“ in the qucsstudio diode library. These are the properties

silicon schottky diode
60V, 50mA, 0.1ns, 2.2pF @0V
Manufacturer: ST Microelectronics





This is the simulated output spectrum scaled in „dBV“ .

Please compare it now to the simulation in the Time Domain, followed by an FFT (found on the previous page)

Voltage sources properties:

a) RF input
peak voltage = 10 mV
frequency = 137 MHz

b) LO signal
peak voltage = 2 V
frequency = 37 MHz

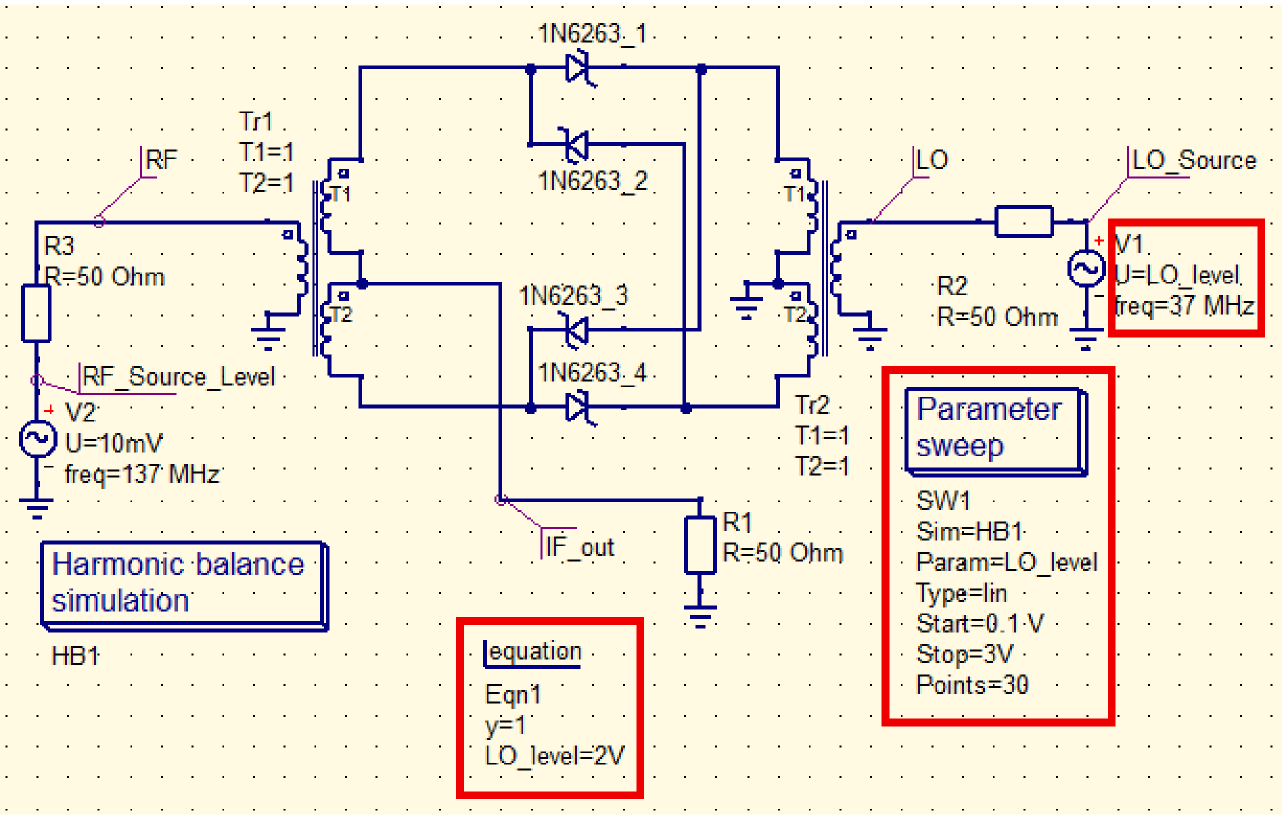
Harmonic Balance is really a fine toy...

6.3. Harmonic Balance Parameter Sweep of the LO Level

6.3.1. Output Spectrum for different LO Amplitudes

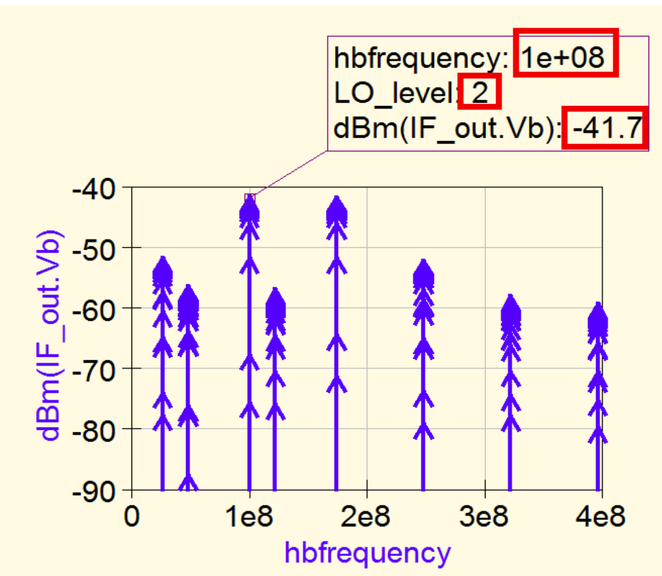
For a successful simulation we need a variable for the internal voltage of the LO signal source and name it „LO_level". Using equation „Eqn1" the start value is **$\hat{U}_{LO}$ = 2V.**

The parameter sweep starts at a **minimum voltage value of $\hat{U}_{LO}$ = 0.1 V and rises up to 3 V in steps of 0.1 V.**



Now start the simulation and show **„dBm(IF_out.Vb)" for a range of -90 dBm up to -40 dBm in the frequency range from 0 to 400 Mhz.**

Use a marker for the LSB signal.



The spectral lines look like old christmas trees but this is easy to explain:

The different simulation results at a regarded frequency are written „one upon the other". You see that if you shift the marker and read the entries in the marker field.

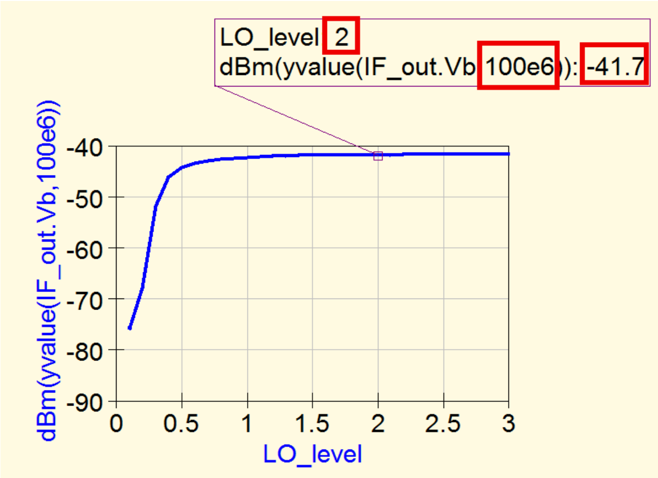
Shown example for the used marker position

LSB frequency= 100 MHz

LO level = 2 V

Level of „IF_out.Vb" for f = 100 MHz is -41,7 dBm

You can also indicate the **LSB level at f = 100 MHz versus the LO level.**



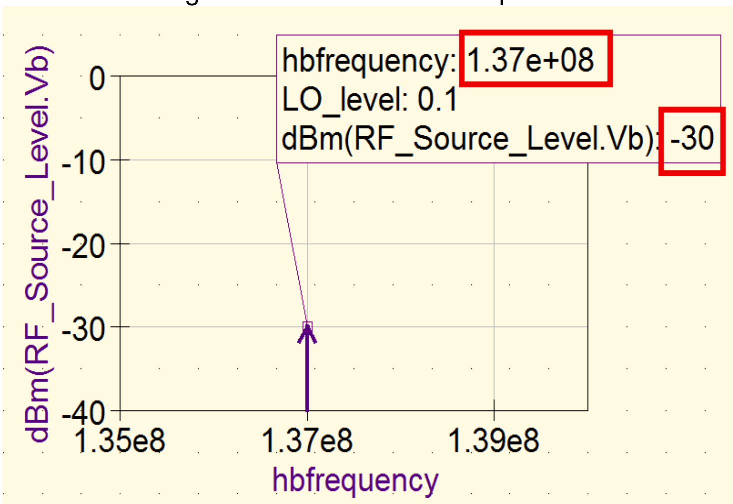
Therefor use the equation for the Graph properties:

dBm(yvalue(IF_out.Vb,100e6))

The marker is set again to a LO voltage value of 2 V. Please compare to the last diagram...

6.3.2. Presentation of the Conversion Loss

The internal voltage of the RF source has a peak level of 10 mV. This can be converted to dBm by the simple equation



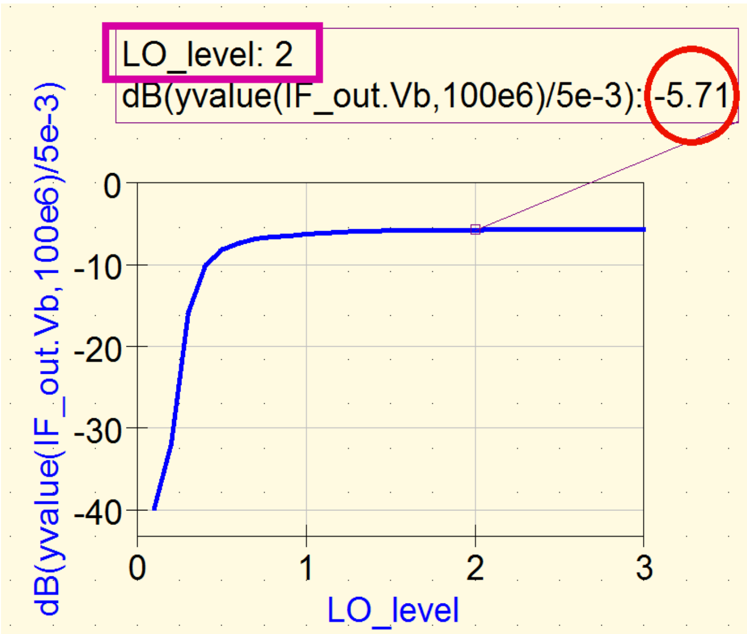
dBm(RF_Source_Level)

and the result is a level of -30 dBm at f = 137 MHz(see diagram)

The Incident Wave can be calculated by the well known relationship

„Incident Wave = Uo / 2 = RF_Source_Level/2"

Thus the Incident Wave Level (which arrives at the mixer input pin) must be 6 dB down and has a value of **-36 dBm (= 5 mV)**



Now write the equation to calculate **S21** for different LO levels:

dB(yvalue(IF_out.Vb,100e6)/5e-3)

You see:
The conversion loss is 5,71 dB for a LO peak voltage value of 2 V (See the marker field).

The „IF_out" - level for f = 100 MHz must now be

-36 dBm – 5,71 dB = -41,7 dBm

See previous page....

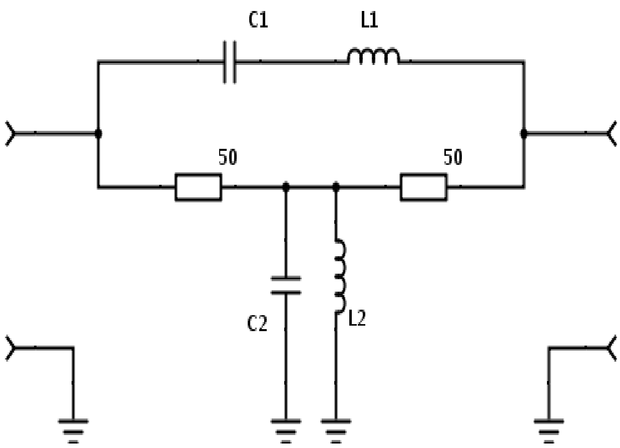
6.3.3. Addition of a Diplexer to the Mixer Output

Double Balanced Mixers equipped with diodes must have a perfect broadband 50Ω termination at every port to reduce distortions and to show the specifications given in the data sheet.

This can be done by a circuit named „Diplexer“ with the following properties:

a) a **band pass filter characteristic** for the **IF-Signal output** (here: f = 100 MHz), but

b) a perfect 50Ω termination for the mixrer output – for the complete regarded frequency range



Online calculators for diplexer can be found on the WWW.

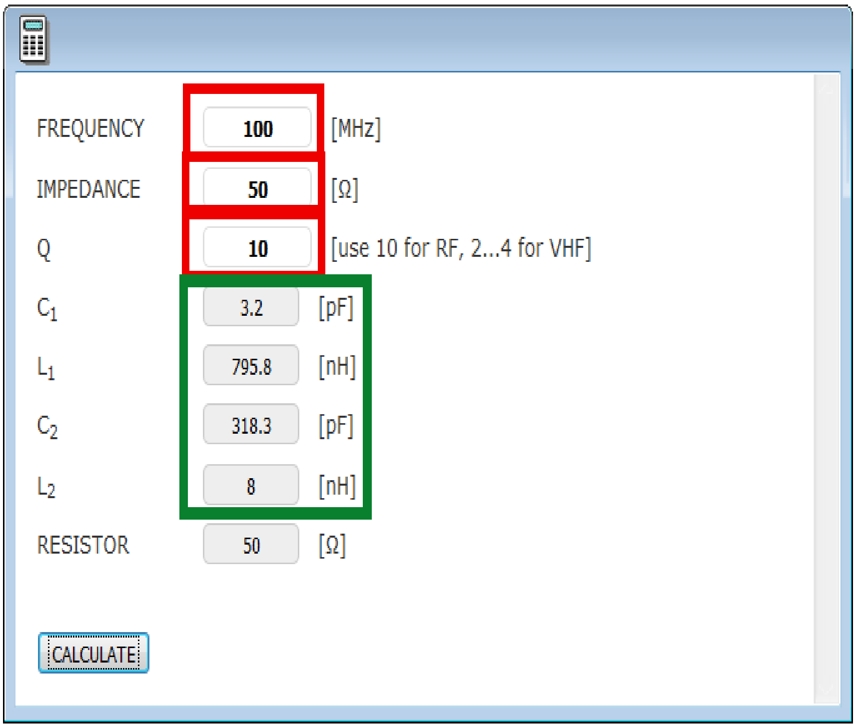
This design comes from

http://www.changpuak.ch

At first enter the IF frequency of 100 MHz

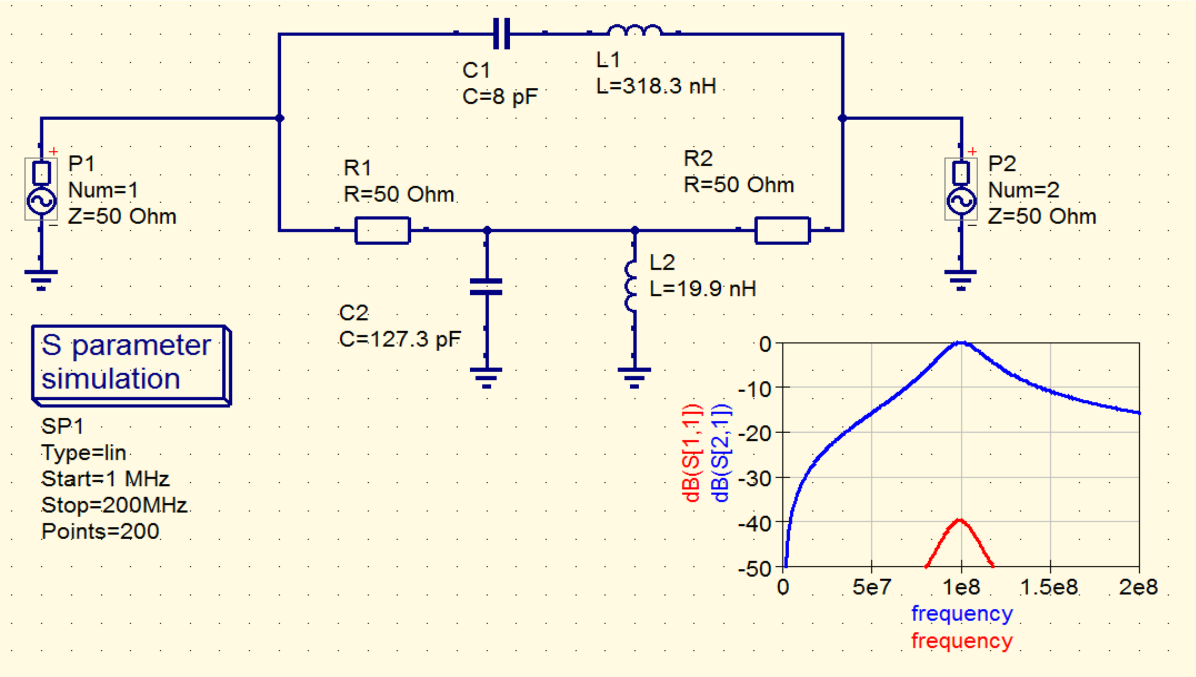
then the characteristic system impedance of Z = 50 Ω

and at last the quality factor Q = 10 as proposed by the calculator for RF circuits.

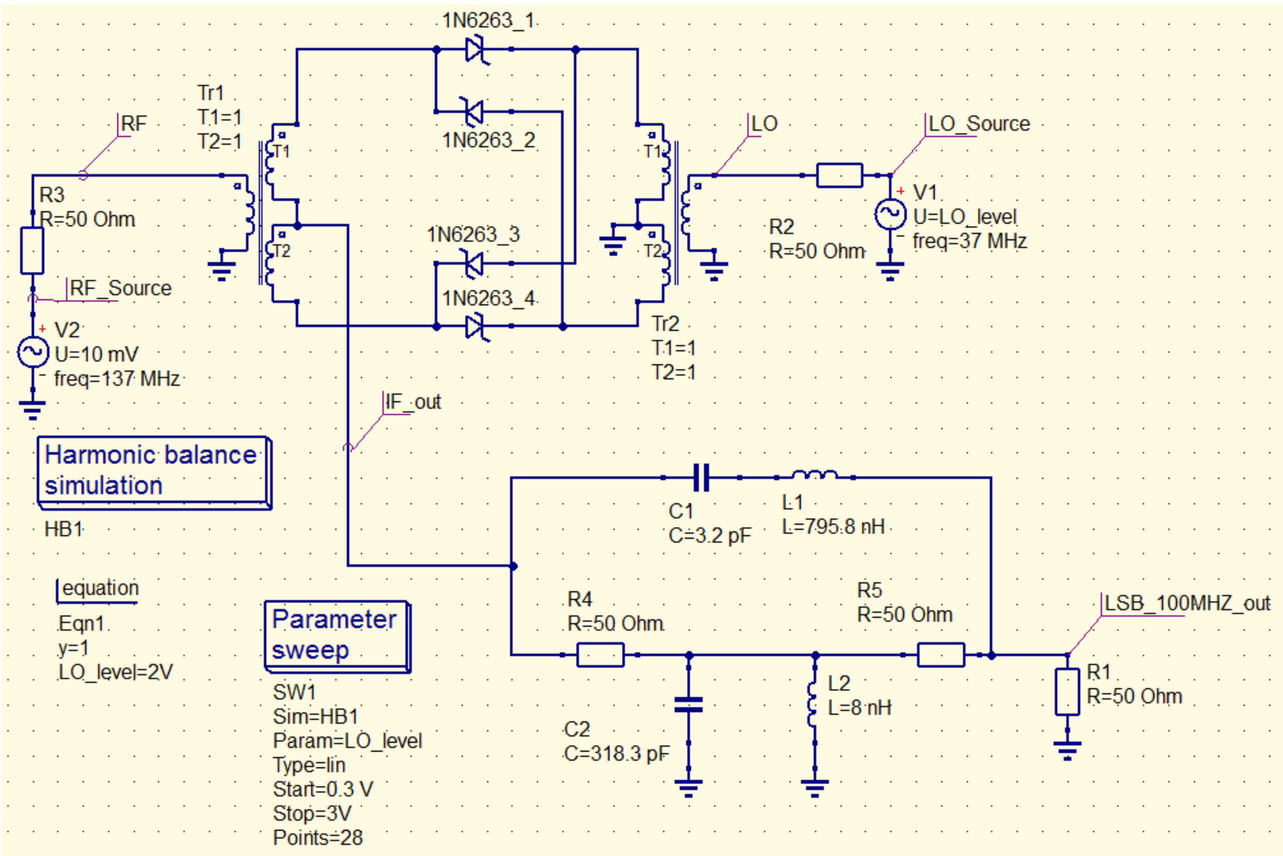


Then draw a schematic to simulate the diplexer with qucsstudio.

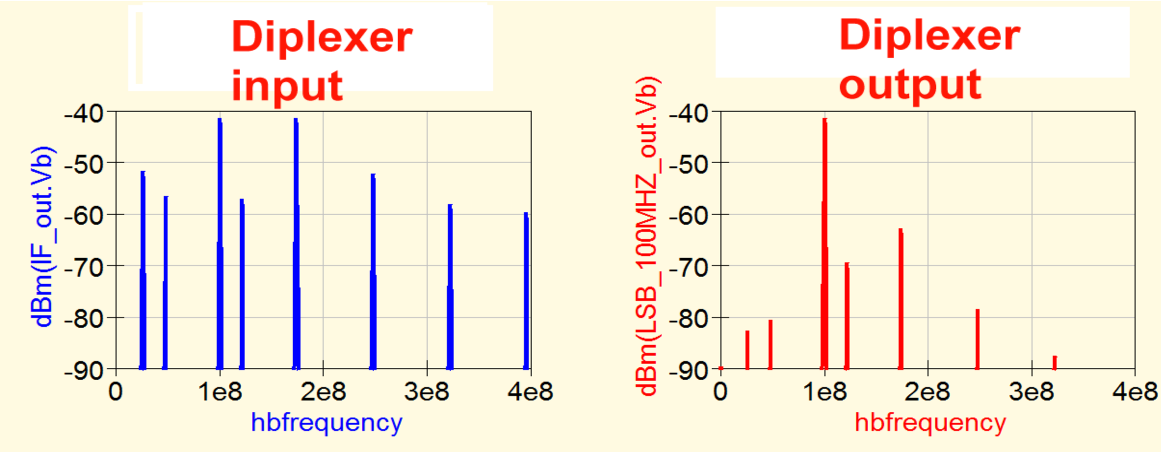
Simulate the S parameters S11 and S21 from 0 to 200 MHz.



The Input Reflection S11 is below -40 dB in the complete frequency range. And if we could use lossless parts, S21 would be 0 dB for f = 100 MHz.
Now combine the mixer and the diplexer.



The simulated spectres for the input and the output of the diplexer can be found on the next page. A full success....

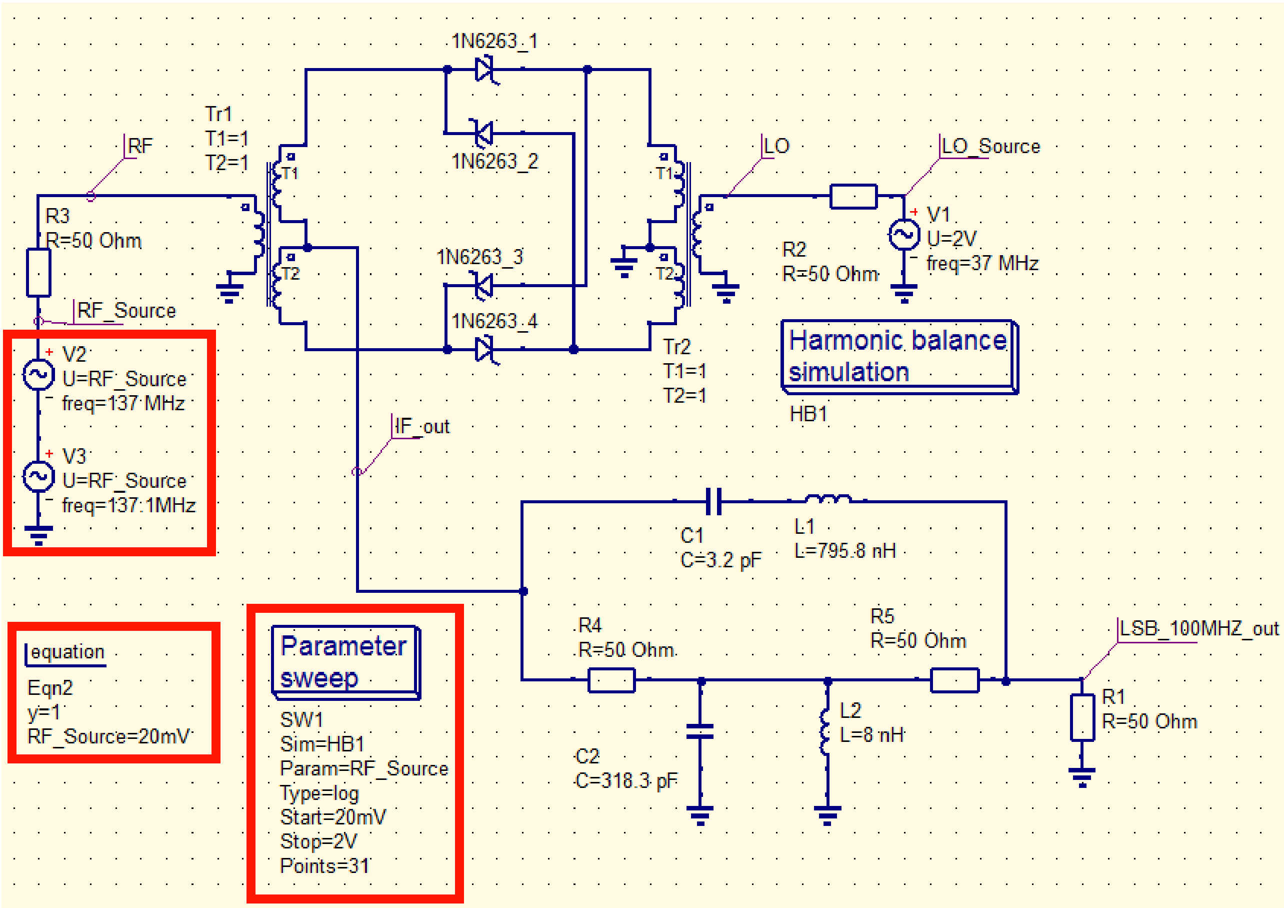


6.3.4. Finding the IP3 Point for a LO Peak Voltage of 2V

This problem was solved for an amplifier in chapter 4.8. and we use the same procedure for the mixer circuit.

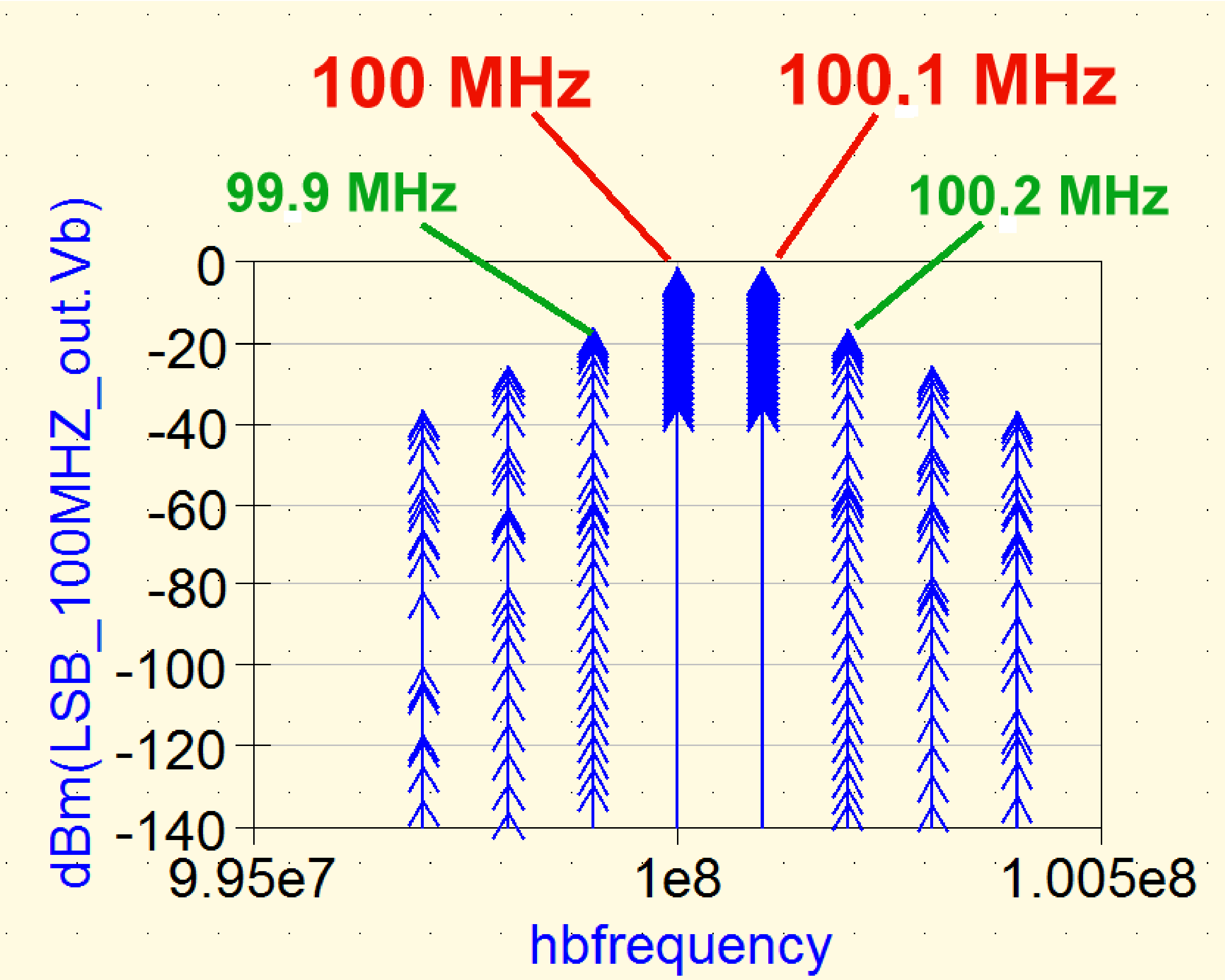
a) At the RF input two voltage sources are connected in series. The sources use the **same peak voltage value „RF_Source" = 20 mV as starting value.** The input frequencies are **137 MHz and 137.1 MHz – thus we find a frequency difference of 0.1 MHz.**

Now a HB parameter sweep for „RF_Source" = 20 mV ......2 V is started.



This simulation is time consuming but the result is fine.
Please show the simulated **spectrum around f = 100 MHz at the diplexers output by using this equation and a frequency range from 99.5 to 100.5 MHz for the horizontal axis:**

dBm(LSB_100MHz_out.Vb)



The **LSB signals with f = 100 MHz and 100.1 MHz** can easily be found in the diagram. Examining the „tops of the arrows“ we see that the value range starts at -37 dBm and rises up to nearly 0 dBm.

The **„IP3 intermodulation products“ are found at 99.9 MHz at 100.2 MHz**. Their amplitude values are nearly invisible for small values of the input voltage level „U_RF“, but thenD

To see the details we regard only **the IP3 product at 99.9 MHz and the adjacent LSB signal at 100MHz** for the parameter sweep.

Then followes the game „how to find the approxmation straight lines for the simulated curves“ using four different equations:

a) for the **LSB signal at f = 100 MHz**: **dBm(yvalue(LSB_100MHZ_out.Vb,100e6))**

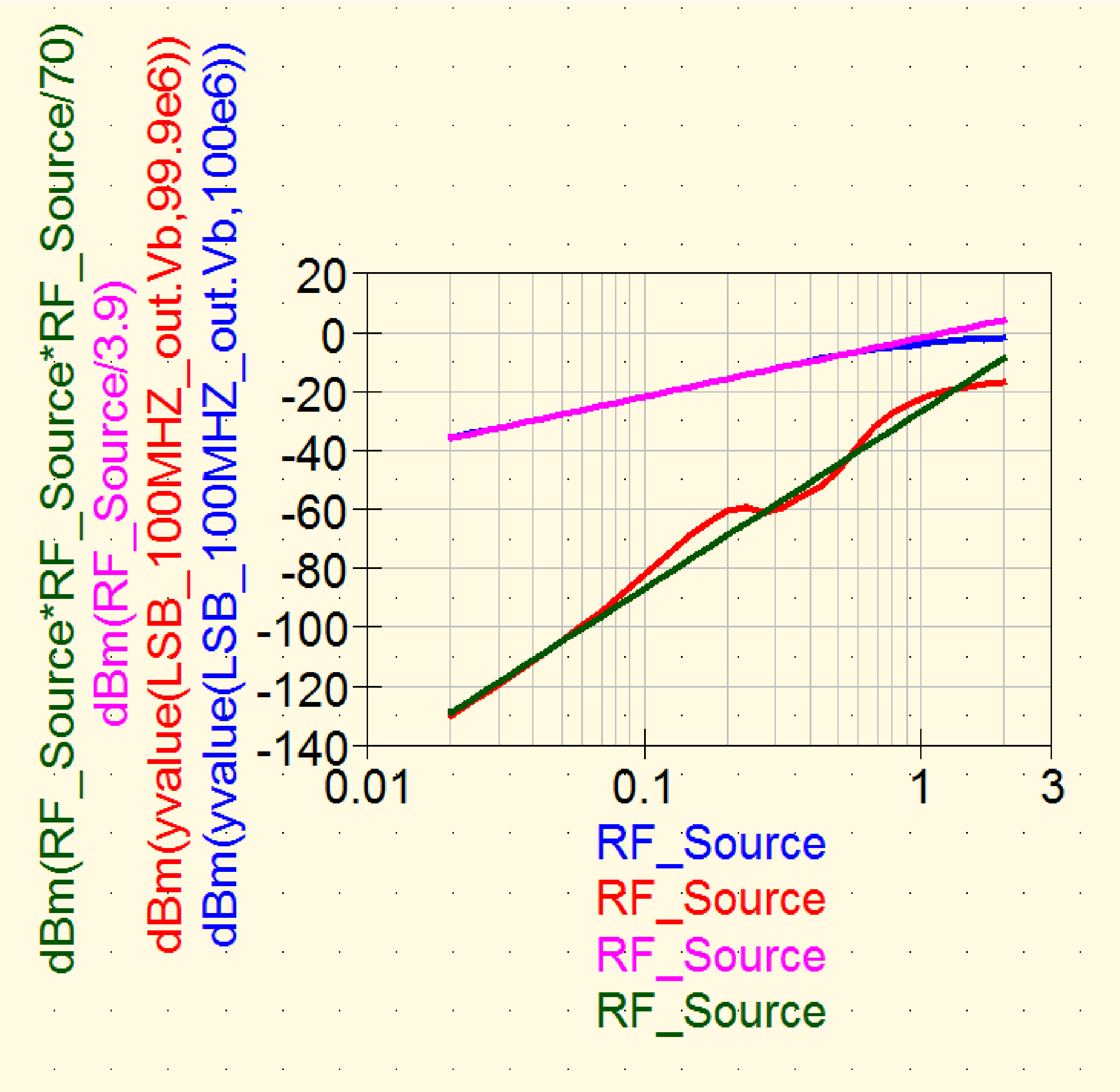
b) for the associated approximation straight line for this curce: **dBm(RF-Source/3.9)**

c) for the IP3 product at **f = 99,9 MHz**: **dBm(yvalue(LSB_100MHZ_out.Vb,99.9e6))**

d) for the associated approximation straight line for this curve:

dBm(RF_Source*RF_Source*RF_Source)/70)

The result:



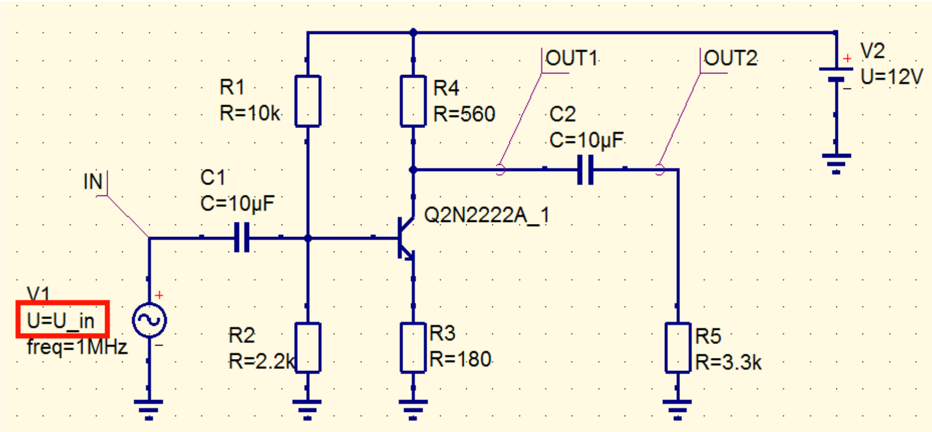
The break even point of the straight lines lies at the right hand side at more than „U_RF" = 3V (peak value). The associated output level is ca. +10 dBm at the left vertical axis.

This means that the output IP3 point has a value of ca.

+10 dBm

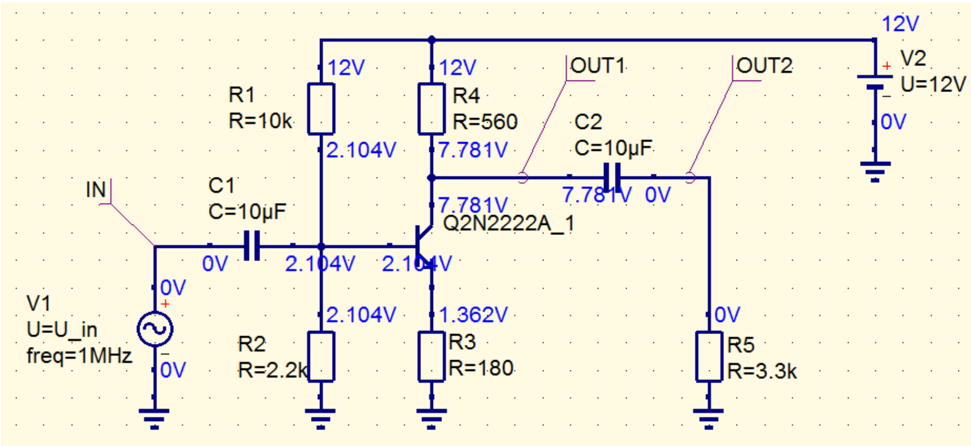
7. Summary: Harmonic Balance Simulation in Question and Answer for an Amplifier Stage

7.1. Which Circuit?



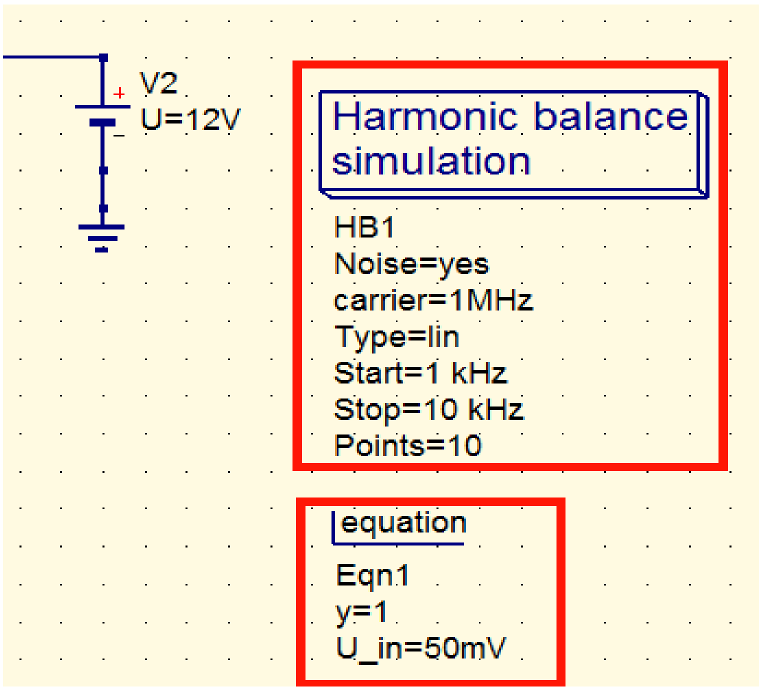
We use the example „**simple_amplifier.sch**“ as found in the qucsstudio homepage (which is a good choice).
But the amplitude value of the input signal must be replaced by a variable **„U_in“.**

7.2. How can I quickly find every DC Voltage of the Operating Point?



In the task bar you find the well known gear wheel for the start of the simulation.
At its left hand side the same gear wheel but over-printed in red with „DC“.
A left hand click on it calculates all DC voltages (See this illustration).
A further click on to the gear wheel lets all entries disappear.

7.3. What are the necessary Preparations for a successful HB Simulation?



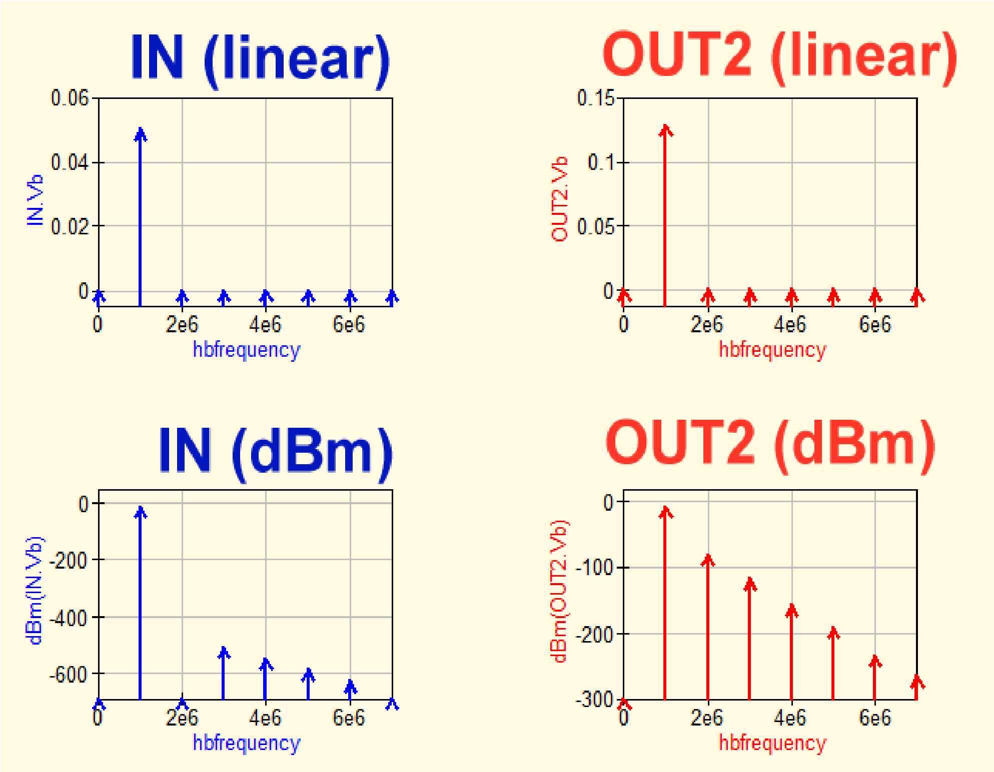
You need at first the „Harmonic balance directive“ with the necessary entries for a successful noise simulation (see this illustration).

Then write an equation „Eqn1“ with the **initial value of 50 mV for the input voltage „U_in“.**

.

Now start the simulation.

7.4. How can I present the Input Voltage and the Output Voltage including the gain (= linear / in dB / in dBm)?



Open a cartesian diagram and enter „**IN.Vb**“ as graph property.

Repeat this and enter the following equations :

OUT2.Vb

dBm(IN.Vb)

dBm(OUT2.Vb)

And now to the gain:

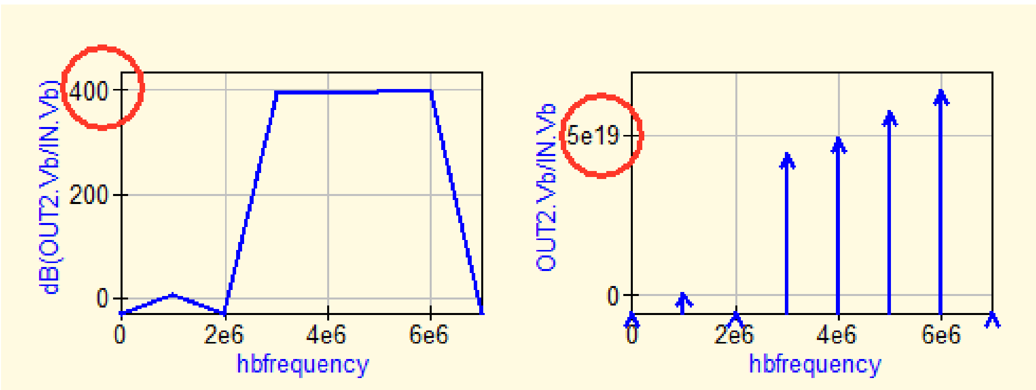
Use two cartesian diagrams. Write the graph equation

OUT2.Vb / IN.Vb

and

db(OUT2.Vb / IN.Vb

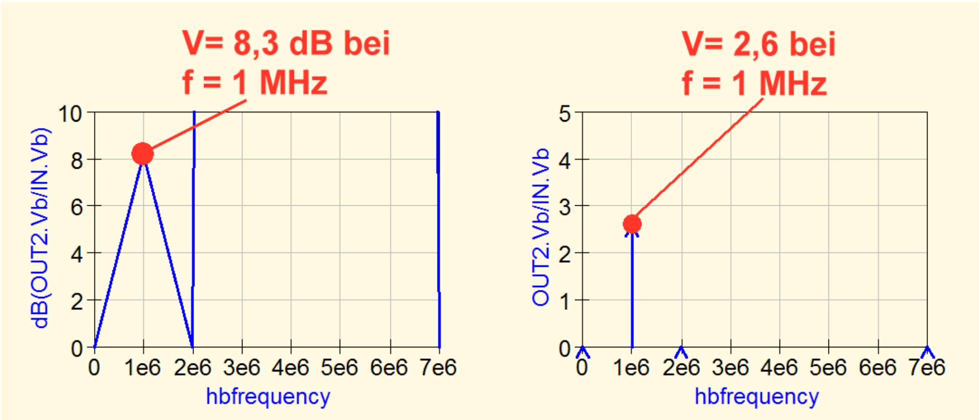
But the result is very irritating:



We did not expect a maximum value of 400 dB respectively a maximum value of „5e19“ ...

The reason for this phenomenon is very simple:
In the „Harmonic balance property list“ we have entered a **maximum harmonic order of N = 8**

Every Harmonic frequency is marked by an arrow on the horizontal diagram axis and for every Harmonic frequency the amplitude will be calculated. But these amplitudes decrease with the order „N“ and thus the calculated gain for the Harmonics rises with N and rises and rises.....



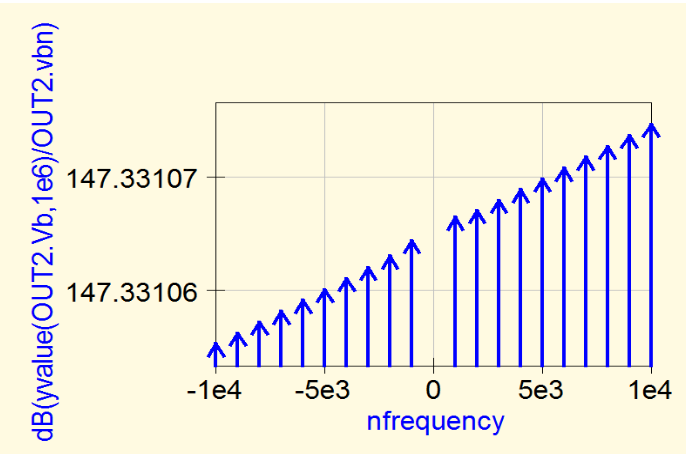
The remedy is simple:

Reduce the scaling at the vertical diagram axis (under „Limits“) to 0...5 for linear presentation and to 0....10 dB for dB-presentation.

7.5. I want to see the Signal to Noise Ratio for f = 1 MHz at the Output in db

Use this equation:

dB(yvalue(OUT2.Vb,1e6 / OUT2.vbn)



In chapter 7.3 we have set the start frequency for noise simulation to 1 kHz and the end to 10 kHz using 10 steps.

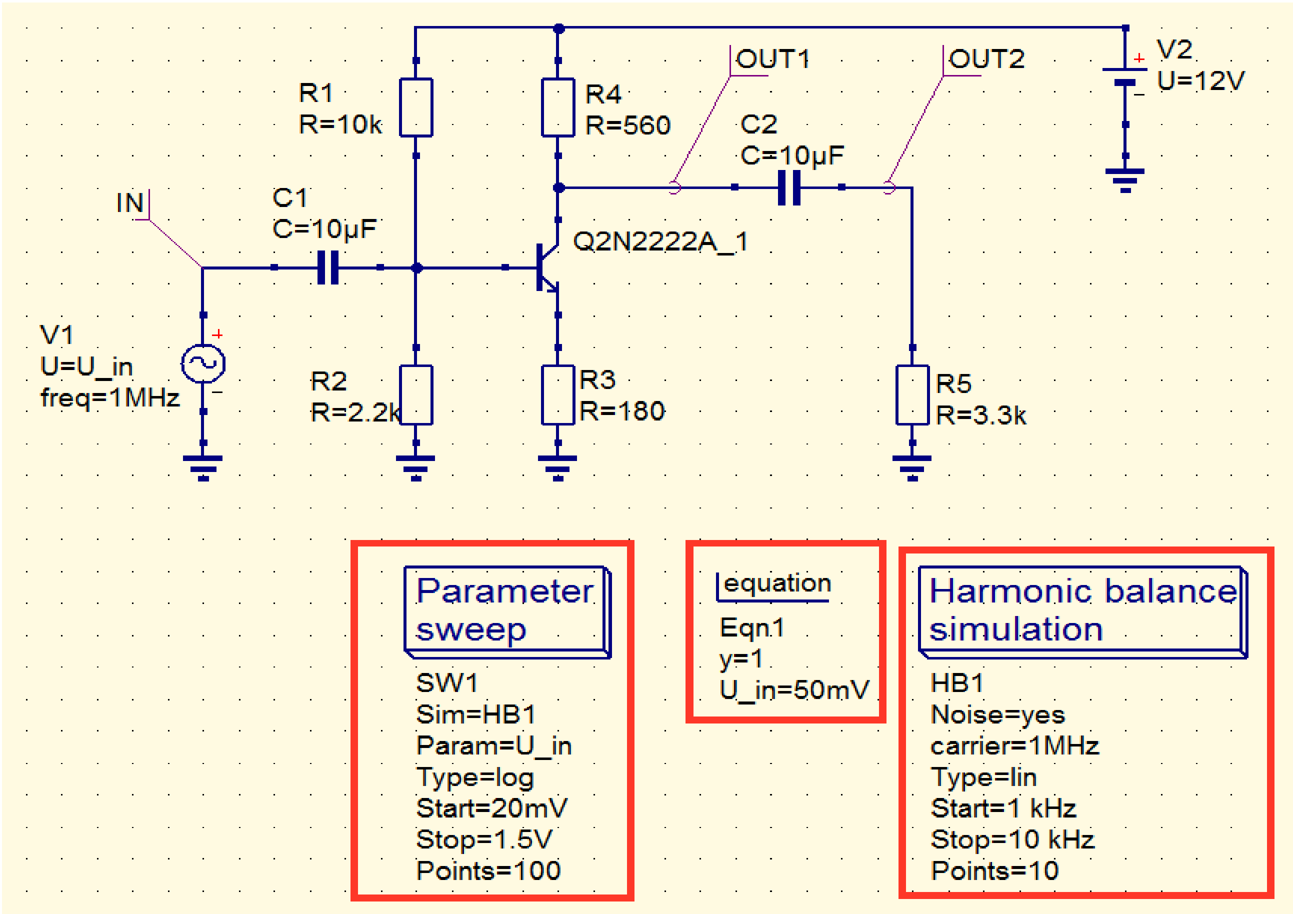
Don't be worried by the „slope“ of the amplitudes and have a look at the scaling of the vertical axis:

this is the fith position after decimal point....

And always remember:
Harmonic balance can only handle and calculate a collection of discrete frequencies...but never a continuous spectrum!

7.6. I want to simulate with the Input Voltage as Parameter. How does the Schematic look like for this Task?

That is it:



7.7. Parameter Sweep of the Input Voltage: I want to see the Output Voltage and the Transfer Function

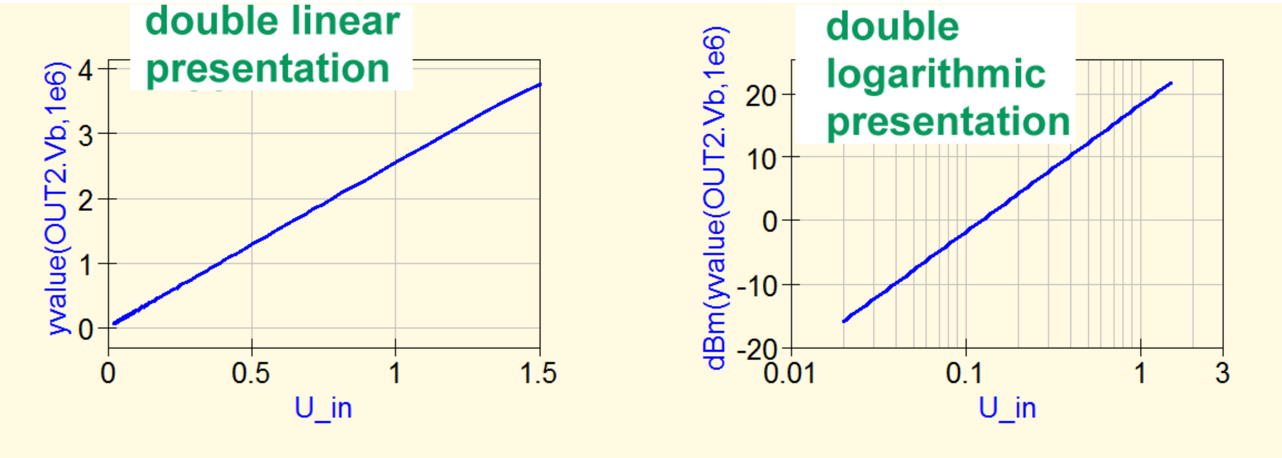
Write a parameter sweep declaration and sweep logarithmic from

U_in = 20 mV to **U_in = 1,5 V**

using 100 points at a frequency of f = 1 MHz.

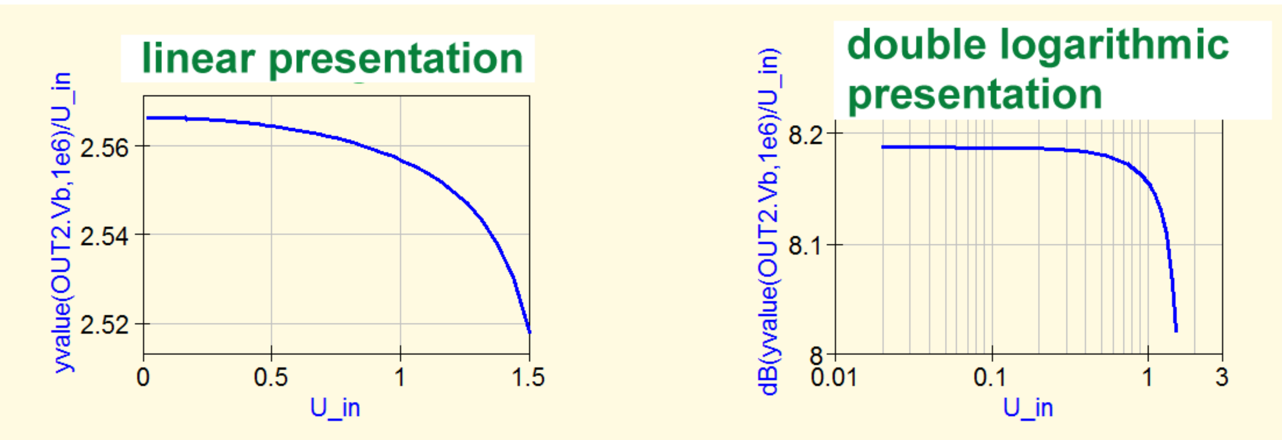
Solution for the output voltage:

yvalue(OUT2.Vb,1e6) **dBm(yvalue(OUT2.Vb,1e6))**



And this is the transfer function in linear and double logarithmic form:
Gleichungen:

yvalue(OUT2.Vb,1e6)/U_in **dB(yvalue(OUT2.Vb,1e6)/U_in)**



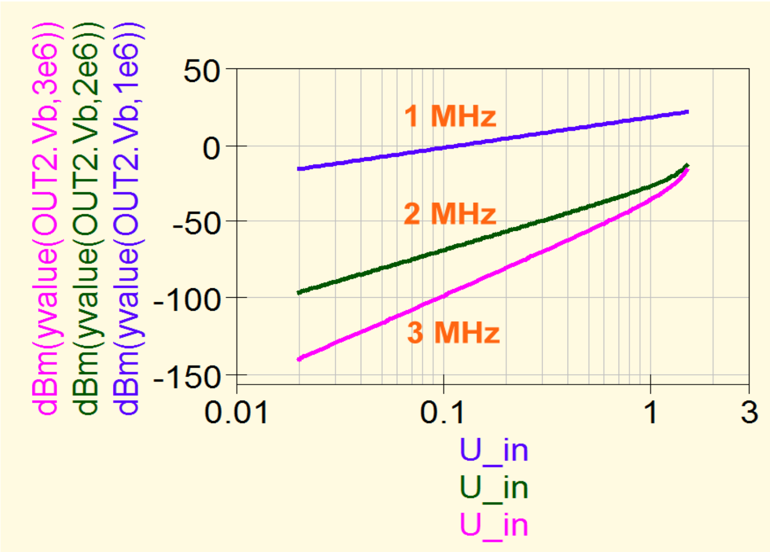
7.8. Parameter Sweep of the Input Voltage: I would like to see the Fundamental Frequency and two Harmonics in dBm at the Output

For this purpose we use the double logarithmic presentation and write three graph property equations

dBm(yvalue(OUT2.Vb,1e6))

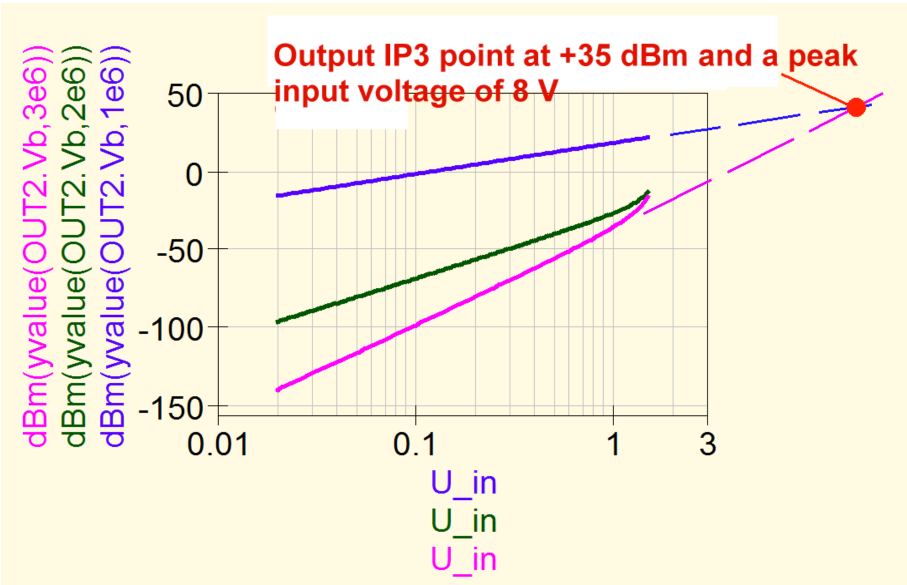
dBm(yvalue(OUT2.Vb,2e6))

dBm(yvalue(OUT2.Vb,3e6))



7.9. Parameter Sweep of the Input Voltage: how can I evalute the Output IP3 Point?

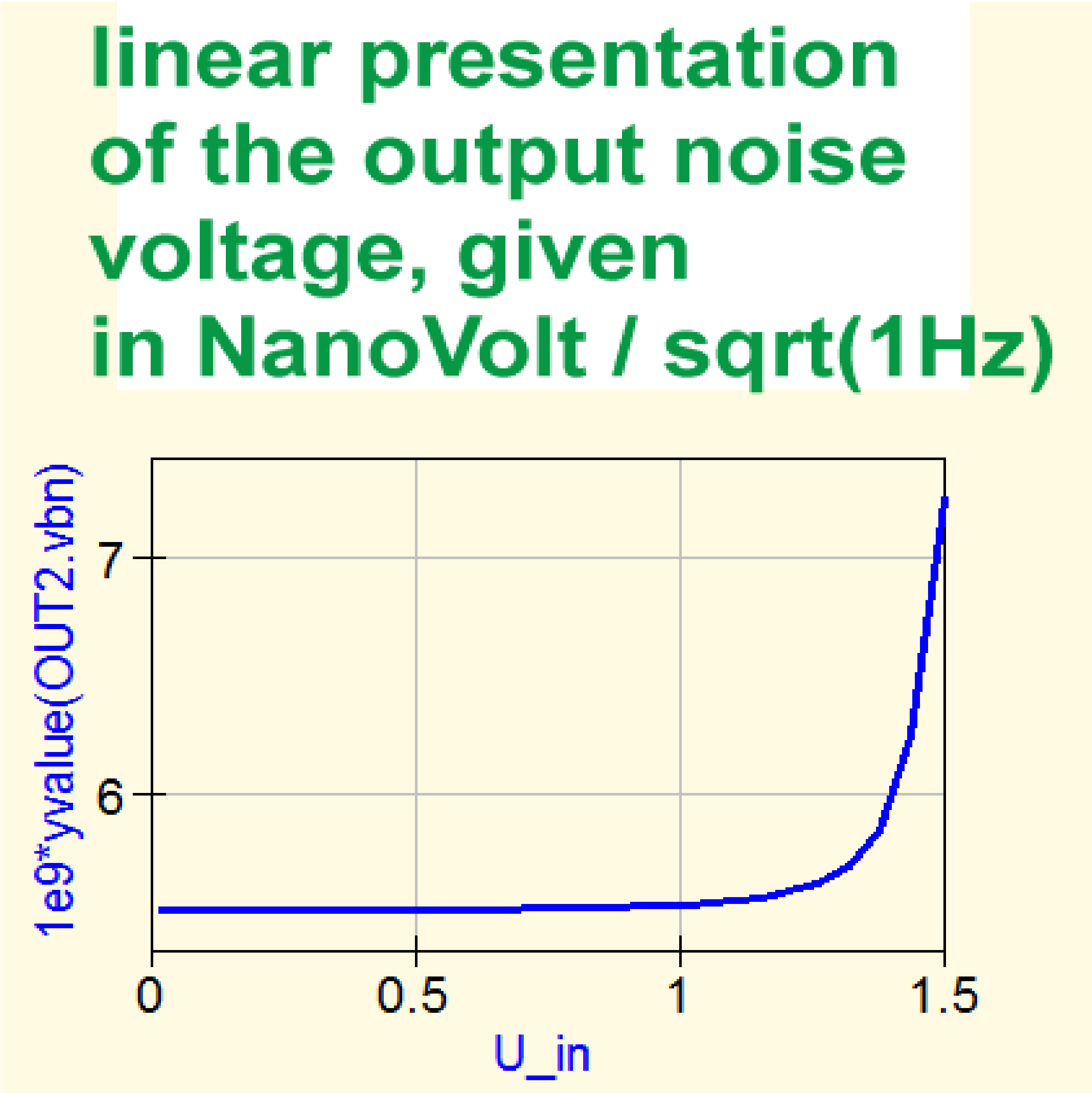
We use the same graph as above and extend (in our brain) the associated approximation straight lines of the curve for 1 MHz (= fundamental frequency) and the curve for 3 MHz (= third order product). Then we evaluate the break-even point:



7.10. Parameter Sweep of the Input Voltage: how does the Noise Voltage at the Output vary with an increasing Input Voltage?

Never forget:
Noise voltages are always given as **„Volts per square root of 1 Hz“** and this is the noise voltage resp. noise power which can be measured in a bandwidth of 1 Hz. The amplitude values are extremely small and thus they are normally given as „NanoVolts“. **But 1 Volt is identical to „10^9 NanoVolts“** and thus the graph equation can be written as:

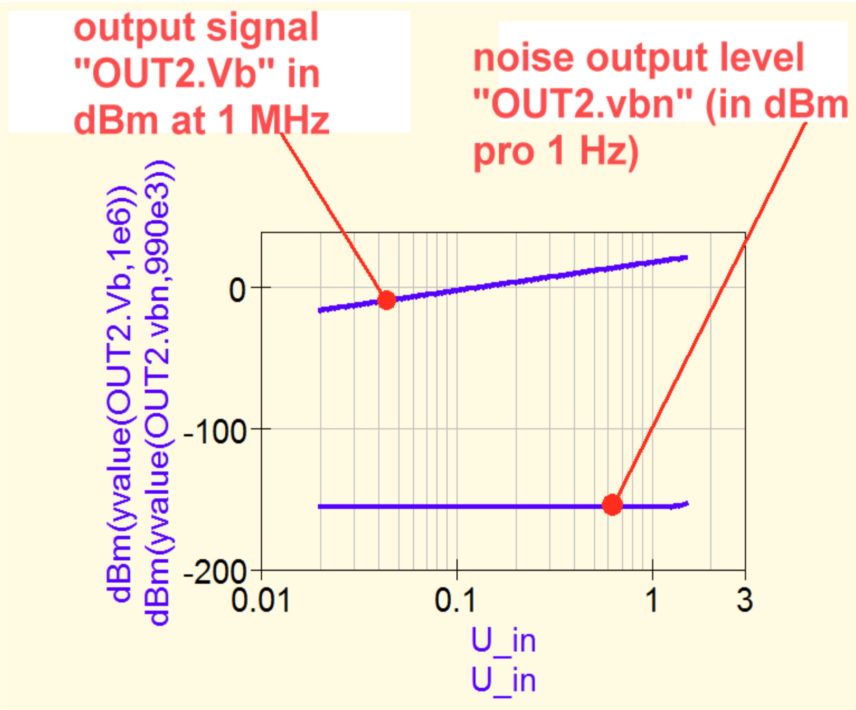
1e9*yvalue(OUT2.vbn)



7.11. Parameter Sweep of the Input Voltage: how can I simulate the Signal to Noise Ratio in dB at the Output?

You have two options

a) Present the output signal „OUT2.Vb" in dBm and the output noise voltage in dBm in the same diagram. Then determine the level difference between the curves for a desired input voltage.

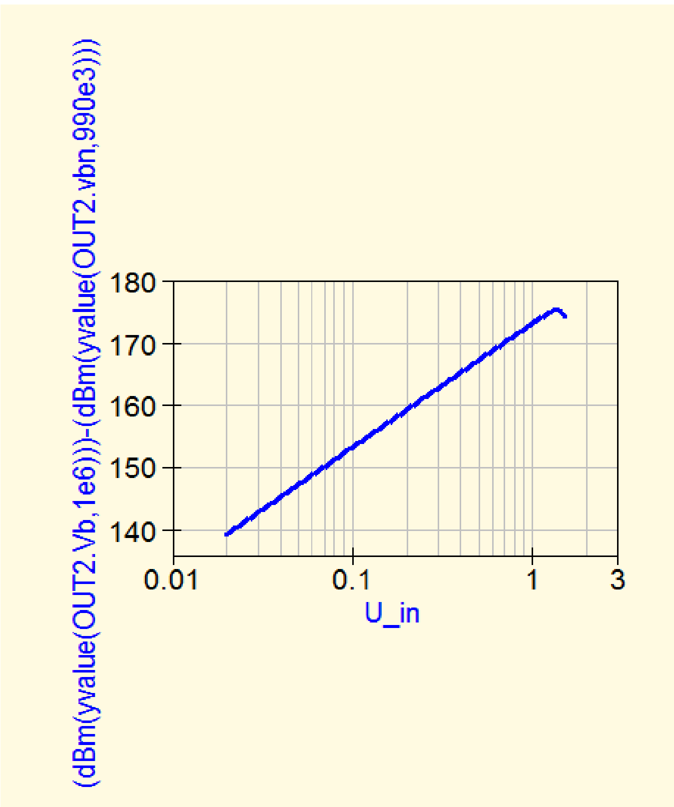


Remark:
The output signal has a frequency of f = 1 MHz.

The noise signal is calculated at a frequency which is 10 kHz lower (f = 990 kHz).

equations:

dBm(yvalue(OUT2.Vb)
und
dBm(yvalue(OUT2.vbn)



b) Write an equation which calculates the dBm difference between the two curves (bandwidth = 1 Hz for the noise power):

(dBm(yvalue(OUT2.Vb,1e6)))-(dBm(yvalue(OUT2.vbn,990e3)))

7.12. Parameter Sweep of the Input Voltage: how can I determine the Signal to Noise Ratio in dB for a Bandwidth of 1000 Hz?

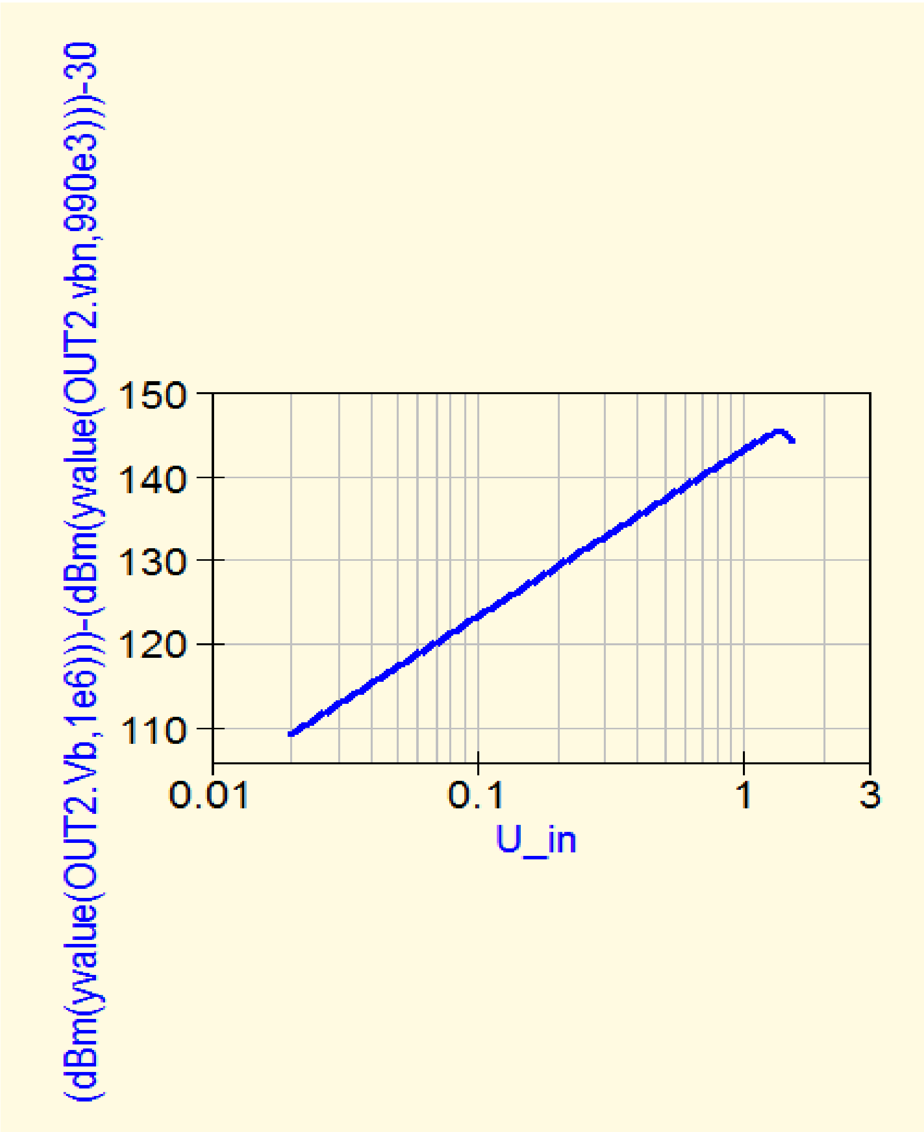
Very easy:

The noise power increases linearly with the bandwidth. Working with „dB“ or „dBm“, then the following expression must be subtracted from the S / N ratio:

10*log10(regarded bandwidth / 1 Hz)

For a bandwidth of 1 kHz you get a correction factor of **-30 dB for the curve of the last example (in chapter 7.11.)**
This is the new graph equation:

(dBm(yvalue(OUT2.Vb,1e6)))
-(dBm(yvalue(OUT2.vbn,990e3)))
-30



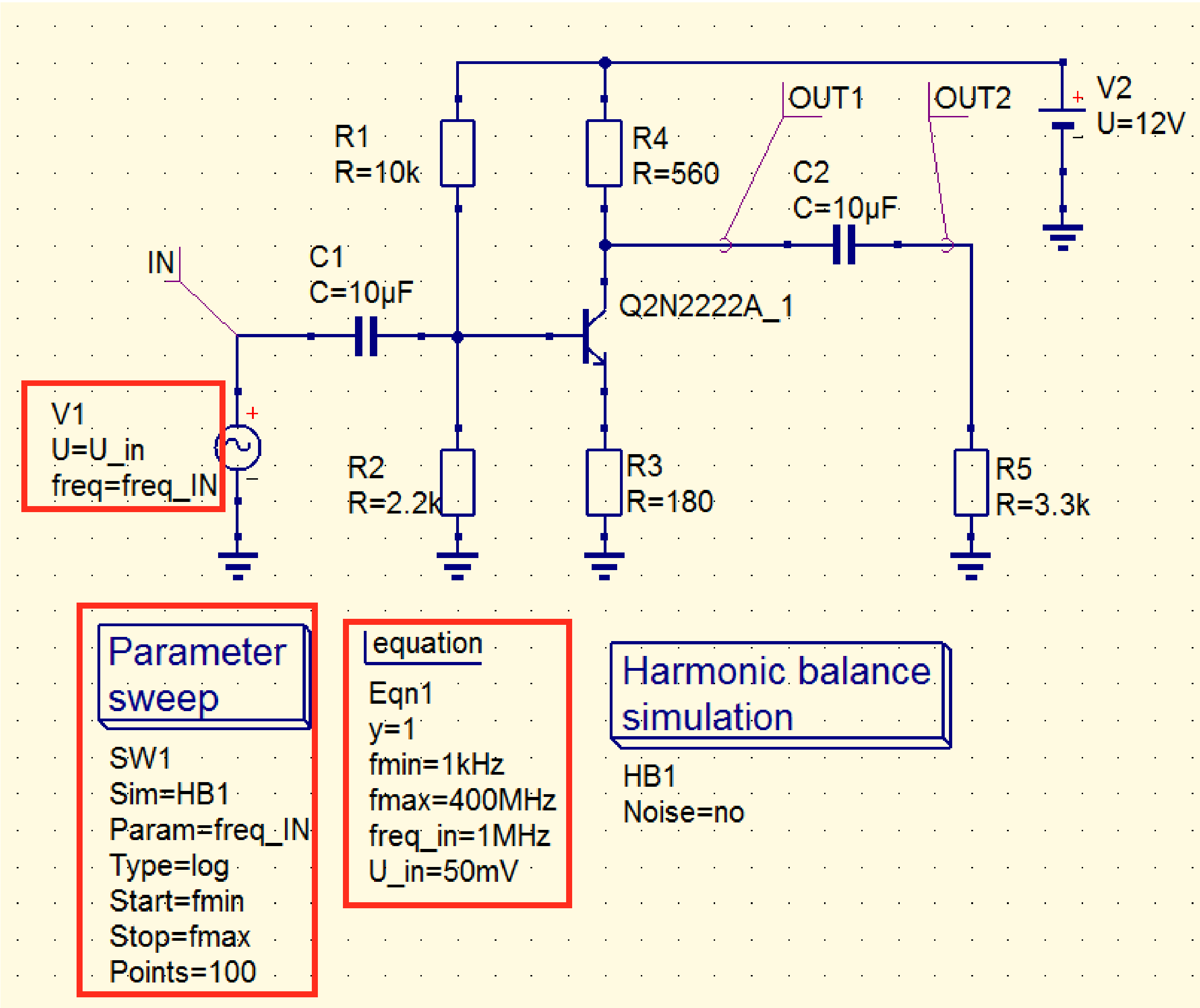
7.13. Parameter Sweep of the Frequency: Preparation of the HB Simulation

This simulation needs some time for the preparation:

a) **Use two variables in the property menu of the input voltage source (= „U_in“ and „freq_in“)**

b) Use „**equation**“ to enter
fmin = 1 kHz
fmax = 400 MHz
freq_in = 1 MHz
U_in = 50 mV

c) Write the properties for the parameter sweep:
Sim=HB1
Param=freq_in
Type =log
Start=fmin
Stop=fmax
Points=100

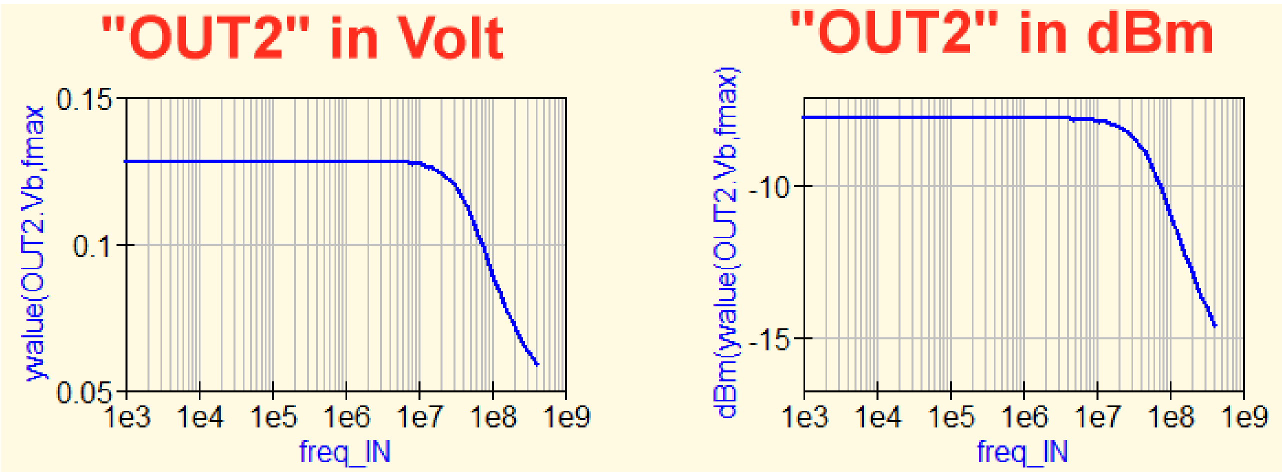


7.14. Parameter Sweep of the Frequency: curves of Output Voltage and Gain

At first write the correct equations for the result of a frequency sweep:

Linear presentation of the output voltage | **Output voltage in dBm**

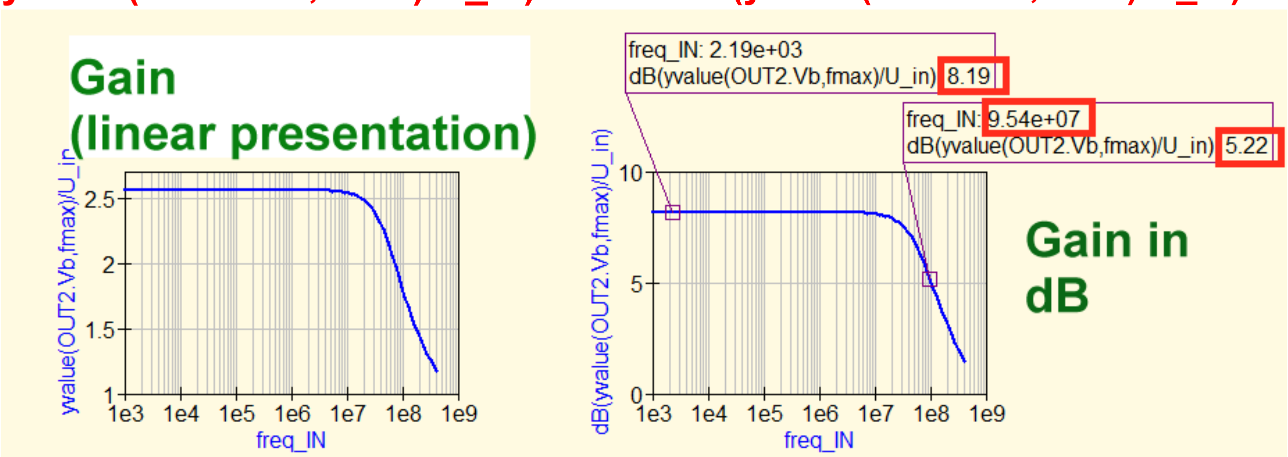
yvalue(OUT2.Vb,fmax) | **dBm(yvalue(OUT2.Vb,fmax)**



Now gain versus frequency isn't a great problem

Gain (linear presentation): | **Gain (in dB)**

yvalue(OUT2.Vb,fmax)/U_in) | **dB(yvalue(OUT2.Vb,fmax)/U_in)**

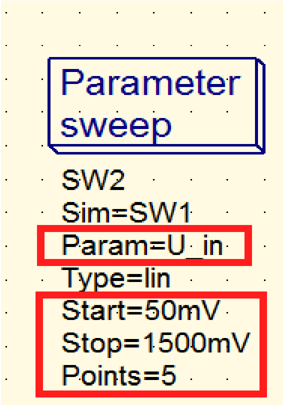


In the right diagram you can see how the „upper 3 dB cutoff frequency“ was determined by two markers.

The left marker indicates a gain value of 8.2 dB at low frequencies.

The right marker has been shifted to 95.4 MHz. There the gain dropped to 5.2 dB (= 3 dB reduction).

7.15. Simultaneous Parameter Sweep of Frequency AND Input Voltage



Simply add a parameter sweep SW2 for the input voltage „U_in“.

The input voltage is increased from 50 mV up to 1500 mV (five steps).

The result is really impressive: