OrCAD - Capture + PSPICE Version 16.2

Tutorial for succesful simulations with the free demo-version (delivered by FlowCAD)

Tutorial-Version 1.2

Copyright by
Gunthard Kraus, Prof. em. at the Elektronikschule Tettnang, Germany
Email: krausg@elektronikschule.de and Gunthard.Kraus@gmx.de

Homepage: www.elektronikschule.de/~krausg

September 10th, 2009

Contents: **Page**

1. Product Description and Installation Informations **3**
1.1. Program Concept 3
1.2. Program Installation 3

2. How to organize an ORCAD PSpice - Project (RC - Low Pass Filter) **4**

3. Drawing Schematics with ORCAD – Capture **6**
3.1. Placing, Rotating, Mirroring and Deleting of Parts 6
3.2. Wiring 12
3.3. Changing a Part Value 12
3.4. Programming the Voltage Source for Transient Analysis 13

4. First Simulation (RC – LPF fed with a Square Wave Voltage) **14**
4.1. Preparing the PSPICE – Simulation Project 14
4.2. Presentation of the Simulation Results 15

5. Other Transient Simulations **17**
5.1. RC – LPF fed with a Sine Wave 17
5.2. RC – LPF fed with a Triangle Wave 19
5.3. Change the Grid and use a Cursor 19
5.3. For Communication Specialists only: RC – LPF fed with a FM – Signal 21

6. Simulation of non-repetitive signals **22**
6.1. Step Response h(t) of a Low Pass Filter 22
6.2. Impulse Response g(t) of a Low Pass Filter 24
6.3. Modification of the Result Diagrams 27

7. Sweeping the Frequency Response of a RC-LPF **28**
7.1. Linear Presentation 28
7.2. Presentation of the Frequency Response in dB 30

8. Simulation of simple Power Supply Circuits **31**
8.1. Preparing the Transformer 31
8.2. One Pulse - Rectifier 33
8.3. Bridge Rectifier Circuit 34

9. Two Stage Transistor Broadband Amplifier with Feedback **35**
9.1. Time Domain Simulation 35
9.2. Frequency Domain Simulation 37

10. Using POWER – Symbols **38**

11. A Sine-Oscillator and its Spectrum (= FFT Application) **39**

12. OPA Circuits **42**
12.1. Inverting Amplifier 42
12.2. Analog Adder 45
12.3. Active Filter Circuit 47

13. Simulation of Digital Circuits **51**
13.1. Checking a J-K_Flipflop 51
13.2. NAND Gates 53
13.3. Four Stage Binary Counter 54

14. 100MHz Chebyshev- LPF **55**

Appendix: Impulse Response g(t) and Transfer Function of a System **59**

1. Product Description and Installation Informations

1.1. Program Concept

Around the Simulation Kernel „PSPICE“ You find a Desktop and a lot of toole for Develloping and Testing of Eelctronic Circuits. Bur we only want to have a look at the Simulation Machine which is organized in this manner:

At first draw Your Circuit with „**CAPTURE**“.

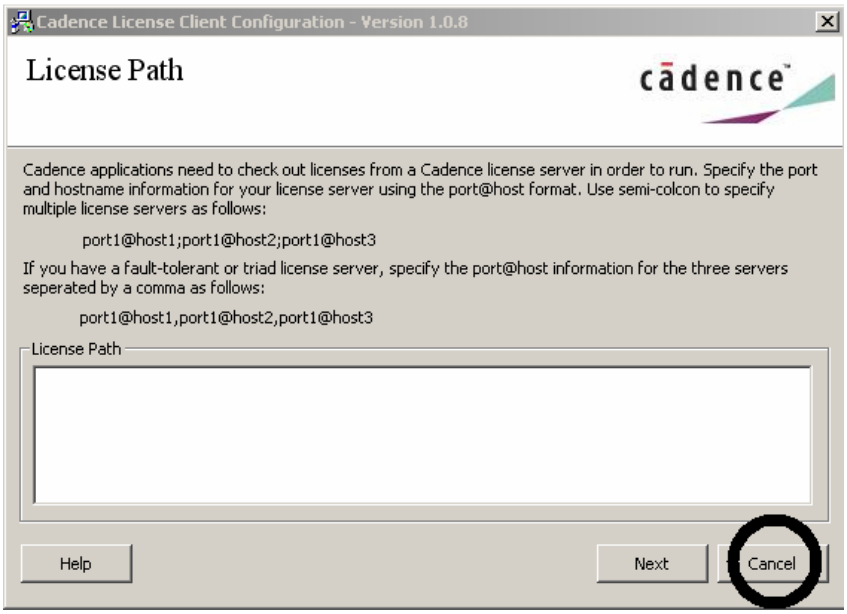
Then simulate with „**PSPICE**“.

And at last call „**PROBE**“ to show the desired results (Voltages, Currents, Frequency Responses, Spectrae).

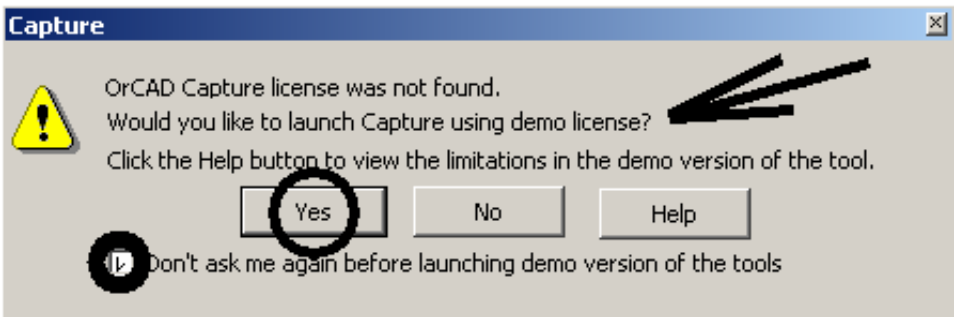
A nice feature -- Sorry, only in the full licensed version -- is CIS“ = Component Information System. With this program You can search missing part models in the Internet (in „help \ OrCAD Capture help \ CIS help” You find all necessary informations.

1.2. Program Installation

Getting the free Demo Version seems to be very simple., but: the download of 2,7Gigabyte can be a problem. So it is better to ask for the free Demo CD at www.flowcad.de.

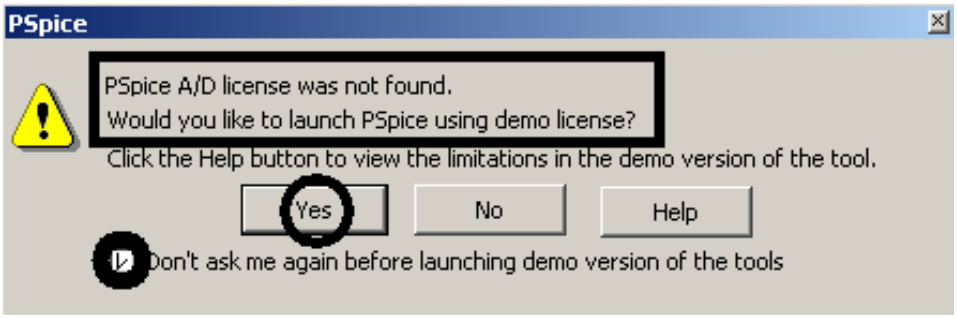


Installation is no problem, but during the process You are asked to enter the Licence file, but this is not necessary for the Demo Version.
So press “cancel”, but the problem will come again when Yuo will lauch the installed program for the first time. Then the problem must be solved.



So please click „**Yes**“ to start the program “Capture”.

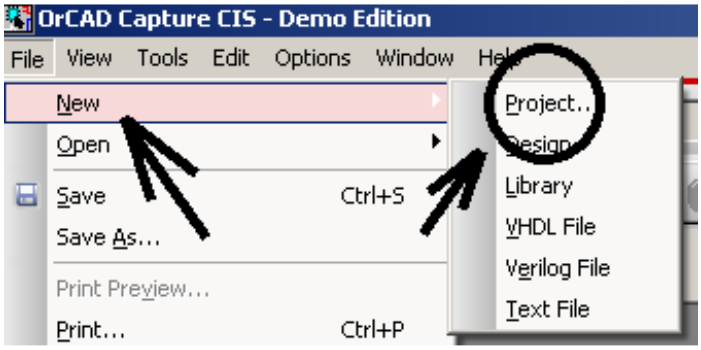
Also mark **“Don`t ask me again for launching demo version of the tools**“ to avoid this message in the future.



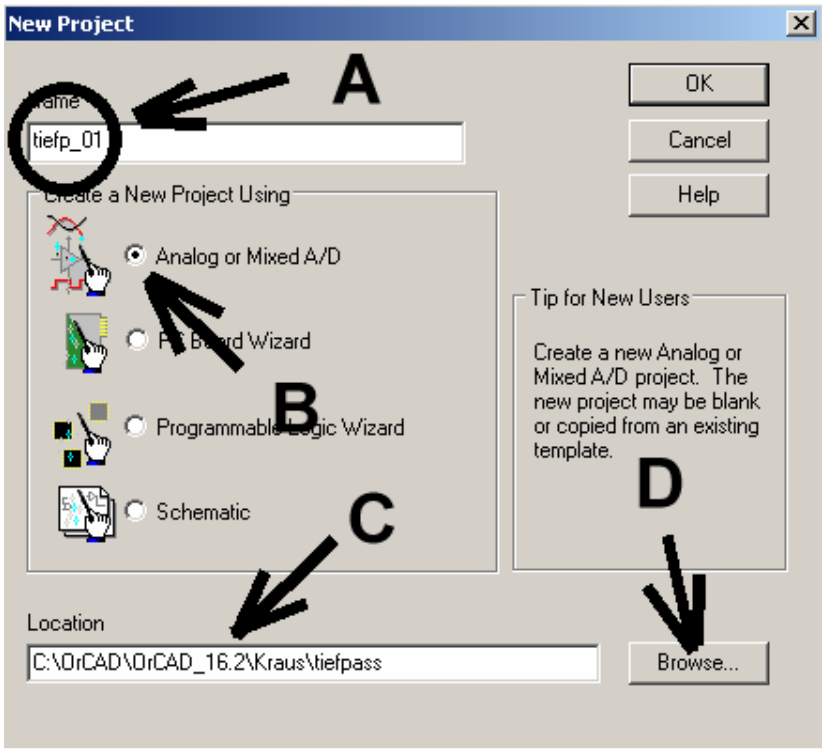
When You have finished Your circuit drawing procedure and You start „**PSPICE**“, there comes the same question and the same marking procedure to work with the Demo version.

Remark:
In the “doc”- folder You find two nice pdf-Documents which are helpfull:
„ORCAD_install_guide and PSPICE User’s Guide.
==========

2. How to organize an ORCAD PSpice – Capture - Project (RC – Lowpass - Filter)



Start the program „**Capture CIS**“. Press „**File**“, then „**NEW**“ and „**Project**“



At first enter the Project Name „**tiefp_01**“at „**A**“.

At „**B**“ mark „**Analog or Mixed A/D**“.

„**C**“ will show the **correct path and location** of our project. So press at „**D**“ the button „**Browse**“ to install this new project.

This opens a little program „**Select Directory**“ and in the well known Windows manners You can enter the location and the path of our project.

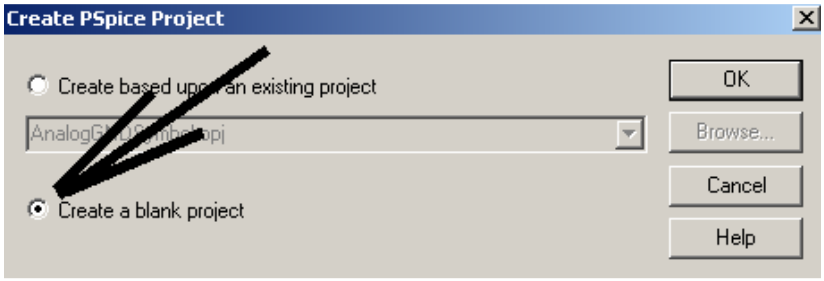
(See next page).

It is the best to create a new directory (f.i. with Your name) in the OrCAD_16.2-folder and therein a folder for Your first project ("tiefpass" means "lowpass filter" in English).

In this case the correct path would be:

C:\ OrCAD \ OrCAD_16.2 \ Kraus \ tiefpass

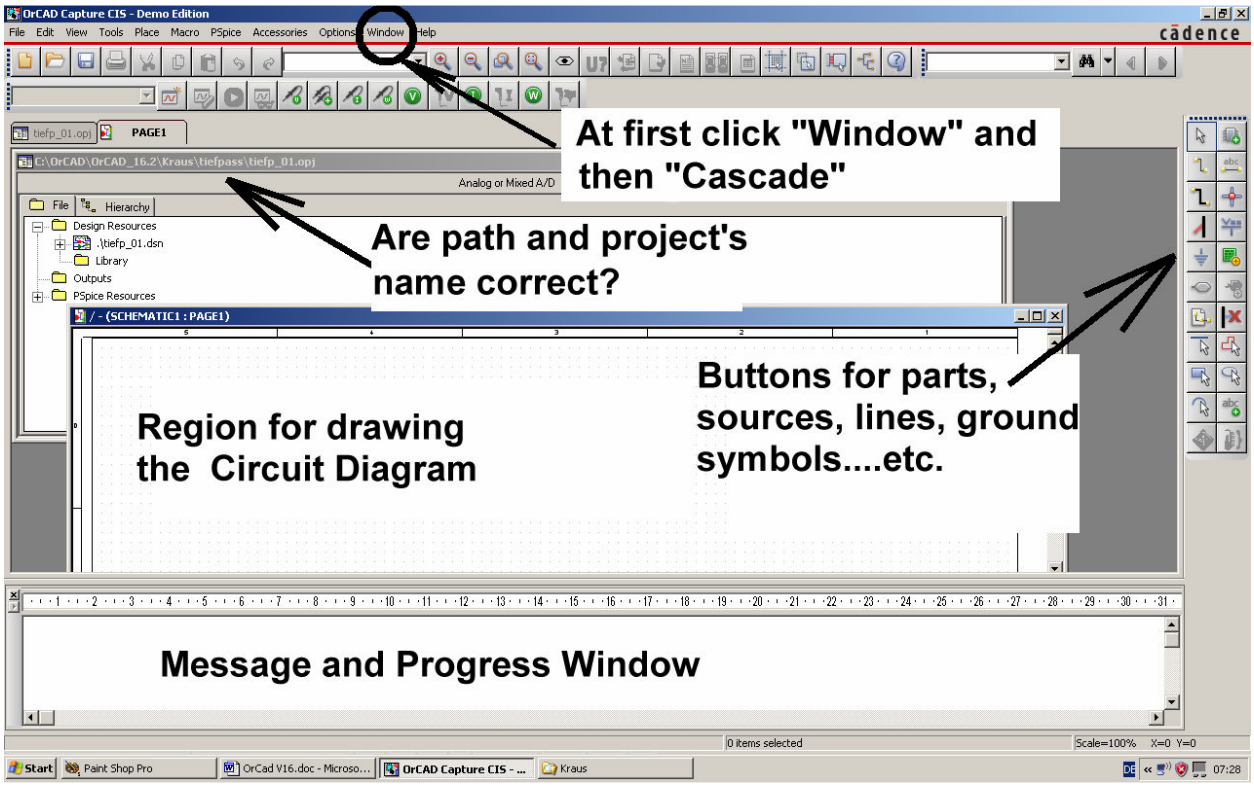
When this is done and correct, press **OK.**



Now we have to decide whether a already existing project shall be used or a new one shall be created.

We want a new one, so we choose **„Create a blank project"** and close with **„OK"**.

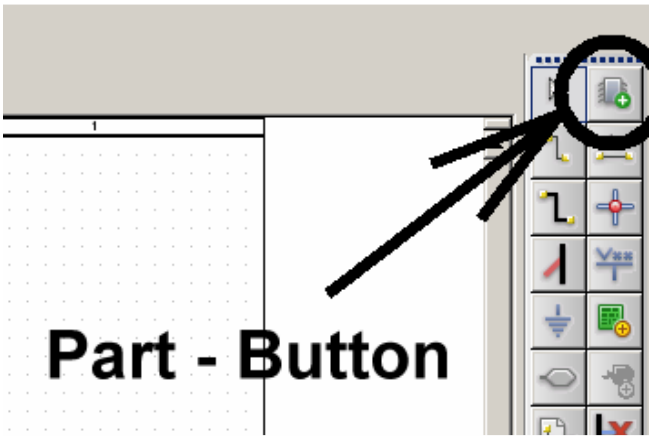
Now the game is running. So press „**Window**" and then „**Cascade**" to get this screen.



Finally enlarge the Drawing Region of the Editor and start with the Schematic Drawing.

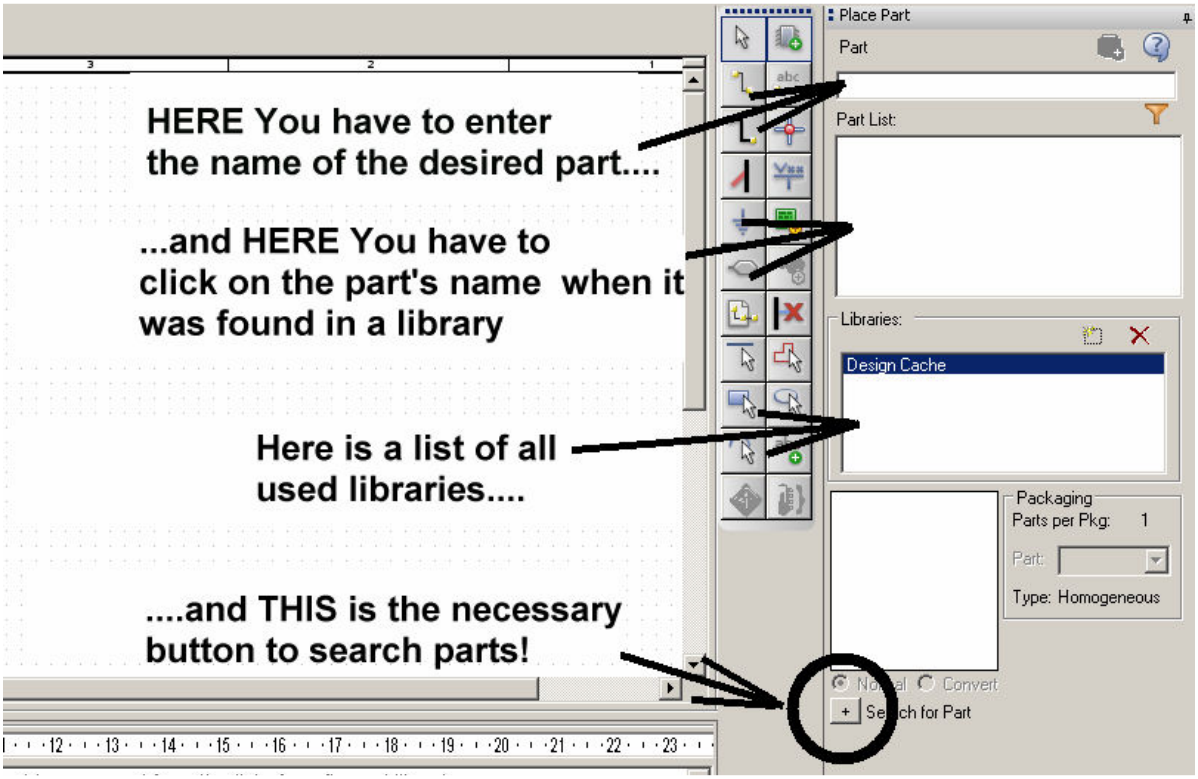
3. Drawing Schematics with ORCAD - Capture

3.1. Placing, Rotating, Mirroring and Deleting of Parts



At first we need a **capacitor** and press the „**Part-Button**“ (right upper corner of the menu bar). This is the way to all part libraries.

But now: High Life….



So we start with entering „**C**“ in the uppermost field „**Part**“ and press then the button **„Search for Part“**.

Attention:
In the field “Search for...” also the letter „C“ must be entered.

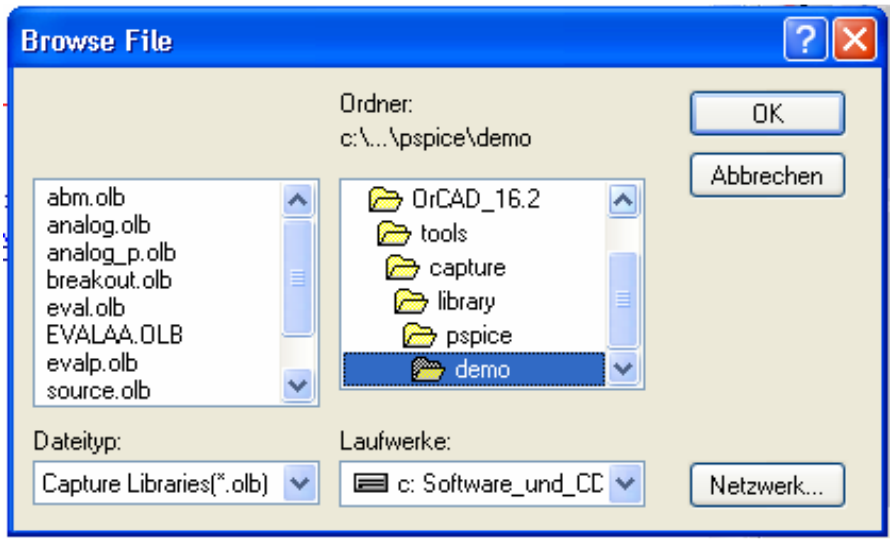
Then we try to find the correct library.

Warning:

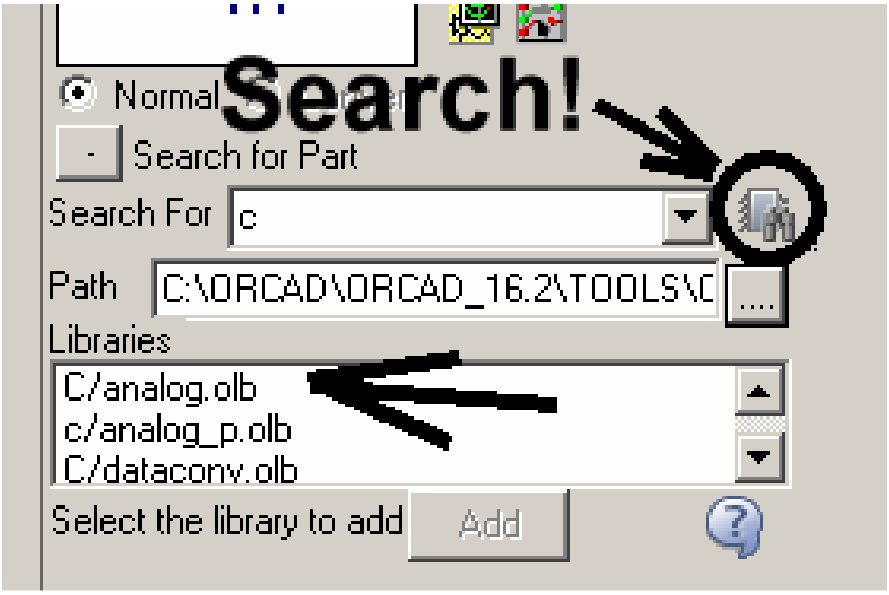
In the Demo-Version you find all libraries of the full licensed version, but most of them are EMPTY!!
Also very iritating is the fact that you can use parts of some libraries, but afterwards -- when you start a simulation -- you recognize that the SPICE Models are missing...

So please remember:
Always use only this one library path:

OrCAD \ Orcad_16.2 \ TOOLS \ CAPTURE \ LIBRARY \ PSPICE \ Demo



If it looks like this picture on Your PC screen, press OK.

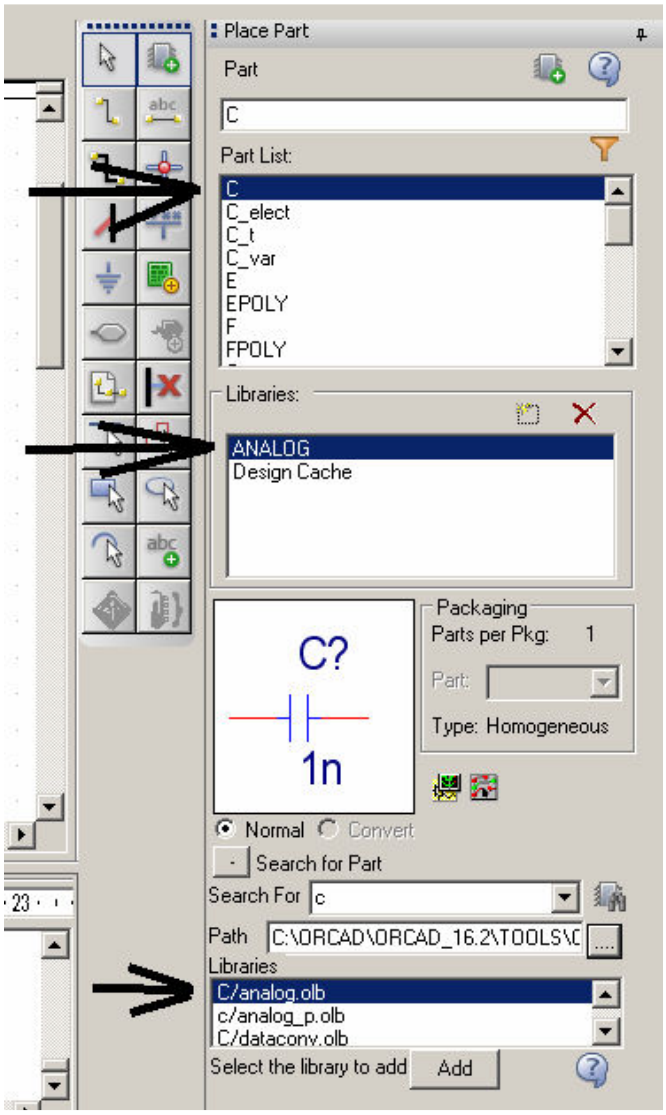


If You have already entered „C“ in the line “Search for…”, press „**Search!**“ and wait for the result.

Under the listed results we click on

C / analog.olb

and we can see how this library is selected and used for the part search.
Now automatically the desired part appears in the list in the upper half of the window.



So it looks like now and the desired part „C“ is hanging on the cursor.
Please place it in your schematic. If necessary turn it by 90 degrees (= press “R”).

Now follows a short list how to handle already placed parts:

If you click on the symbol of a part, it will be marked and changes its colour to “**red**”.

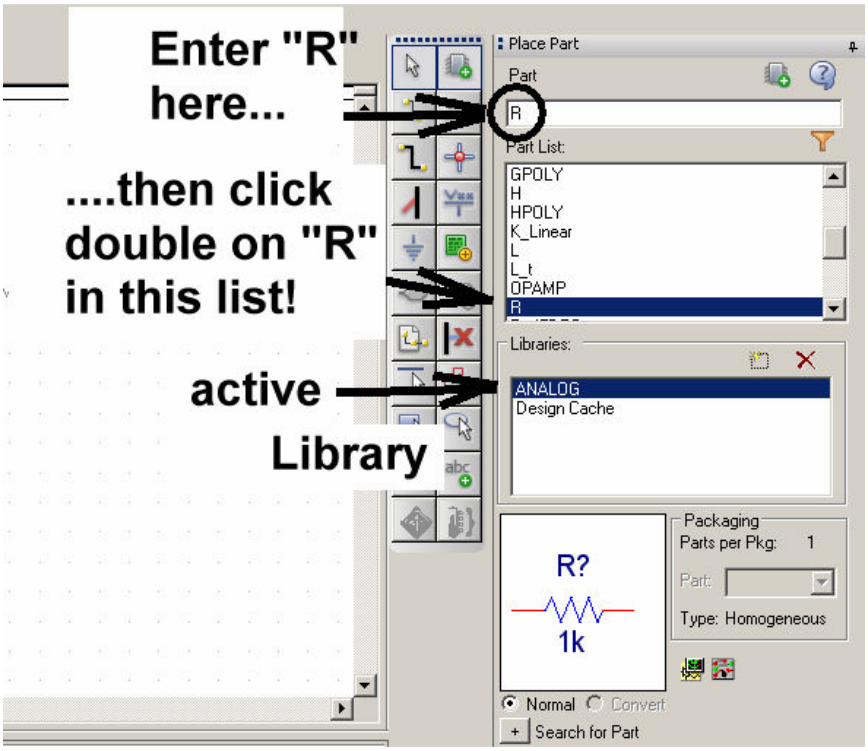
When marked, you can turn it by 90 degrees by pressing **<R>** on the Keyboard.

Using „**Drag and Drop**“ you can move the part within the schematic.

And with the **Delete-Key** you can delete this part from the schematic.

Mirroring is a little bit complicated. First click on the symbol to mark the part, then open the **menus “Edit” and “Mirror”**. The rest is self-explaining.

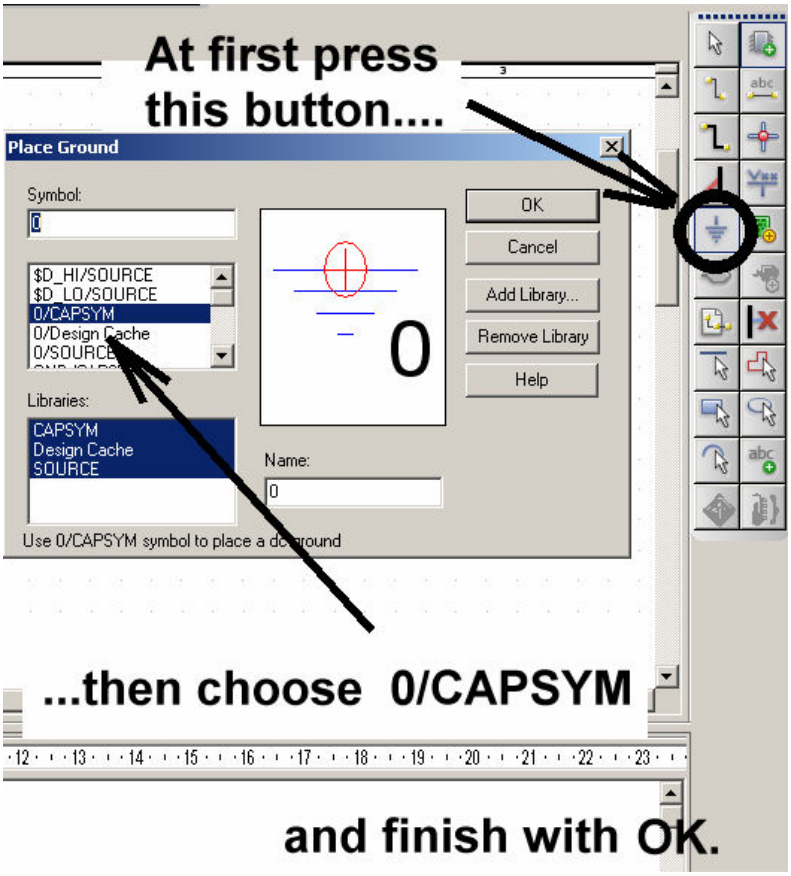
Part marking is finished by pressing the „ESC“-Kay or by clicking left with your mouse on an empty place in your schematic.



The procedure must now be repeated to get a **resistor „R**“.

This part is found in the same library as the capacitor “C”, so You only have to enter the letter “**R**” in the uppermost window and click double on “**R**” in the appearing list.
Then the resistor is hanging on the cursor and can be placed in the schematic.

Afterwards press ESC to be free of the resistor’s symbol on the cursor.



Then we need two Ground Symbols. They have an own button and when this button is pressed, you have to choose

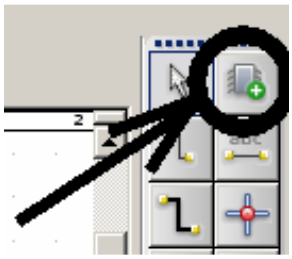
„**0 / CAPSYM**“:

The ground symbol is then hanging on the cursor and can be placed (two times) on the schematic.

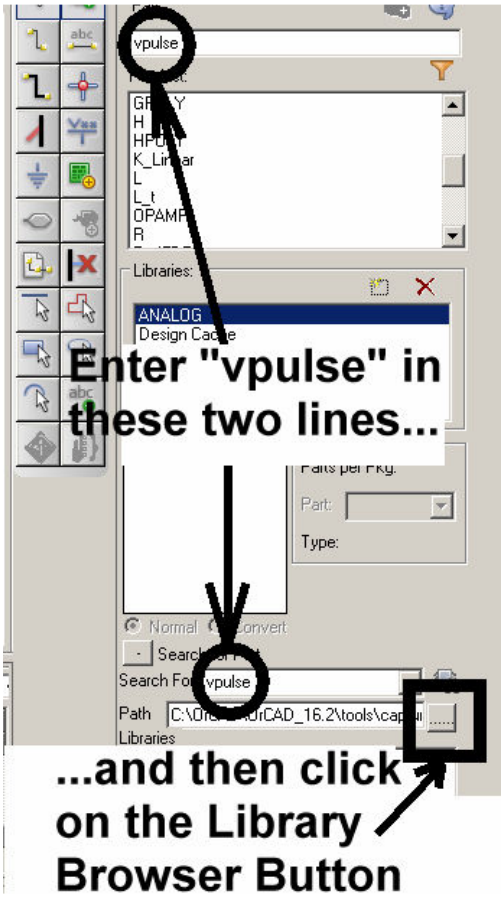
Finish this step with OK.

At last we need a Pulse Voltage Source to feed our circuit. Remember the following steps:

Step 1:



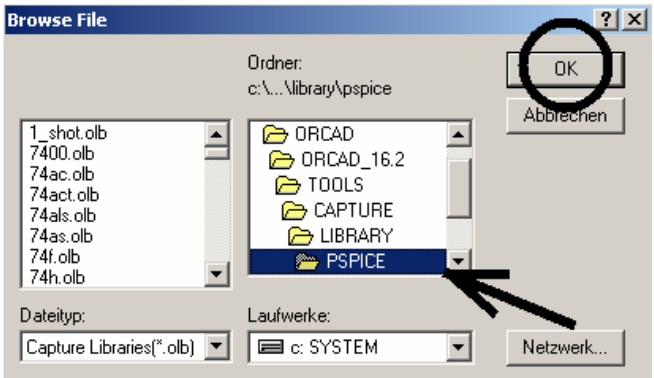
Press the „**Search Part“-Button** on the right hand side of the screen



Step 2:
Enter

vpulse

in the marked two lines and then open the Library Browser.

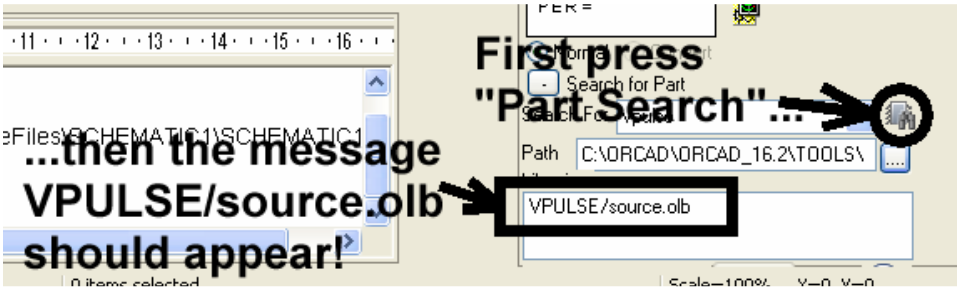


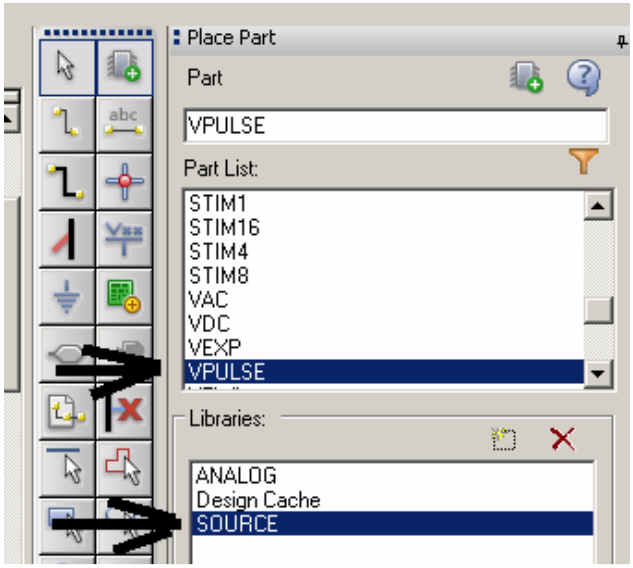
Step 3:
Carefully choose the path

C: \ ORCAD \ ORCAD_16.2 \ TOOLS \ CAPTURE \ LIBRARY \ PSPICE \ DEMO

and click on the folder “**DEMO**”
Finish with **OK**.

Step 4:
In this manner you can now search for the Pulse Voltage Source “vpulse2:





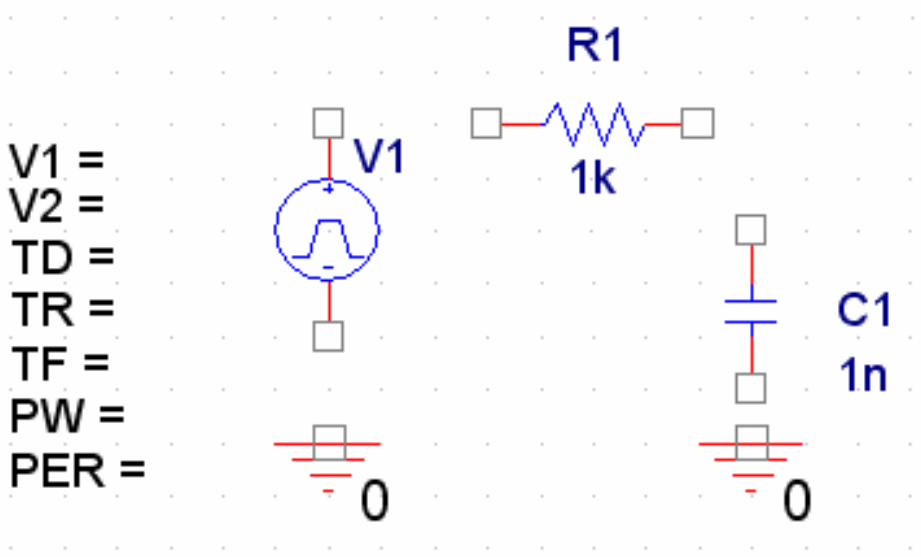
Step 5:

When we double-click on

„**VPULSE / source.olb**“,

then most of the work is done.

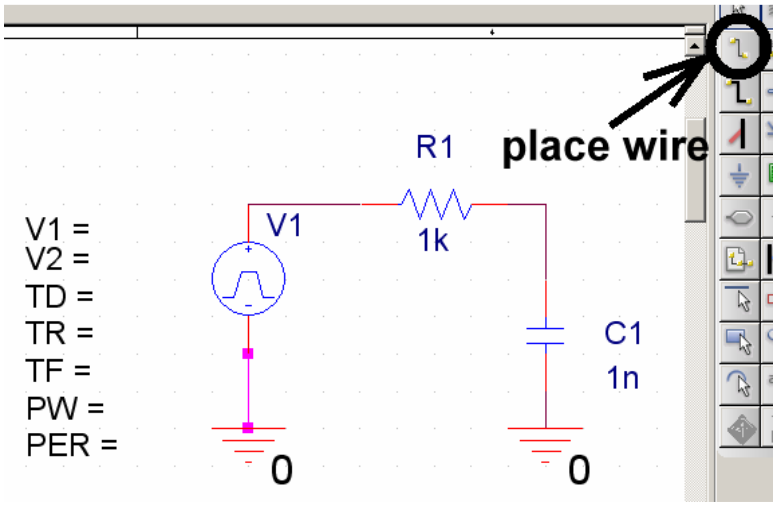
“Vpulse” is now hanging on the cursor, the „SOURCE.olb“ – Library is added to the list and the model of „VPULSE“ is activated.



So the schematic should now look like.

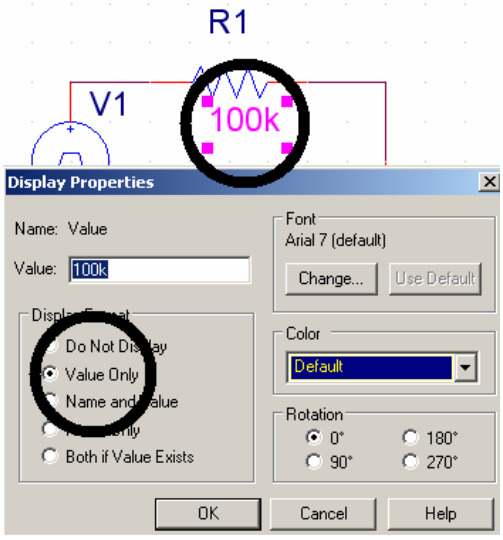
3.2. Wiring

Connections between parts of the schematic are produced by the **„WIRE"- Method.**
At first move all parts by "Drop and Drop" to their desired position on the screen and press the „**Place Wire"-Button.**
If You now klick on a part's connection, a wire will be soldered at this point. Then move the mouse (Attention: do NOT press the left mouse button!!) and You can see how this wire follows the way of the mouse cursor. If you reach the connection of the other part, click left and the wire is soldered to the other part. If the wire is there not cutted automatically, You can do that with ESC.
Attention: when You roll with the mouse and suddenly click left, You produce a BEND and this is sometimes important to change the wire's direction. So please try!



This must be the result of our work.

3.3. Changing a Part Value



First press ESC to be shure that no other function is active. Then click double on the part value. The value should now change its colour to "RED" or "Magenta" and the part's Property Menu (= „**Display Property Window**") will popen. Enter now the desired value of 100k.

Attention:
If also the part's description "R1" has changed its colour You can stop that my marking "**Value only**" in the menu.

Please repeat now the procedure for the capacitor and enter a value of **10 Nanofarad** (= 10nF).

But always remeber:

1 Ohm	is entered as	**1**		
1 Milli - Ohm	is entered as	**1m**		
1 Kilo - Ohm	is entered as	**1k**		
1 Mega - Ohm	is entered as	**1MEG**		
1 Picofarad	is entered as	**1pF**	or	**1p**
1 Nanofarad	is entered as	**1nF**	or	**1n**
1 Mikrofarad	is entered as	**1uF**	or	**1u**
1 Farad	is entered as	**1F**		

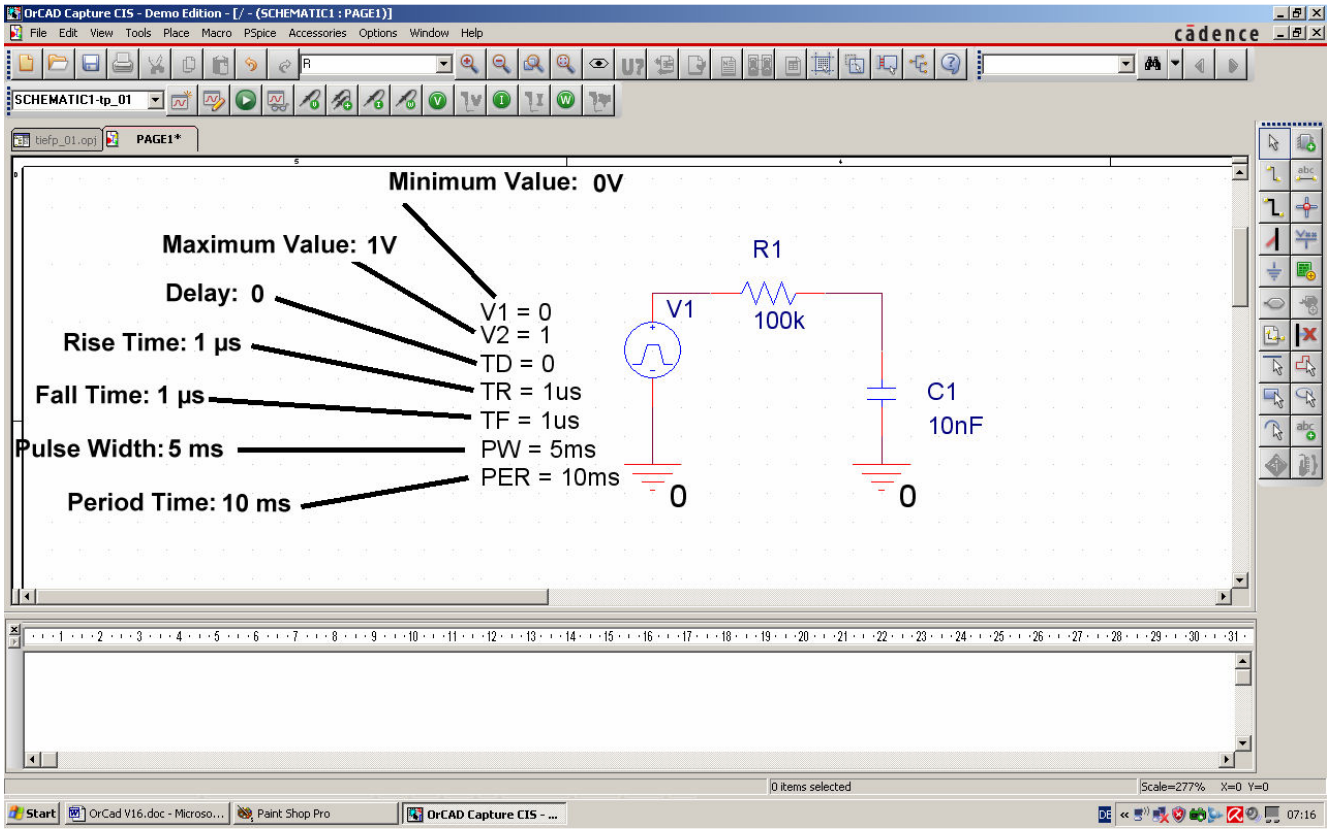
3.4. Programming the Voltage Source for Transient Analysis

The Input Voltage or our LPF shall be a Pulse Voltage with the following properties. These values must be entered in the value lines beside the symbol:

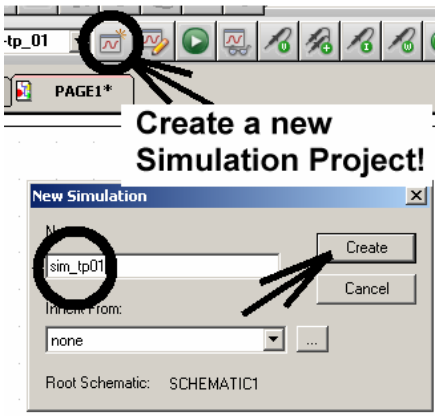
Minimum Amplitude	**V1**	=	0V
Maximum Amplitude	**V2**	=	1V
Delay	**TD**	=	0
Rise Time	**TR**	=	1us
Fall Time	**TF**	=	1us
Pulse Width	**PW**	=	5ms
Period Time	**PER**	=	10ms

IMPORTANT: There must Never be an empty space between the value and the unit of a value!!!!!

This should now be the correct schematic:



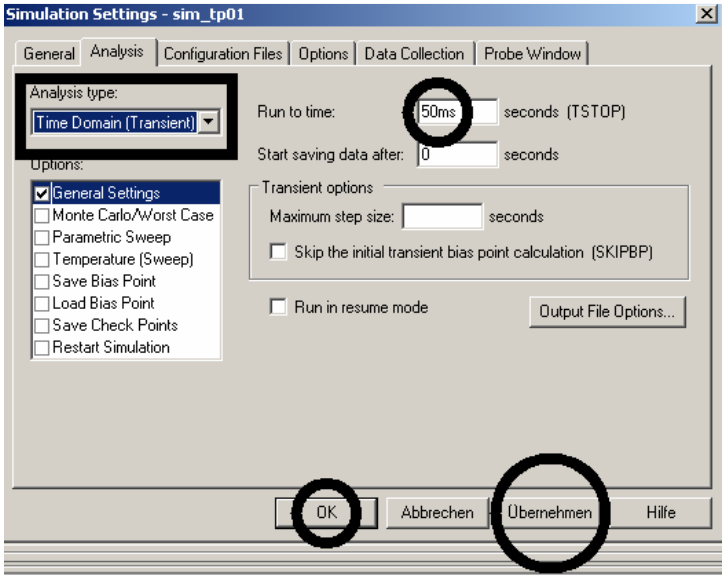
4. First Simulation (RC – LPF fed with a Square Wave Voltage)
4.1. Preparing the OrCAD PSPICE Simulation Project



At first a new Simulation Project must be created for the simulation of the LPF!

So click on the marked Button and a window "**New Simulation**" will open.
Please enter a name for the simulation task (here: „**sim_tp01**") and press „**create**".

Now follow the „**Simulation Settings**".



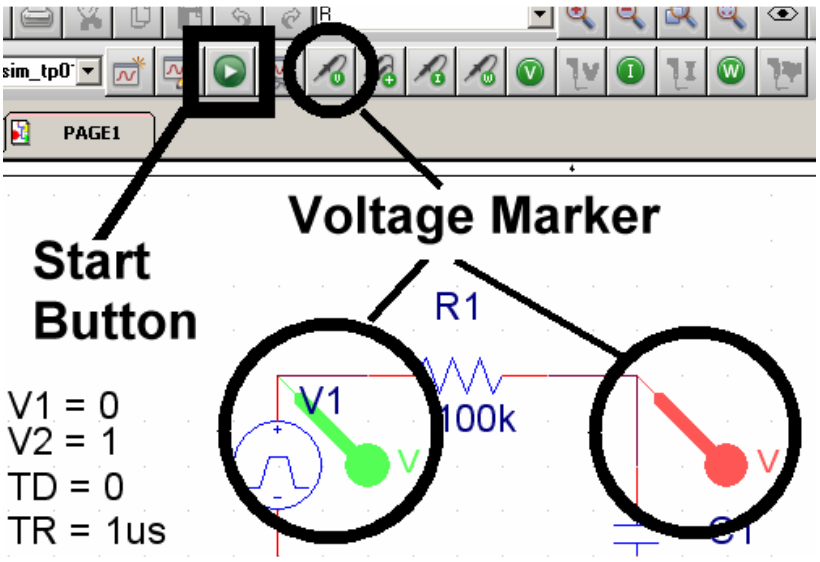
a) Please check wether the menu „**Analysis**" is selected.

b) Please check wether „**Time Domain (Transient)**" is already selected as Analysis Type.

c) Please enter **50ms** at "**Run to time**".

d) Press „**Accept" = „Übernehmen**" and finally

e) press „**OK**".

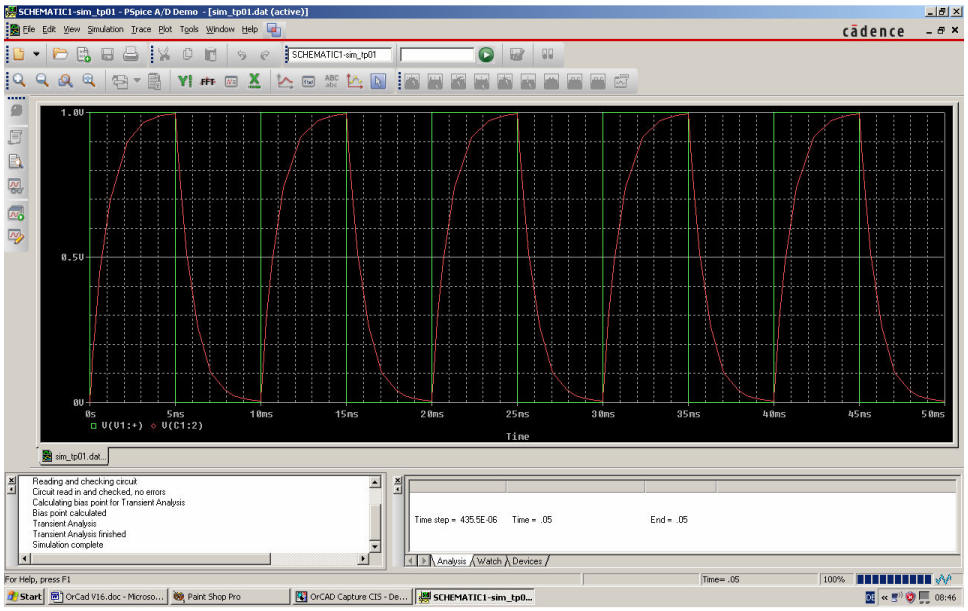


Now we need **Voltage Markers** for the interesting nodes of the circuit **(= input and output)**.

So click on the **Marker Button** and place the marker, which then hangs on the cursor, at the input. Repeat the procedure to set also a marker at the output. If You want to turn such a marker, use **<Control> + <R>**.

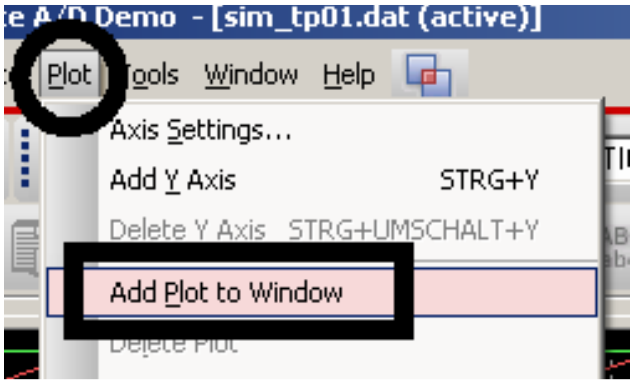
Finish this work with ESC and click on the Start Button.

4.2. Presentation of the Simulation Results.



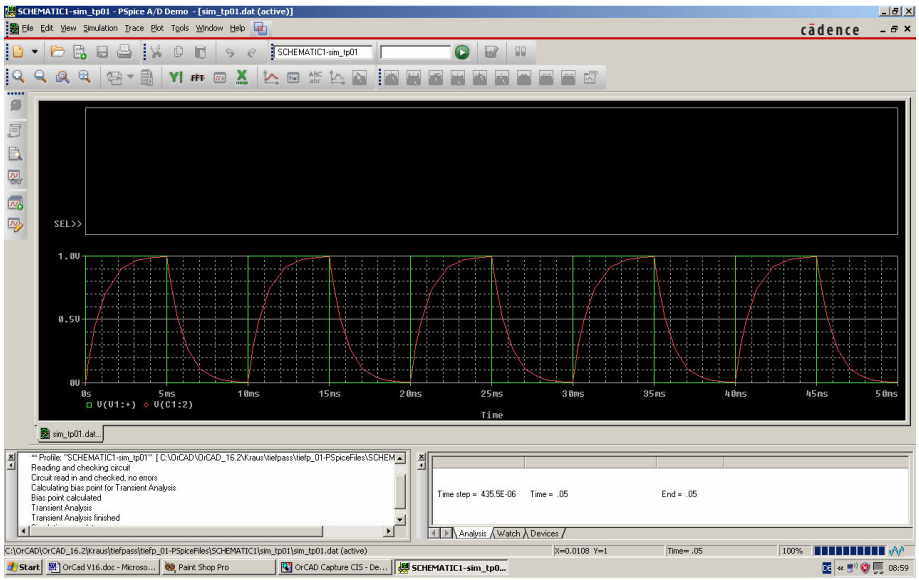
The screen shows not only a diagram with the two voltages but also the „**Output Window**“ and the „**Simulation Status Window**“

If You don’t need the Output and the Simulation Status Window, simply close them (To open again, go into the Pulldown Menu “View”).



Now we want to present the two voltages in different diagrams. Please remember the exact names “**V(V1:+)**” for the Input Voltage and “**V(C1:2)**” for the Output Voltage (which are indicated -- together with the colours -- in the lower left corner of the window).

Then the menu „**Plot**“ is opened and the “**Add Plot to Window**” is selected.

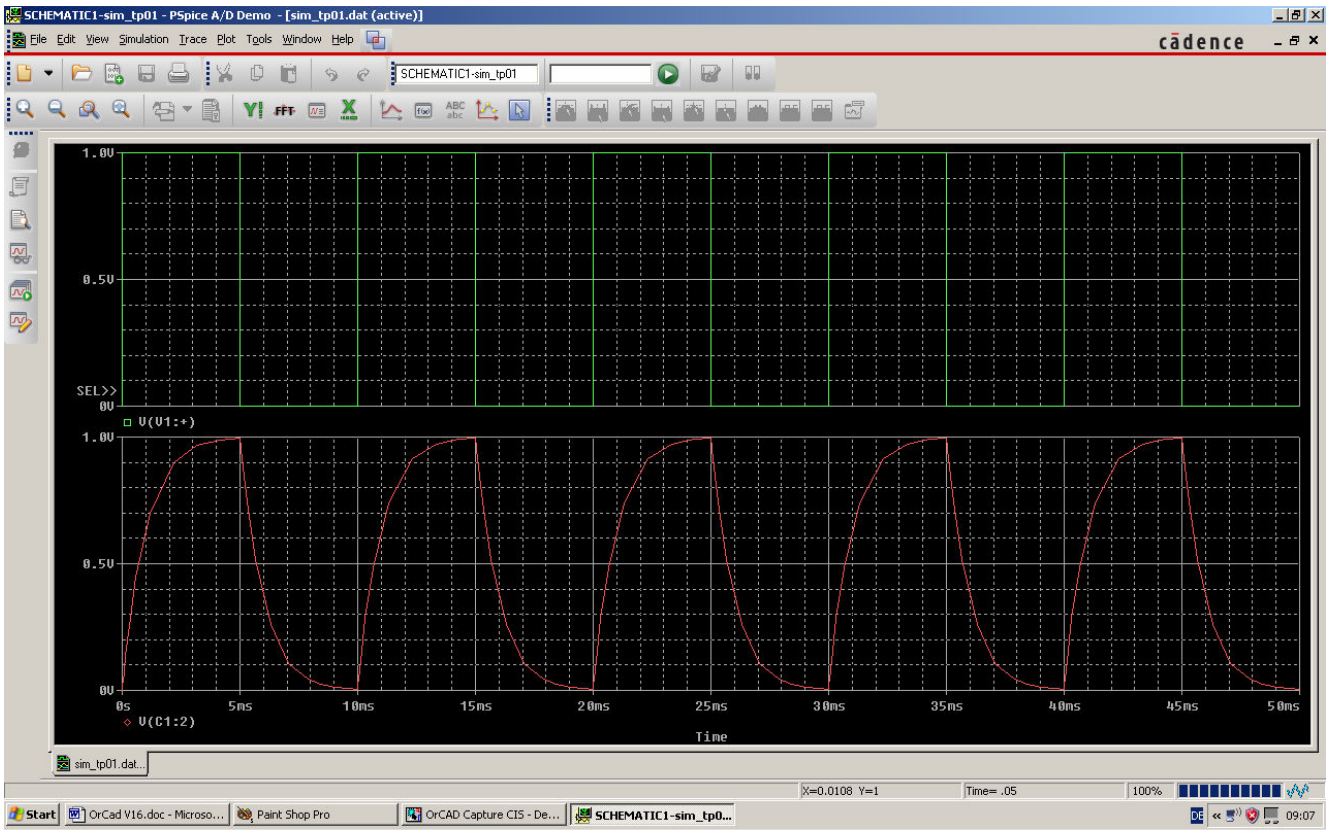


Suddenly we have two diagrams and the new empty one is marked by „**SEL**“ (= selected).

So go into the Pulldown Menu „**Trace**“, press „**Add Trace**“ and choose **V(V1:+)** from the appearing list. At once the Input Voltage is presented in the upper diagram.

To delete this curve from the lower diagram, click on his vertical axis to select it. Then click on “**V(V1:+)”** in this diagram to mark the curve and with the DELETE Key the delete process can be finished.

So the screen looks now like:

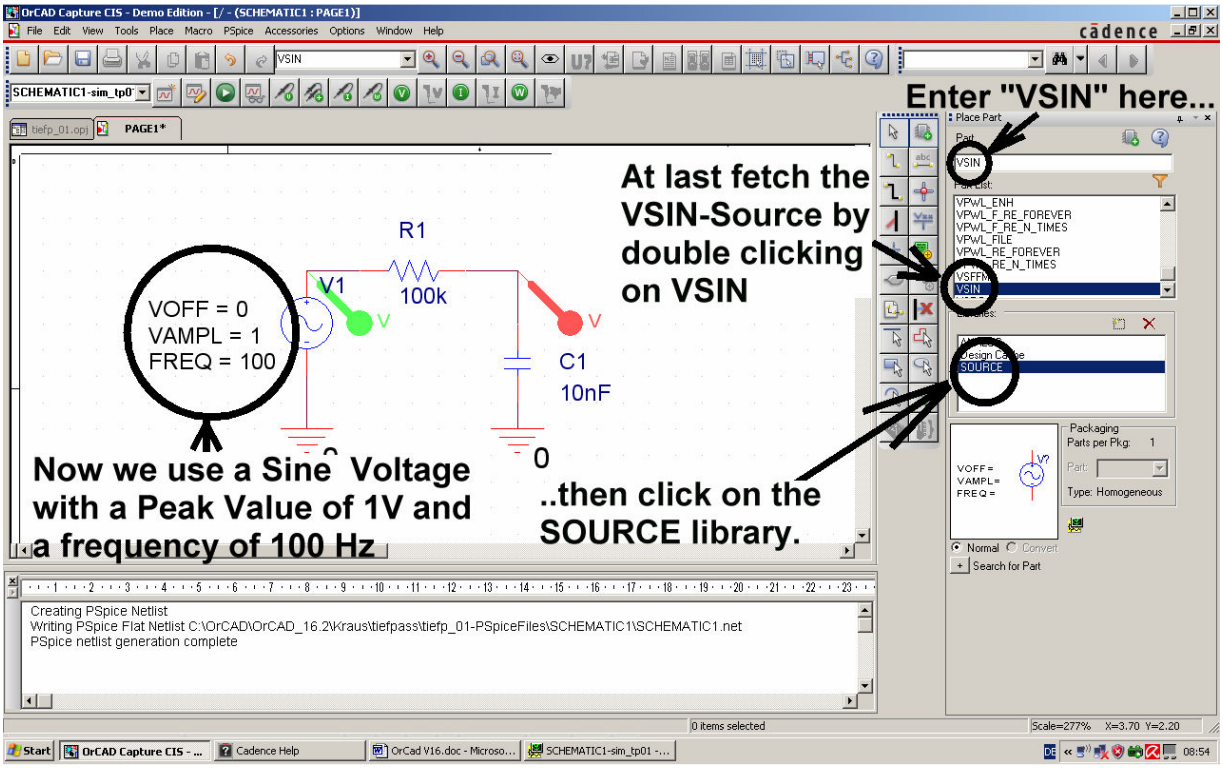


If You want to repeat the simulation with other values, switch to the editor screen, do the wanted or necessary changes and press again the Simulation Button with the triangle.

5. Other Transient Simulations

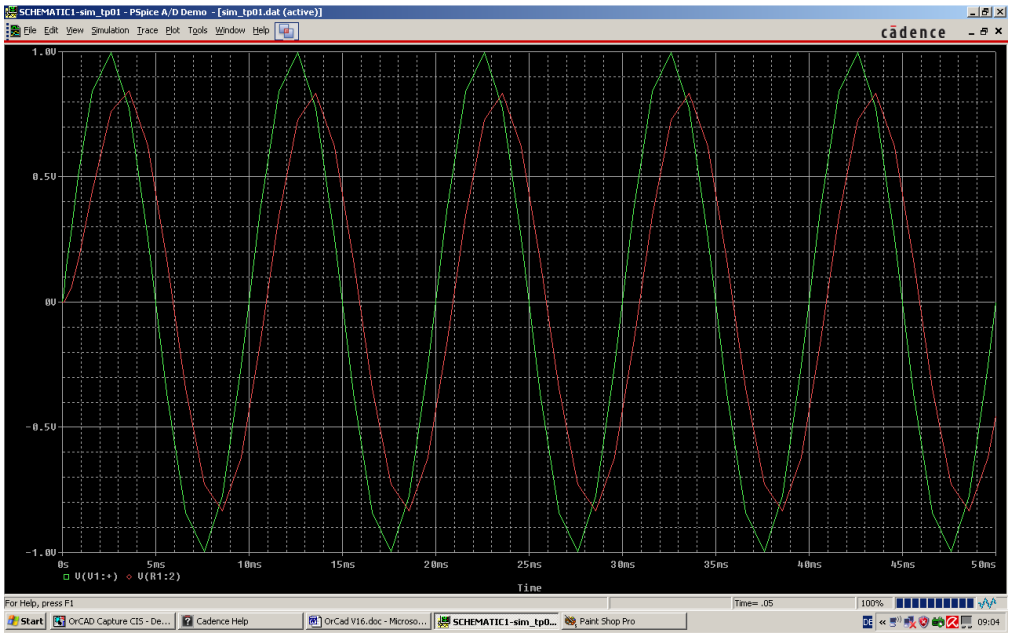
5.1. RC – LPF fed with a Sine Wave

We delete the Pulse Voltage from the schematic (= mark it and afterwards press DELETE on the keyboard). Now we open the part library and clcik double on „**SOURCE.olb**“ after entering „**VSIN**“ in the Part Window. The Sine Voltage Source hangs now on the cursor and can be inserted into our circuit.

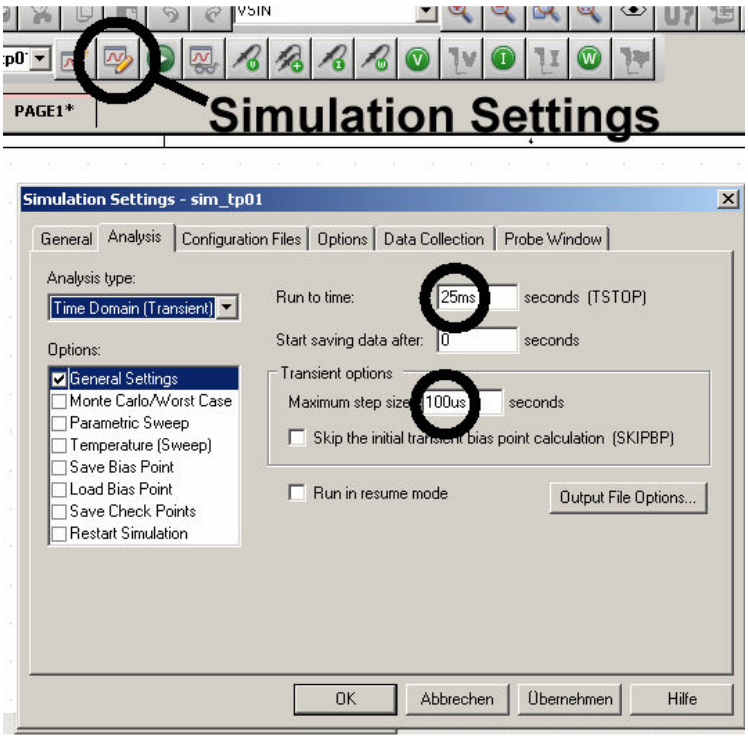


Programming the Sine Voltage Source is not difficult: enter the values for the **DC Offset Voltage** (= 0V), the **Peak Value** (1V) and the **Frequency** (100 Hz) after double clicking.

The „Probe-Window“ will now be seen as follows:

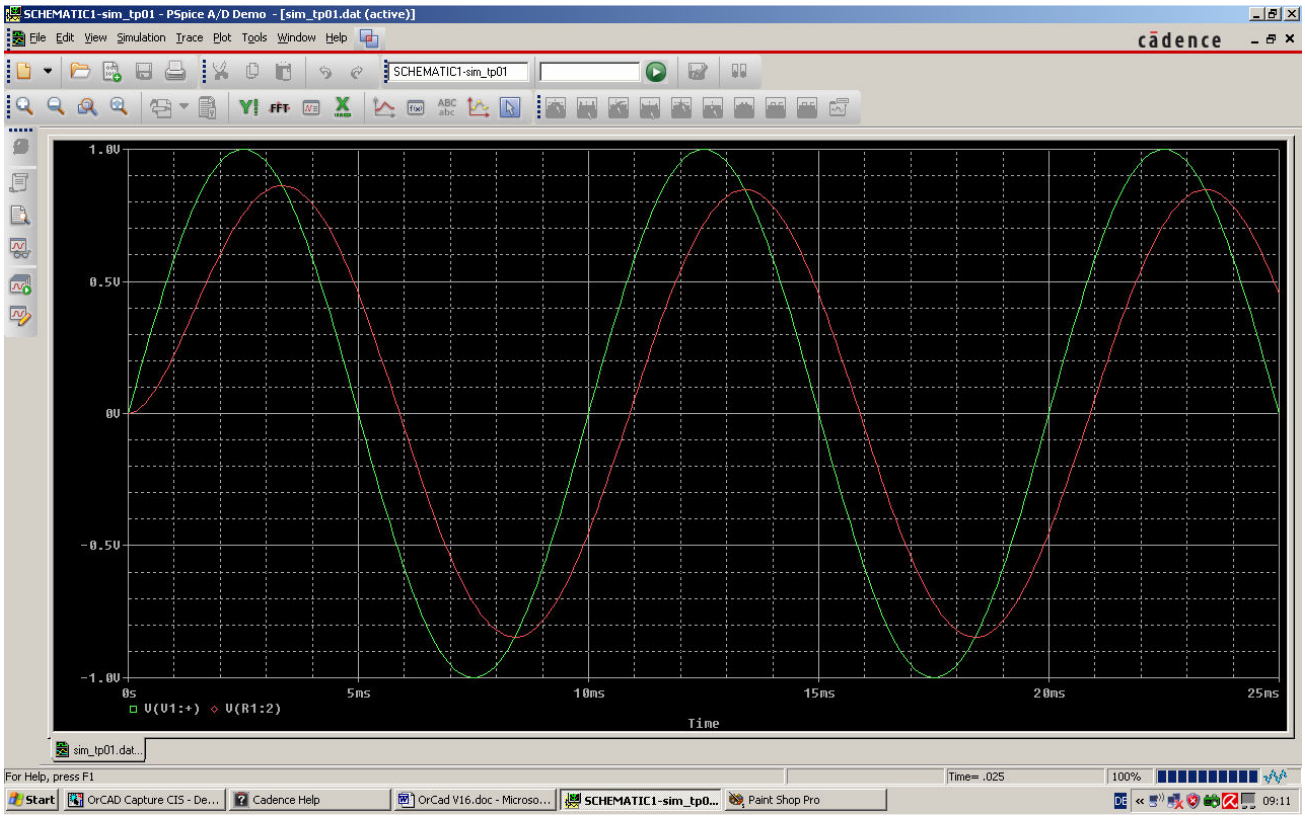


If we examine the curves, we see a lot of corners. That can be eliminated by increasing the numers of Simulation Points. Also the Simulation Time Intervall should be increased to 0...25ms



So open the „**Simulation Settings**“ and enter 25ms at „**Run to time**“.
Set the Maximum Stepp size to **100 Mikroseconds.**

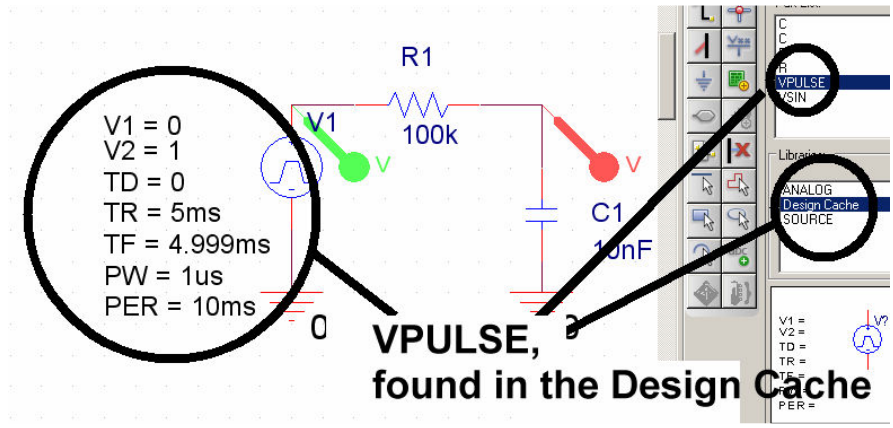
And this is the result which looks much more better.



Additional Task:
a) Try to show the Voltage at the Resistor in a separate diagram. Use for this task the „**Differential Voltmeter Marker“**.

b) Calculate the **Cutoff Frequency of the LPF and simulate this case.**
Check the Phase- and Amplitude Differences at this frequency in the Simulation Result Diagram

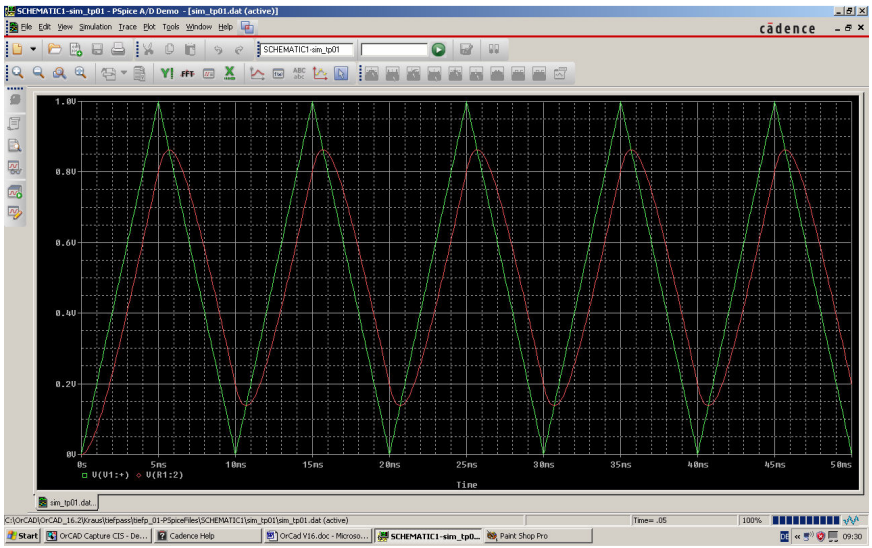
5.2. RC – LPF fed with a Triangle Wave



We need not to change the part values but we replace the VSIN-Source by the PULSE Voltage Source (see first simulation) which ca be found in the „**Design Cache**“ .

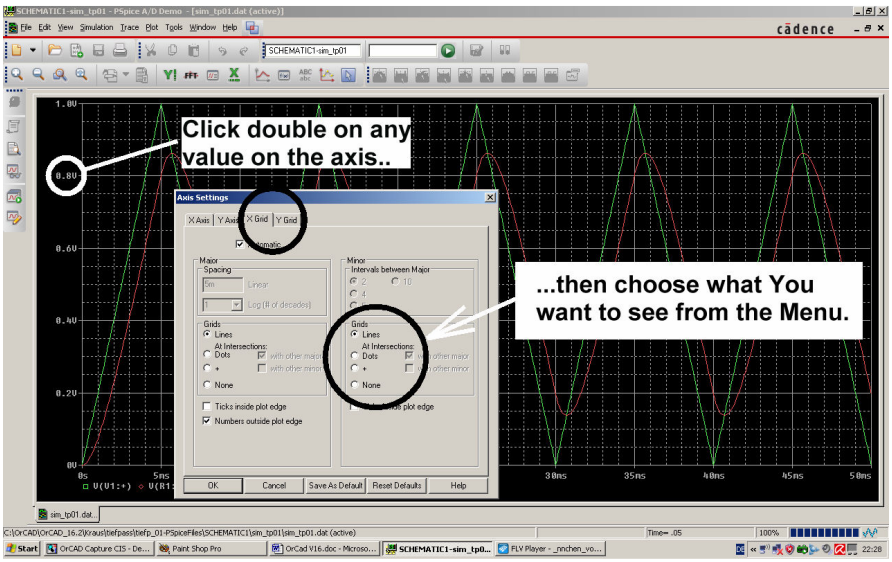
To produce a TRIANGLE Voltage Source, we choose a long rise time (5ms), a fall time of 4,999 ms and a very short pulse width (1 Microsecond).

So the frequency is set to f = 100 Hz.



For a Run Time of 50 ms we get this result.

5.3. Change the Grid and use a Cursor



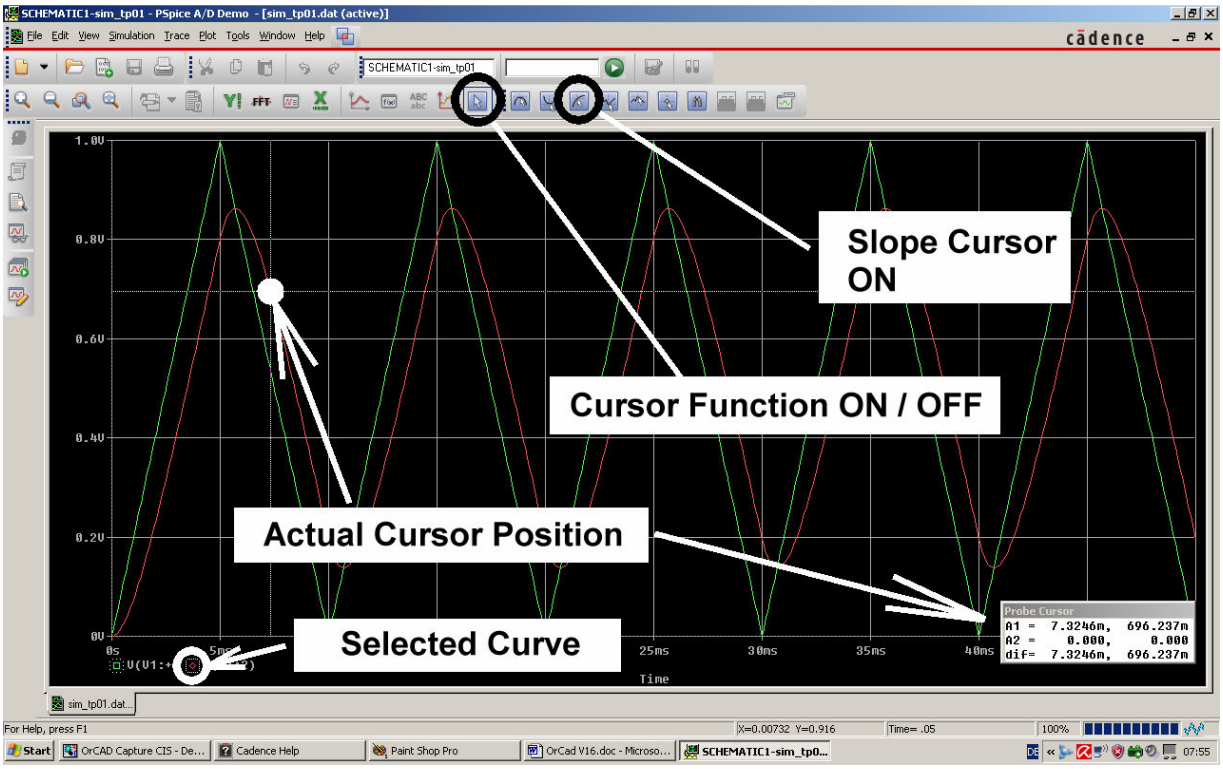
A **grid** over the result diagram is very usefull for checking the amplitudes at dedicated points.
But where can it be found….? And how can it be switched off when no longer necessary...?

So have a look at this screenshot an do what You want.

Very usefull is the „**Probe-Cursor**“ to determine exact Time or Amplitude Values:

a) **Switch off the X-Grid and the Y-Grid by activating „None“ in the menu** (See last page). So You can better watch the cursor's task.

b) Now activate the Cursor Function and press the **CURSOR ON – Button**. Then use the „**Slope-Cursor**“:



Press the left Mouse Button and roll the mouse to shift along the curve. The actual Cursor Position (Time + Amplitude) is now always indicated in a window (right hand side of the screen).

To change the selected curve click on the preceeding little symbol of the curve's name.

Attention:
If You click with the right Mouse Button anywhere in the diagram, a second cursor is activated. So you can easily measure Amplitude – oder Time – Differences.

Task:
Play with the different Cursor Functions which are available.

Delete all cursors by simply pressing the Cursor Function ON / OFF – Button.

5.4. For Communication Specialists only: RC – LPF fed with a FM – Signal

In a FM-Signal the Carrier Frequency is periodically varied by a second signal (= Information Signal). So linear distortions of a system can be identified by a simple look at the OUTPUT Signal.

Let us use the well known LPF for this purpose.

a) The LPF consists of the **100 KΩ - Resistor** and the **10 nF-Capacitor**.

b) Delete the Voltage Source from the schematic and replace it by the FM Voltage Source
c)

VSFFM

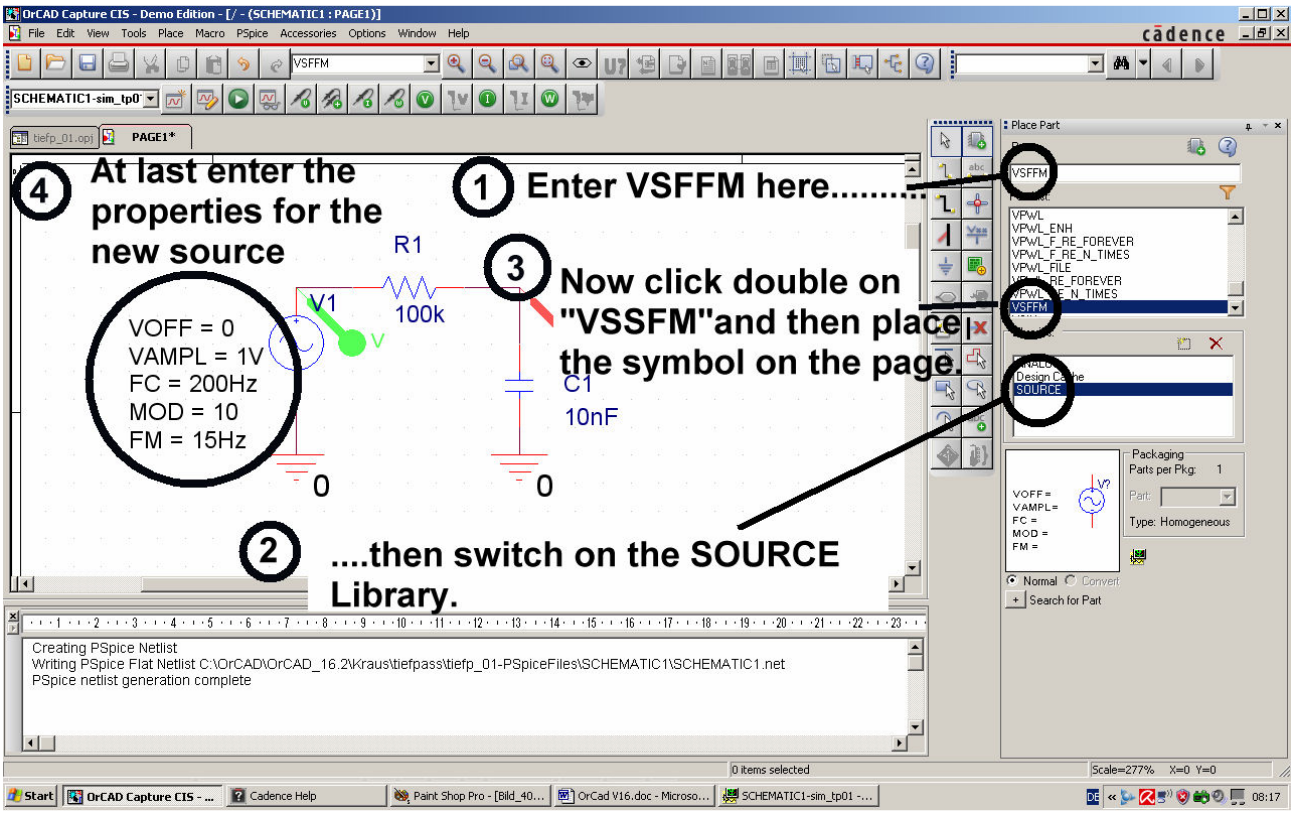
(which can be found in the Source-Library)

Now we set the Source's properties as follows:
a) No DC OFFSET. It means that **Voff = 0**.

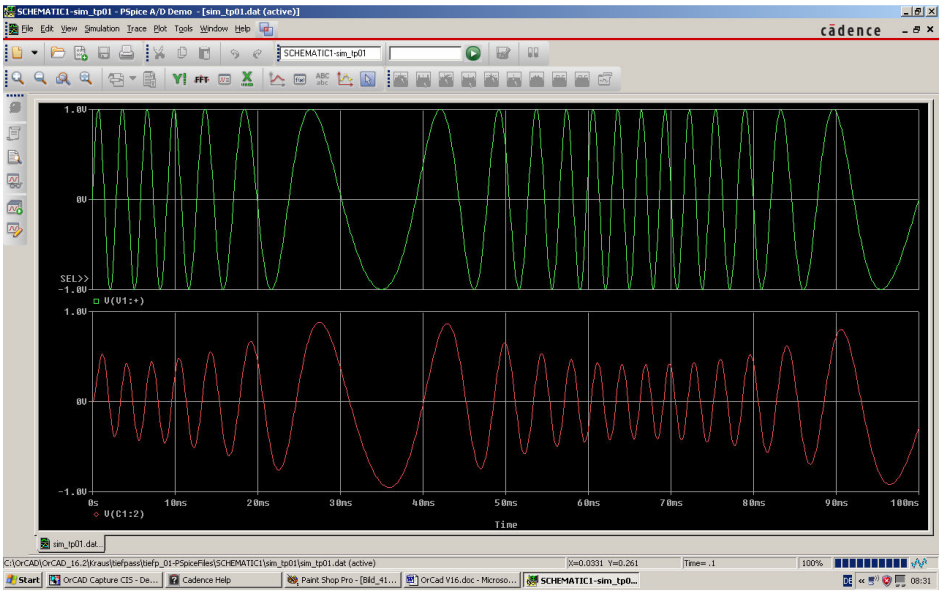
b) **Amplitude and Center Frequency of the unmodulated FM-Signal are VAMPL = 1V and Fc = 200Hz.**

c) **The frequency of the modulating signal is 15Hz**.

d) **Modulation Index MOD = 10.**
(The Modulation Index gives the maximum Phase Devation in Radians, caused by the modulating signal in comparison to the unmodulated case. So one period of maximum deviation produces **MOD = 360 degrees = 2π = 6,28 in both directions)**



Now enter a Simulation Run Time from 0…100ms (in the Simulation Setting Menu) and start the simulation. You should get the following result:



---

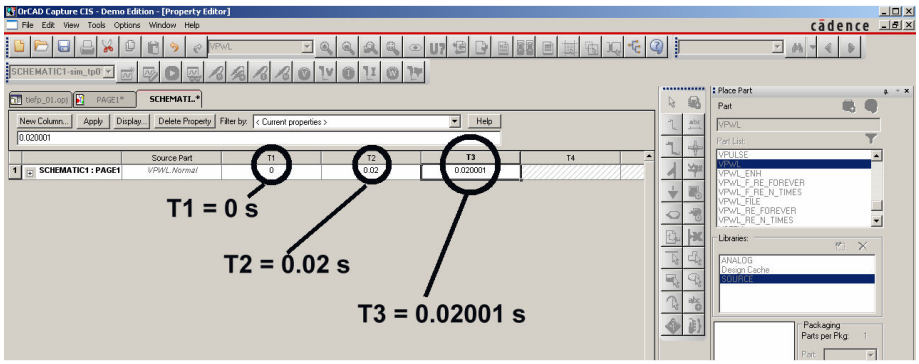
6. Simulation of non-repetitive signals

6.1. Step Response h(t) of a Low Pass Filter

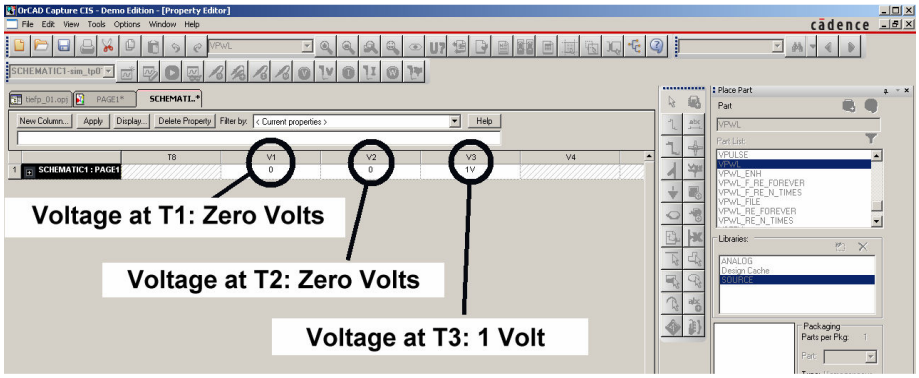
Therefore we use the **VPWL – Source form the Source Library to generate the desired waveform by adding short linear pieces.**

We start at a value of 0V for 20ms, then let the voltage rise to 1V during an intervall of 1µs. The LPF itself stays unchanged (R = 100k / C = 10nF). Now we double-click on the source's symbol to set the properties. Scroll the opening window wide to the right side to see where to enter the voltage values at the desired time points. And then we start to program the curve form:

At T1= 0 V1 = 0
At T2 = 0.02s V2 = 0
At T3 = 0.020001s V3 = 1V

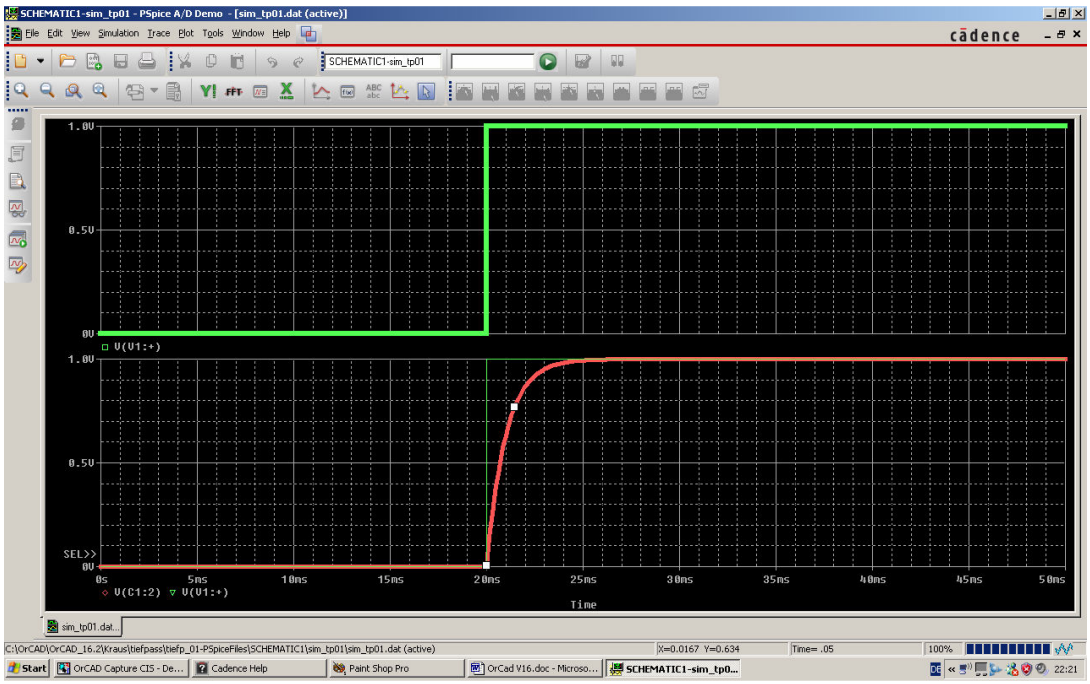


These are the 3 times…



…and here we have to enter the voltage values at these times°

Finally choose in the “Setup Settings” a Run Time of 0…..50ms and simulate. Then modify the result diagram to get two separate diagrams and enlarge the line width (in the Property Menu after a right mouse click on a curve) to get this nice picture.



Additional Task:

Let the voltage stay on 1V for 20ms and then switch back to 0V. So You see the switching ON and OFF process.

6.2. Impuls Response g(t) of a RC - LPF

In the modern Developement World a lot of work is done by **Fourier -, Fast - Fourier- and Laplace - Transformation**. So You can analyze and preview a system's response, if You know the **Transfer Function** of the System.

The Transfer Function is the ratio of the Output Signal to the Input Signal in the complete regarded frequency range (measured as „Magnitude and Phase" but mostly expressed in complex form).

So let us have a look at the Impulse Response g(t) and her task.

If you feed an unknown system with a „Dirac-Pulse" at its input, then the output signal = Impulse Response g(t) contains all informations to determine the Transfer Function.

And when You then know the Transfer Function, it is possible to calculate the output signal for ANY INPUT SIGNAL WAVEFORM by „convolution"!

The **Dirac – Impulse is** a square wave pulse with an infinite amplitude and a pulse length with goes to zero, So his area is „1" and seems to be nonsense. But it is without any problems possible to replace it by a pulse with a big real amplitude and a very very short real pulse length -- but the same area! A recommendation for practical use is that the time constant of the regarded system should be 100…1000 times larger but the Dirac Pulse Length. In this case you get the same results as when using a "true Dirac Signal".

Let us examine that at our LPF

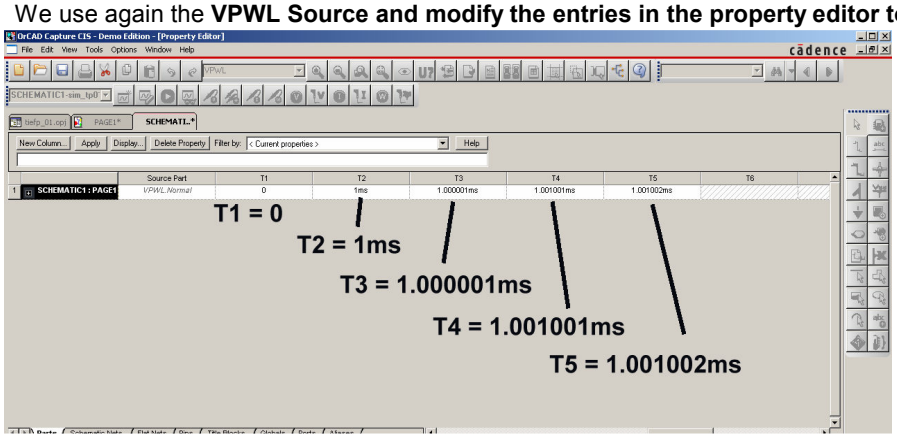
Task 1): Feed the input of the LPF (R = 100kΩ, C = 10nF) with a pulse with the following properties: **Vampl = 1 Megavolt / Delay - Zeit = 1 ms / Rise Time = 1 Nanosecond / Pulse Length = 1 Microsecond / Fall Time = 1 Nanosecond.**

This gives an area of **1 Megavolt x 1Microsecond = 1Voltsecond**

Use a run time of 5 ms.

Task 2): Reduce the **Amplitude to 100 Kilovolt** and therefore **enlarge the Pulse Width to 10 Microseconds to get the same area**. Simulate and compare the result with task 1.

Task 1:
We use again the **VPWL Source and modify the entries in the property editor to enter the "Dirac Data"**



Amplitude: 1Megavolt,

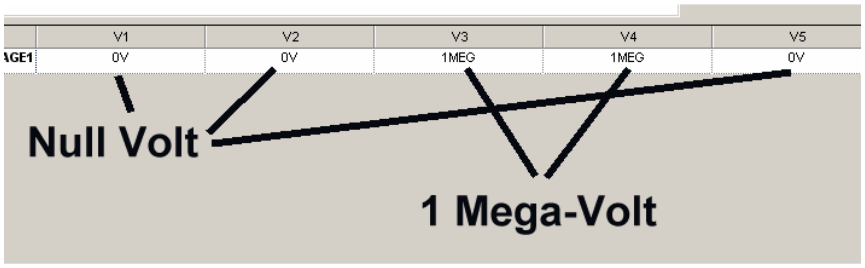
Pulse Length: 1µs

Rise Time: 1Nanosecond

Fall Time: 1 Nanosecond

So we have to enter the Time Values

0 1ms 1.00001ms 1.001001ms 1.001002ms



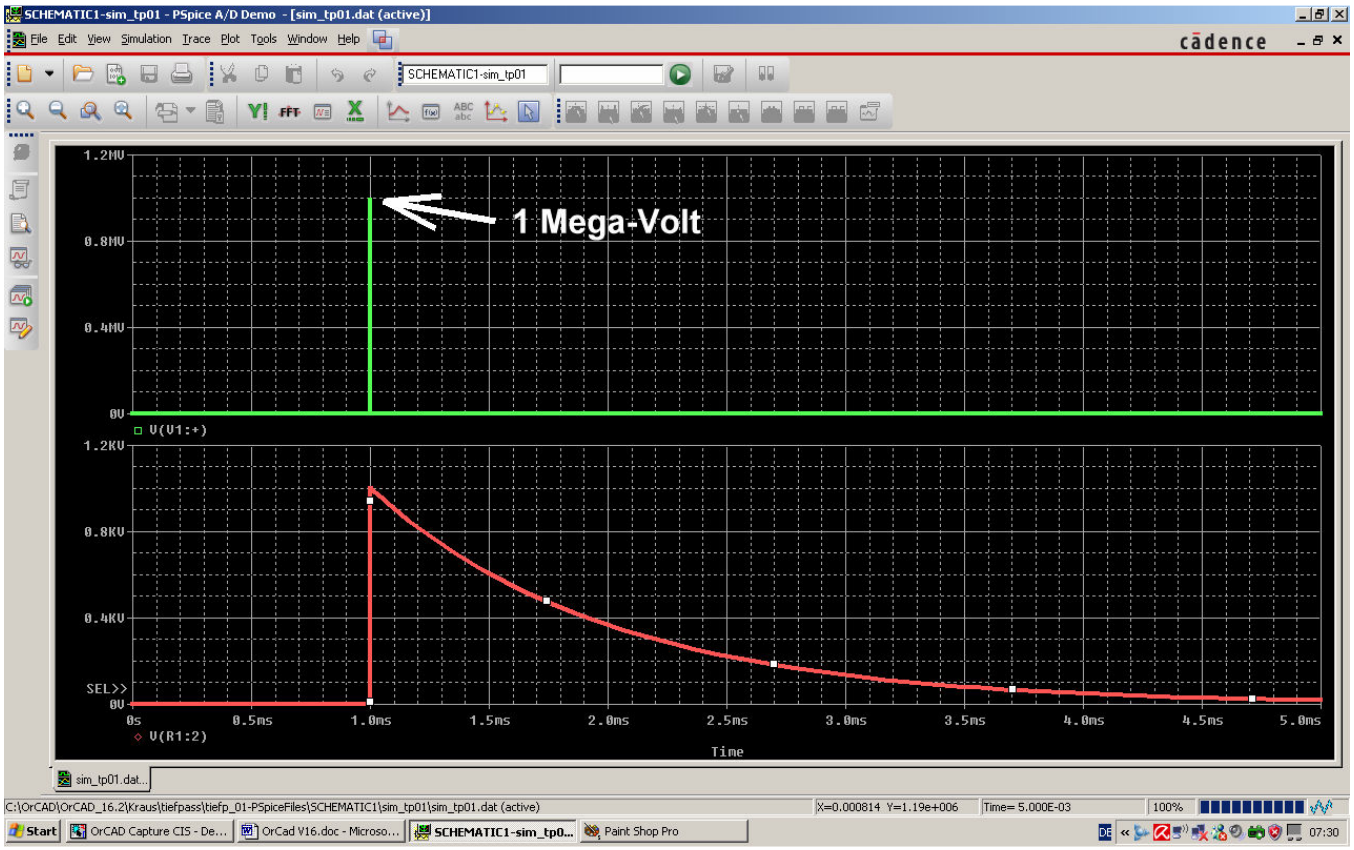
And the Voltage Amplitudes at these 5 times can be seen in the left screenshot.

In the Simulation Settings we enter a Run Time from 0…**5 ms** and simulate

Then present the result in two separate diagrams because there are big differences between the two maximum amplitudes:

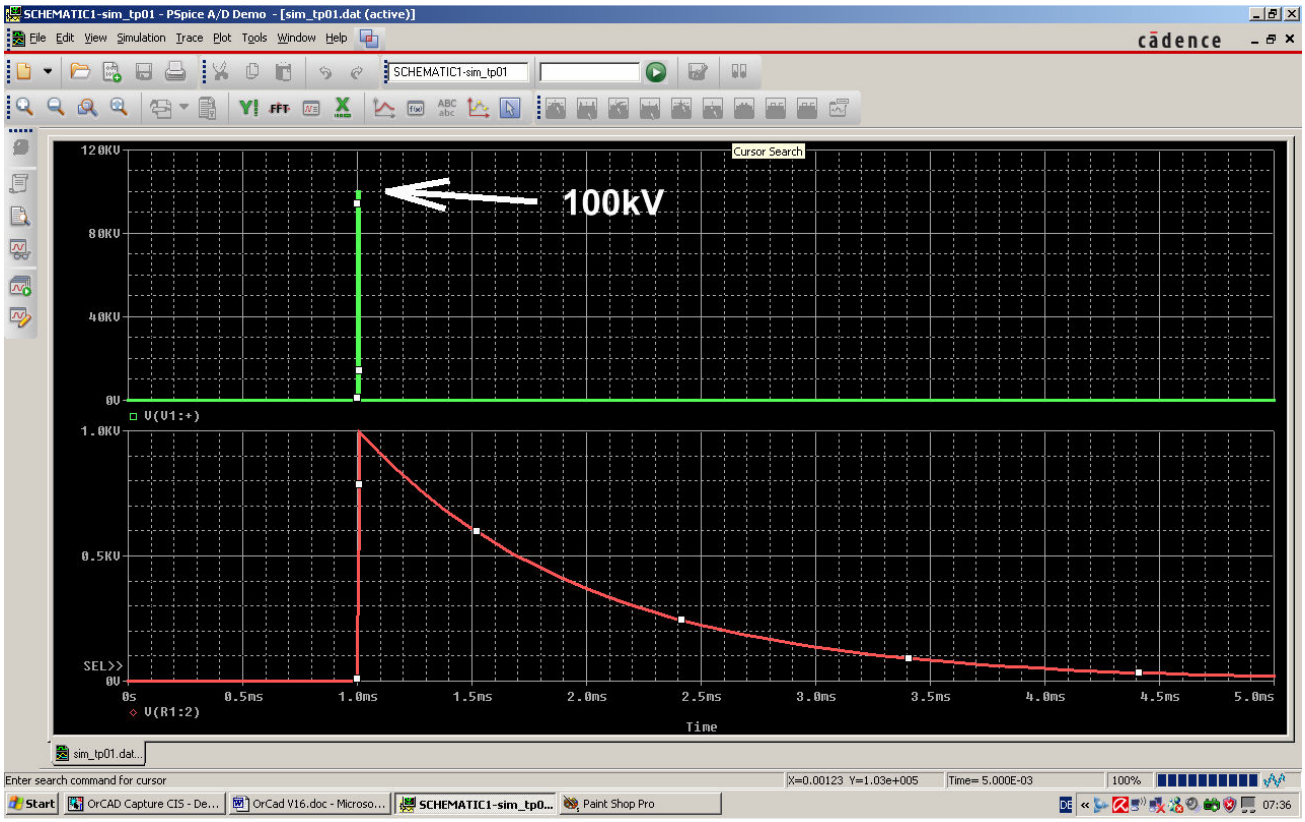
Input: 1 MV, Output: 1kV

Also enlarge the line width of the two curves:



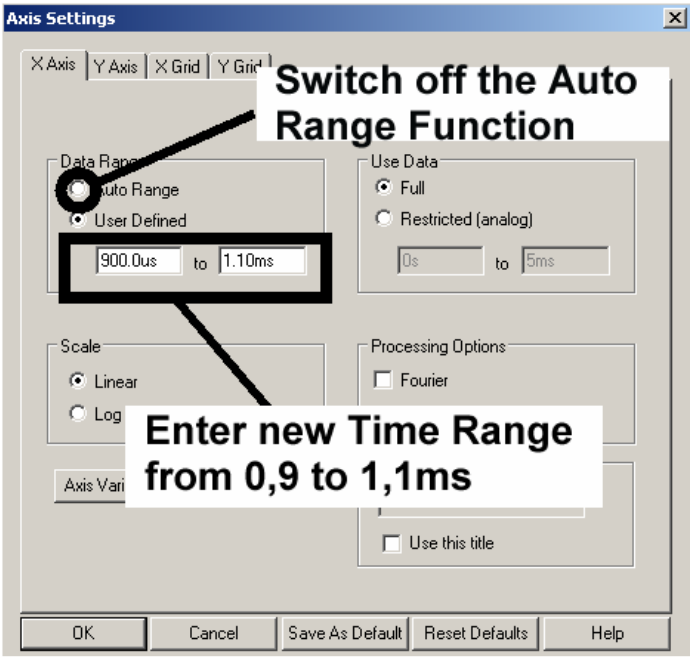
In task 2 the **Amplitude** of the Dirac Pulse is reduced to 100kV, but the Pulse Length enlarged to 10µs.
These are the new properties for the Voltage Source:

T1 = 0		**V1 = 0**
T2 = 1ms	**/**	**V2 = 0**
T3 = 1,000001ms	**/**	**V3 = 100k**
T4 = 1,010001ms	**/**	**V4 = 100k**
T5 = 1,010002ms	**/**	**V5 = 0**



The result is the same as in task 1.

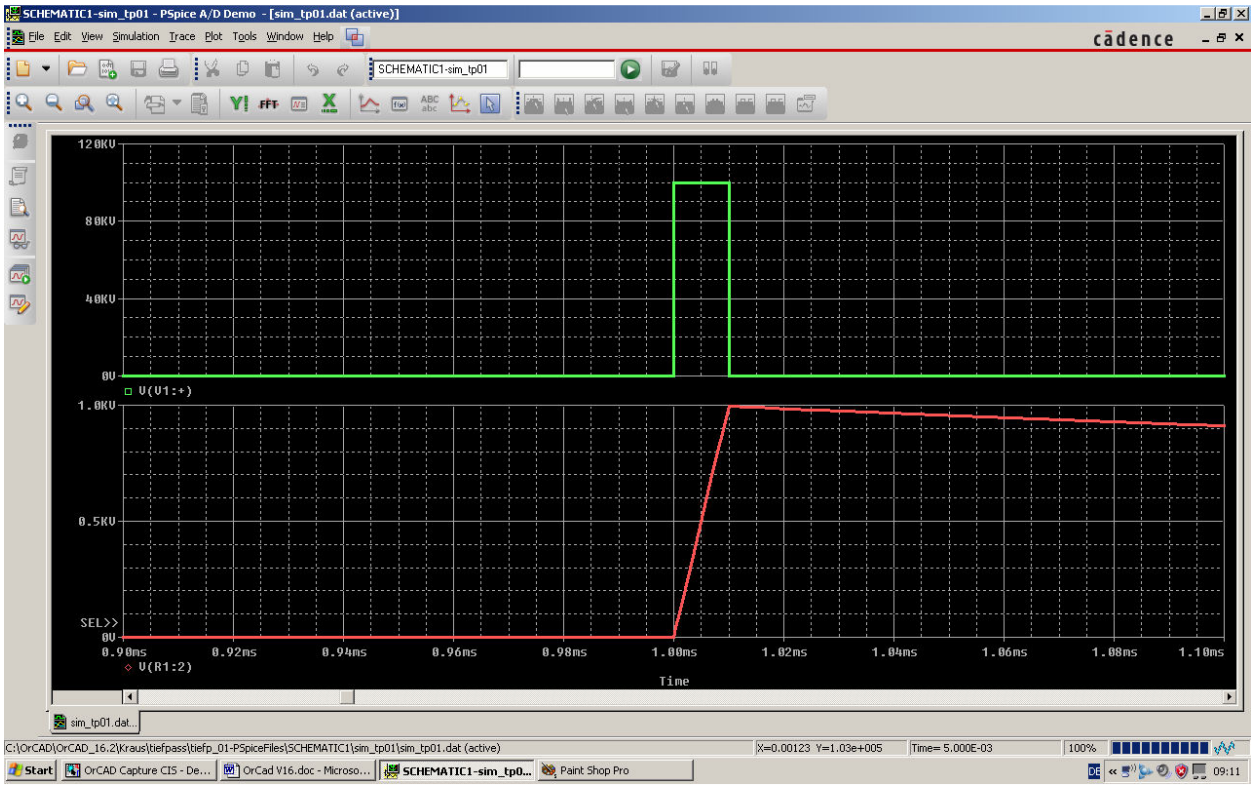
6.3. Modification of the Result Diagrams



To examine the details of the varied Dirac Pulse we choose only a Time Range from 0,9 to 1,1ms.

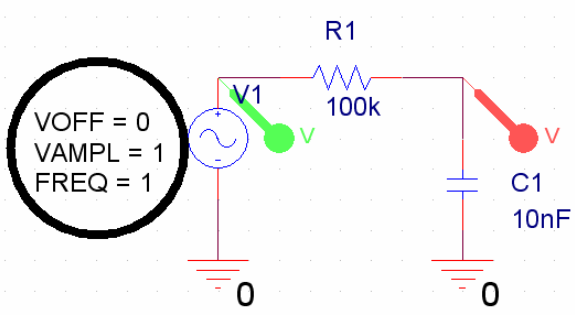
So we click double on a number on the vertical axis. At once a new menu is opend. First we switch off the Auto Range Function and then we enter the desired Time Range from **0,9 Milliseconds to 1,1 Milliseconds**.

Now the details of the programmed VPWL-Voltage can be seen.



7. Sweeping the Frequency Response of a RC-LPF
7.1. Linear Presentation

Attention:
„Frequency Response“ is a special case of the Transfer Function for a LTI-System (…LTI means „linear and time-invariant“). When simulating the FREQUENCY RESPONSE, You are using an ideal SINE WAVE at the Input .. which has started some hundred years before and will not end until 2350…
The Transfer Function is generally the ratio of the output’s answer to any input signal form.

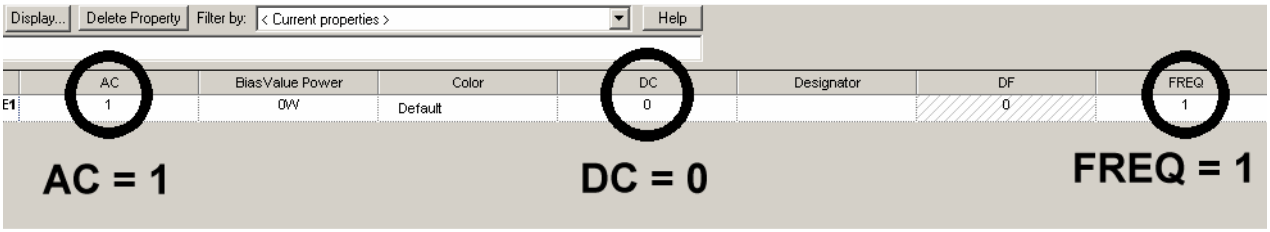


So let us apply a Sine Wave to the input of our well known LPF.
So fetch the VSIN-Source from the Cache or from the Source Library and install it in our schematic.

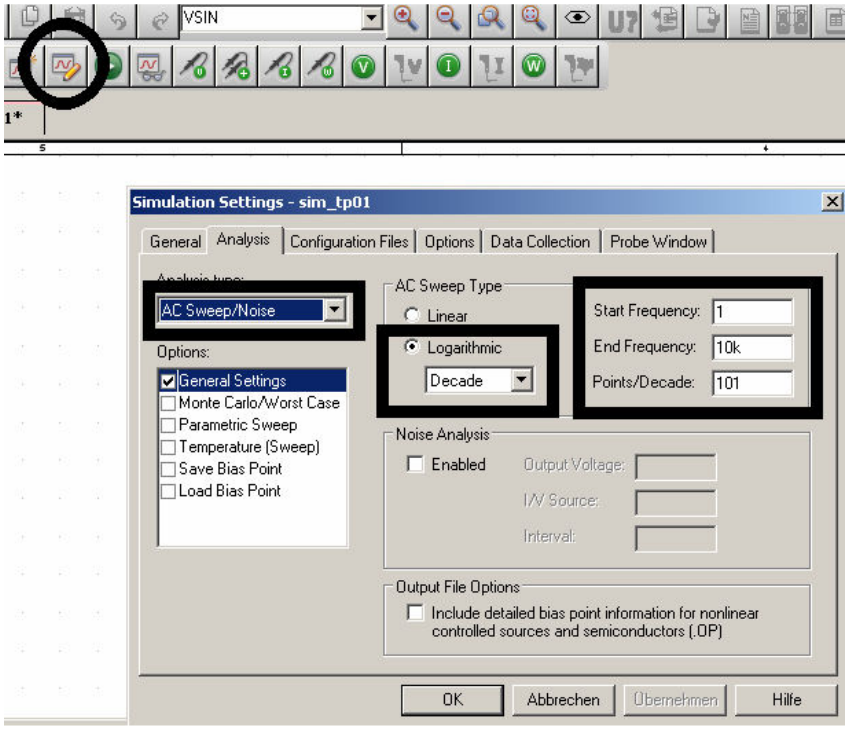
Enter the following properties at the source:

VOFF = 0
VAMPL = 1
FREQ = 1

Then click on the symbol of the source to open the Property Editor. Please enter:



Now open the Simulation Settings:



In the Analysis Window please switch to **Sweep / Noise**“ and then enter the necessary data:

Logarithmic Sweep

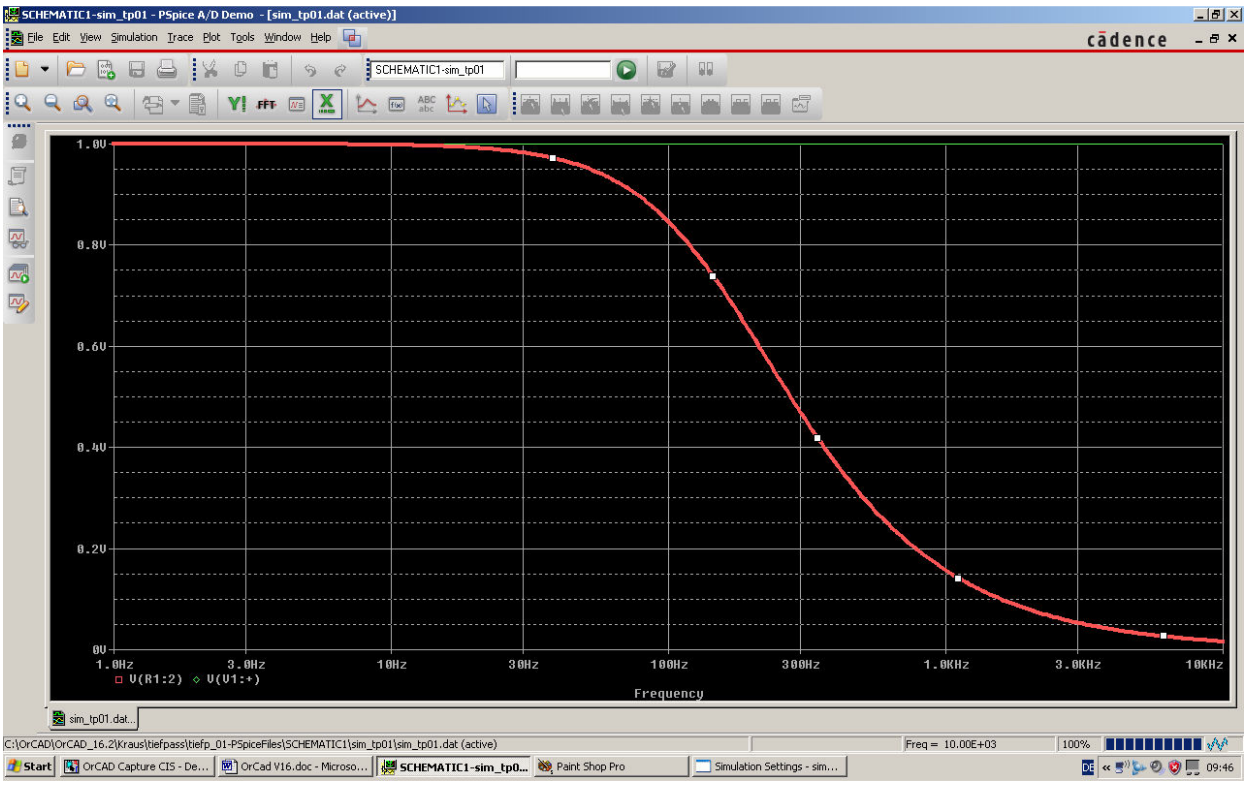
Decade

Start Frequency = 1Hz

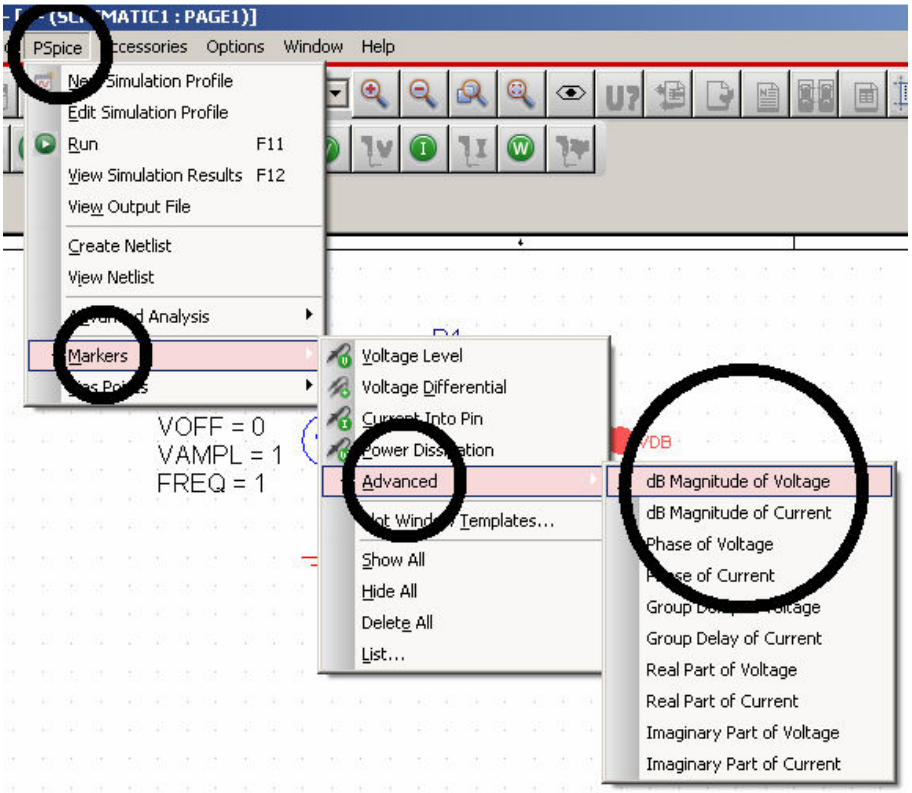
Stop Frequenz = 10kHz

101 Points per Decade

Finally we say “OK” and simulate (with Voltage Markers at Input and Output).
This is the result:



7.2. Presentation of the Frequency Response in dB



At first delete all Voltage Markers from the schematic. (= mark the symbol and click “DEL”).

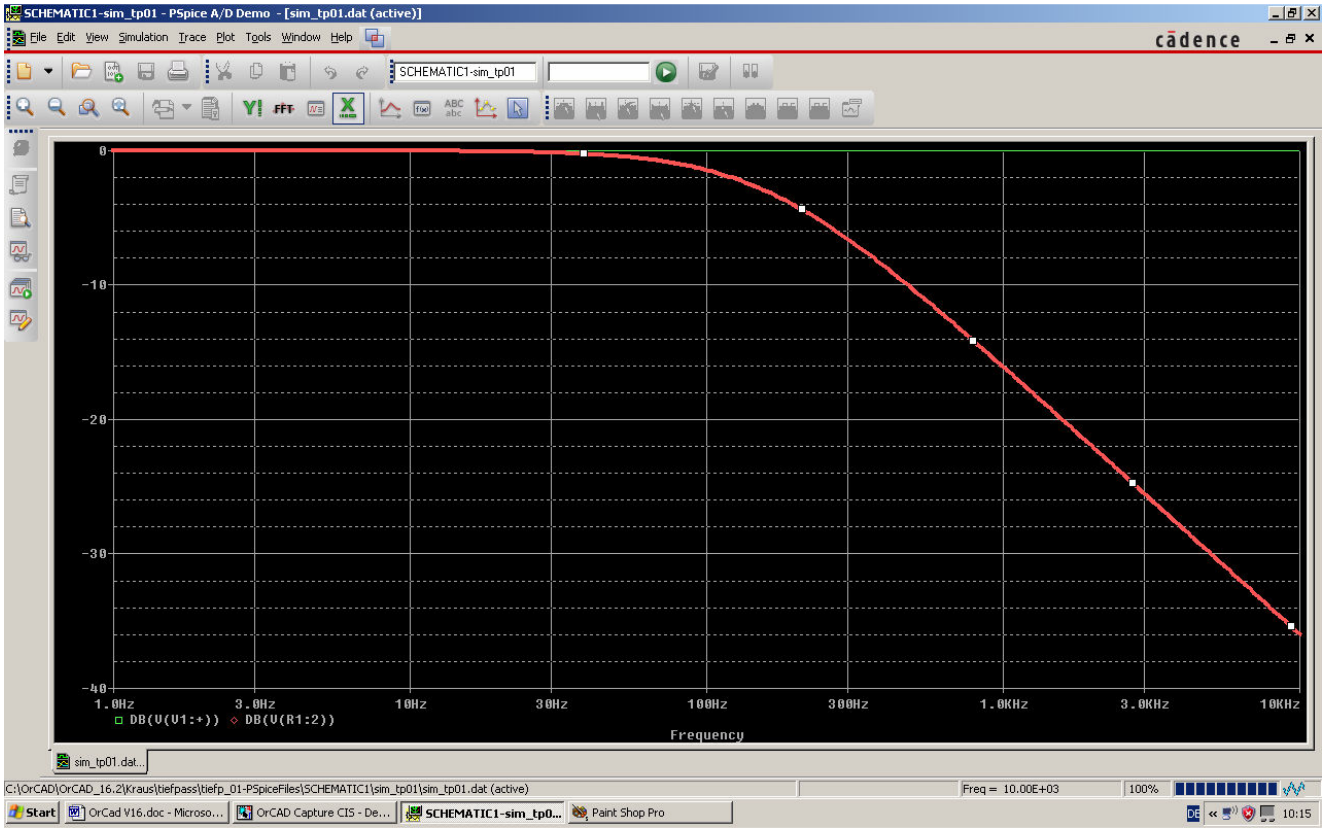
Now follow the path

Pspice / Markers / Advanced

to the option

db Magnitude of Voltage.

Two of these dB-markers are place left and right from the resistor in the schematic. Then the simulation can be started.

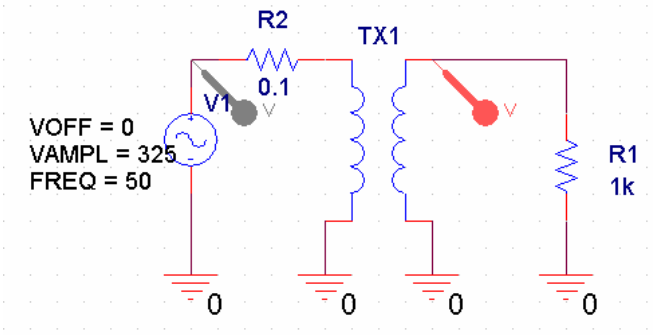


This is what we wanted to see.

8. Simulation of simple Power Supply Circuits

8.1. Preparing the Transformer

Close the LPF-Project and start a new project. Then fetch the following parts from the Libraries:



a) the transformer „**XFRM_LINEAR“ from „ANALOG.OLB“**

b) two resistors „**R“ from „ANALOG.OLB“**

c) the Sine Voltage Source „**VSIN“ from „SOURCE.OLB“**

d) two Ground Symbolsl **0 / CAPSYM**

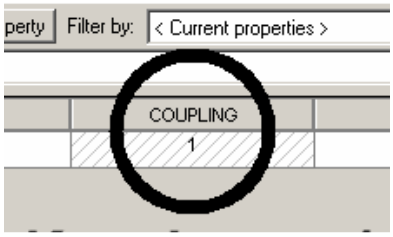
e) Two **Voltage Markers**

So please draw with them this schematic, wire all and place Voltage Markers at the input and the output.

Important:
Resistor R1 must never be missing, but can have a very small value (f. i. 1 Milli-Ohm). This is a fundamental rule when connecting a coil or a transformer to a Voltage Source (…the reason is very simple: otherwise the current in the coil could -- theoretically! -- rise to Infinite….).
If You forget to add this resistor you have the chance for a cryptic error message or a simulation breakdown.

But now let us start with the settings and begin with the line voltage. So click on the symbol of the voltage source and enter the following properties:

Frequency f = 50Hz A peak value **VAMPL = 325V** gives an RMS value of 230V. The DC Offset **VOFF is Zero.**



Programming the transformer is a little more difficult. You have to start with the **magnetic coupling** of the primary and secondary winding. A coupling of “1” (= 100%) means that we have ideal conditions and the complete magnetic field of the primary winding is running through the secondary winding. There it induces the secondary voltage. In our example we want to work with this ideal transformer (**COUPLING = 1**).

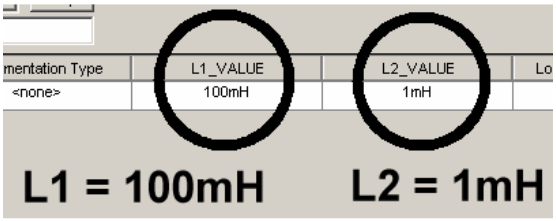
With a double click on the transformer’s symbol we open the properties and scroll right to enter this coupling value.

Attention:
In PSPICE it is not possible to work with the Voltage Transfer Ratio “r = U1 / U2” of a Transformer, because the Software works with parts and their real physical properties.
So we must use the primary and secondary inductance of the transformer, combined with the magnetic coupling of both.
But how to tackle this problem? Very simple. Take a real primary inductance (let’s say: L1 = 100mH) and calculate the corresponding secondary inductance by the formula

$$L2 = L1 / r^2$$

Example.
We need a Voltage Transfer Ratio of **r = 10**. With L1 = 100mH this gives

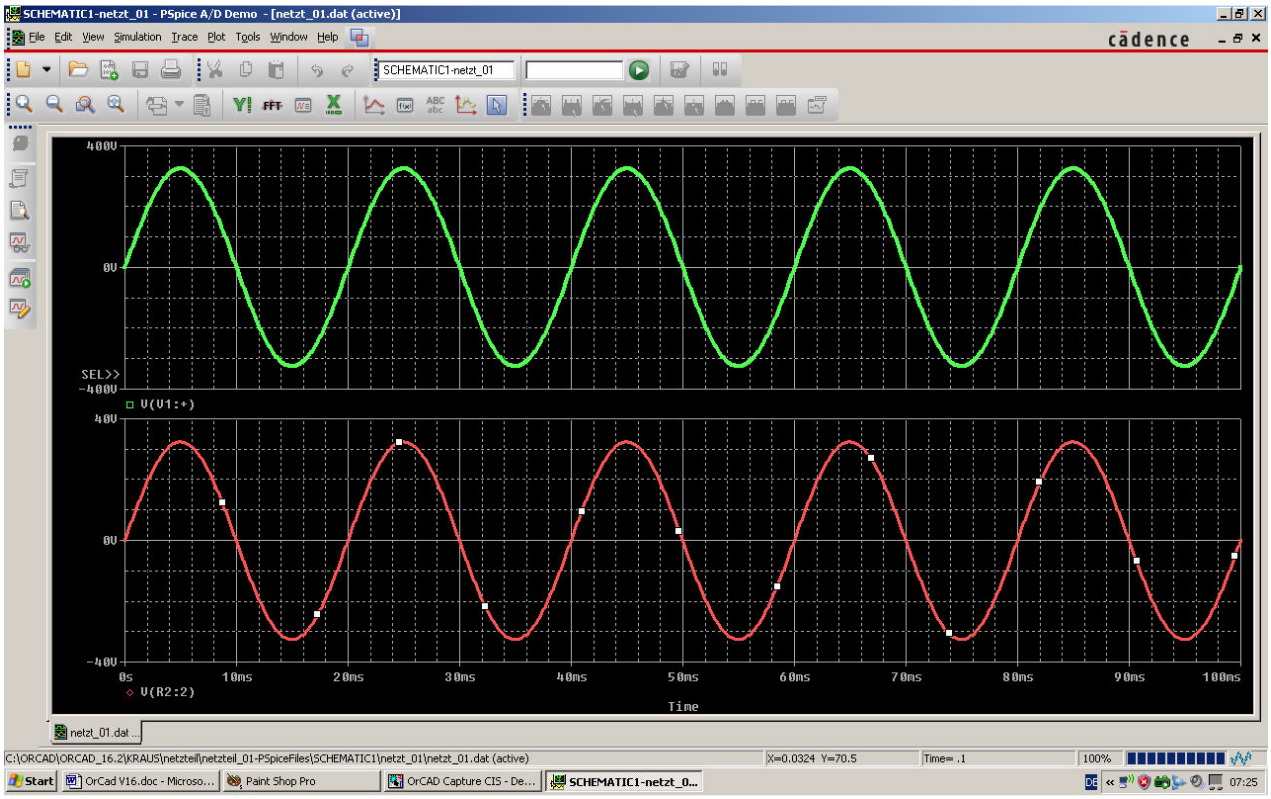
$$L2 = 100mH / 100 = 1mH$$



So open the Property Editor of the transformer and enter these Inductance Values.
Also check the correct value for the coupling (= 1).

Finally use Simulation Settings for a Run Time from 0 to 100ms with a step width of 0.1ms. Chose two different diagrams for the result presentation and enlarge the line widthes.

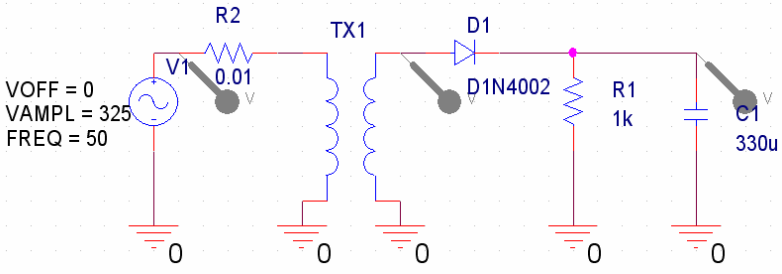
Now we se, that the Input Voltage is 10x larger but the Output Voltage -- as desired.



Additional Task:
Modify the circuit to get a Phase Shift of 180 degrees between these two voltages…..

8.2. One Pulse Rectifier

Now modify the Transformer Circuit to a One Pulse Rectifier with Capacitive Load.



Let the Sine Voltage Source unchanged (the source can be found as „VSIN“ in the “SOURCE.olb” – Library.

Remember:
If You search for parts and have only a Demo Version, always use the path

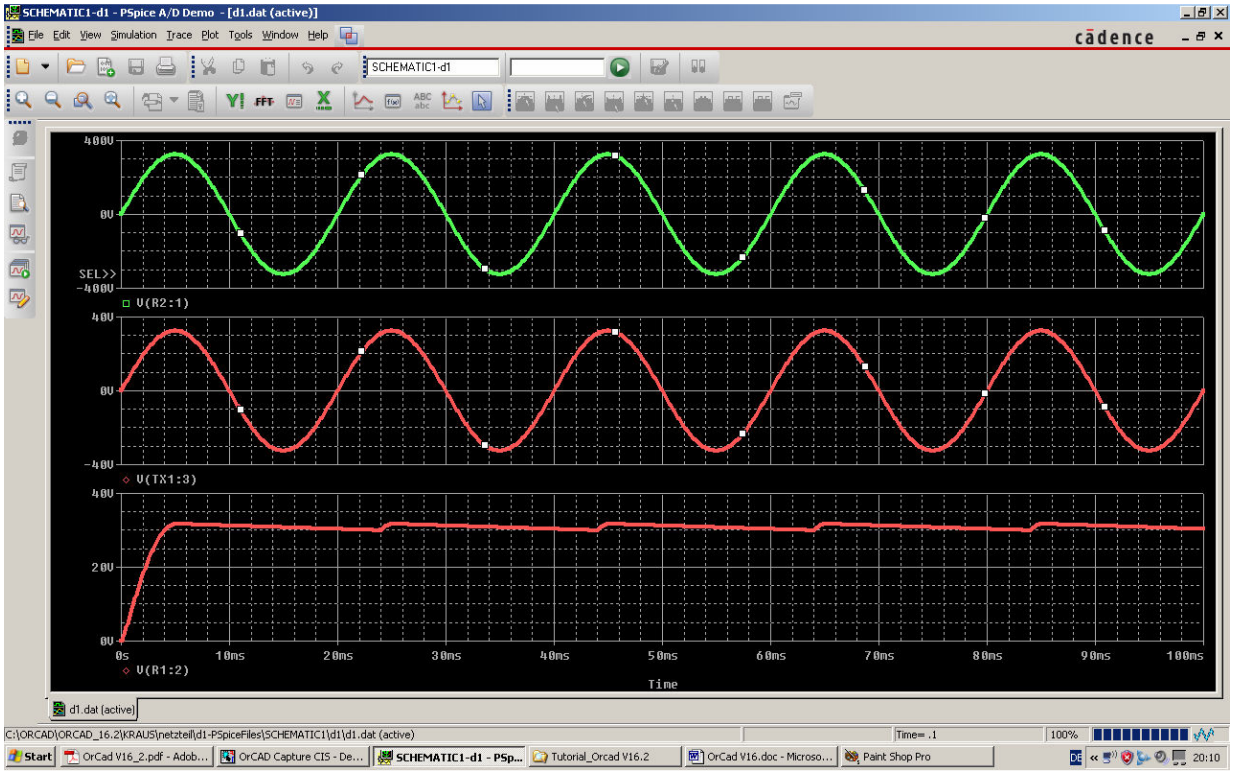
C: \ ORCAD \ ORCAD_16.2 \ TOOLS \ CAPTURE \ LIBRARY \ PSPICE \ DEMO

The Capacitor, the Resistor and the Transformer can be found in the **ANALOG.OLB – Library.**

Let also the Properties of the **Transformer** unchanged (**part“XFRM_LINEAR” in the ANALOG-Library / coupling = 1 / L1 = 100mH / L2 = 1mH**).

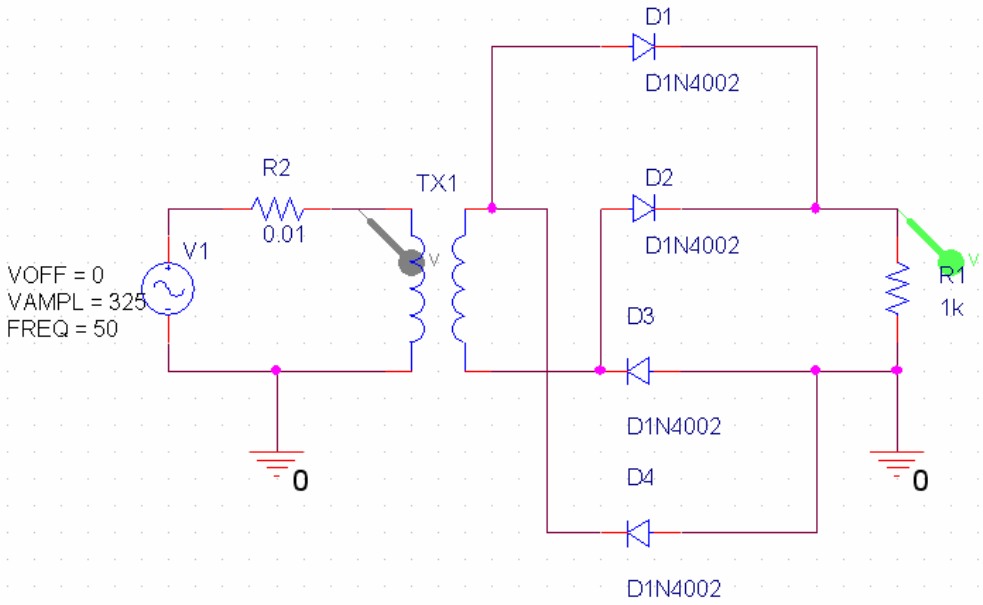
Use “**Part Search**” to find the Diode „**D1N4002**“ in the “**EVAL.olb”-Library.**

Simulate the 3 marked voltages and show the result in 3 separate diagrams (run Time = 100ms, maximum step size = 0.1ms).



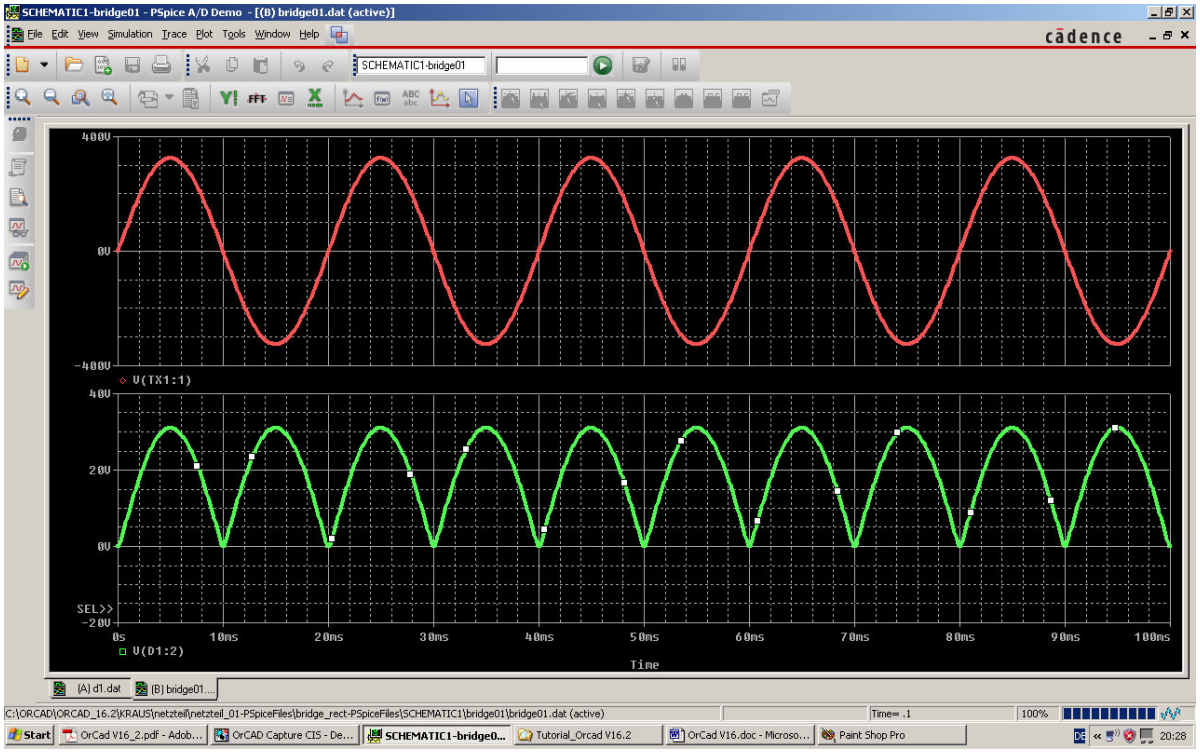
Additional Task:
Reduce the value of the capacitor and repeat the simulation. At last delete this part from the schematic. Show the Diode Current in a separate diagram.

8.3. Bridge Rectifier Circuit



Draw the schematic of this Bridge Rectifier without Load Capacitor and simulate the marked voltages.

Use the same Voltage Source and the same Transformer as in the last examples.



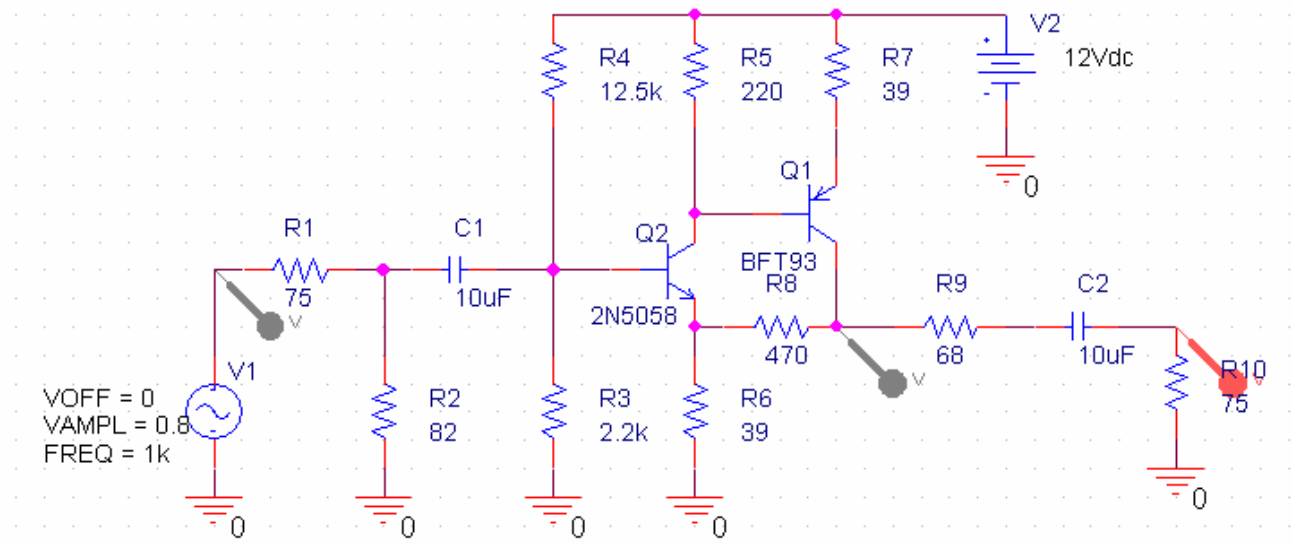
Very nice, indeed…

Additional Task:
Add a Load Capacitor (330µF) in parallel to the Load Resistor R1 and repeat the simulation for other values of this capacitor.

9. Two Stage Transistor Broadband Amplifier with Feedback

9.1. Time Domain - Simulation

Draw and simulate this schematic:



Circuit Description

The circuit uses two Emitter-Grounded Transistor Stages with Direct Coupling. The Output of the second stage is connected to the Emitter of the first stage.Due to the phase difference of 180 degrees at these two points this reduces the total gain. But this “Negative Feedback” reduces also the Nonlinear Distortions, stabilizes the Operating Point in the stages, reduces the Output Resistance and gives a higher upper Cutoff Freqency.

Input and Output Resistance are 75Ω, a Supply Voltage of +12V is needed.

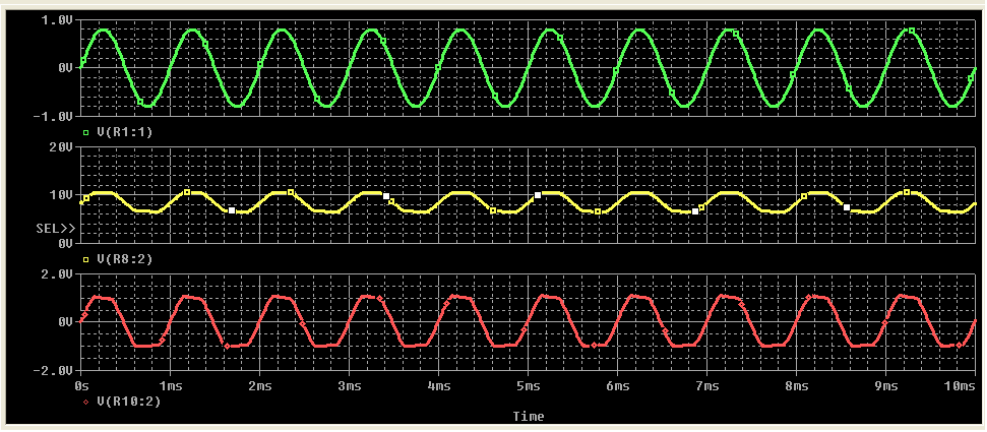
As Input Signal we use a Sine Wave Source **VSIN** with a peak value of **800 mV** at a frequency of 1kHz (Attention: this peak value will already overdrive the amplifier...).

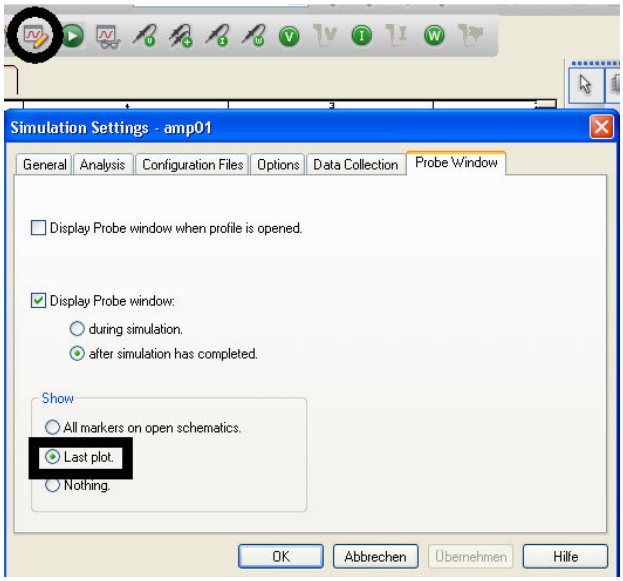
So please organize a new ORCAD - Project (f. i. „amp01”) and draw the schematic. Resistors and Capacitors can be found in the „ANALOG.OLB“, both voltage sources in the „SOURCE.OLB“ and the transistors in the EVALAA.OLB.

Now use the following Simulation Settings:

Time Domain (Transient) **Run Time 0....10 ms** **Maximum Step Width = 10 Microseconds**

and present the 3 marked voltages in different diagrams.

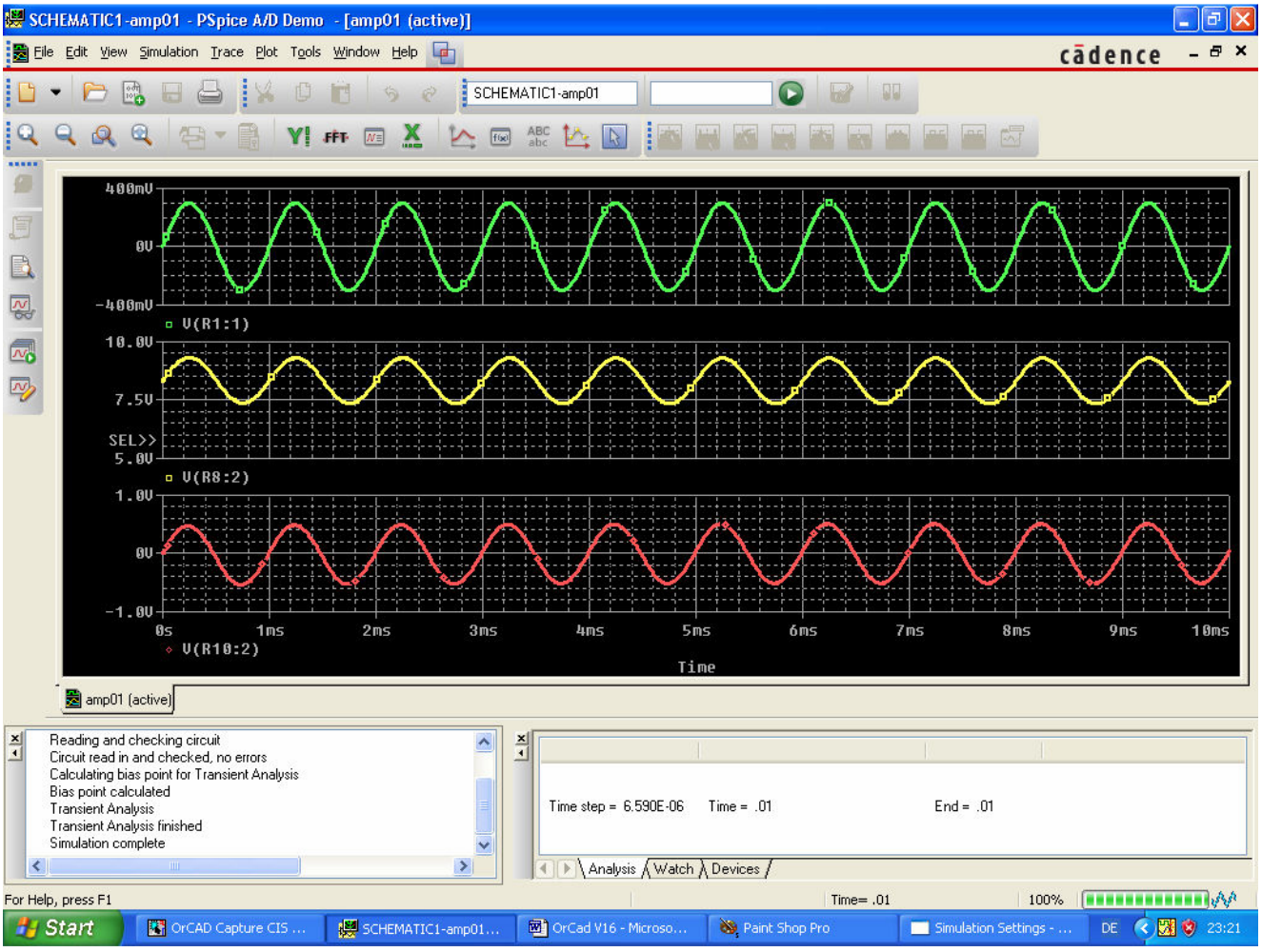




Now reduce the amplitude of the Input Voltage exactly to a point where no more „cutting“ can be observed at the Output Voltage (f. i. at an Input Peak Value of 0,3V).

To see the 3 different plots again after the next simulation, please open the „Simulation Settings“ and activate „**Show Last Plot”.**

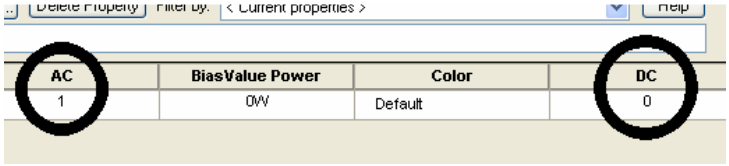
You’ll then get this result:

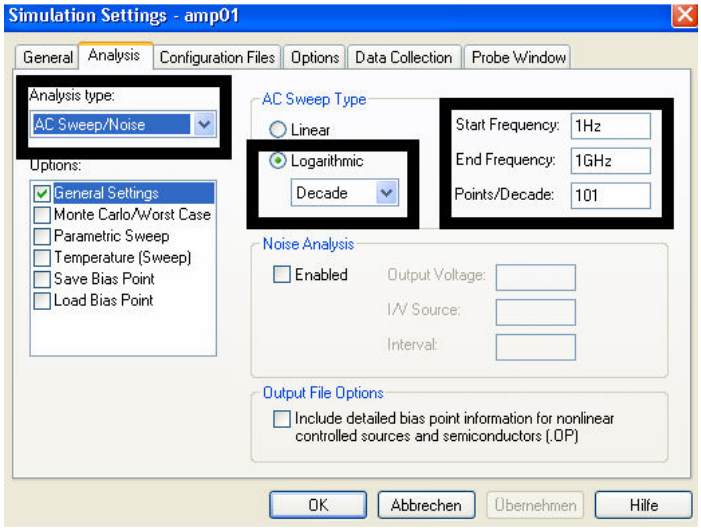


9.2. Frequency Domain - Simulation

Now we want to sweep the gain over the desired frequency range.
Double click on the **symbol of the sine input voltage** an enter the following values:

AC = 1 DC = 0 FREQ = 1





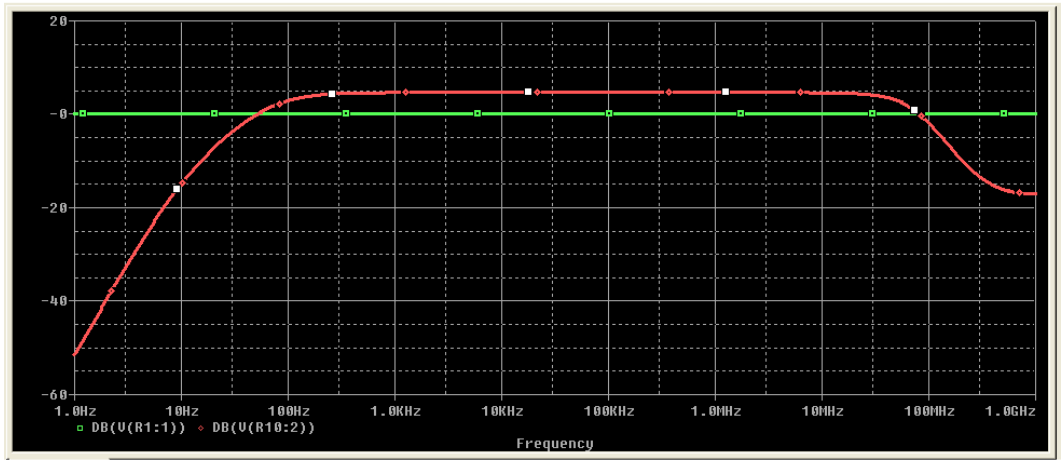
Open the Simulation Settings and change the Analysis taype to „**AC Sweep / Noise“**

Let „**General Settings“** at „Options“ unchanged

As „AC Sweep Type“ choose **Logarithmic** and **Decade.**

Enter a Start Frequency of **1 Hz, a Stop Frequency of 1 GHz** and **101 Points per Decade**.

Please delete the two Voltage Markers in the schematic and replace them by two „**dB Magnitude of Voltage“ – Markers (menu: PSPICE / Marker / Advanced**). After the simulatio You get this result:

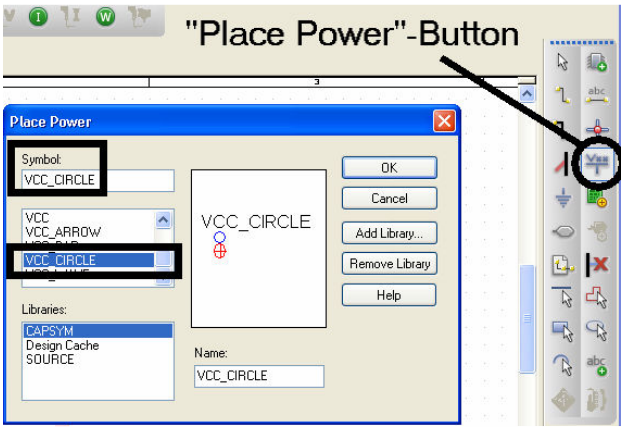


Task:
Activate the Cursor Function and determine the gain at 1kHz in dB. Also find the upper and lower Cutoff Frequency.

Change the values of the Coupling Capacitors C1 and C2 to check the variations of the upper and lower Cutoff Frequency.

10. Using POWER - Symbols

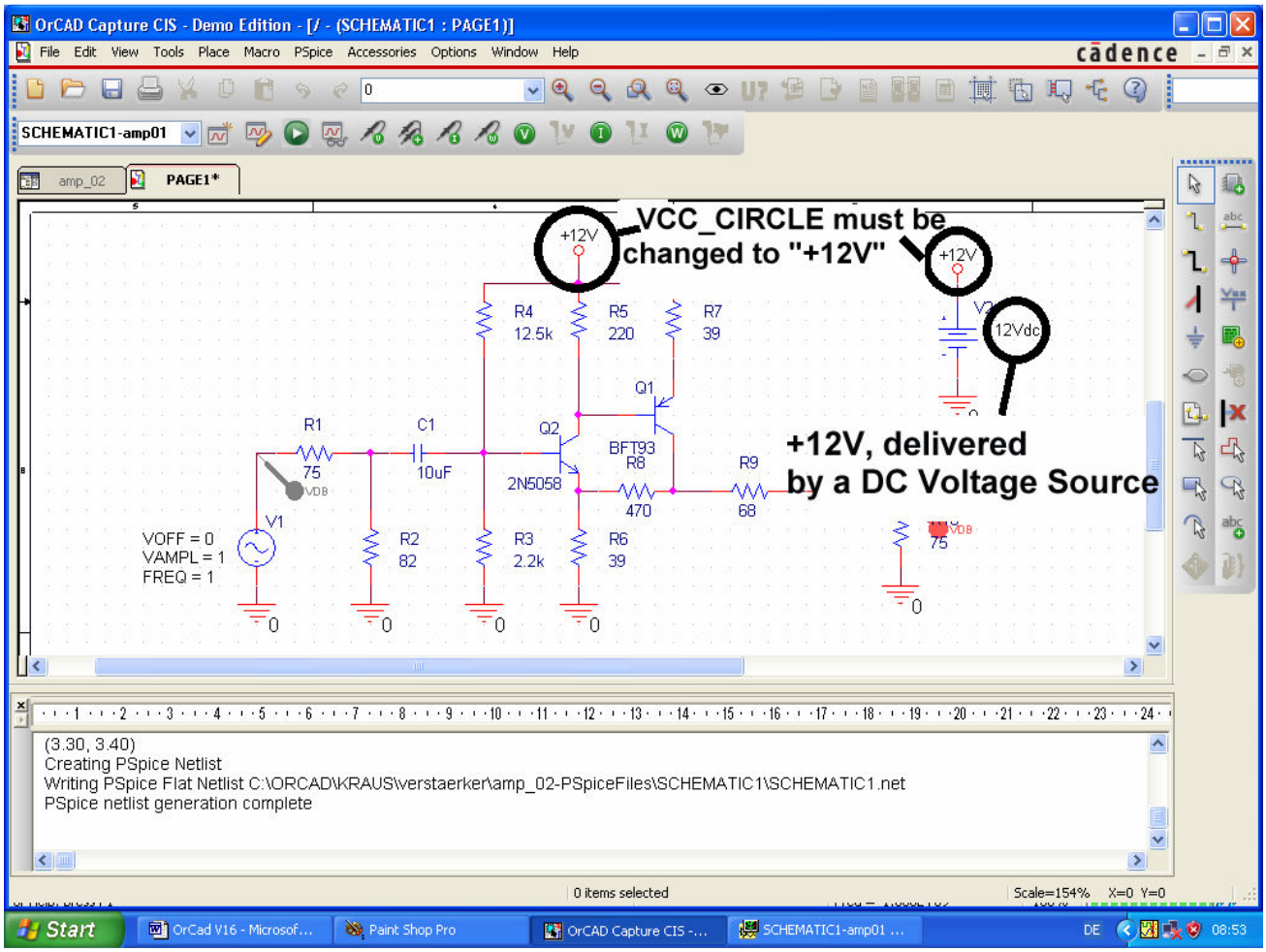
If a schematic ist getting more and more complicated, replace the wiring for the Supply Voltage by **Power Symbols**. So the Overview is improved.
This is shown at the last Amplifier Example.



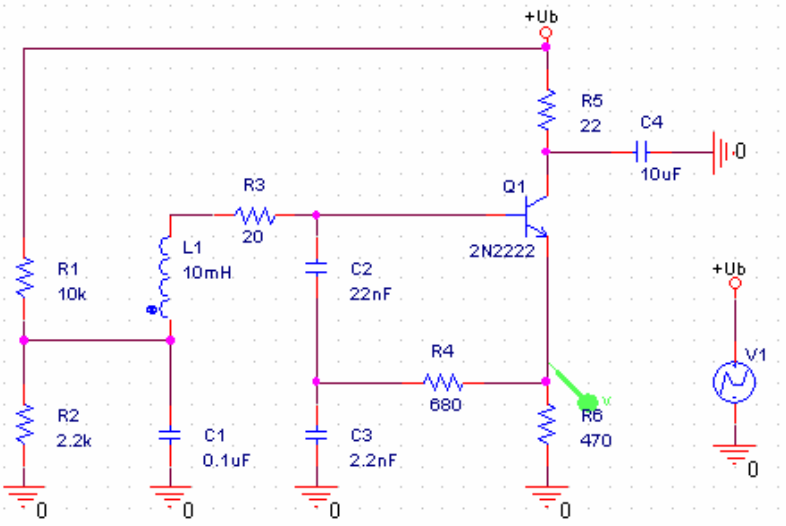
Open the **Power Library** by pressing the **„Place Power“-Button**.

Then search the “**VCC_CIRCLE**” – Symbol, click OK and place it in the schematic. You need two symbols for the amplifier.

Modify now the schematic in this manner:



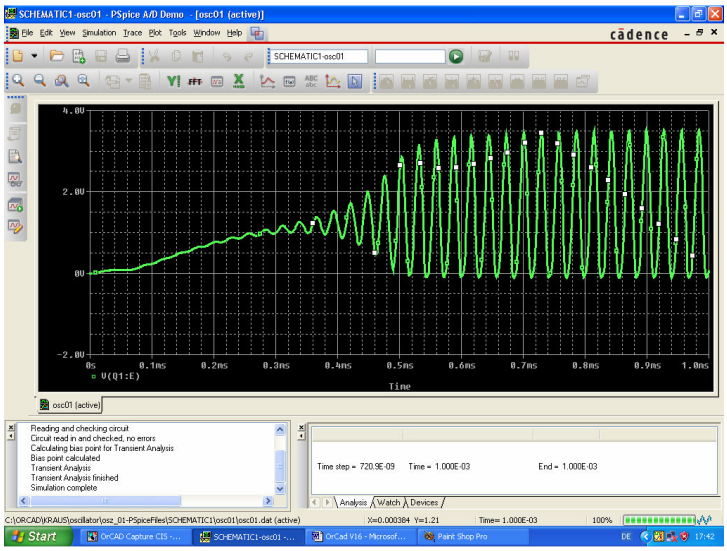
11. A Sine-Oscillator and its Spectrum (= FFT Application)



Let us examine a Sine Oscillator Circuit using a Collector-Grounded Colpitts Oscillator with an NPN-Transistor.

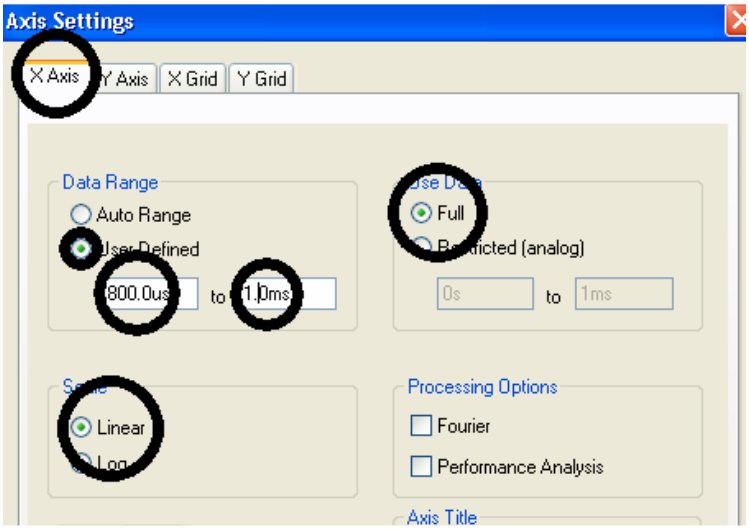
To start the oscillating process You must program a little "**Starting Push**". **So let the Supply Voltage rise from 0 to +12V in 1 Microsecond using the VPWL-Source from the SOURCE Library.**

Let the Run Time be **0…. 1ms, but do not forget to limit the „Maximum Time Step" to 1 Microsecond. (Otherwise the SPICE program would not recognize the start of the oscillation).**



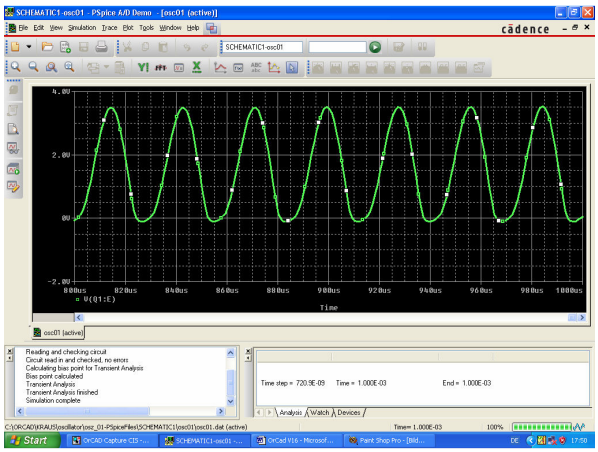
Now the start of the oscillation can analyzed without any problems.

To see more details of this AC Voltage (when it rised to the final maximum value) please click on „**Plot**" and then on „**Axis Settings**".



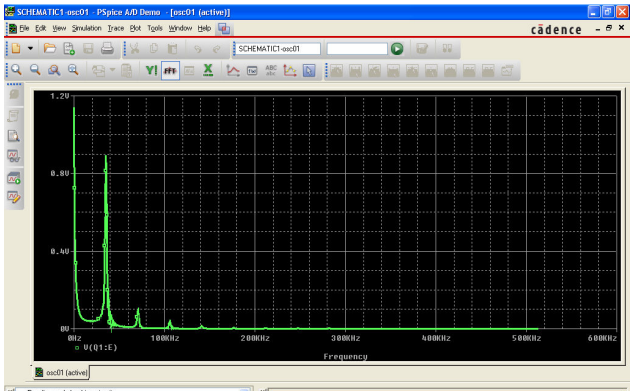
On the X Axis Card swith on „User Defined" and enter a Time Intervall from **0.8ms to 1.0ms**.

Set „**Scale**" to „**Linear**" and „**User Data**" to „**Full**".



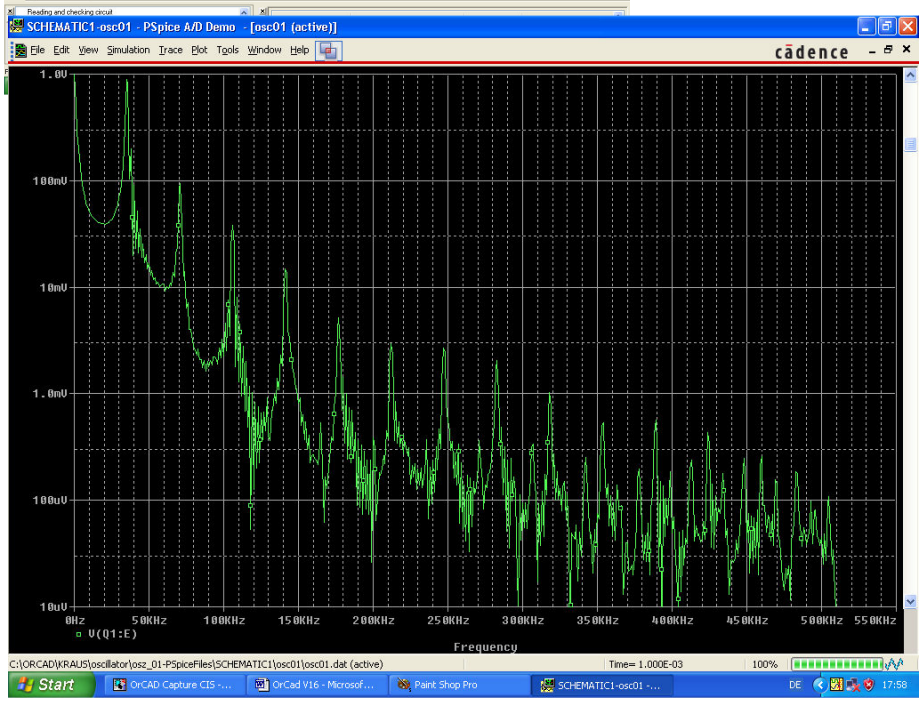
After clicking on OK we get this zoomed view and if You activate a cursor, it is easy to determine the Period Time by the indicated Values A1 and A2 resp. their Time Difference.

At last let us have a look at the Frequency Spectrum of the signal using the „**Fast Fourier Transformation**“. Please search this Button and press it (**in the „Trace“ Menu).**



The first look is not very convincing due to the linear scaling of the Vertical Axis.
So open the „**PLOT“ Menu** and then **„Axis Settings**“. Finally choose the „**Y-Axis**“ card.

With „**User defined**“, „**logarithmic scaling“** and a Voltage Range from **10µV to1V** we get that what we want to see.



In this nice diagram You can measure the „Attenuation“ of every Harmonic Frequency in relation to the Fundamental Oscillation Frequency (35kHz).

Attention:
At the right hand side of the “Help“-Button You find a a nice little picture. Clicking on it gives a “**Full Screen Presentation”.**
In the „Trace“ Menu You find the FFT Button to return again to the Time Domain.

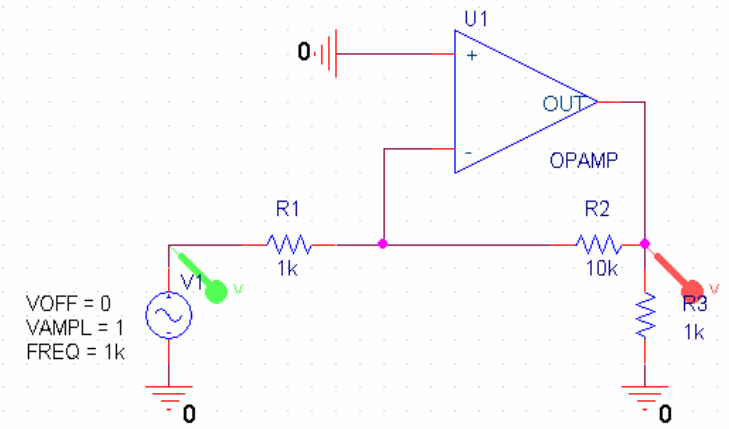
Interesting / Important:
Resistor R4 / 680Ω produces the „Positive Feed Back“. A part of the Emitter AC Current flows over this resistor into the Resonant Cicuit, compensates the losses and so enables the oscillation.

Increasing the resistors value gives less feedback and so the curve form of the Output Signal is improved (= more attenuation of the Harmonics). But there is a limit for the resistor value. When it is too high, the feedback cannot maintain the oscillation any longer and the circuit stops with oscillating.

Task:
Try to find out this R4 – value for stopping the oscillation. Also watch the Output Curve Form and the Output Spectrum when increasing R4.

12. OPA Circuits

12.1. Inverting Amplifier



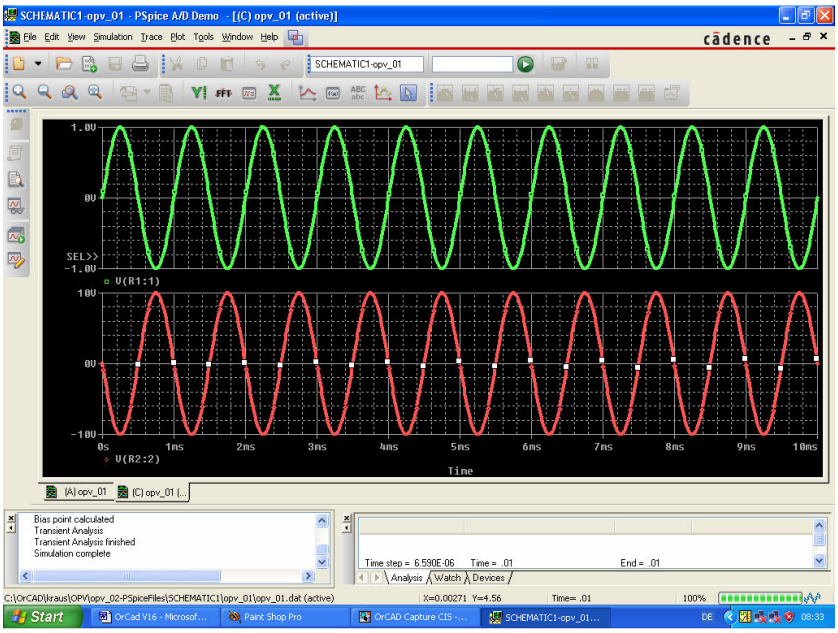
Task:
Draw this schematic with the Editor. Apply a Sine Wave (peak value = 1V, frequency = 1kHz) at the input.

Use two separate diagrams for the presentation of the input and the output voltage.

Calculate the Voltage Gain.

An ideal OPA You can find as OPAMP in the ANALOG.olb – Library.

Solution:



Formula:

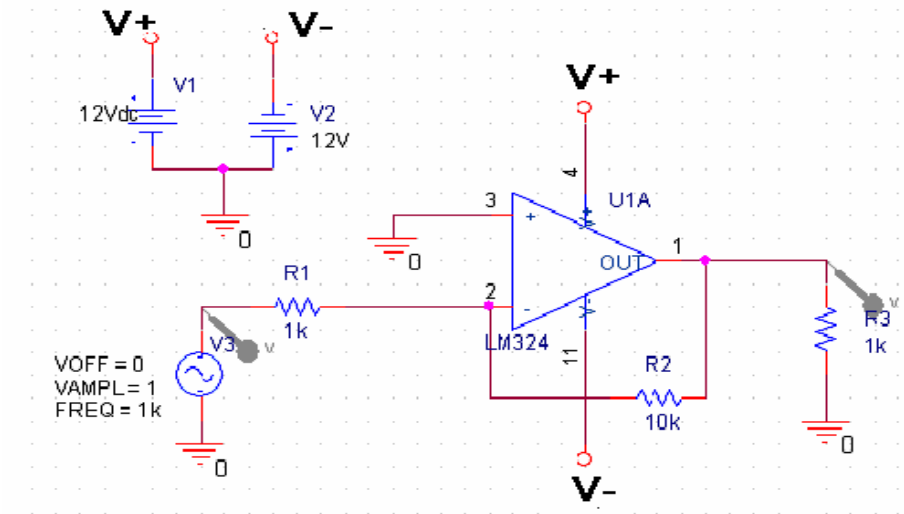
V_u = - (R2 / R1)

Additional Task:

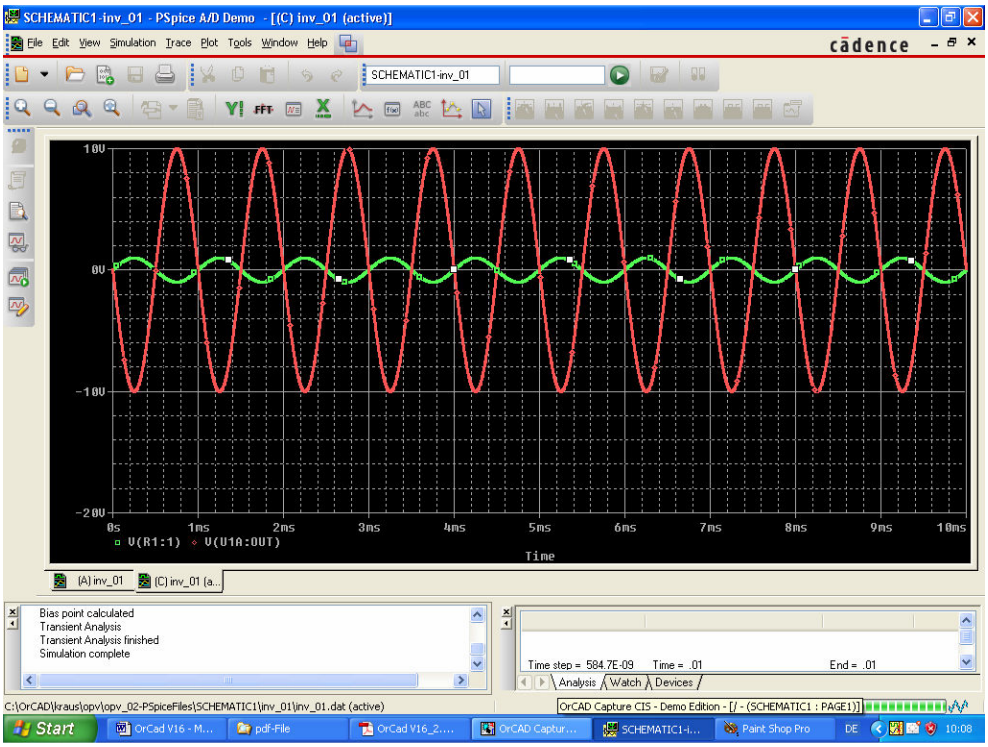
Replace the ideal OPA by a **LM324**. This part can be found in the **EVAL.olb – Library**.

Use two **Power Supplies (+12V and -12V) with „Power Symbols“ (= VCC_CIRCLE / CAPSYM, to be found in the menu behind the Ground Button)** and repeat the simulation.

Correct Solution:



Simulation Result:



Additional Task:

Simulate the „Frequency Response“ and use the Cursor to determine the lower and the upper Cutoff Frequency.

Preparations:
At first open the Simulation Settings and switch to „**AC-Sweep**“ with the following parameters:

Start Frequency = 1 Hz
Stop Frequency = 10MHz
Decade Sweep with 101 Points per Decade

After a double click on the symbol of the voltage source enter

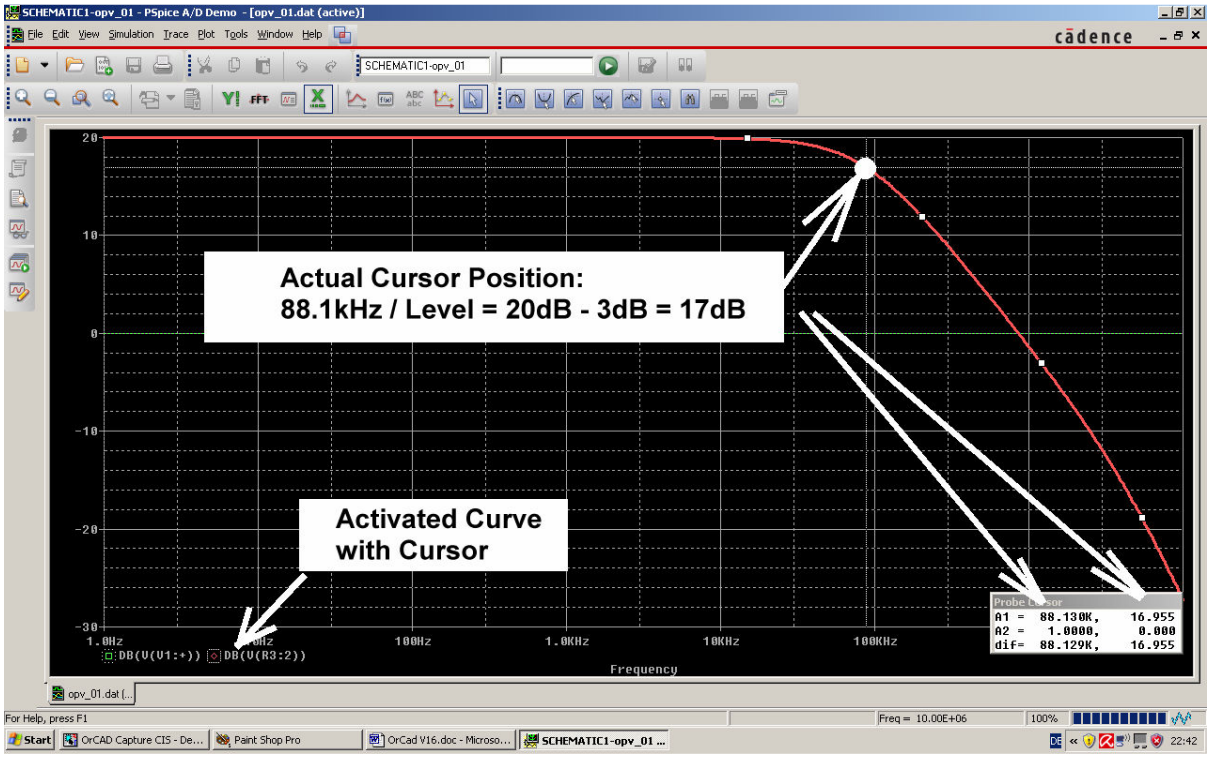
AC = 1 DC = 0 FREQ = 1

At last delete the two Voltage Markers and replace them by two markers of the

dB Magnitude of Voltage

You'll find them in the menu **PSPICE \ Markers \ Advanced**

When the simulation is done, please activate the cursor (menu: **Trace \ Cursor \ Display**) and roll to the point where the gain has decreased from 20dB to 17dB (diffference = -3dB).
This is the Cutoff Frequency.

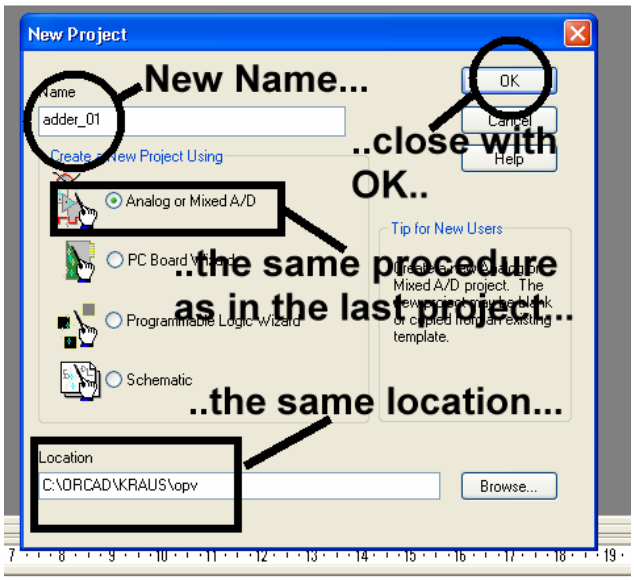


12.2 Analog Adder

You can also use this circuit to add Analog Signals. Here an example for two Input Signals.

Signal 1: **Sine Wave with f = 1kHz and a Peak Value of 0,5V**

Signal 2: **Sine Wave with f = 5kHz and a Peak Value of 0,2V**

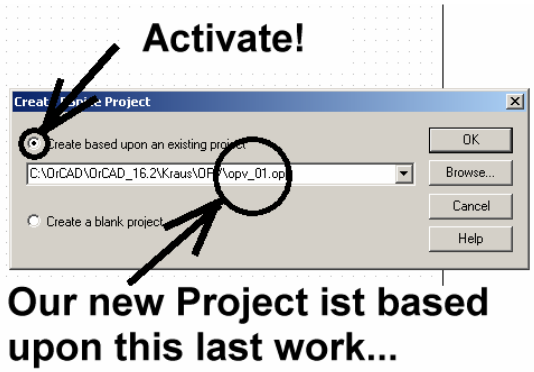


Let use see how to use the last project for this purpose:

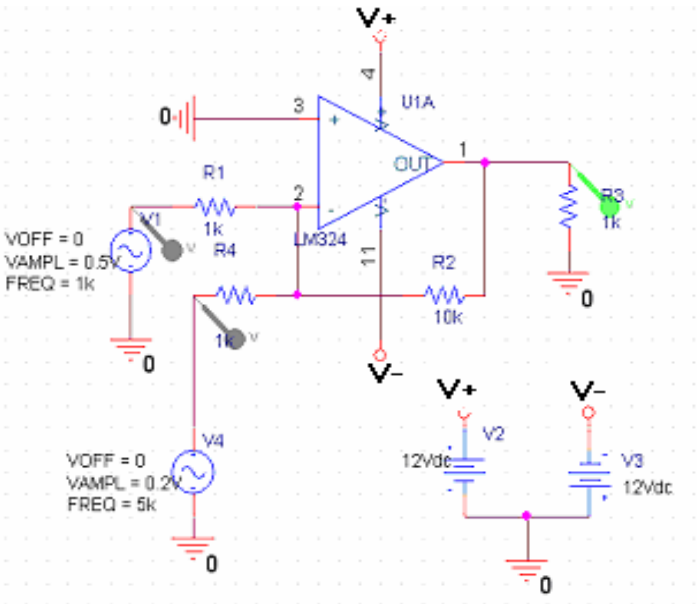
Step 1:
Close the last project and open a new one. It is named

adder_01

and is saved in the same folder. See the left screenshot.

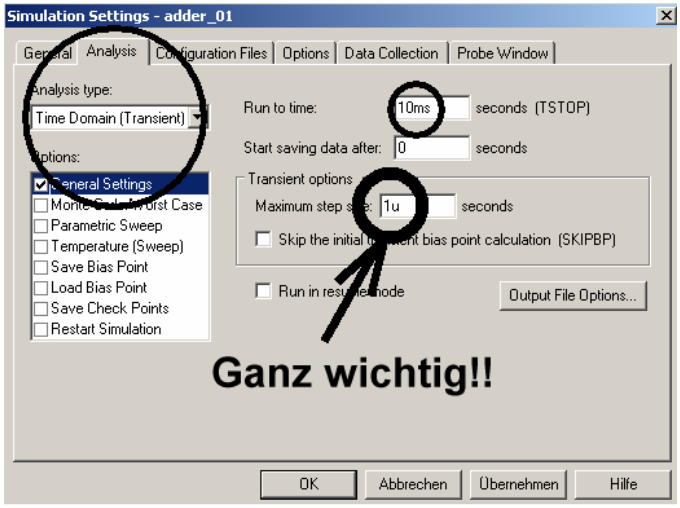


Step 2:
Tell OrCAD that the new work is based upon this old project.



Step 3:
Click Ok and do the changings in the schematic.

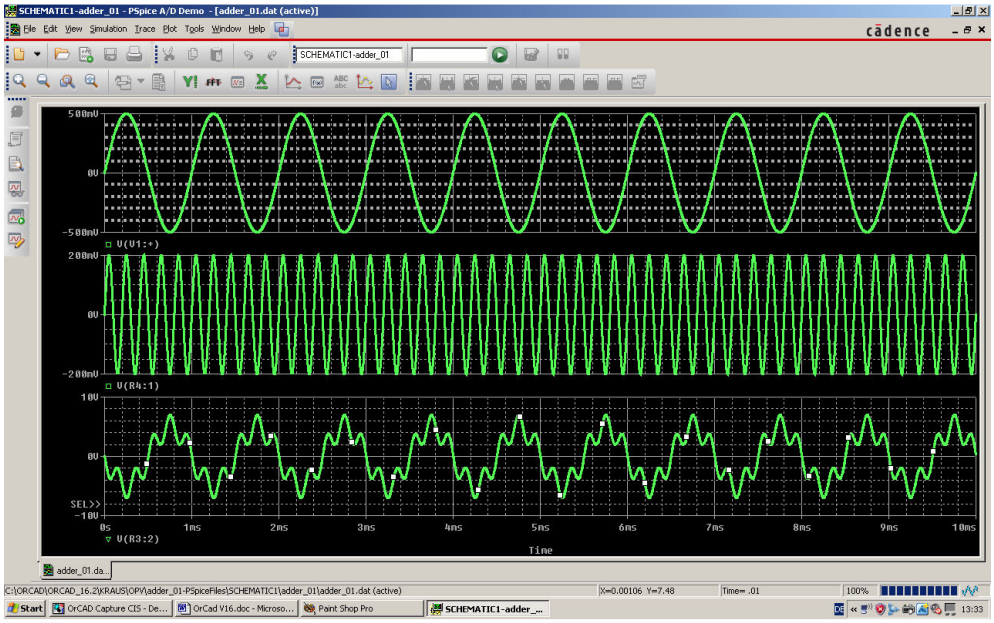
This is the goal: now we have two different input voltages which are added by the OPA.



In the Simulation Settings we prepare a **Time Domain Analysis for 0…10ms with a Maximum Step Width of 1 Microsecond.**

If You forget to set the Maximum Step Width to this value, You get problems and a breakdown.

(Reason: some signals at some nodes do not converge...)



Successfully done!

12.3. Active Filter Circuit

Active Filter with OPA are now not only used in the Audio Range but also for RF Filtering up to 100 MHz.
And these are the advantages:

-- Using only OPA's with R + C - Feedback
-- Much cheaper because no coils and / or Alignments are necessary
-- Very small dimensions when using SMD Parts
-- Gain is adjustable in a wide range
-- Missing nonlinear distortions when using strong Feedback and low overall gain

The Fundamentals and Formulae can easily be found when using the Internet and searching „Active Filters OPA". There are also lot of free programs to do all necessary calculations during the developement work (See f.e. the homepages of Texas Instruments or Burr Brown or National Instruments).

Example : Sallen – Key – 4th Order Lowpass Filter

From the mentioned programs You get all what You need if You enter the following parameters:

Kind of Filter: **Lowpass**

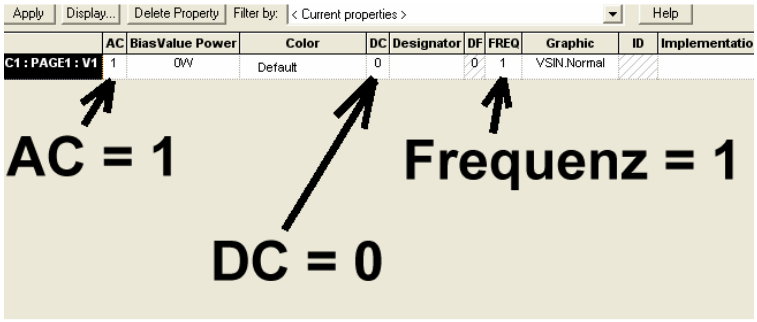
Type of Filter: **Tchebychev**

Cutoff Frequency: **3400 Hz**

Passband Ripple": **0,5 dB**

Order of Filter: **4**

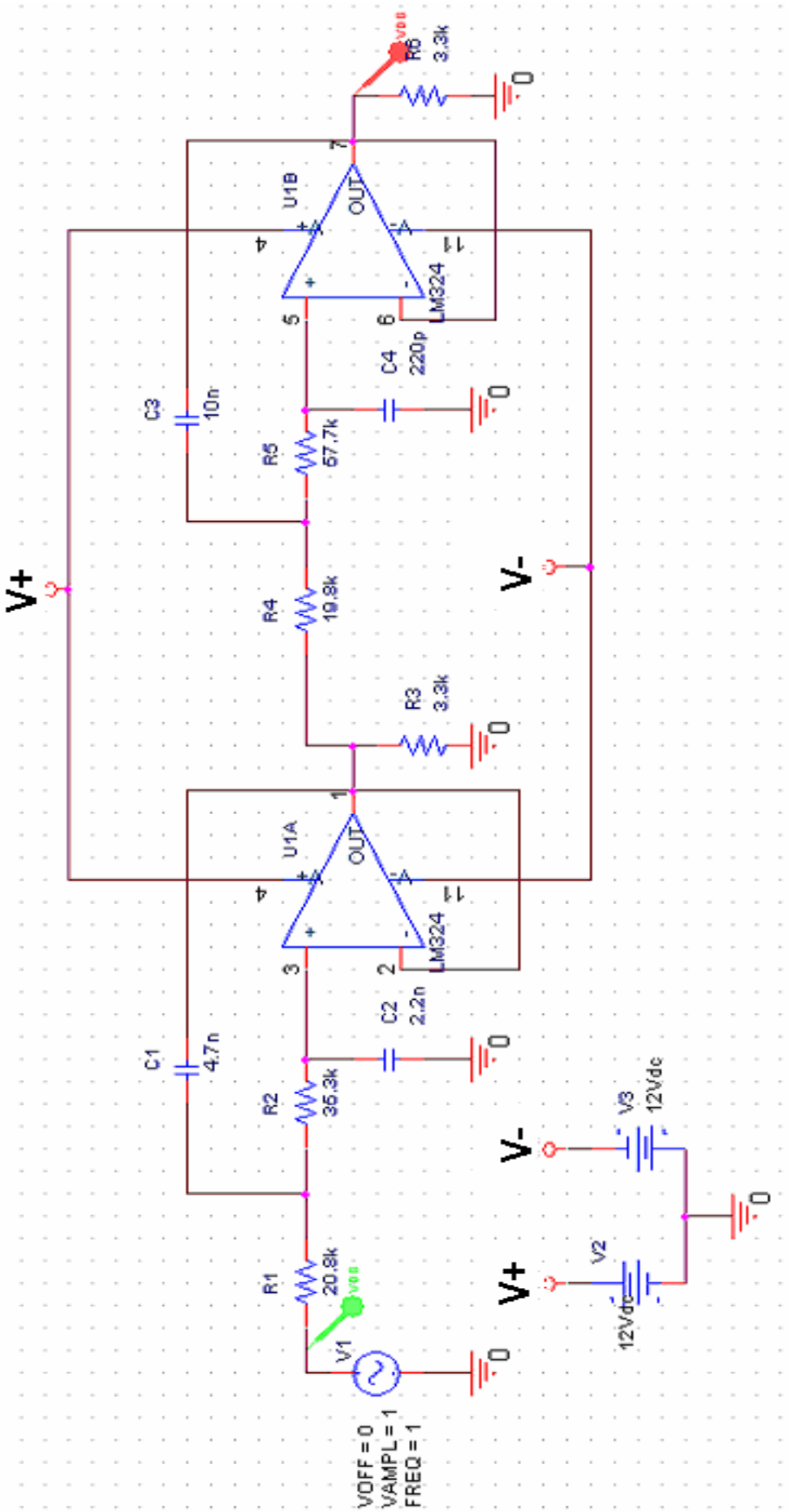
See the complete schematic on the following page
Please use two different DC Power Supplies with Power Symbols for V+ = +12V and V- = -12 V (....do You remember: „**VCC_CIRCLE / CAPSYM**".behind the button with the Ground Symbol…).
Do not forget to add the **„dB-Markers**" at the Input and the Output.

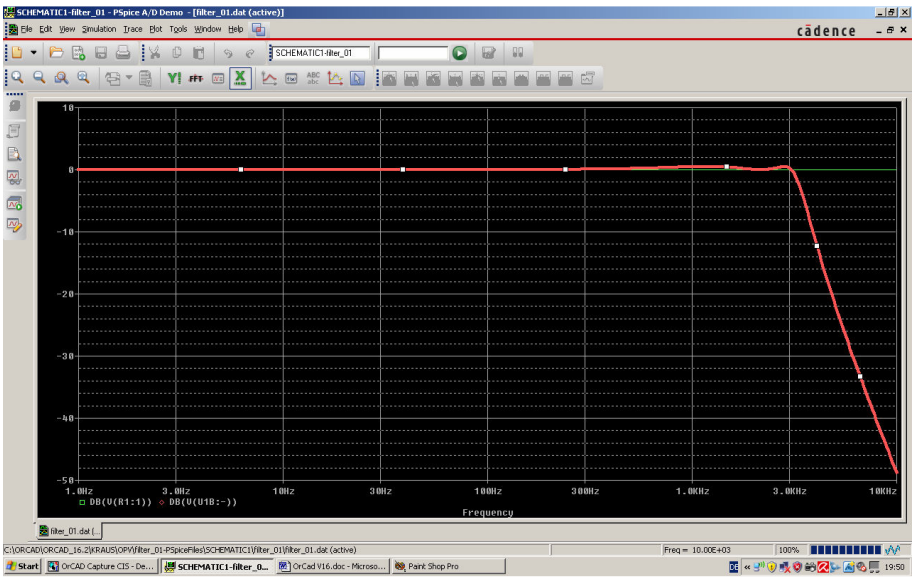


Now apply a Sine Wave **(VSIN/SOURCE") with these properties** at the input.

At last program an **AC Sweep** with these properties
AC – Sweep
Logarithmic
Decade

Start Frequency: **1 Hz**
Stop Frequency: **10 kHz**
101 Points per Decade

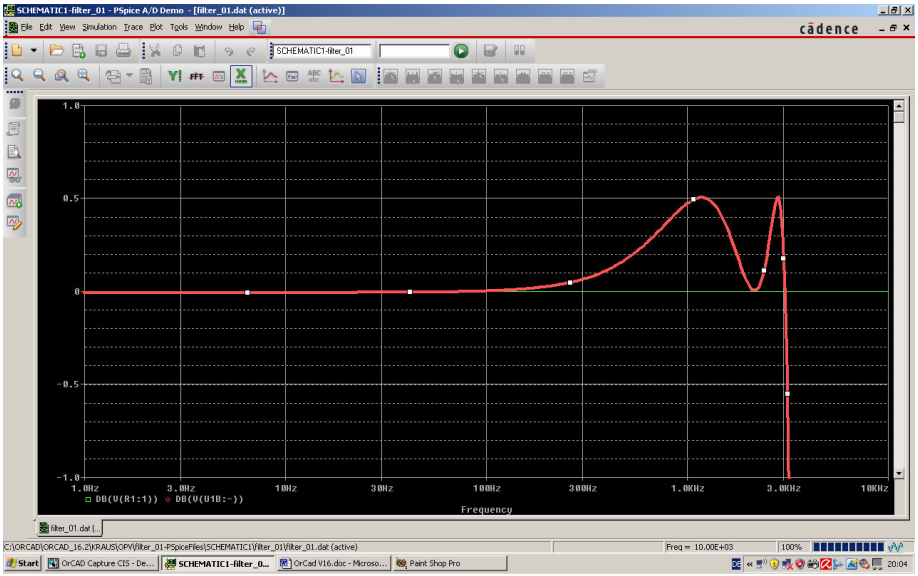

R6
3.3k
0
U1B
OUT
LM324
C4
220p
C3
10n
R5
67.7k
R4
19.8k
V+
V-
R3
3.3k
U1A
OUT
LM324
C2
2.2n
C1
4.7n
R2
36.3k
V3
12Vdc
V2
12Vdc
R1
20.8k
V1
VOFF = 0
VAMPL = 1
FREQ = 1



Passband and Stopband can be identified without any problems.

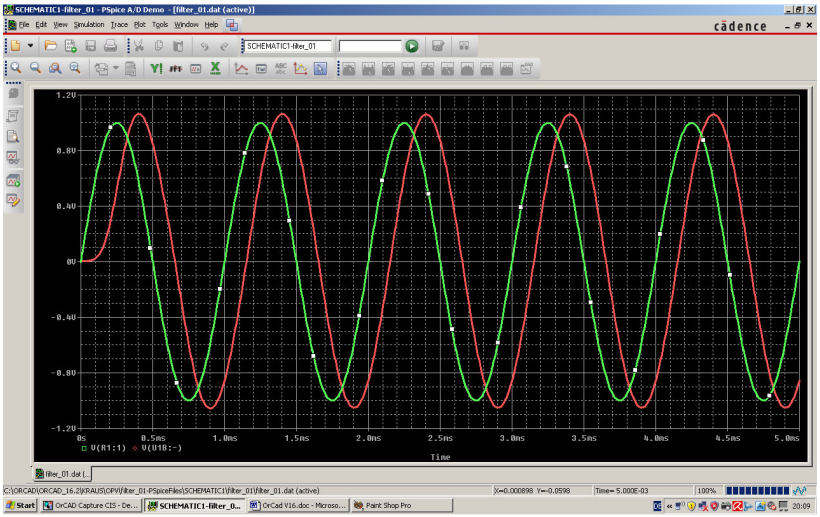
But how can We check wether the maximum passband ripple is really only 0.5dB?

So click on a value on the Vertical Axis and choose **„User defined“ in the menu**.



With a new range from **-1dB to +1dB** everything is allright.

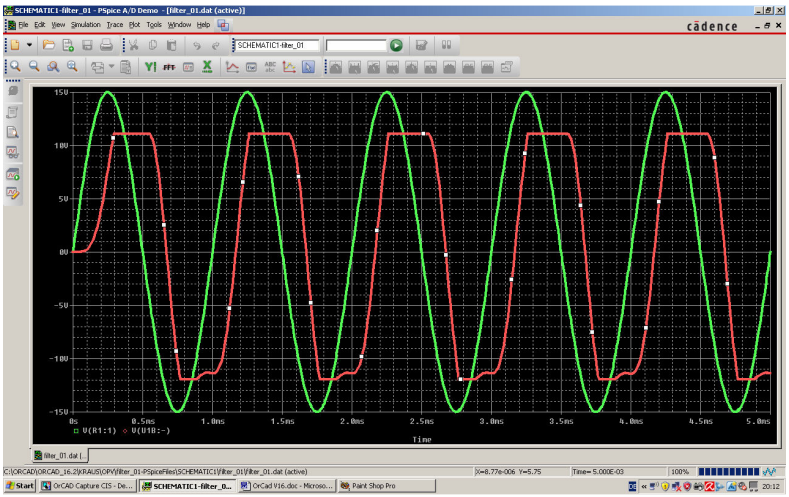
Additional Task:



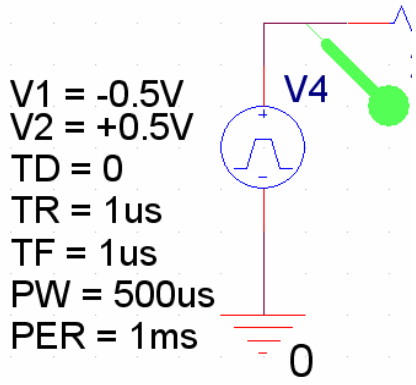
Now organize a „Time Domain / Transient“ – Simulation and apply a Sine Wave (f = 1kHz, Peak Value = 1V) to the input.

Replace the **VdB-Markers by two Voltage Markers.**

Simulate the Input and the Output Voltage at this frequency.



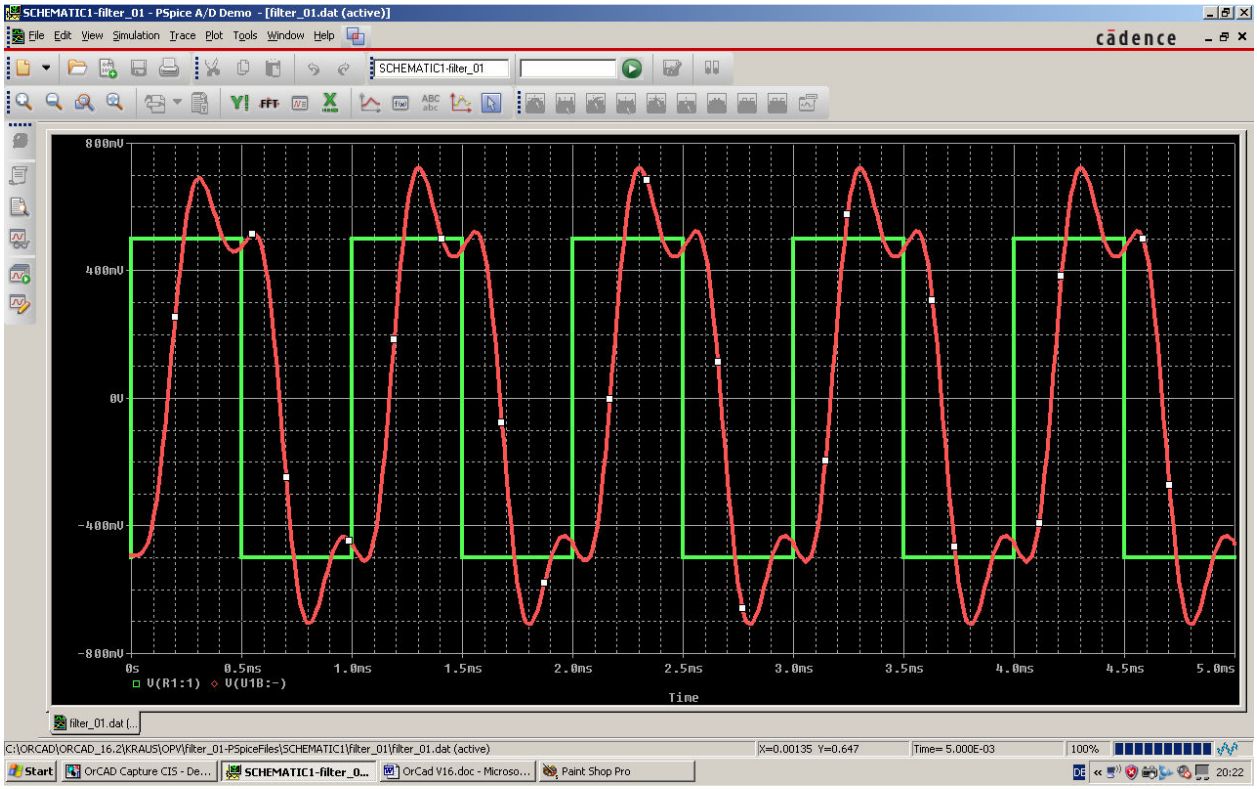
Increase the Input Voltage Peak Value to 15V to see the „overdriving effect“.



Now replace the Sine Wave at the input by a Pulse Voltage **(VPULSE / SOURCE)** and test the circuit how this signal will be

Pulse Voltage Data:
f = 1kHz
positive peak value = 0,5V
negative peak value = - 0,5V
Rise Time = Fall Time = 1µs.
Pulse Widht = 500µs
Periode Time = 1 Millisecond

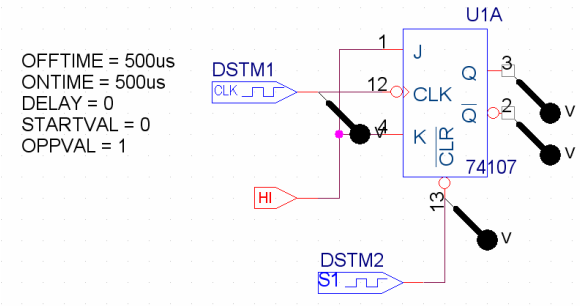
Solution:



13. Simulations of Digital Circuits

13.1. Checking a J-K-Flipflop

This is „exploring unknown territories“ and so comes here the complete schematic for the simulation:



Step 1:
Press the „Place Part“-Button in the menu at the right hand side of the screen.
Then search for part **74107** using the path
C: \ ORCAD \ ORCAD_16.2 \ TOOLS \ CAPTURE \ LIBRARY \ PSPICE \ DEMO

Do You still remember this procedure ….?

A little help:
The 74107-part can be found in the **7400.olb – Library**....

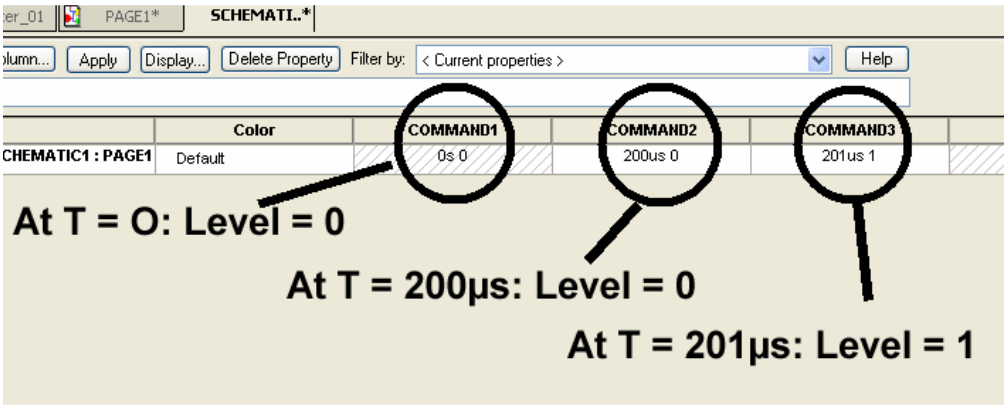
Please olace it on the screen when You have been succesful.

Schritt 2:
Look for the **DigClock – Signal in the „SOURCE“-Library** .
Connect it to Pin 12 of the 74107. Beside the symbol there is a little list of values which must be entered step by step:

OFFTIME = 500us
ONTIME = 500us
DELAY = 0
STARTVAL = 0
OPPVAL = 1

Step 3:
Connect Pin 1 and 4 (= J and K) together using a wire and apply **HIGH-Level**. You find this symbol by pressing the Ground Button and looking for **$D_HI**

Step 4:



And now a correct Reset Signal at pin 13 is missing for a good start.

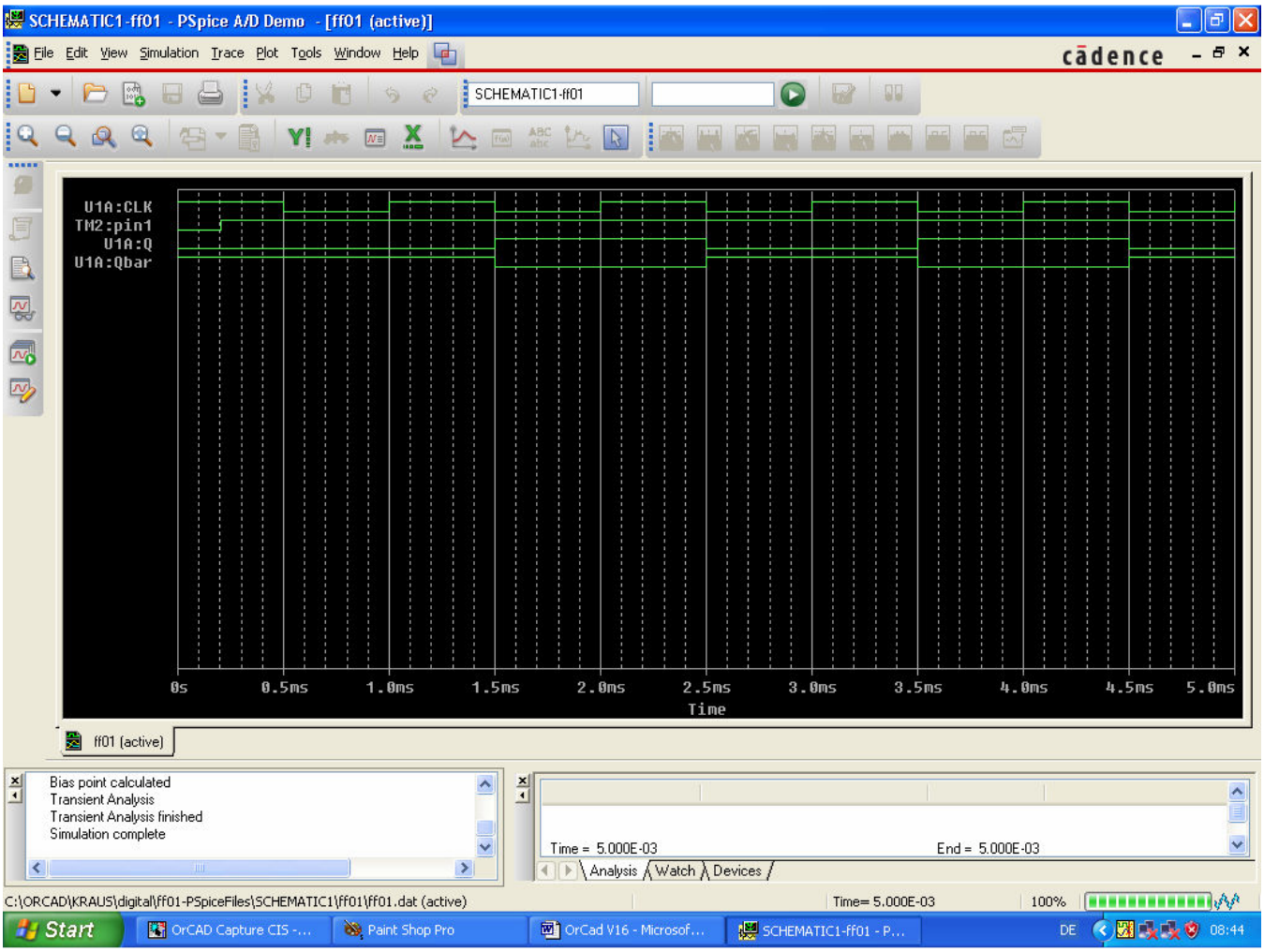
Find it in the **SOURCE Library as STIM1**.

Click double on its symbol and enter the necessary values.

Step 5:
Add Voltage Markers to the Clock Line, the Reset Line and both Outputs.

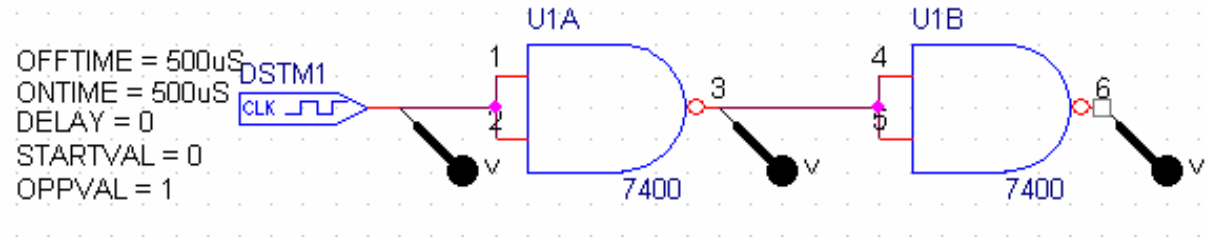
Step 6:
Now simulate in the **TIME DOMAIN for 0.....5 ms.**

If everything was well done, then You get this screen:



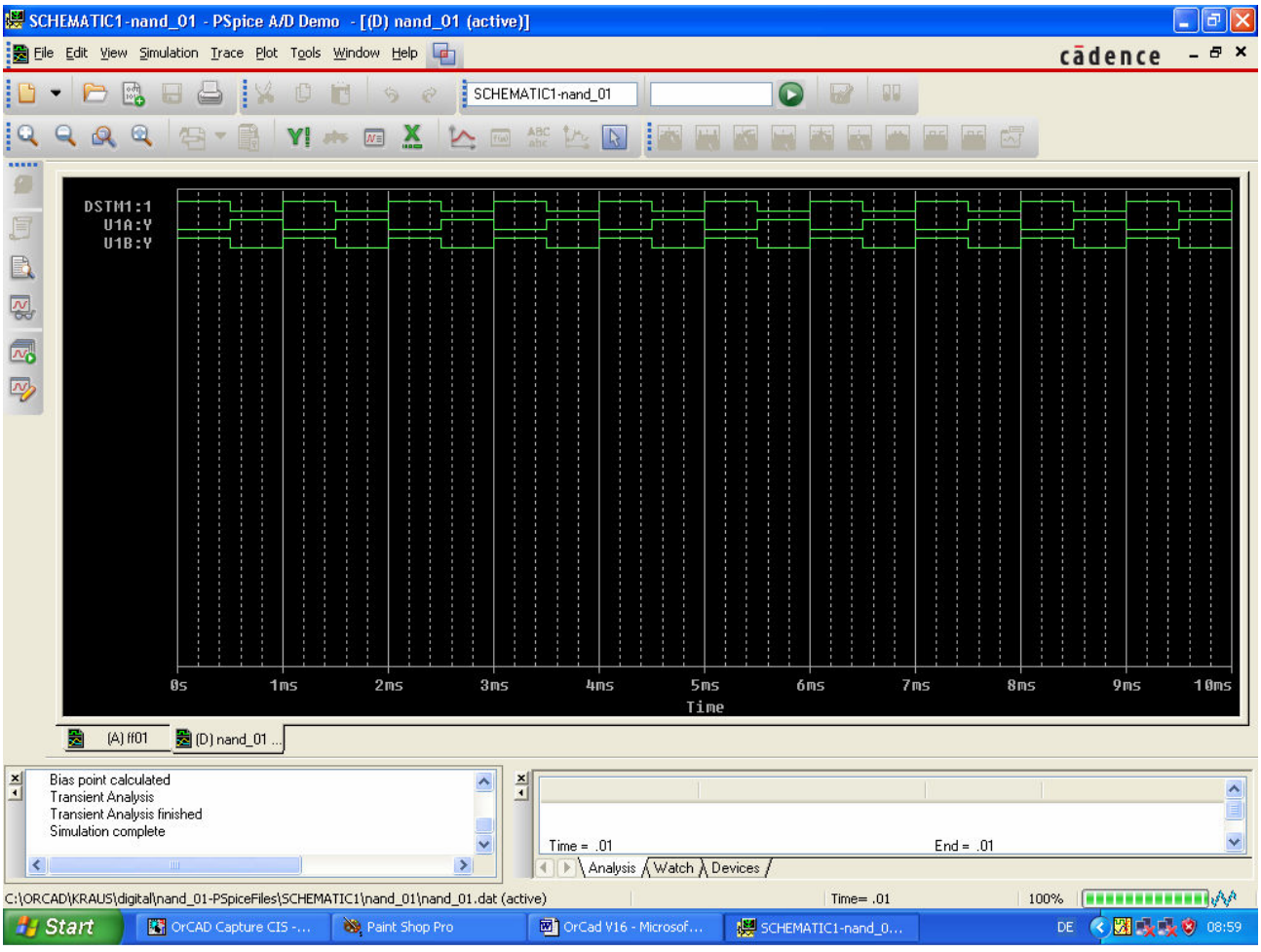
13.2. NAND-Gates

To show how to handle NAND-Gates we connect two of them in series and produce a double inversion of the Input Signal.

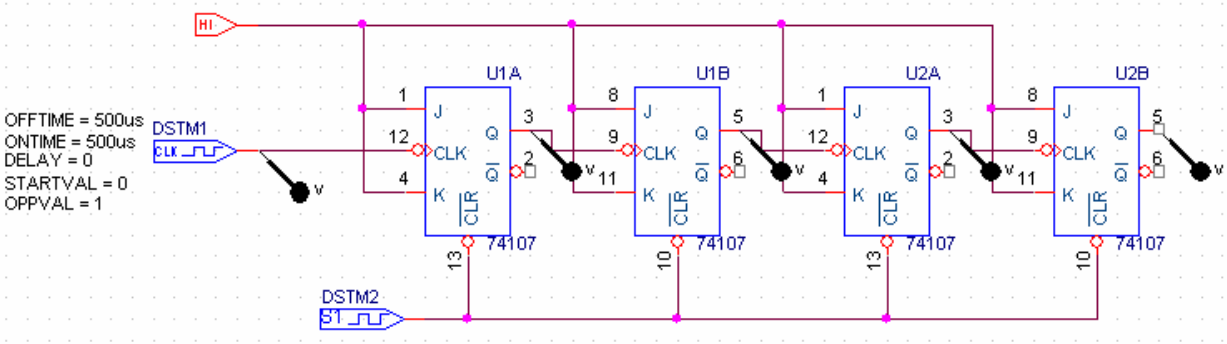


We use a **1kHz Digital Clock (= DigClock, found in the SOURCE Library**). The 7400 comes from the **„7400.olb" – Library.**

So will be the Simulation Result for a **Run Time of 0….10ms**:



13.3. Four Stage Binary Counter



We use the same 1 kHz - Clock Signal “DigStim” as before and apply again the RESET-Signal (= STIM1 in the Source Library) with the 3 Value Couples

0 / 0 200us / 0 201us / 1

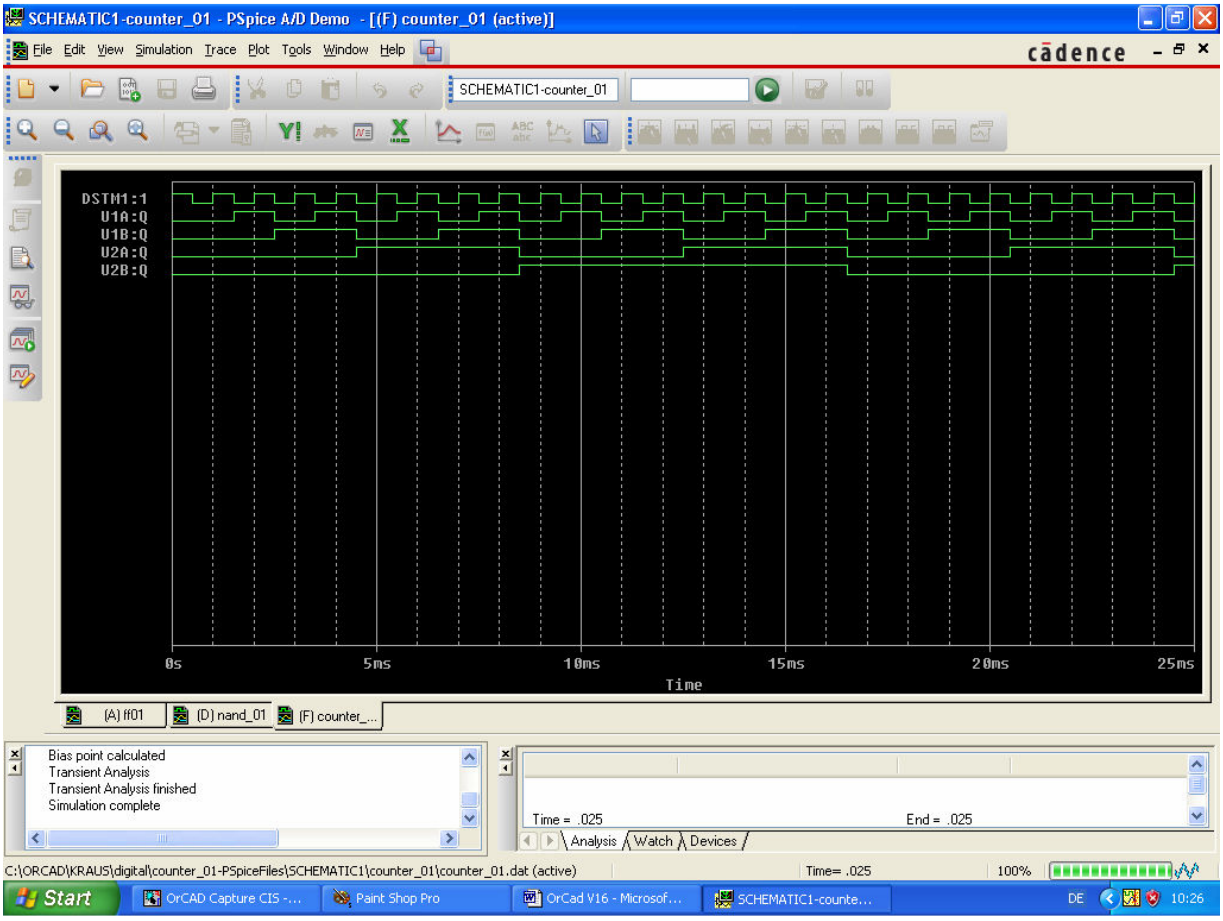
The J- and K-Inputs of all Flipflops are connected together and use

„**HIGH“-Level** = Signal **„$D_HI / SOURCE“**

(You’ll find it after pressing the Ground Button in the appearing menu).

Choose a **Run Time from 0...25ms**.

Result:



14. 100MHz Tchebychev-LPF

At high frequencies it is difficult to measure currents and voltages with high precision -- power measurements are easier!
So from 1MHz upwards another principle is choosen to describe systems and their properties, up to 100GHz and more:

On every place in the system he same „typical resistance“ (= **System Resistance**) is used, normally 50Ω (…for video purposes 75Ω). This is valid for all blocks, their input and output resistances, all terminations and all cables.

This is a well known and simple principle: Power Matching ($R_{in} = R_{out}$) in the complete system!

With Directional Couplers You can now measure the reflected powers and so determine the deviations from the ideal case at the Input and the Output Port. These deviations are listed up as

Reflection Coefficients (= S – Parameters S11 and S22)

Then You measure the Output Power when feeding the Input with a power, called “Incident Wave”. The ratio of these two powers gives the

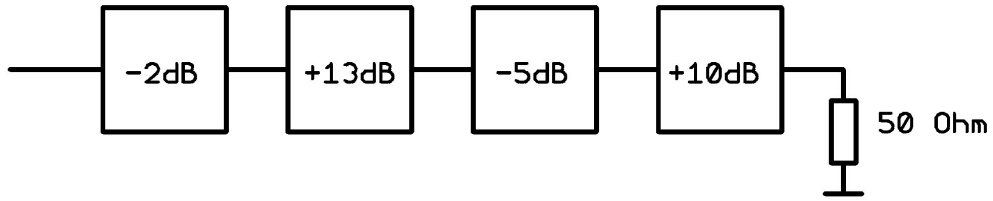
„Forward Transmission“ (= S-Parameter S21)

Feedback existing in such a block is regarded as a power measured at the input, but caused by an output signal. This is the

„Reverse Transmission“ (= S-Parameter S12).

For a complete System connect all blocks in series and there are tons of software to calculate the properties of the total chain using their S-Parameters. As an example: You only have to add the gain values of all stages to compute the total gain to

-2dB + 13dB -5dB + 10dB = 16dB



Now let us simulate a block which is very often used: a **Filter Circuit**.

Definition: by a Filter a small or wide range of frequencies will be suppressed (= Stop Band), all others can pass without problems (= Pass Band).

We use 4 different Filter Types:
Low Pass Filter (LPF) **High Pass Filter (HPF)** **Band Pass Filter (BPF)** **Band Stop Filter (BSF)**

Due to the Transfer Function of the Filter we distinguish

a) **Butterworth- and Besselfilter** with very low Linear Distortions (= low Group Delay Distortions) in the Pass Band, but a very very slow rise of the Stop Band Attenuation with frequency.

b) **Tchebychev-Filtern** have a very rapide change from the Pass Band to the Stop Band with a quick rise of the Stop Band Attenuation. But You pay for that with a „**Ripple**“ of the Pass Band Attenuation.

c) **Elliptic Filters (= Cauer Filters**) have the same Pass Band Ripple as Tchebychev Filters. But the transition from Pass Band to Stop Band is additionally improved. You pay for this property with a Stop Band Attenuation which doesn’t rise to Infinite (like Tchebychev Filters) but have little “breakdowns” in the Stop Band Attenuation.

Also You have to decide wether You want a **Pi - Type Filter (= less coils) or a T – Type Filter (= less capacitors).**

Remember:
All system calculations are here done by using „dBs“....so please repeat this chapter...

Task:
Let us design a Tchebychev LPF with the following propertiesFilter and afterwards check with PSPICE:

Lowpass Filter (LPF)	
Cutoff – Frequency):	**100MHz**
Passband Ripple :	**0,3dB**
Filter Order n (number of parts):	**5**
System Resistance	**50Ω**

To calculate the part values, enter f. e. in Google the following:

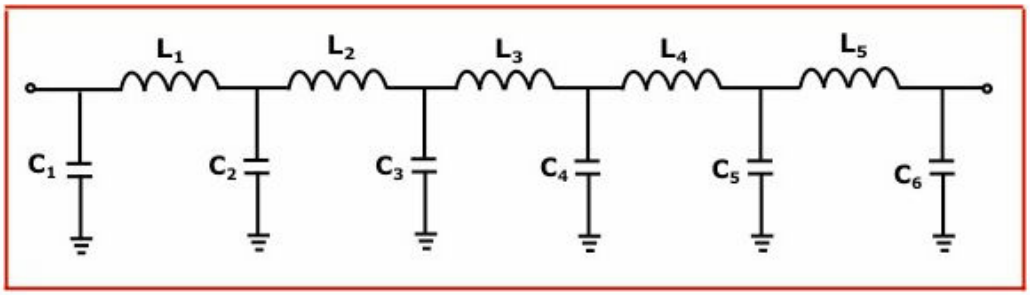
chebyshev pi lc low pass filter calculator

This could be the result:

Chebyshev Pi LC Low Pass Filter Calculator

Ads by Google Low Pass Filter Circuit Air Conditioner Filter Interpolation Filter Color Effect Filter

Enter value, select unit and click on calculate. Result will be displayed.



Enter your values:

Cutoff Frequency:	100	MHz
Impedance Z_0:	50	ohm
Frequency Response Ripple:	0.3	db
Number of Components:	5	(1-11)

Calculate Clear

In the lower half of Your PC screen You get the results:

Results:

Inductance:

Unit : nH

L_1:	103.3877
L_2:	103.3877
L_3:	0.000000
L_4:	0.000000
L_5:	0.000000

Capacitance:

Unit : pF

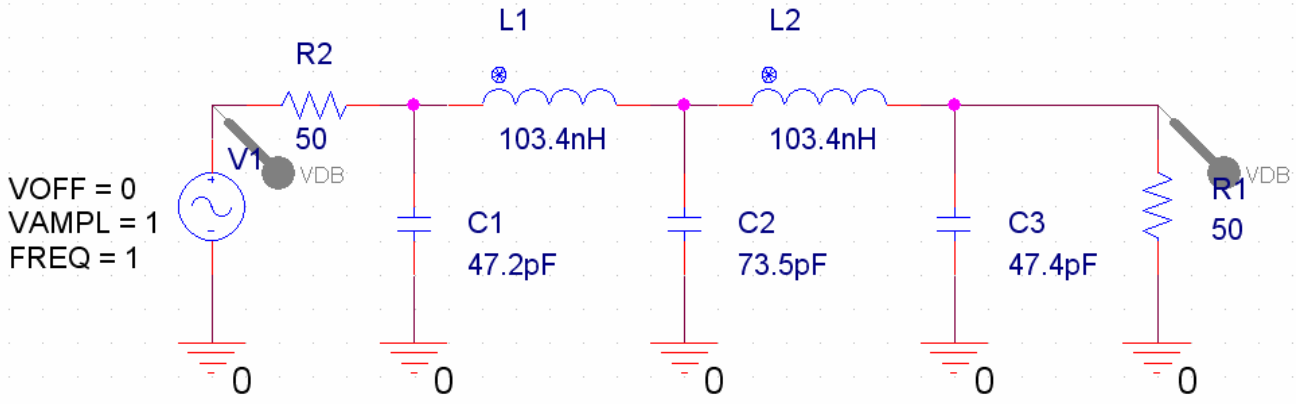
C_1:	47.16323
C_2:	73.51410
C_3:	47.16323
C_4:	0.000000
C_5:	0.000000
C_6:	0.000000

So start a new OrCad-Project, but pay attention to these details:

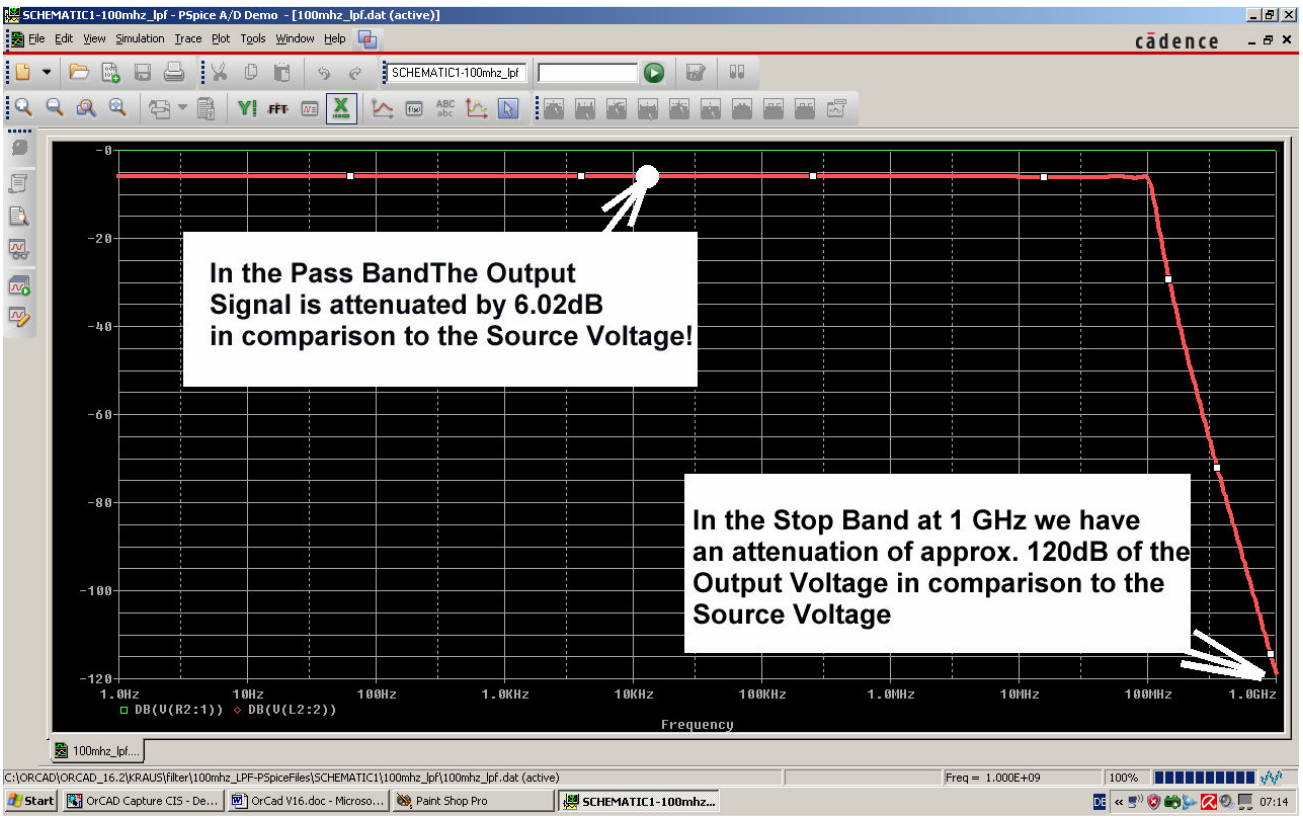
a) The Input Sine Voltage Source must be prepared for an **„AC-Sweep“** with an Internal Resistor of **50Ω**

b) Use two „**dB Magnitude of Voltage“-Markers**“.

c) Finally prepare a **logarithmic AC sweep from 1 Hz to 1GHz with 101 Points per Decade**.



Simulation Result:



But where do the 6.02dB come from?

Remember the Power Matching Principle!
This gives only half the Source Voltage at the Load Resistance….at this means 6.02dB attenuation…

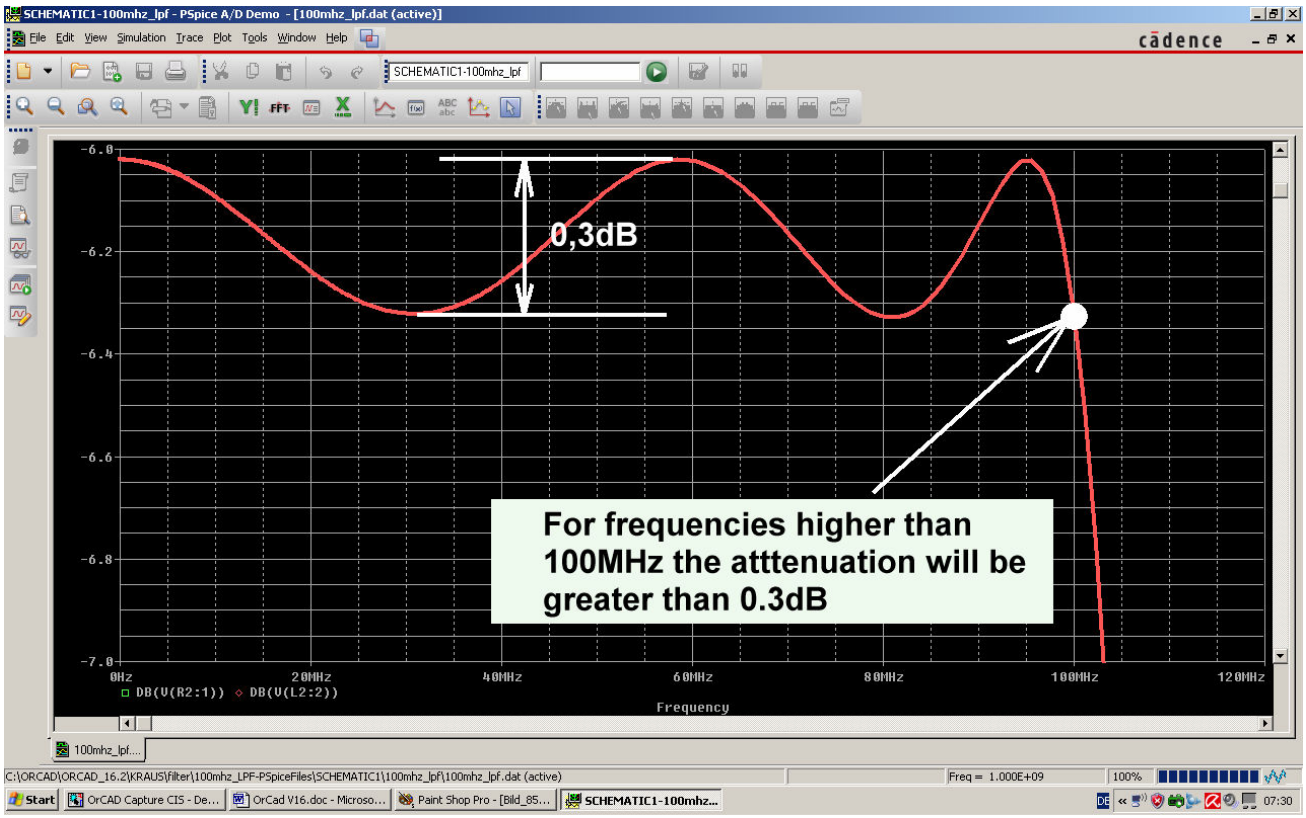
If You want to see the „**Tchebychev-Waves**“ of the Passband Attenuation, use the following Axis Settings:

a) X-Axis **linear**, Frequency Range from **0 to 120MHz**

b) Y-xis: only the small **range from -6db to -7dB shall be seen.**

c) Work with **201 Points per Decade**.

So it must look like and the Cutoff Frequency is to be seen very well.



Appendix: Impulse Response g(t) and Transfer Function of a System

As already mentioned the modern Communication Technique, the Remote Control Technique and the Theory of Systems work very much with Fourier -, Fast – Fourier - and Laplace - Transformation.

So it is possible to preview the System's answer to ANY Input Waveform if you know the Transfer Function.

But how to determine this Transfer Function?

Answer: either in the Time Domain or in the Frequency Domain.

Definition:
The Transfer Function is the Ratio of the Output Signal to the applied Input Signal (regardless to the used curve form). This Ratio is expressed as a complex Function (in general: „Magnitude and Phase Representation")
==========
Method 1 / Frequency Domain:
If You know the Transfer Function, then you get the Frequency Spectrum of the Output Signal by multiplying the Transfer Function and the Frequency Spectrum of the Input Signal.
Using the inverse Fourier- resp. Laplace Transformation gives the Output Signal's curve in the Time Domain.
==========

Direct Measuring or Calculating of the Transfer Function for complicated Systems is often really difficult and so it is better to apply another System Theory Law:

Feed the System with a Dirac Impulse at the Input and watch the Output signal = Impulse Response g(t). With some Mathematics you can then CALCULATE the desired Transfer Function.

But this Impulse Response g(t) is a Time Domain Signal and a very tricky thing, because it is now possible to calculate the Output Signal Curve Form for ANY Input Signal Curve Form by

CONVOLUTION: $U_{OUT}(t) = U_{IN}(t) * g(t)$

This is **Method 2 / Time Domain!**
==========

Remarks:
The **Dirac - Impuls** is a mystery, because it is a „Needle Pulse" with an infinite amplitude (and infinite „dU / dt"). But the Pulse width goes to Zero. This cannot be realized in reality but the signal can be replaced by a „normal" pulse with very large amplitude but very short pulse width.

In this case it is important that the PULSE AREA is and stays = 1 -- even and always if You vary the amplitude and / or the pulse width.

This means that you can increase the pulse width (recommendation: to a maximum of 1% of the system's Time Constant) and therefore decrease the pulse amplitude to get again the same pulse area (= 1) without any problems.

In all these cases you'll get the same Impulse Response g(t) as with an ideal Dirac Pulse.