Esperimentazioni II

Traccia delle esercitazioni PSpice

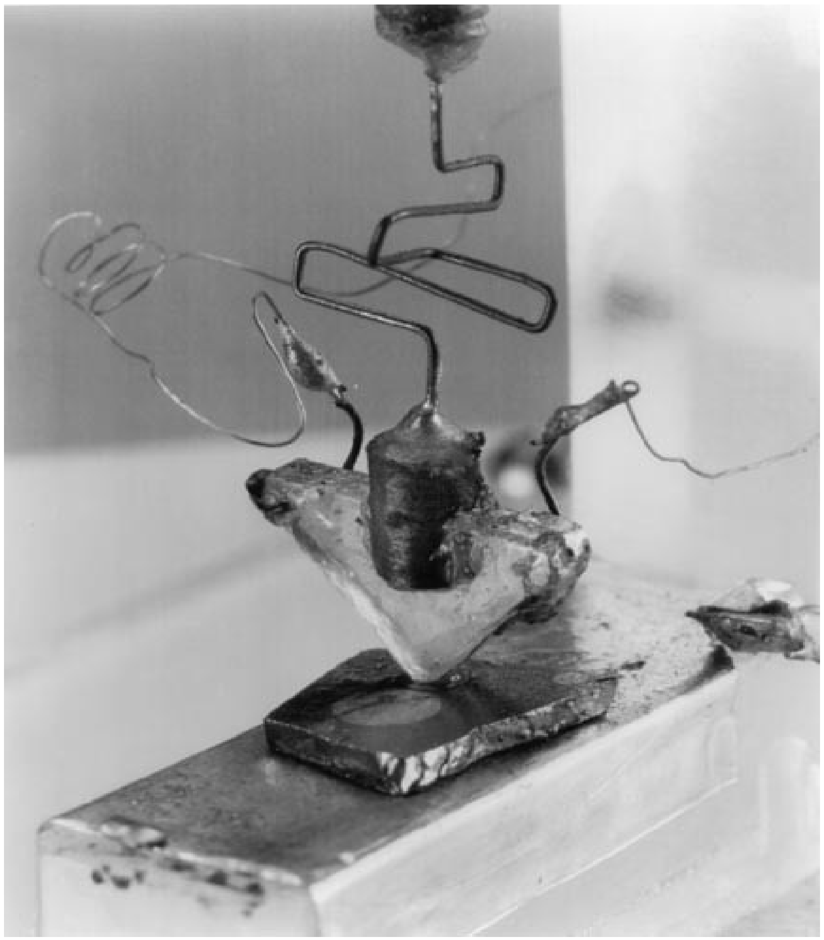
L. Pacher

pacher@to.infn.it

Outline

- the Bipolar Junction Transistor (BJT), look at the blackboard...
- basic usage of PSpice 'on fly'
- lab. experience #8 - BJT characteristic curves
- lab. experience #9 - NPN common-emitter amplifier

The first transistor (1947)



J. Bardeen **W. Shockley** **W. Brattain**

1956 Nobel prize in Physics

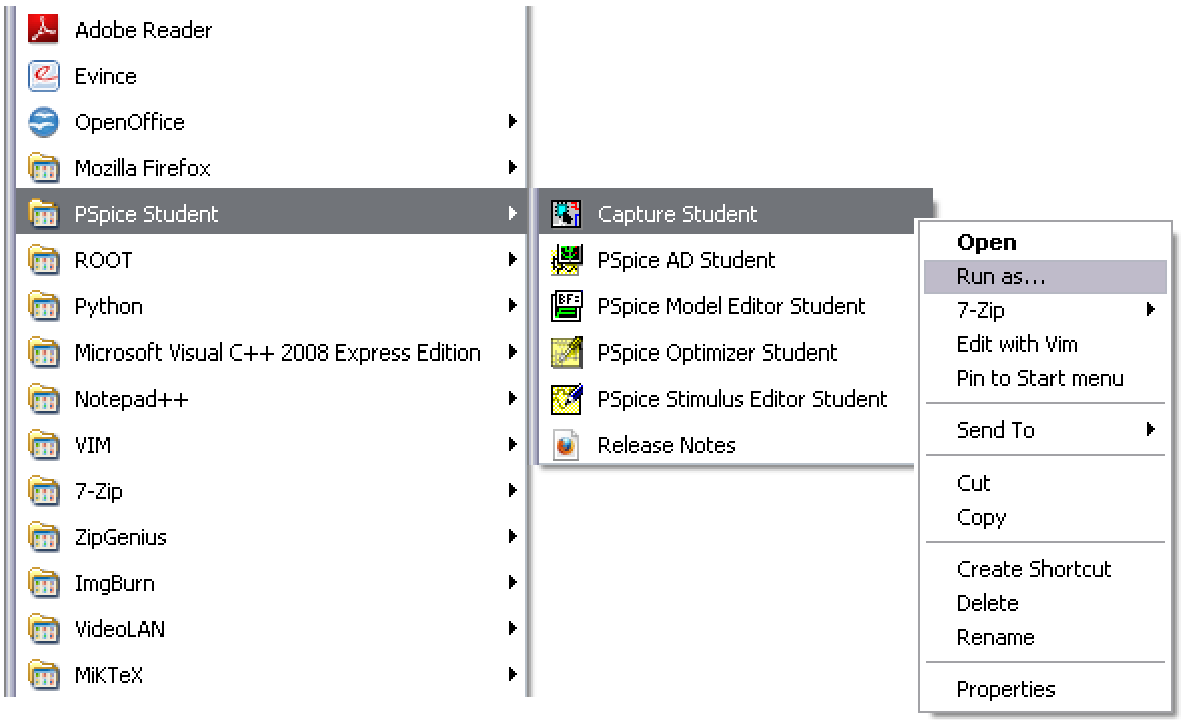
“for their researches on semiconductors and their discovery of the transistor effect”

Uhm....

SPICE and PSpice

- **S**imulation **P**rogram with **I**ntegrated **C**ircuits **E**mphasis, **SPICE**
- '***schematic***' is a meaningless word for a computer, just a human graphical visualization of the circuit
- a sort of interpreted ***'markup' and programming language*** with its own syntax, a plain text input file and another plain text output file
- the description of a circuit using the SPICE language is called ***netlist***
- ***SPICE file = netlist + electrical models + analysis directives***
- netlist and analysis directives are now ***automatically generated*** using ***graphical interfaces***
- **P**ersonal **SPICE**, **PSpice** is commercial graphical front-end for personal computers running MS Windows operating systems
- the ***student version*** is free, of course with some limitations
- installation details are in the backup section

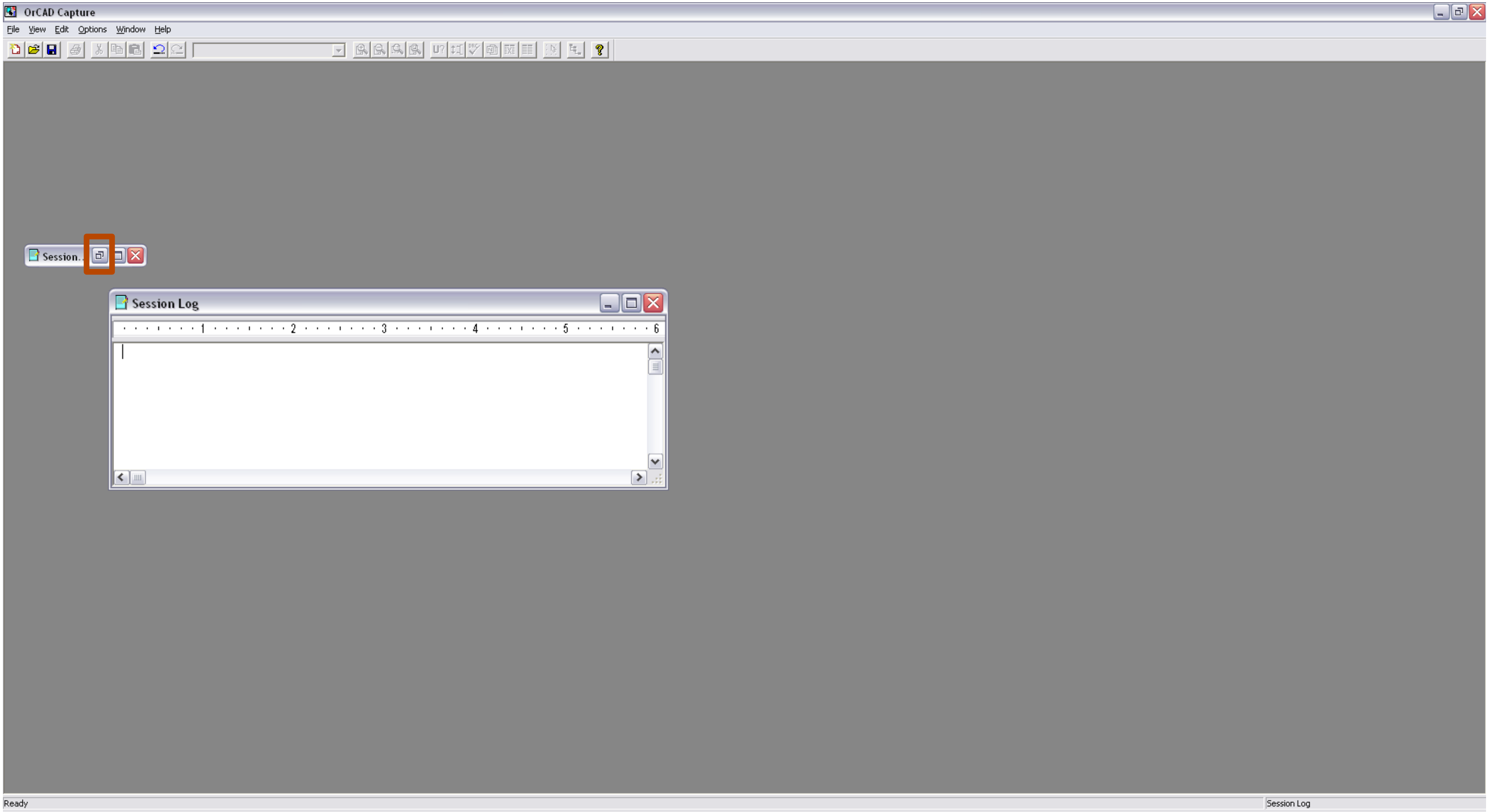
PSpice student



*Be sure to always run Capture with **high privileges**! Pspice simulation files require some (not well defined...) write privileges*

- ***Capture*** - schematic entry tool
- ***PSpice A/D*** - analog, digital and mixed-signals circuit simulator
- ***PSpice Model Editor*** - edit plain text SPICE models or extract models from data sheets
- ***PSpice Stimulus Editor*** – graphical editor for time-based waveforms

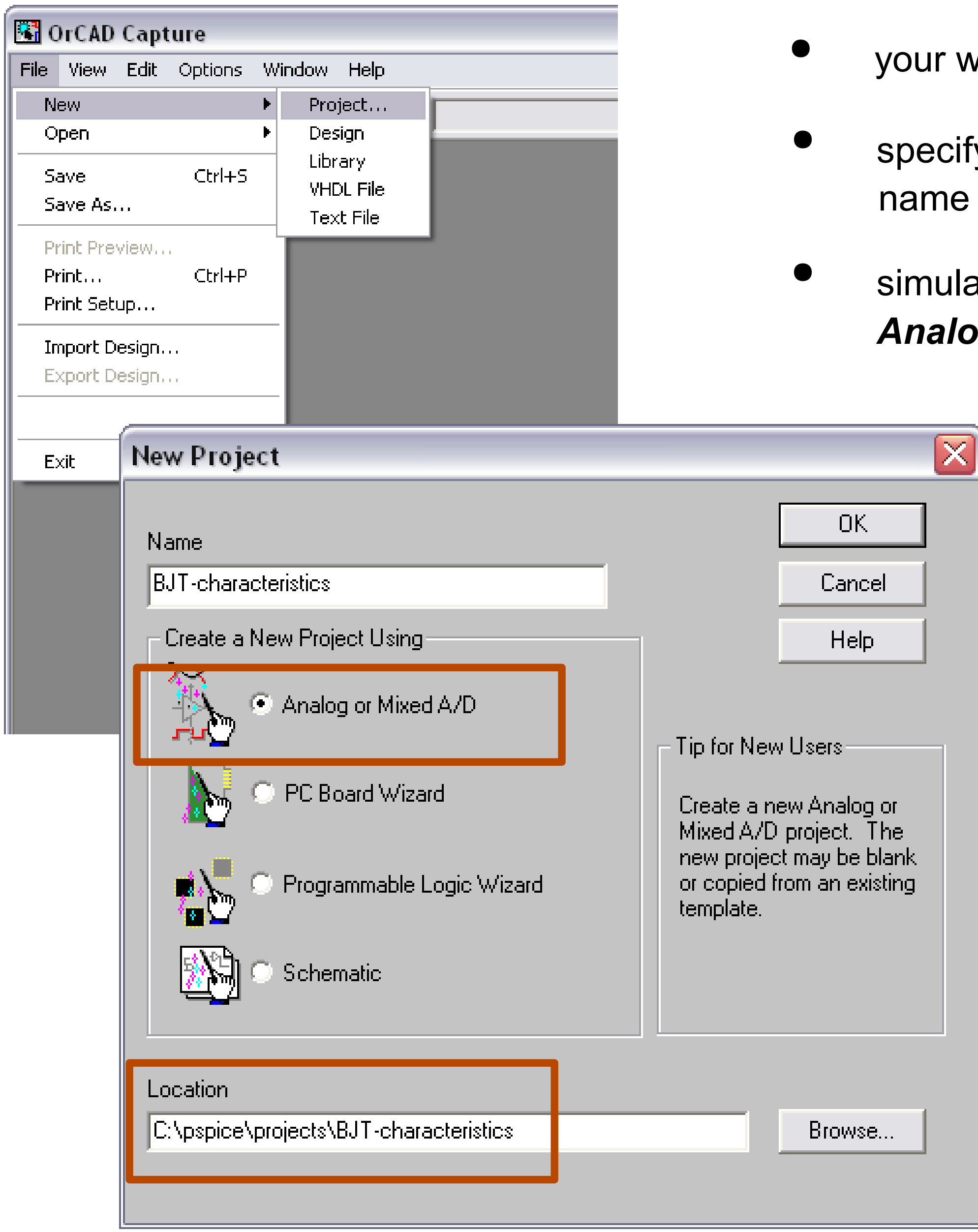
Getting started



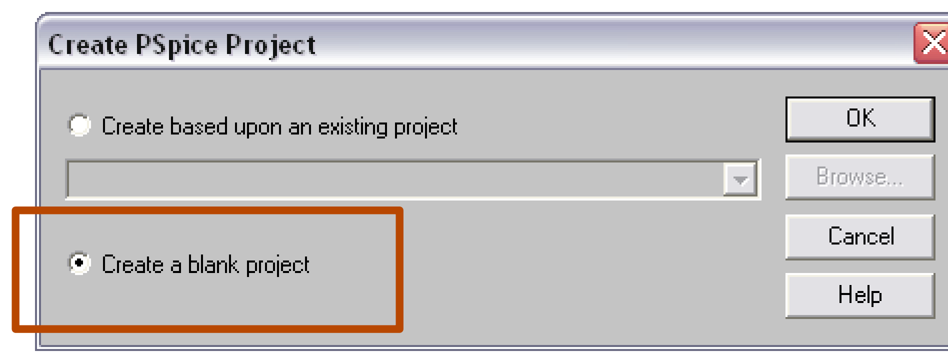
*The **Session Log** keeps trace of your work... and highlights possible **errors** !*

Create a new project

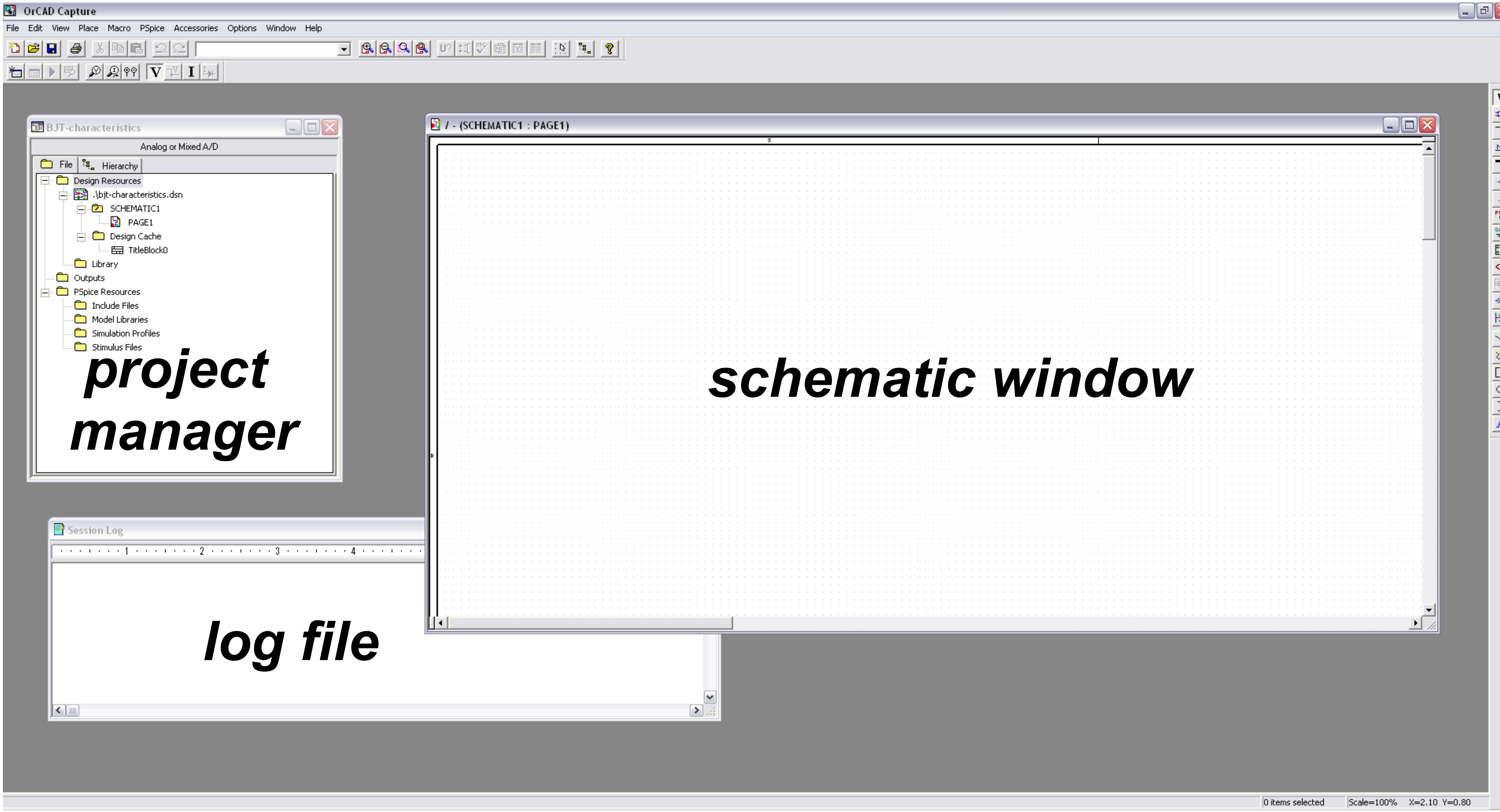
File → New → Project...



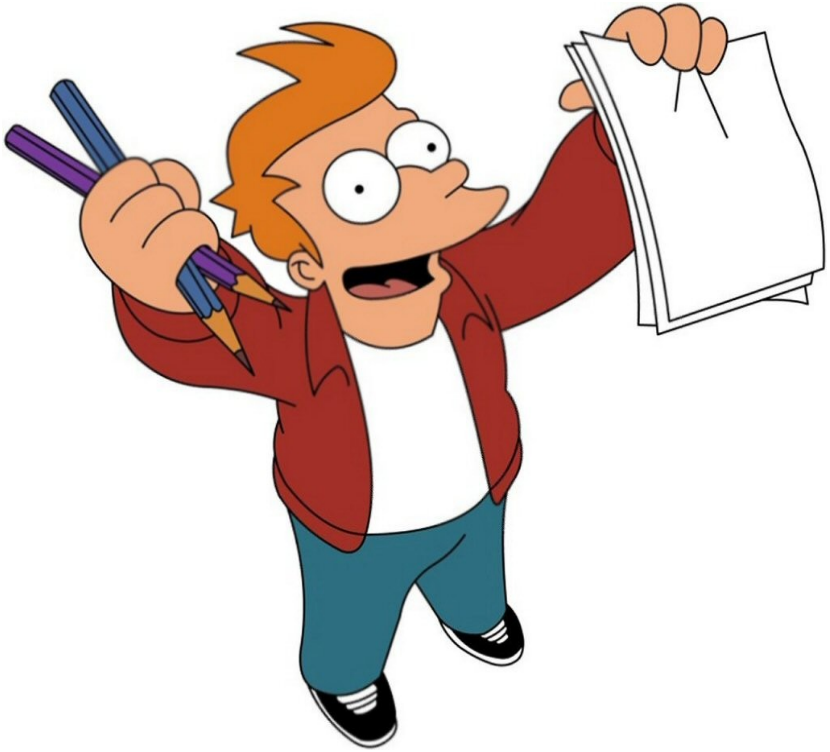
- your work is organized into ***projects*** (**.opj** main file)
- specify a new folder in `C:\pspice\projects` with the same name of the project
- simulations with PSpice are available ***only*** if you choose the ***Analog or Mixed A/D*** option



The graphical interface

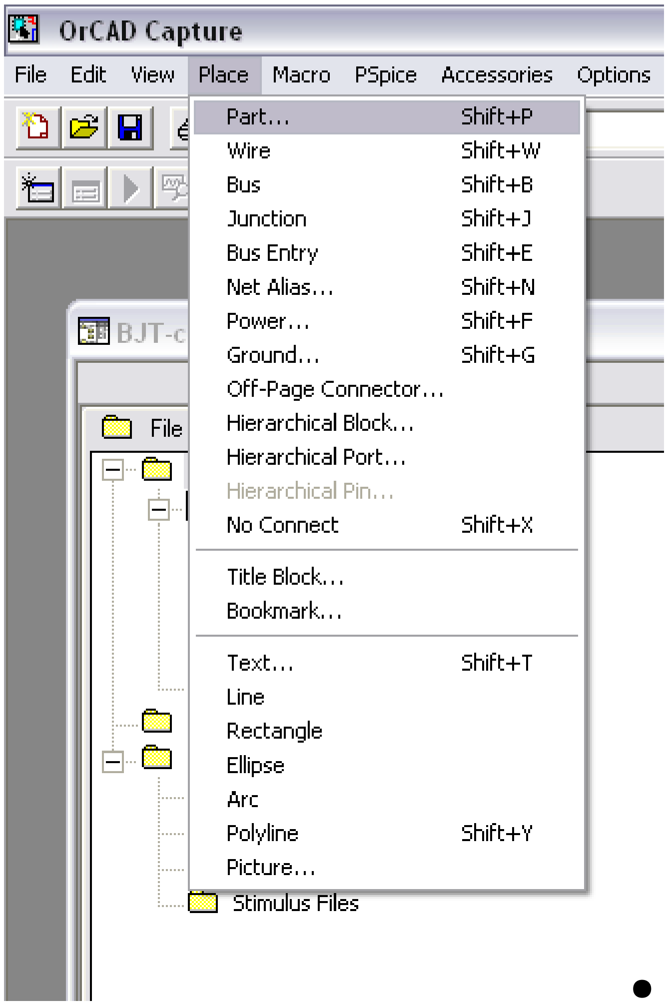


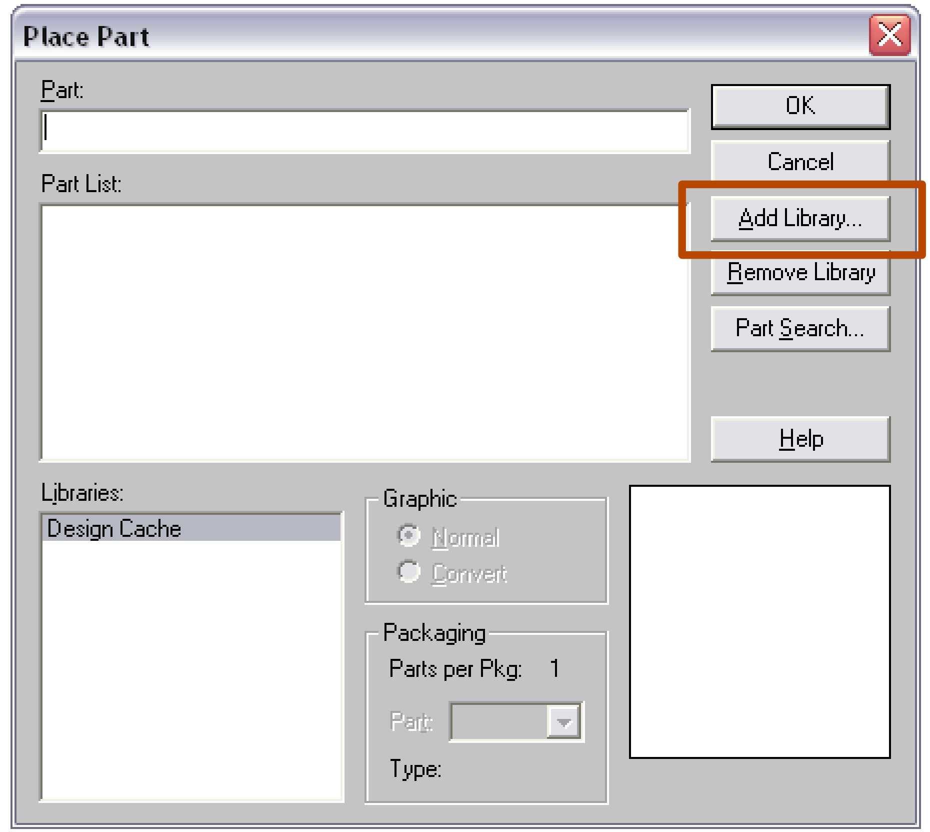
Part I
Schematic entry



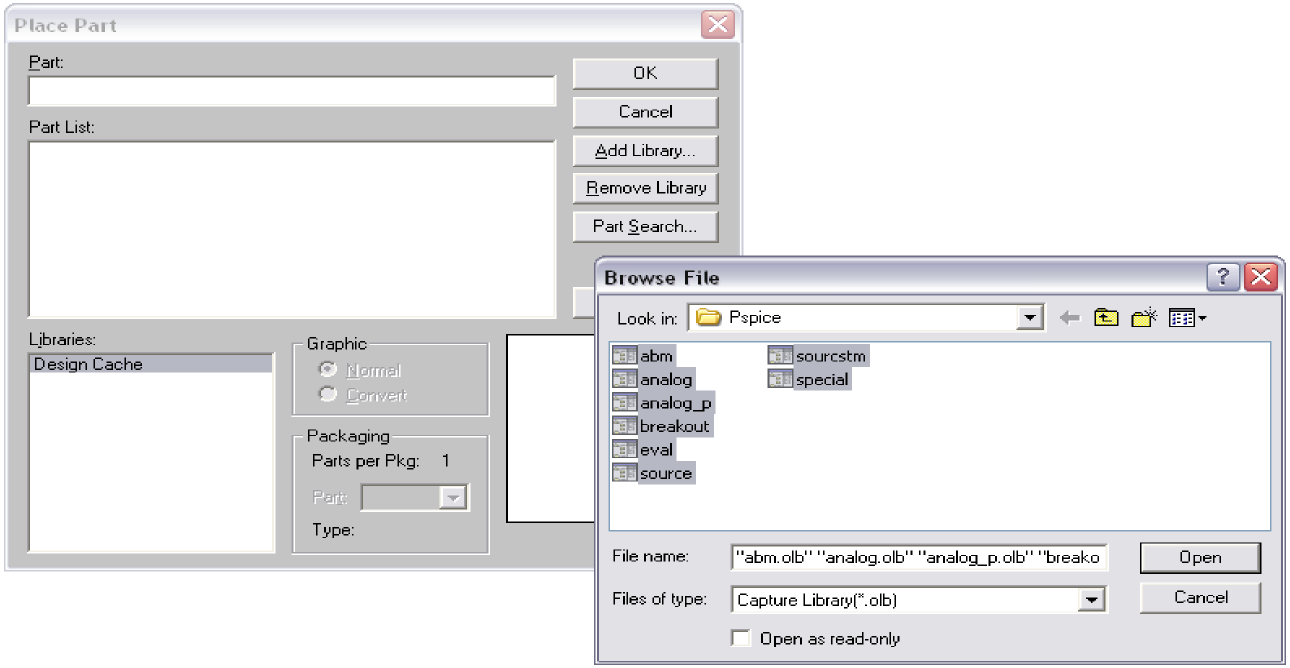
Place components

Place → Part...

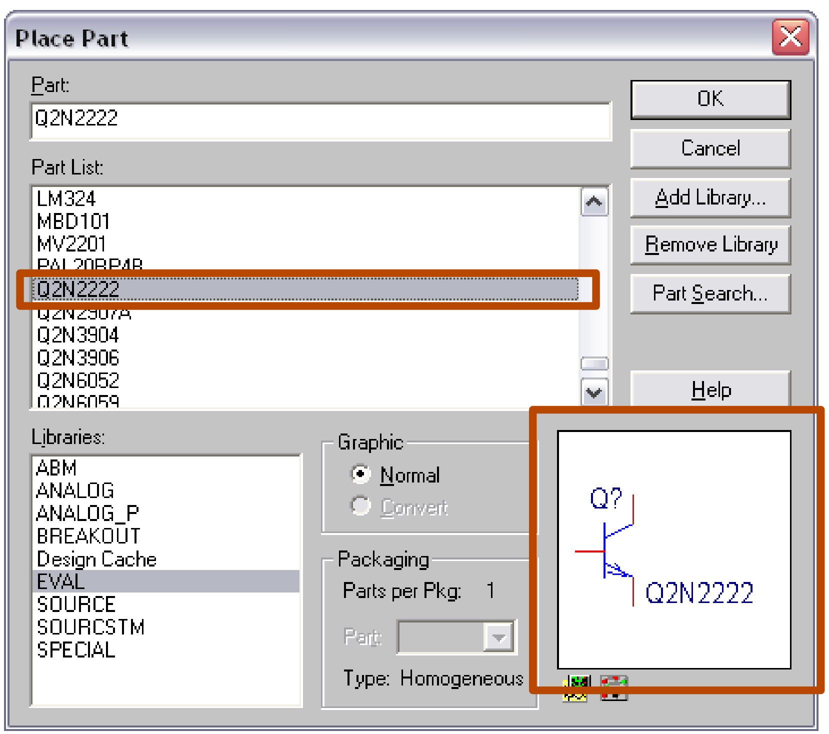




- ***symbols*** of electrical components are organized into ***libraries***
- the first time you are using PSpice you have to import the basic libraries



Basic libraries



- **ANALOG** contains ideal R, L, C and controlled voltage/current sources
- **BREAKOUT** contains ***ideal semiconductor devices*** (transistors, diodes etc.)
- **EVAL** contains a list of ***real semiconductor devices*** (diodes, transistors, logic gates, OPAMP etc.)
- **SOURCE** contains ideal voltage and current sources

NPN 2N2222

Check the datasheet!

Philips Semiconductors **Product specification**

NPN switching transistors 2N2222; 2N2222A

FEATURES

∞ High current (max. 800 mA)

∞ Low voltage (max. 40 V).

APPLICATIONS

∞ Linear amplification and switching.

DESCRIPTION

NPN switching transistor in a TO-18 metal package.
PNP complement: 2N2907A.

PINNING

PIN	DESCRIPTION
1	emitter
2	base
3	collector, connected to case

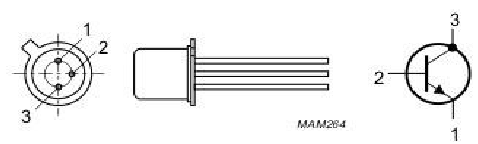
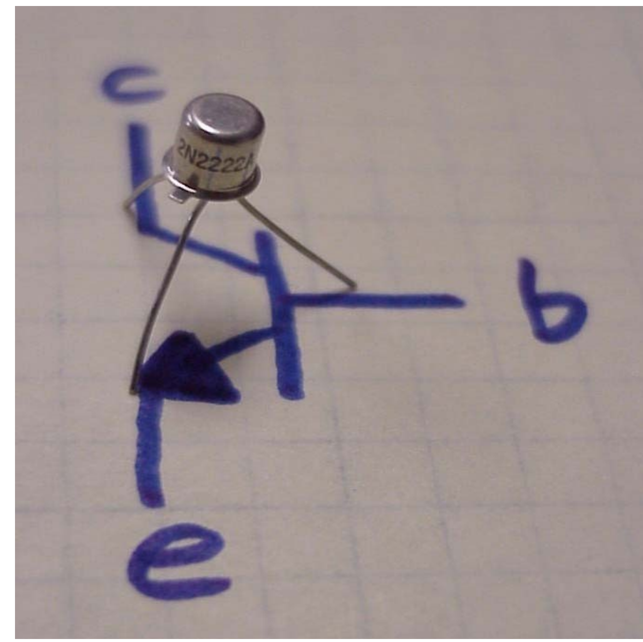


Fig.1 Simplified outline (TO-18) and symbol.

QUICK REFERENCE DATA

SYMBOL	PARAMETER	CONDITIONS	MIN.	MAX.	UNIT
V_{CBO}	collector-base voltage	open emitter			
	2N2222		–	60	V
	2N2222A		–	75	V
V_{CEO}	collector-emitter voltage	open base			
	2N2222		–	30	V
	2N2222A		–	40	V
I_C	collector current (DC)		–	800	mA
P_{tot}	total power dissipation	T_{amb} ″ 25 ϒC	–	500	mW
h_{FE}	DC current gain	I_C = 10 mA; V_{CE} = 10 V	75	–	
f_T	transition frequency	I_C = 20 mA; V_{CE} = 20 V; f = 100 MHz			
	2N2222		250	–	MHz
	2N2222A		300	–	MHz
t_{off}	turn-off time	I_{Con} = 150 mA; I_{Bon} = 15 mA; I_{Boff} = –15 mA	–	250	ns



A general-purpose commercial BJT

NPN TIP31

Check the datasheet again!

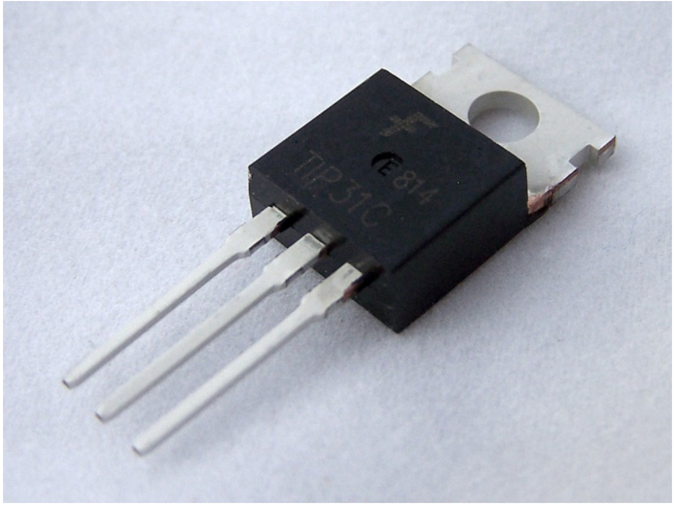
TIP31, TIP31A, TIP31B, TIP31C
NPN SILICON POWER TRANSISTORS

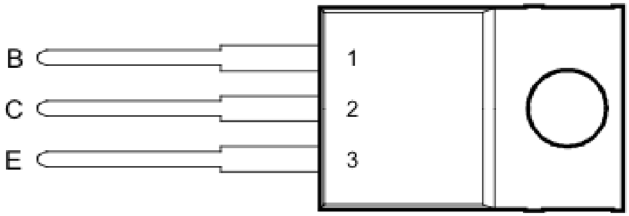
JULY 1968 - REVISED MARCH 1997

electrical characteristics at 25°C case temperature

PARAMETER		TEST CONDITIONS			MIN	TYP	MAX	UNIT
$V_{(BR)CEO}$	Collector-emitter breakdown voltage	I_C = 30 mA (see Note 5)	$I_B = 0$	TIP31 TIP31A TIP31B TIP31C	40 60 80 100			V
I_{CES}	Collector-emitter cut-off current	V_{CE} = 80 V V_{CE} = 100 V V_{CE} = 120 V V_{CE} = 140 V	$V_{BE} = 0$ $V_{BE} = 0$ $V_{BE} = 0$ $V_{BE} = 0$	TIP31 TIP31A TIP31B TIP31C			0.2 0.2 0.2 0.2	mA
I_{CEO}	Collector cut-off current	V_{CE} = 30 V V_{CE} = 60 V	$I_B = 0$ $I_B = 0$	TIP31/31A TIP31B/31C			0.3 0.3	mA
I_{EBO}	Emitter cut-off current	V_{EB} = 5 V	$I_C = 0$				1	mA
h_{FE}	Forward current transfer ratio	V_{CE} = 4 V V_{CE} = 4 V	I_C = 1 A I_C = 3 A	(see Notes 5 and 6)	25 10		 50	
$V_{CE(sat)}$	Collector-emitter saturation voltage	I_B = 375 mA	I_C = 3 A	(see Notes 5 and 6)			1.2	V
V_{BE}	Base-emitter voltage	V_{CE} = 4 V	I_C = 3 A	(see Notes 5 and 6)			1.8	V
h_{fe}	Small signal forward current transfer ratio	V_{CE} = 10 V	I_C = 0.5 A	f = 1 kHz	20			
$\|h_{fe}\|$	Small signal forward current transfer ratio	V_{CE} = 10 V	I_C = 0.5 A	f = 1 MHz	3			

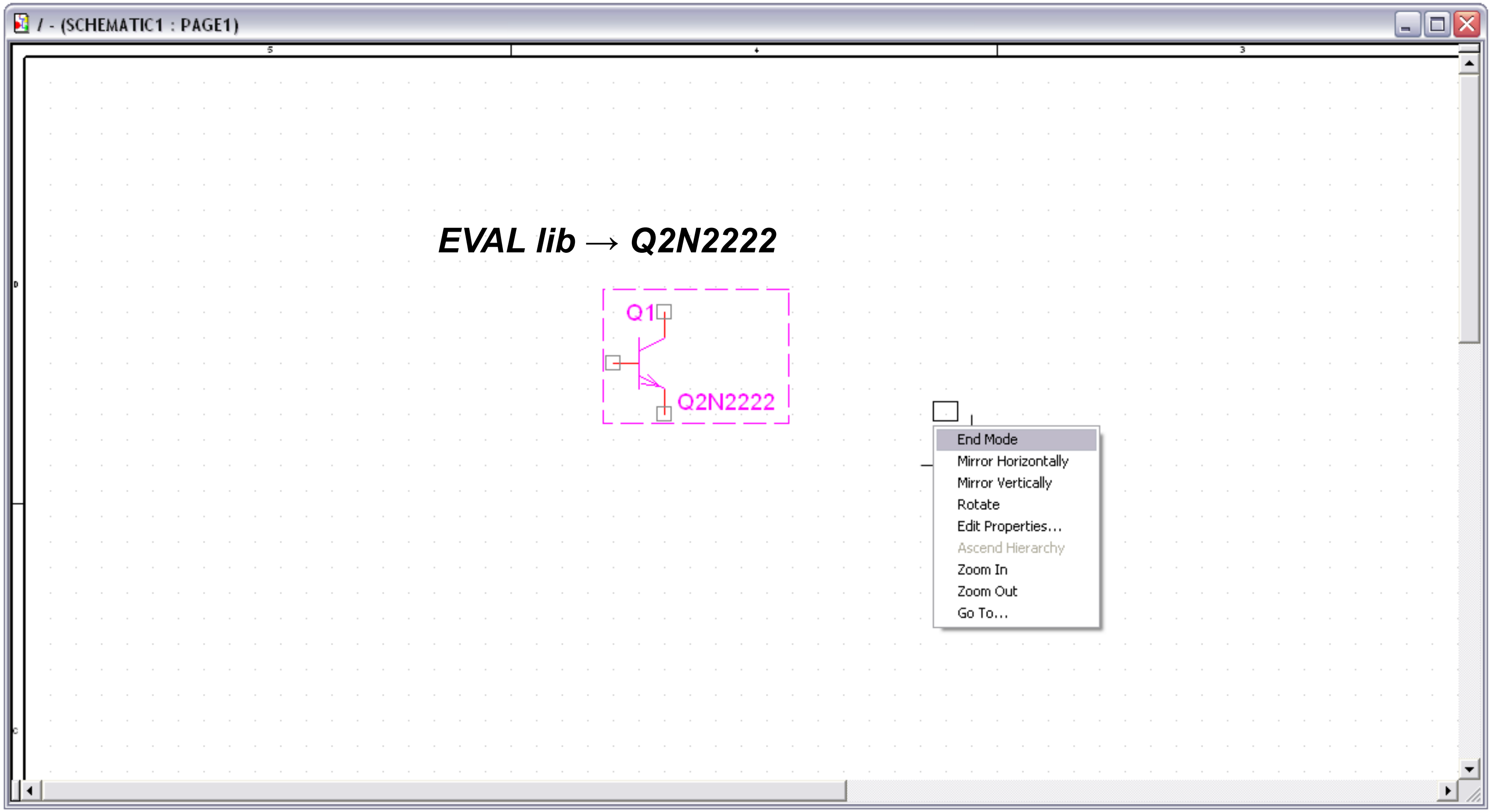
NOTES: 5. These parameters must be measured using pulse techniques, t_p = 300 µs, duty cycle ≤ 2%.
6. These parameters must be measured using voltage-sensing contacts, separate from the current carrying contacts.



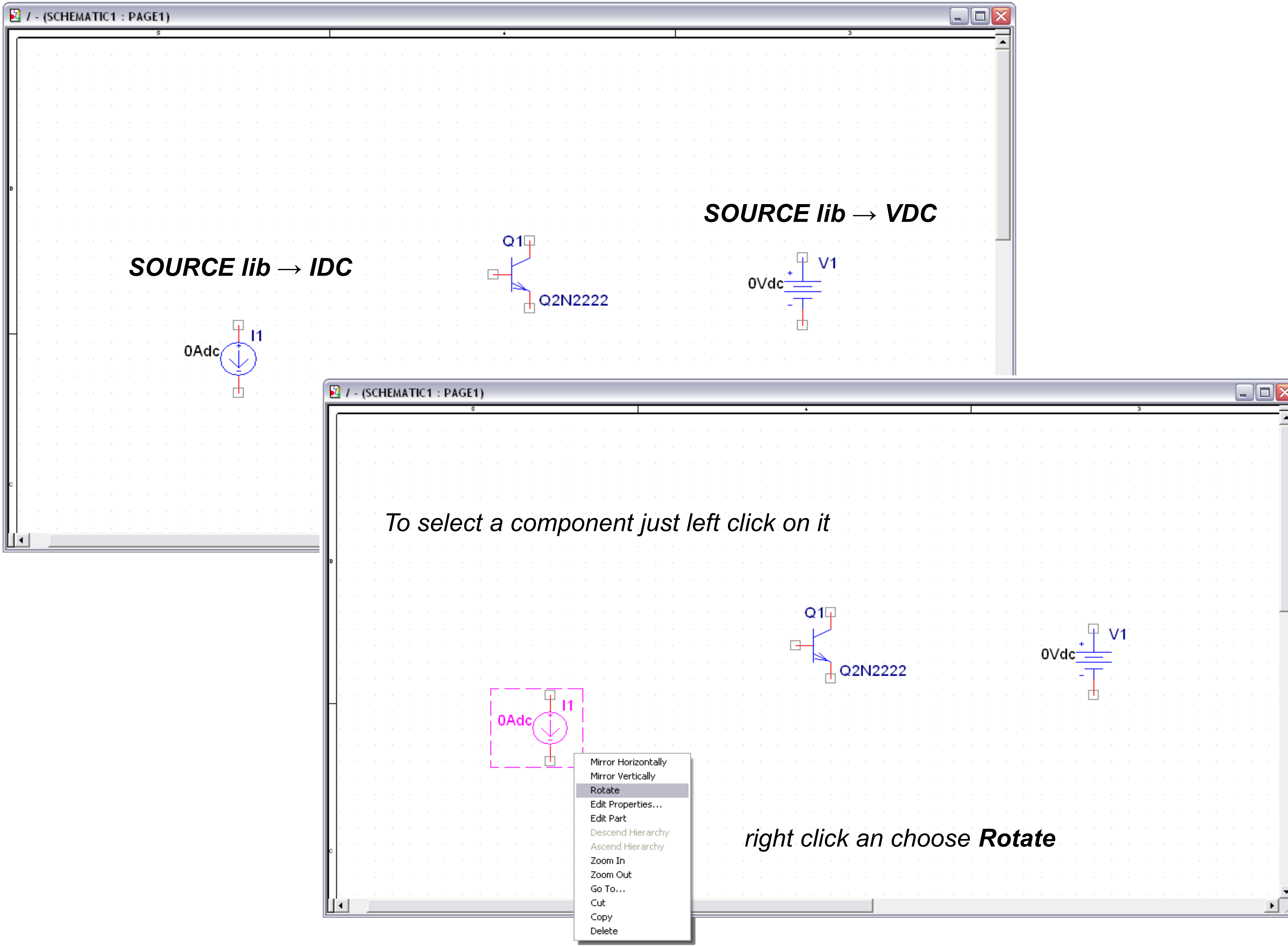


Another general-purpose commercial BJT

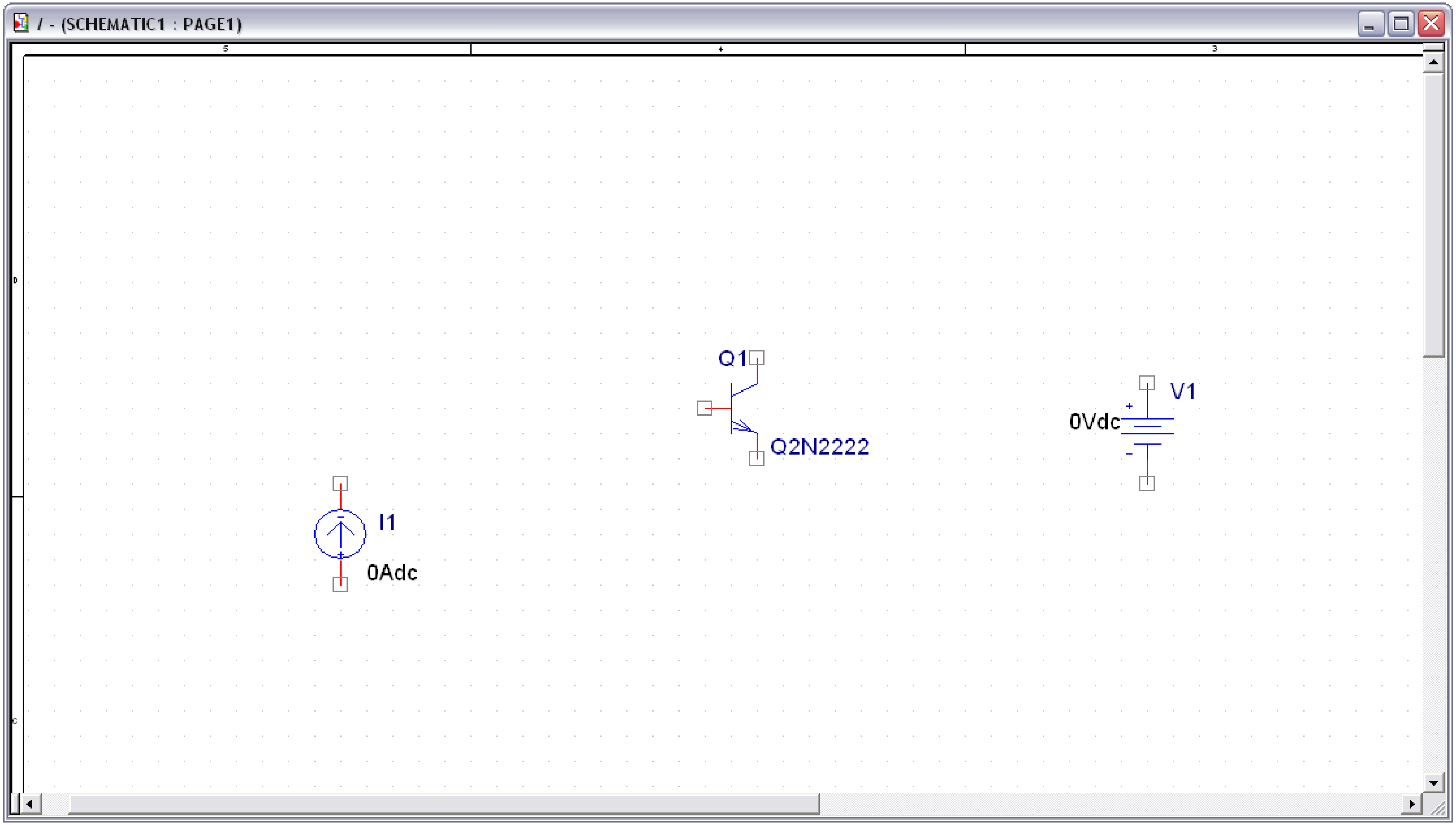
This should be the transistor available for your experimental measurements in lab...



You can place multiple instances of the same component, press ***ESC*** *or* ***right click*** *→* ***End Mode*** *to come back to the select mode*



/ - (SCHEMATIC1 : PAGE1)
SOURCE lib → IDC
SOURCE lib → VDC
Q1
Q2N2222
V1
0Vdc
I1
0Adc
/ - (SCHEMATIC1 : PAGE1)
To select a component just left click on it
Mirror Horizontally
Mirror Vertically
Rotate
Edit Properties...
Edit Part
Descend Hierarchy
Ascend Hierarchy
Zoom In
Zoom Out
Go To...
Cut
Copy
Delete
right click an choose Rotate

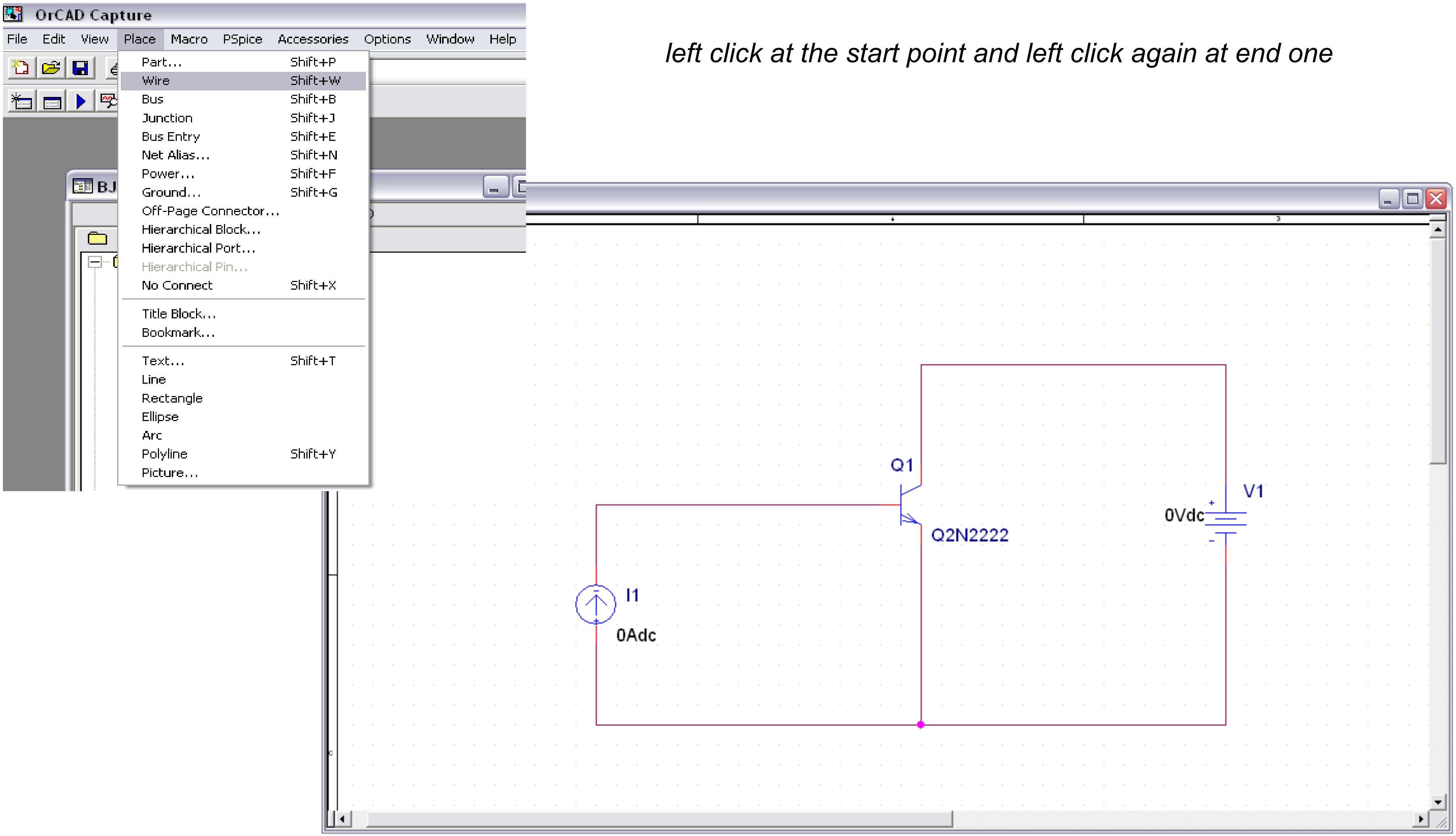


Components are on the table, now we have to make connections...

Make connections

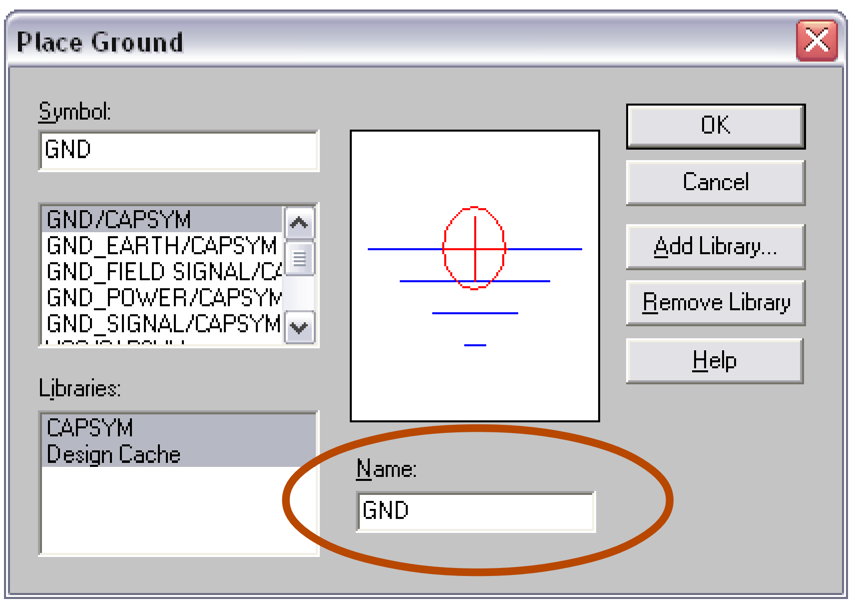
Place → Wire...

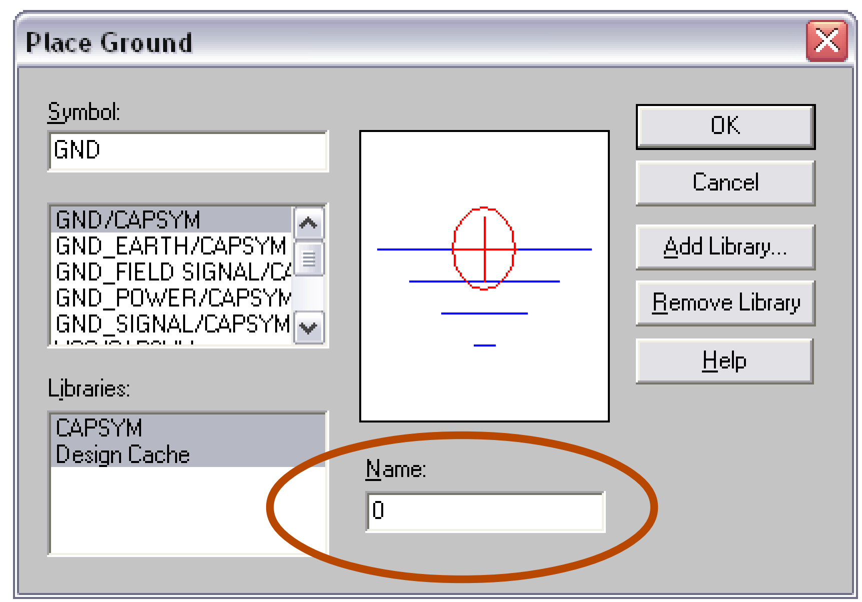
left click at the start point and left click again at end one

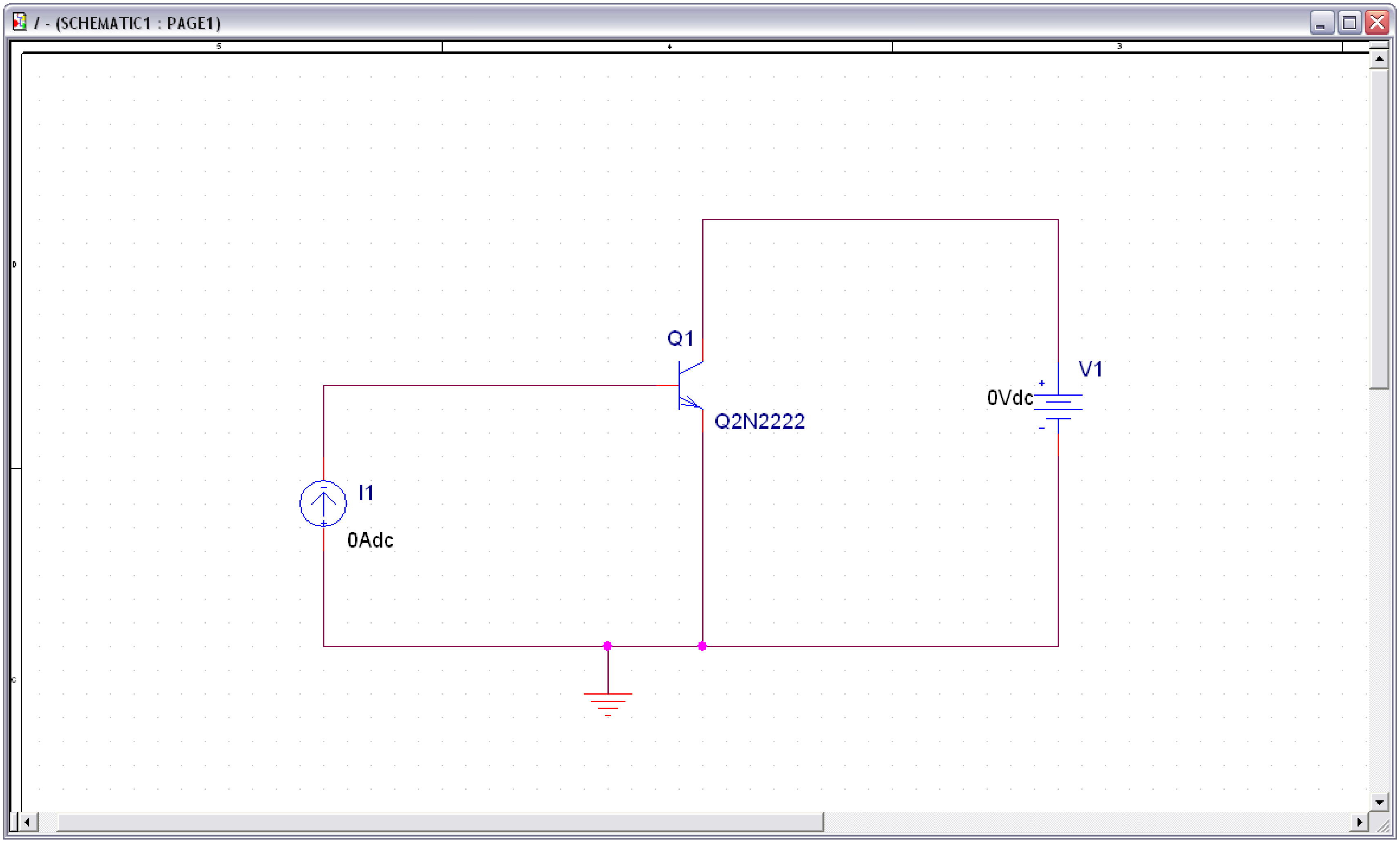


Place grounds

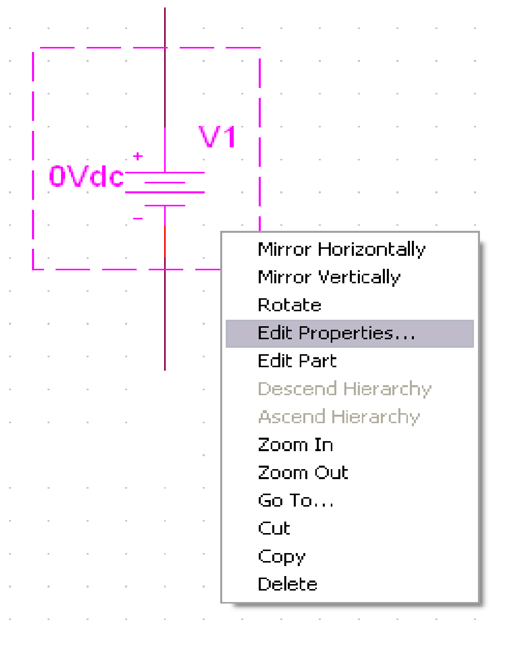
- at least one node ***MUST be named 0*** for the common reference voltage (***floating-node errors*** otherwise)
- ***Place*** **→ Ground...**
- use for instance **CAPSYM /GND** symbol but ***change Name into 0*** *!*

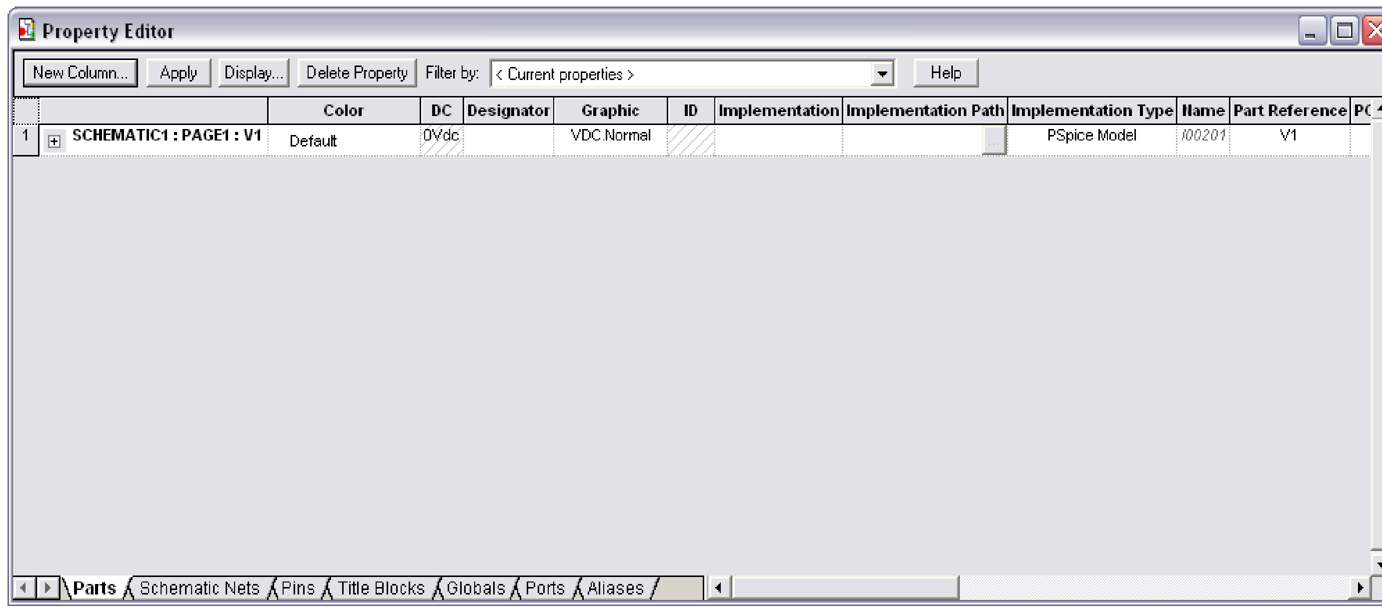




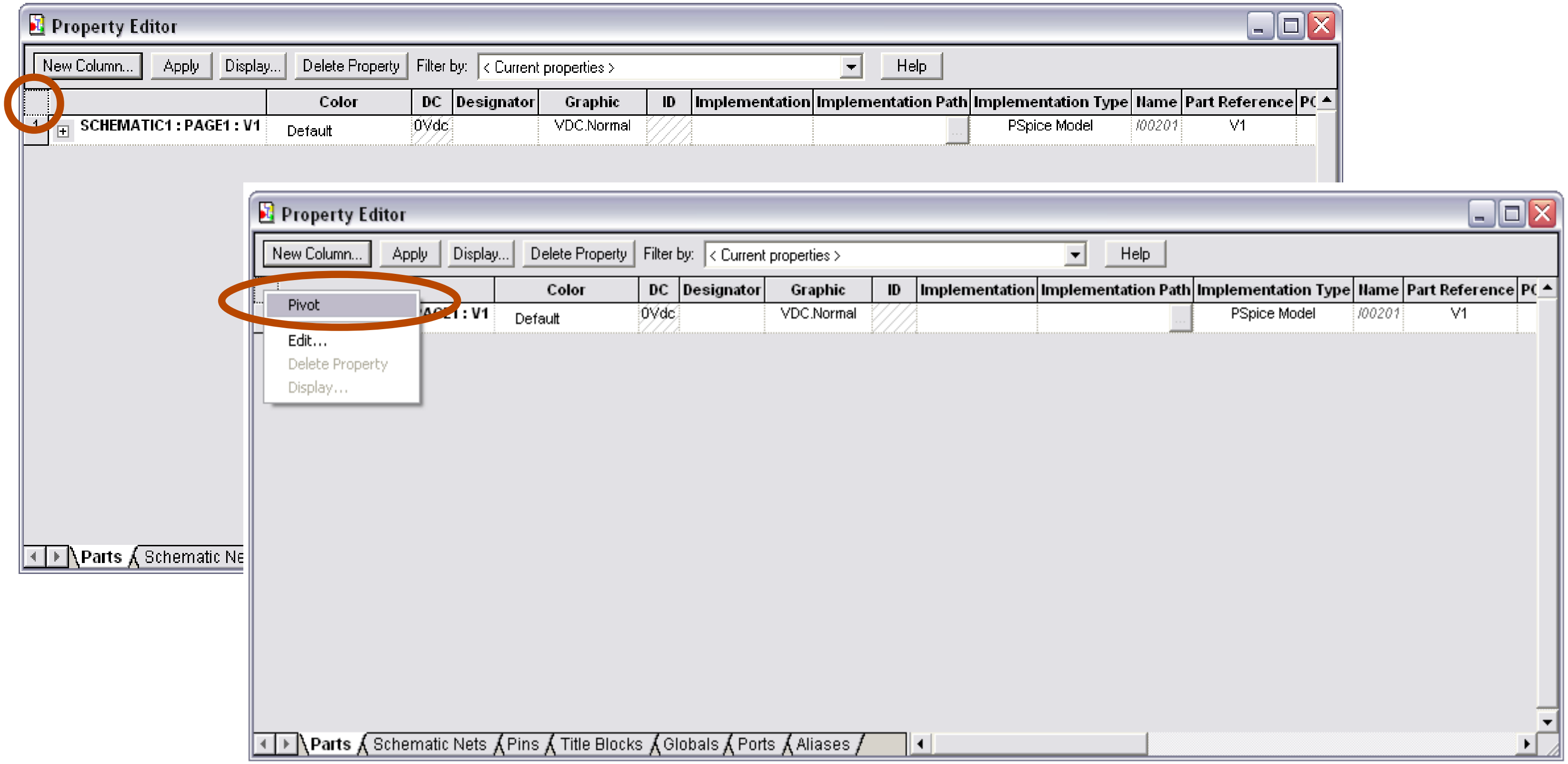

/ - (SCHEMATIC1 : PAGE1)
Q1
Q2N2222
V1
0Vdc
I1
0Adc

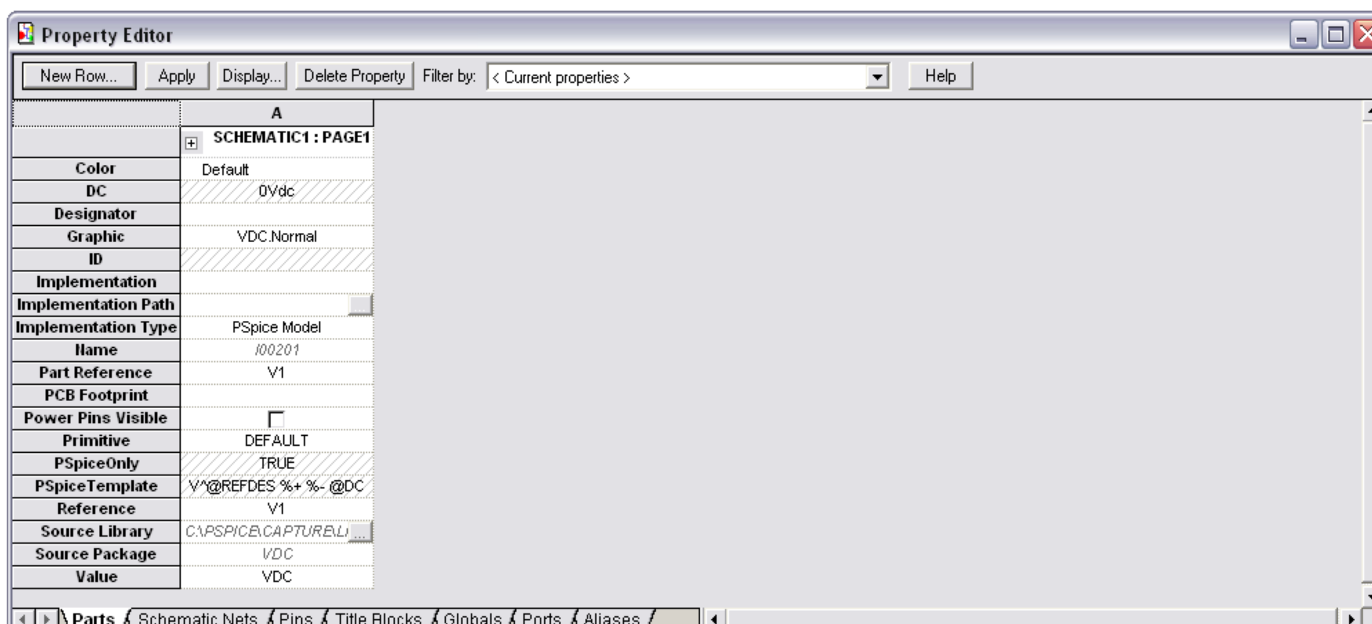
Edit instance properties



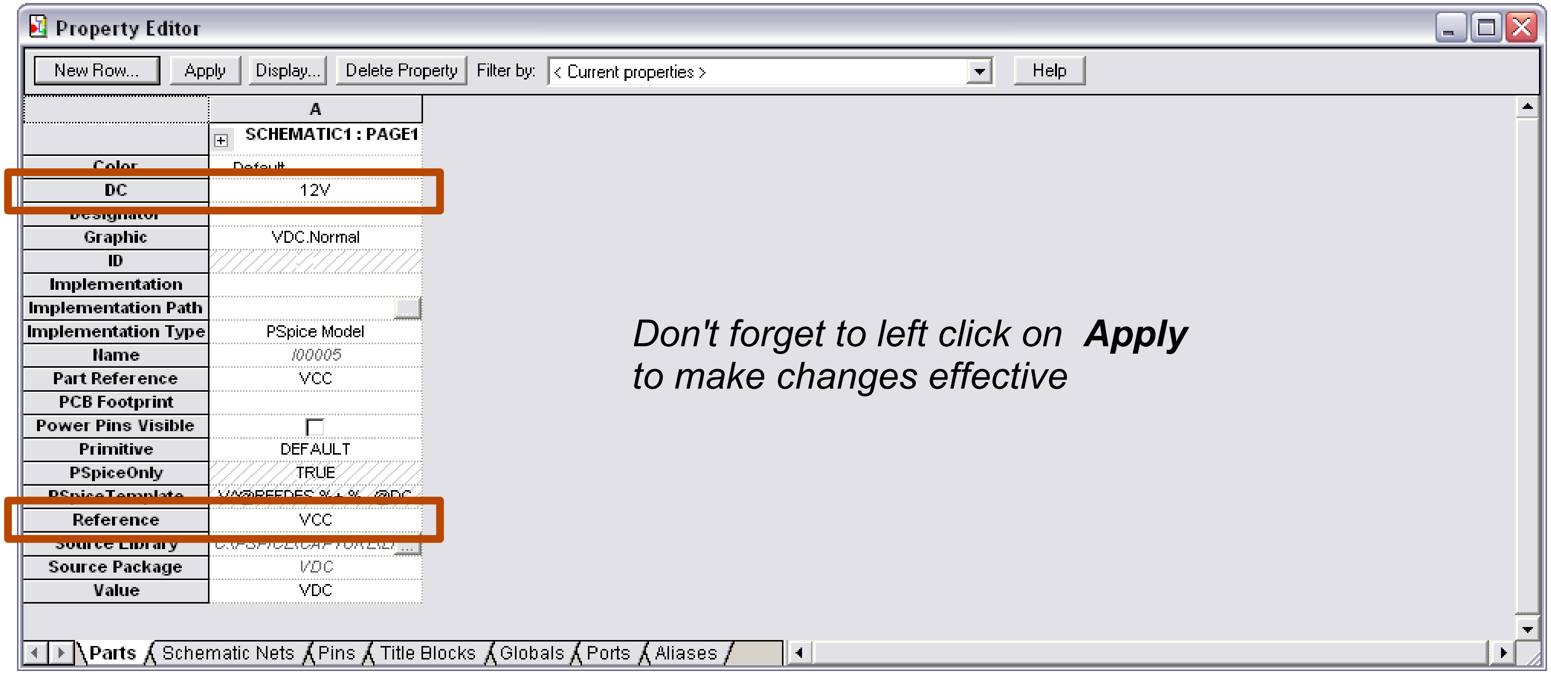


- each component has a set of related ***properties*** and ***parameters***
- a property/parameter has a ***name*** and a ***value***
- some properties names/values are displayed in the schematic, some others by default are shadowed
- left click to select a component, then ***right click*** → ***Edit Properties...***

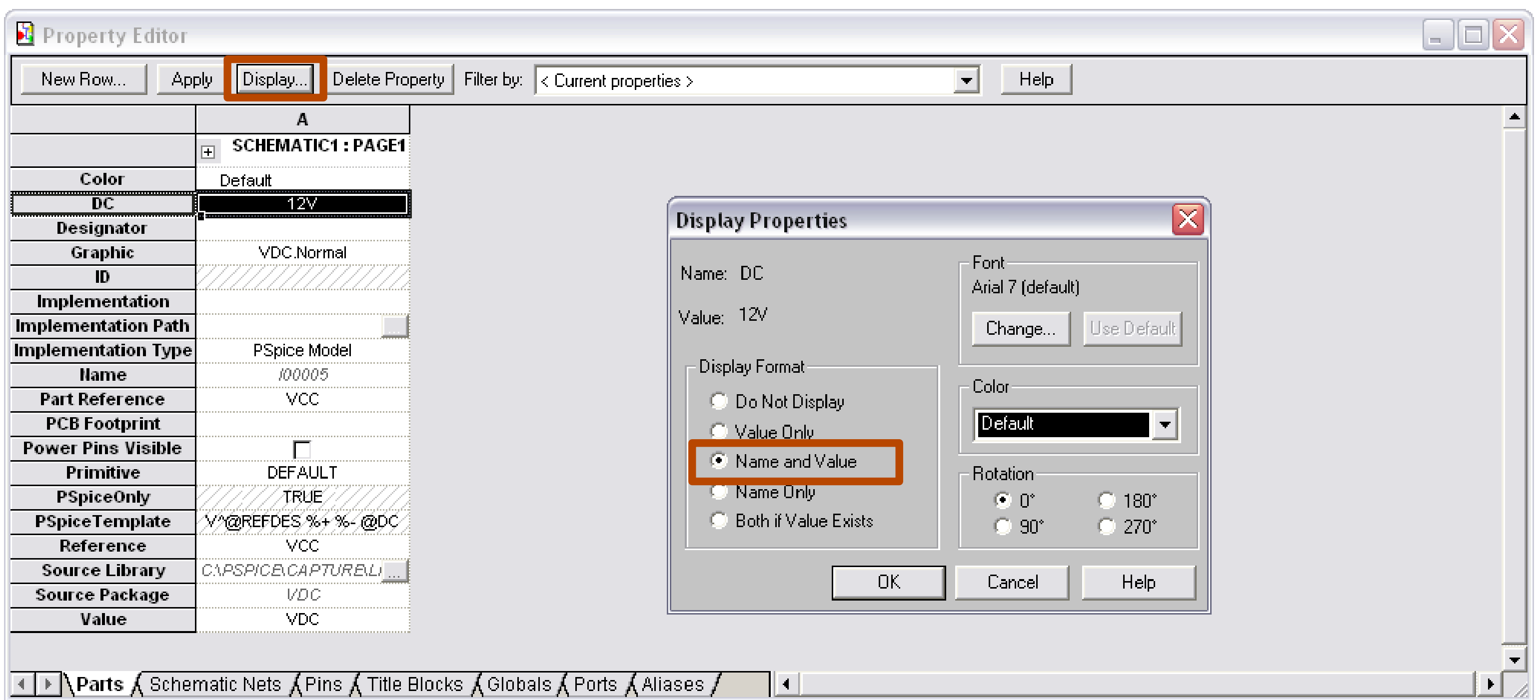


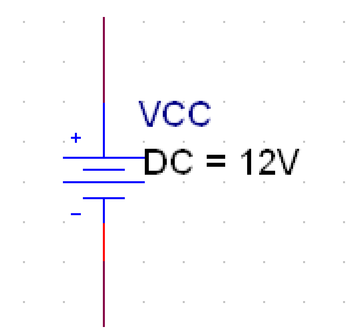


For a better visualization choose the ***Pivot*** view (right click on the top-left square in the table)



*Don't forget to left click on **Apply***
to make changes effective





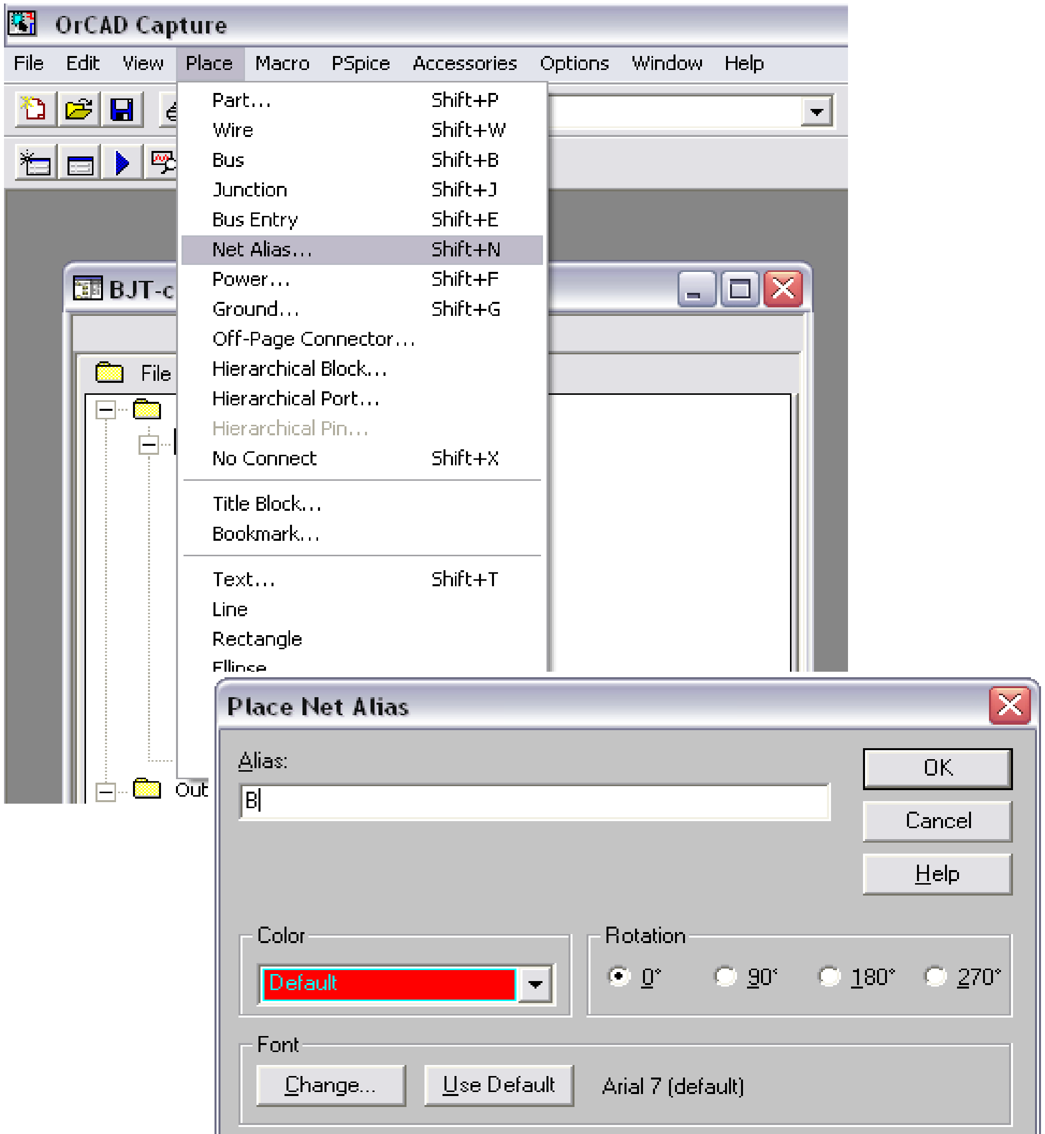
SPICE SI units and prefixes

name	SI	SPICE	C/C++ style
tera	T	T, t	1e12, 1E12
giga	G	G, g	1e9, 1E9
mega	M	MEG, meg	1e6, 1E6
kilo	k	K, k	1e3, 1E3
milli	m	M, m	1e-3, 1E-3
micro	µ	U, u	1e-6, 1E-6
nano	n	N. n	1e-9, 1E-9
pico	p	P, p	1e-12, 1E-12
femto	f	F, f	1e-15, 1E-15

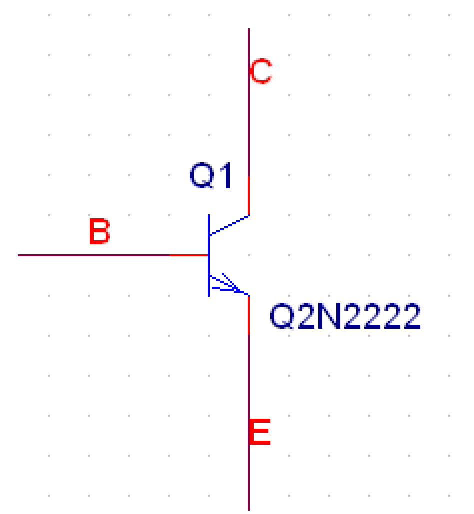
- SPICE is ***not case-sensitive**,* upper case and lower case letters are equivalent
- be careful not to use M for mega! 15Mohm are 15 milliohm for SPICE
- ***the unit name can be neglected***, hence 10 and 10V are equivalent for SPICE
- numerical values and prefixes must be typed without spaces, e.g. ***10uF**, **10u**, **10e-6**, **10E-6f***

Naming nets

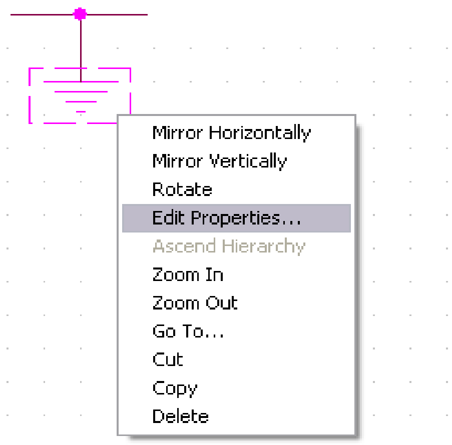
Place → Net Alias...

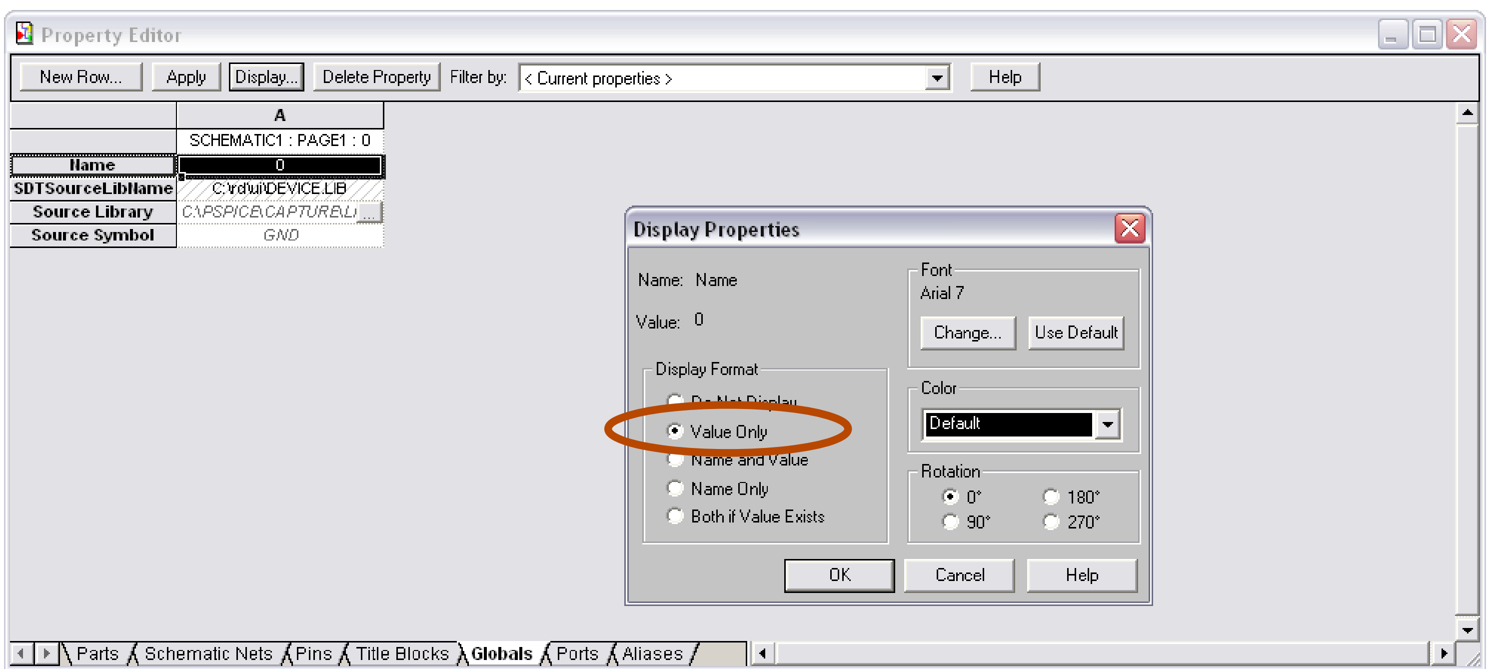


*Give a name (**net alias**) to most significant nodes of the circuit !*



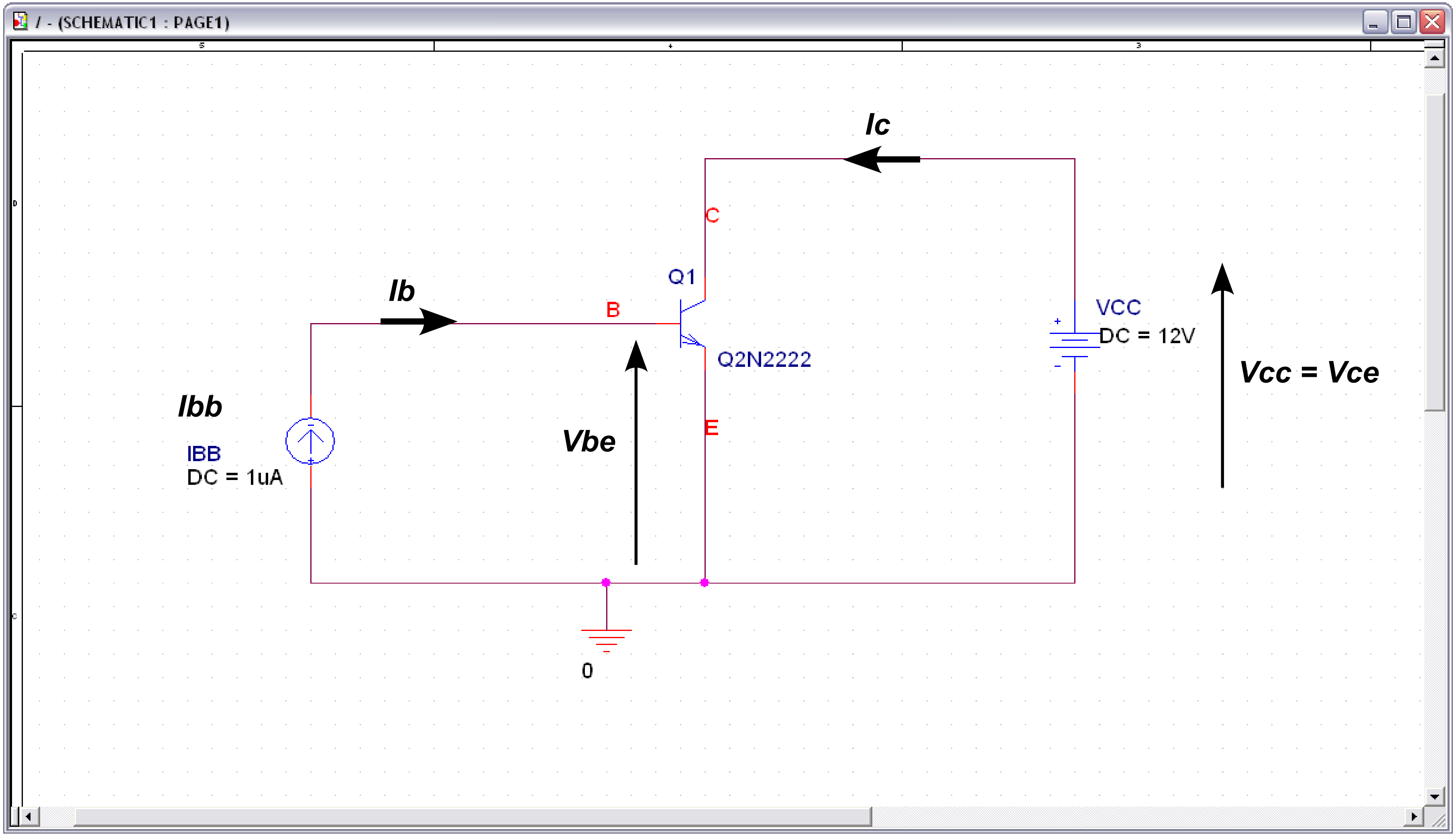
Back to the ground...





Just to remind us that the emitter is tied to zero...

Complete schematic



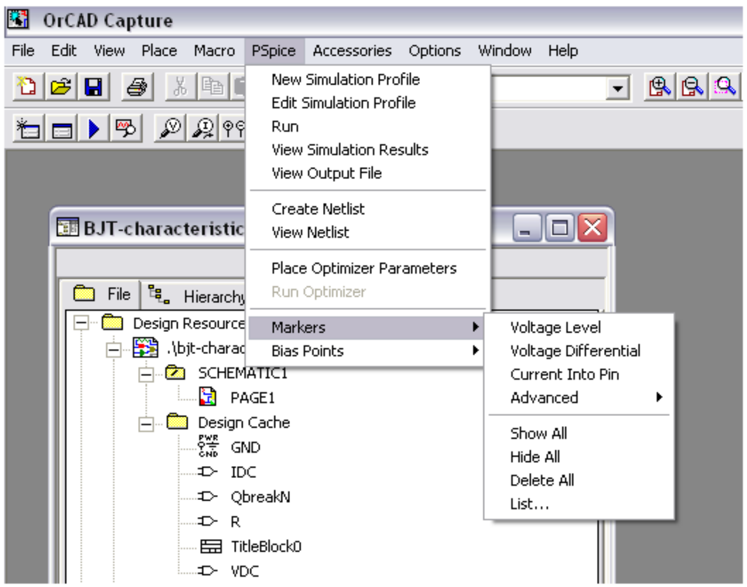
Useful shortcuts

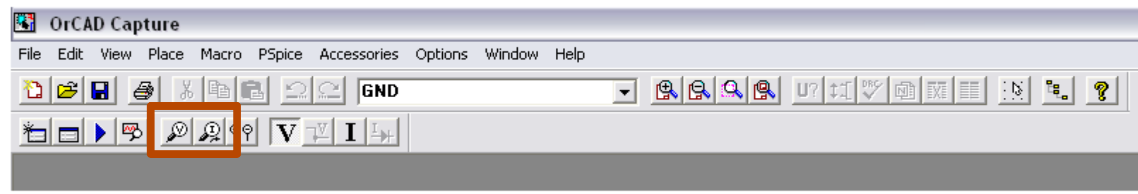
Capture shortcut	description
P	place part
Ctrl + A	add library
G	place ground
F	place power
Ctrl + E	edit component properties
W	place wire
N	place net alias
J	place junction
ESC	end mode
R	rotate component
H / V	mirror horizontally/vertically
T	place text
I /O	zoom in/out
Ctrl + X / Ctrl + V	cut/paste
DEL, **CANC**	delete component

Part II
Simulations ! (and more theory...)

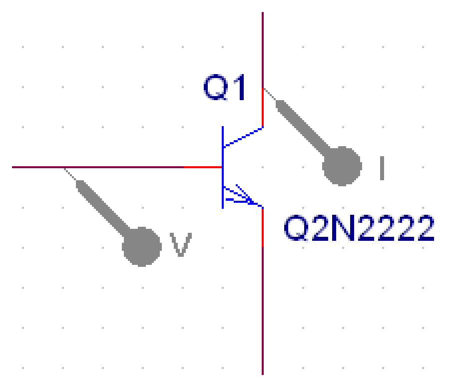
Voltage and current markers

PSpice → Markers → Voltage Level / Current Into Pin



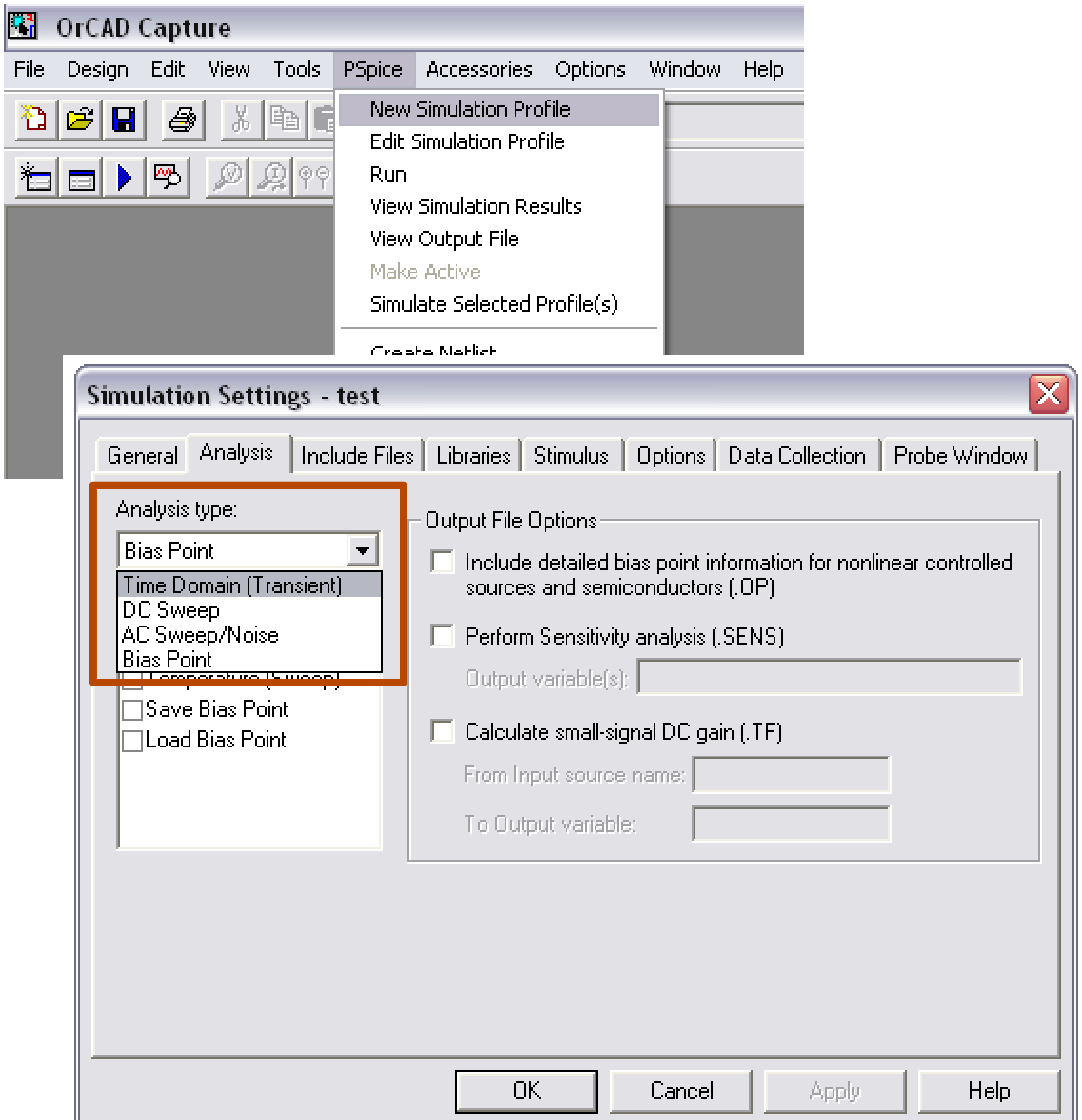


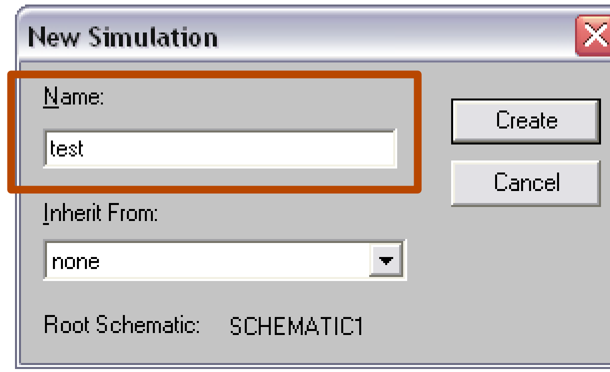
- put ***voltage markers*** on ***wires***
- put ***current markers*** on ***pins***



Simulation profiles

PSpice → New Simulation Profile





- ***transient analysis***
- ***DC sweeps***
- ***frequency (AC) analysis***
- ***bias point***

Bias point

- ***large signal*** DC solution for a particular input voltage/current condition

- ***the time is removed from the circuit***
  - sources with time specifications are set to zero
  - all capacitors are considered open circuits, all inductors shorts
  - DC analysis is a particular case of transient analysis ($dv/dt = 0, di/dt = 0$)

- automatically computed in any other simulation

- simulation results are printed in the text output file and can be visualized in the schematic using ***bias point markers***
  - list of all ***node voltages***, ***currents*** and total ***power dissipation***
  - detailed bias point information for ***semiconductor devices*** (not included by default)

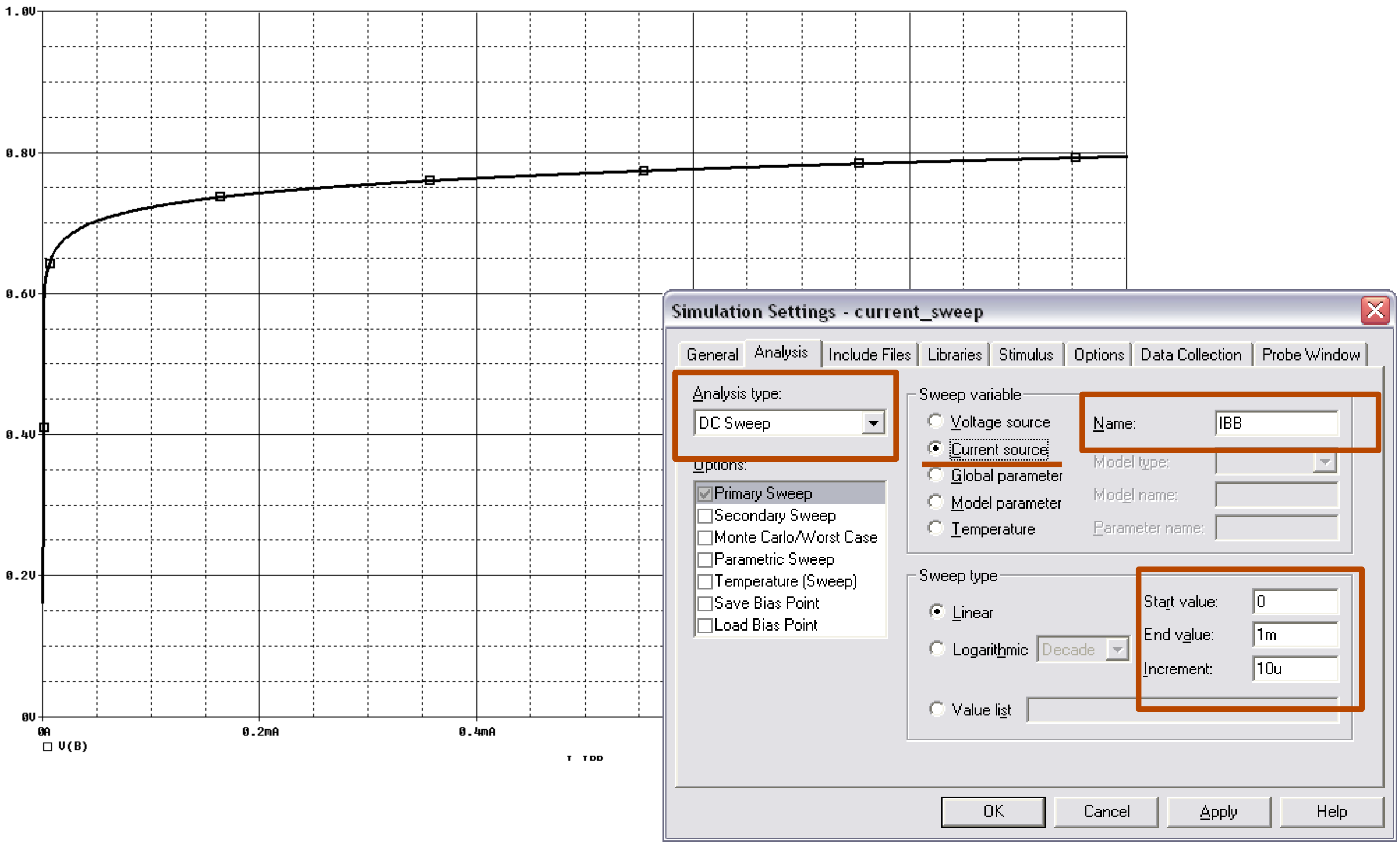
DC sweep

- ***large signal steady-state circuit DC response*** when sweeping a voltage/current source, a global parameter, a model parameter or the temperature ***over a range of values***

- the ***bias point*** is calculated for each value of the sweep

- ***nested DC sweep analysis*** can be performed
  - a secondary sweep variable can be selected after a primary sweep value has been specified
  - ***curve families*** are obtained

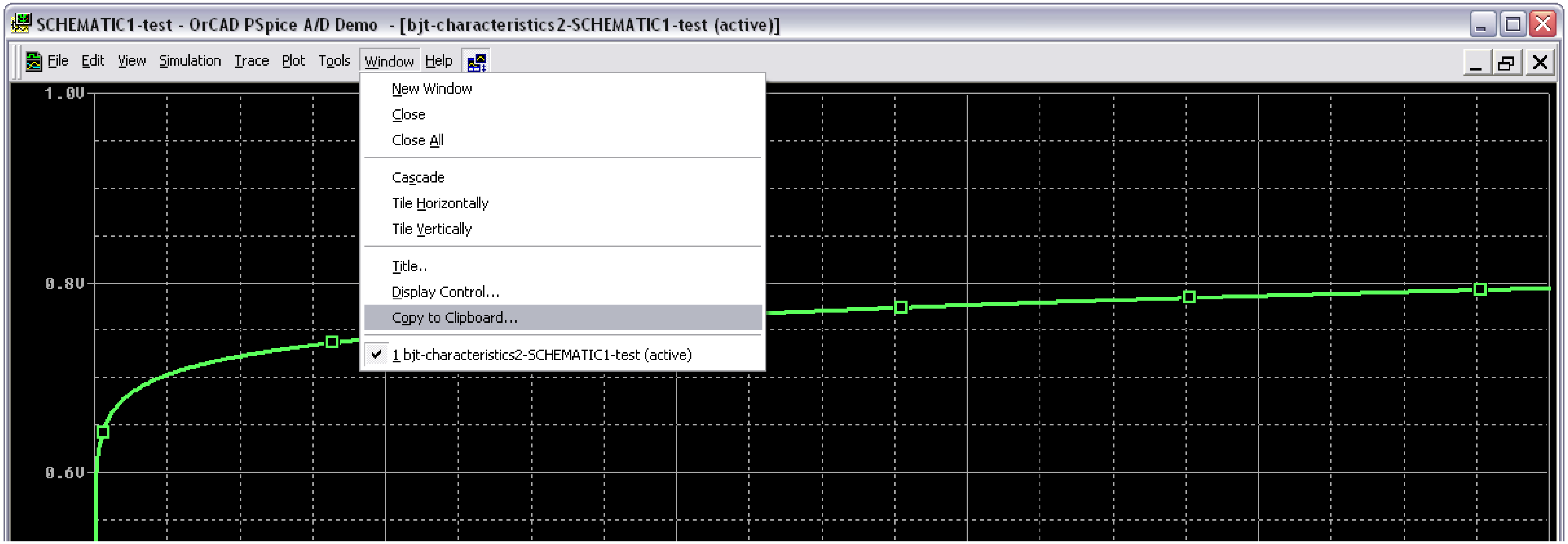
Vbe vs Ibb (Vce = const)

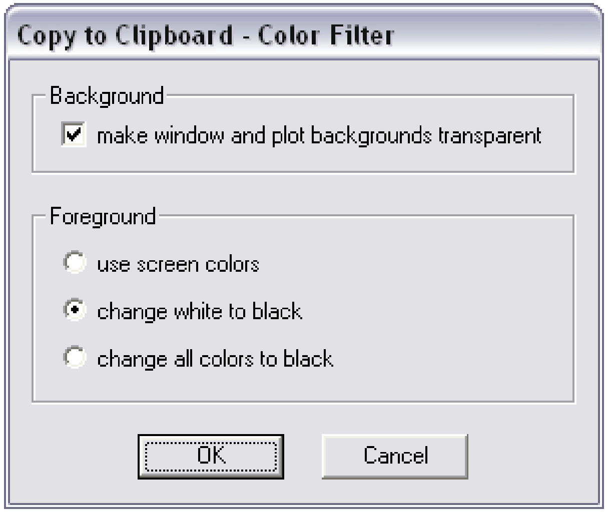


Is it in agreement with our expectations ?

Export the plot image

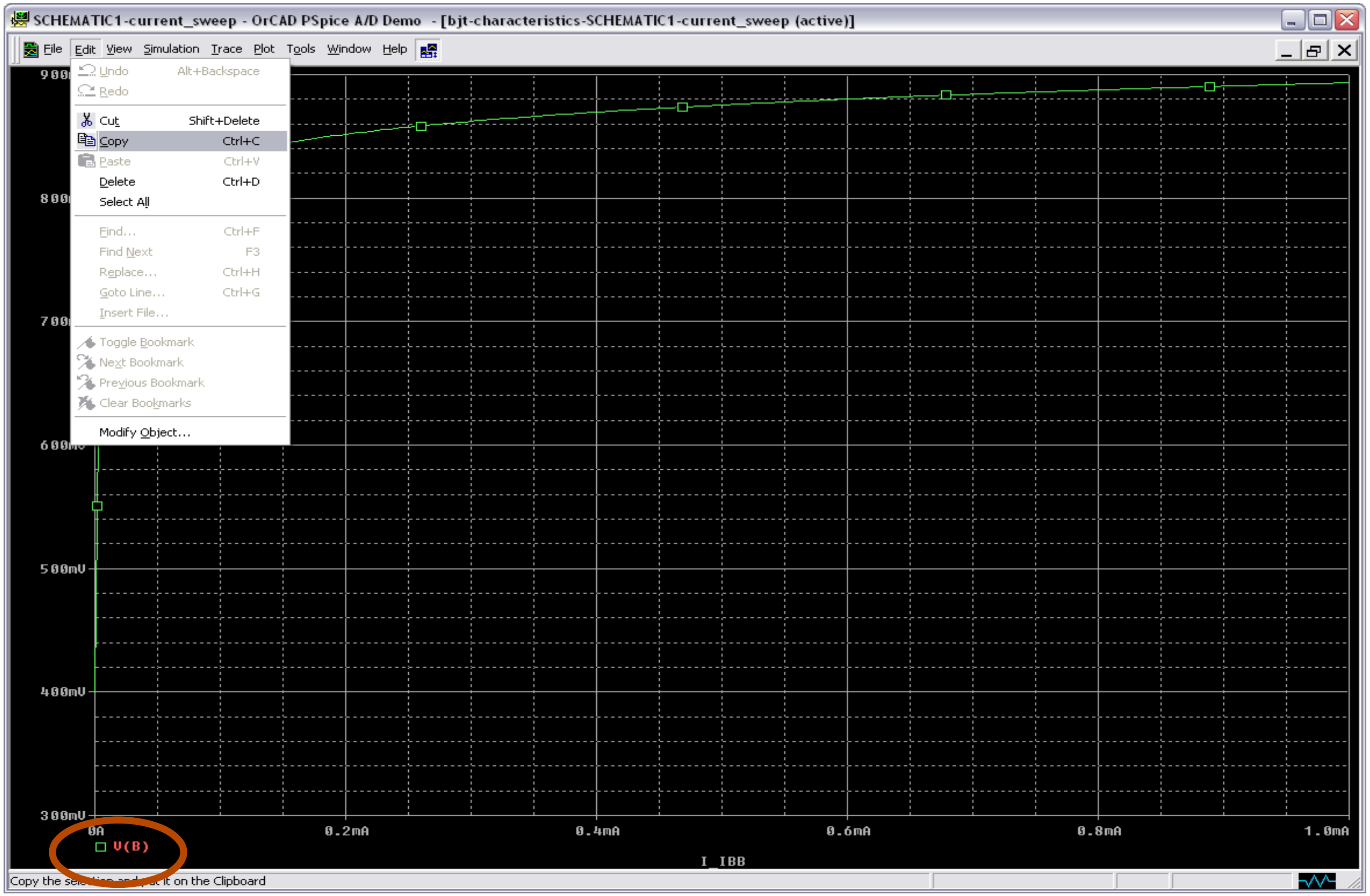
Window → Copy to Clipboard...



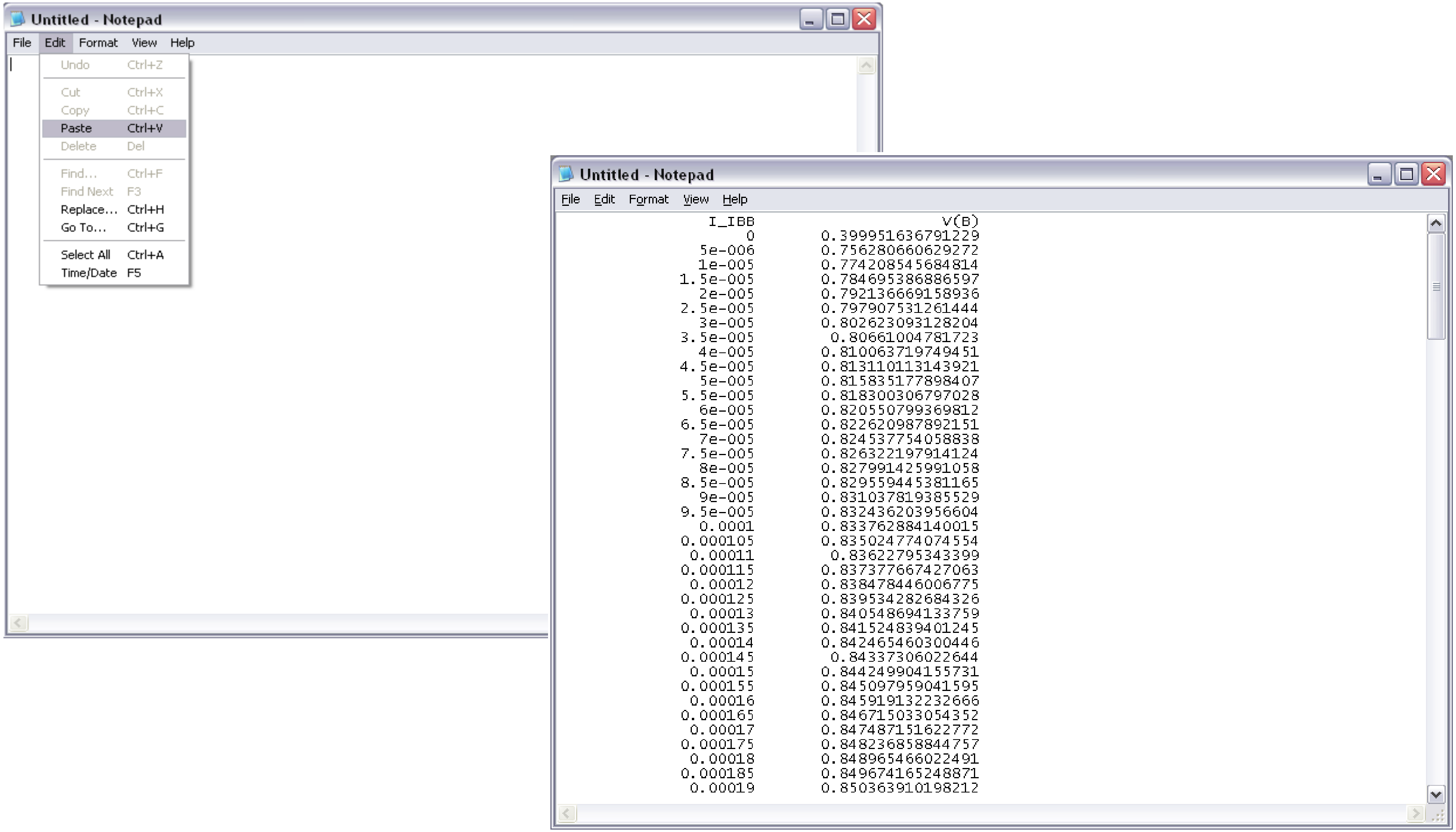


*Choose the color scheme, then open your favorite image editor (**Paint** works fine) and simply do a 'paste' inside it*

Export numerical values

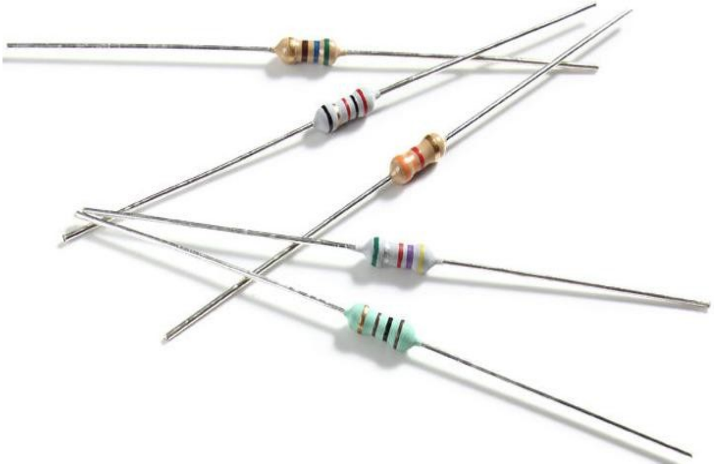


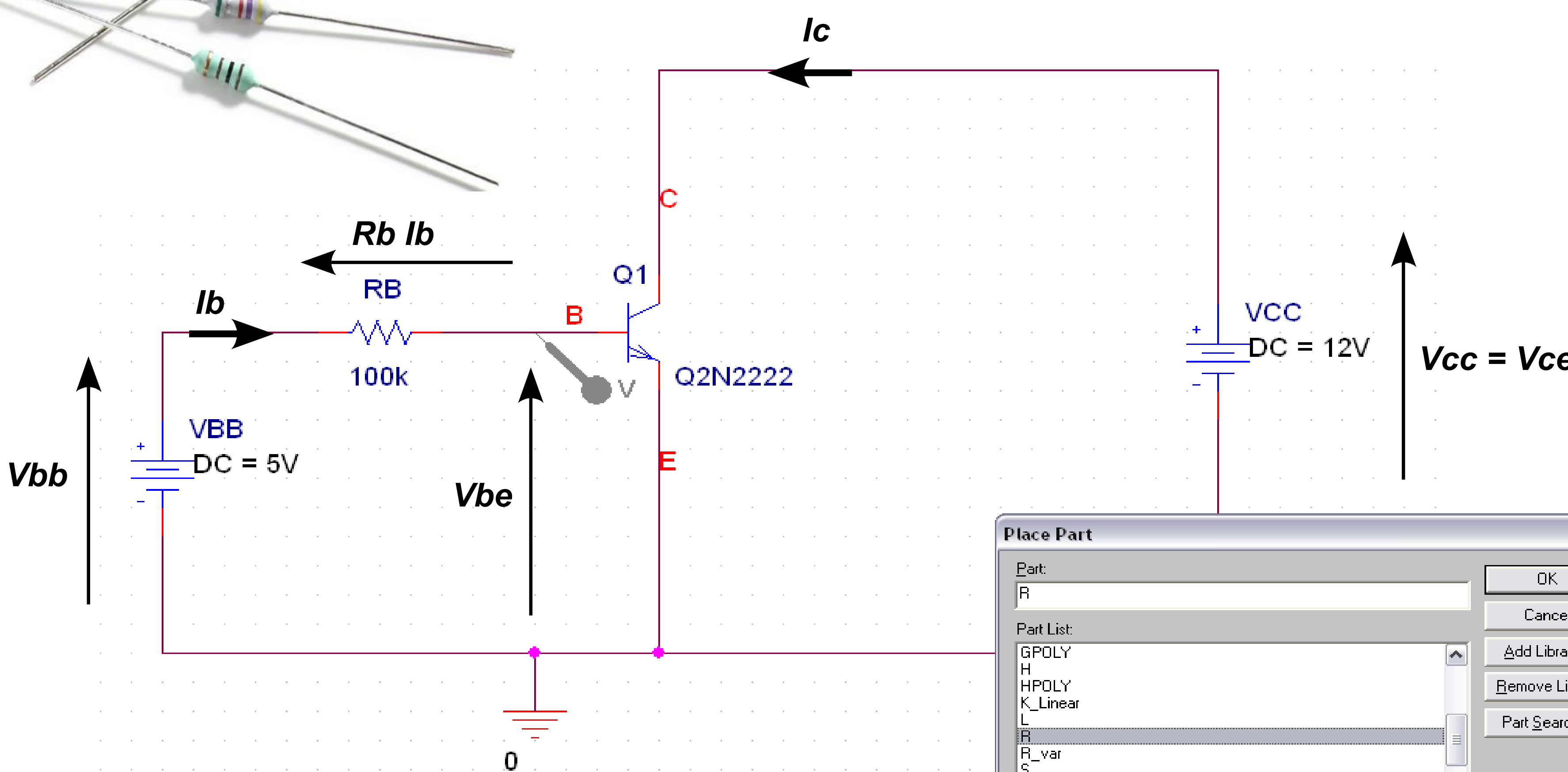
- the waveform can be exported as a set of (x,y) numerical values
- select the waveform name at the bottom-left of the window, then ***Edit* → *Copy***

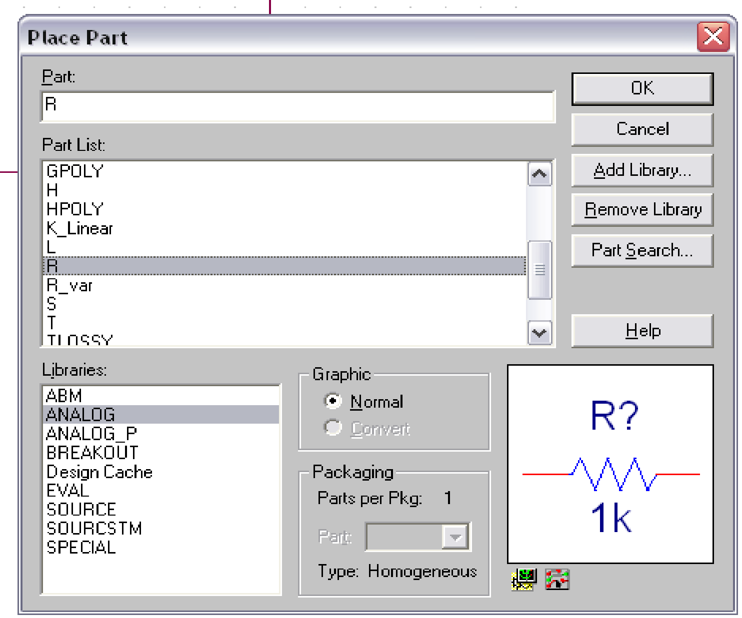


- open a text editor (***Notepad*** works fine)
- simply do a 'paste' inside the text editor
- save the file with the ***.csv*** extension, it can be opened with Excel, Mathematica, ROOT etc.

Welcome to the real world...

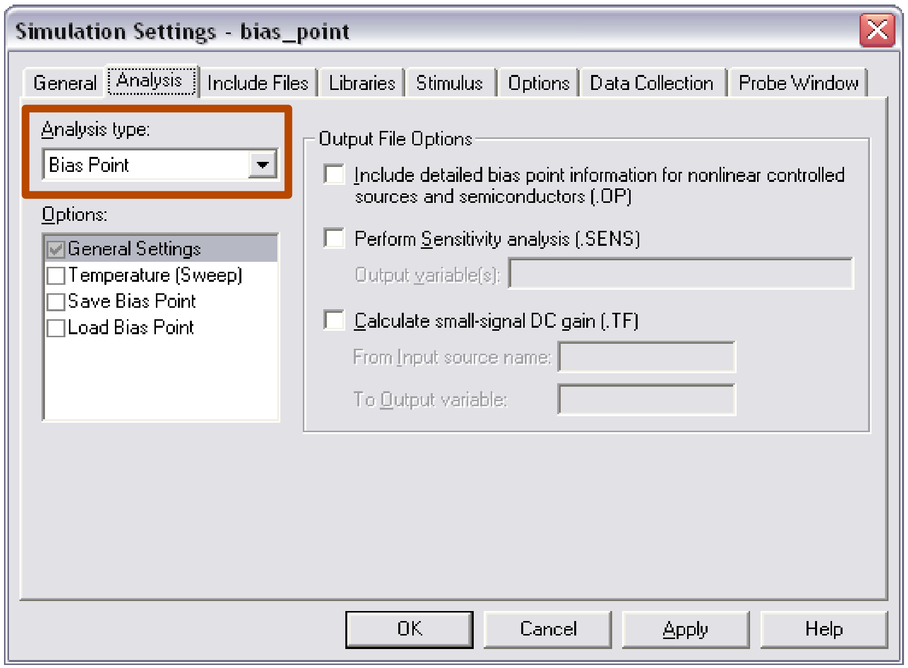


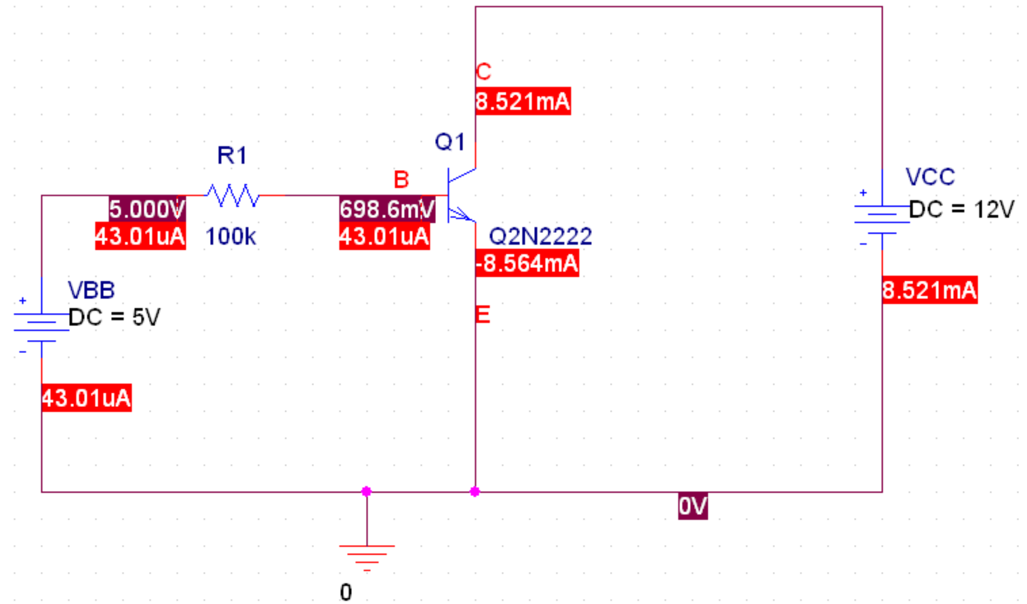


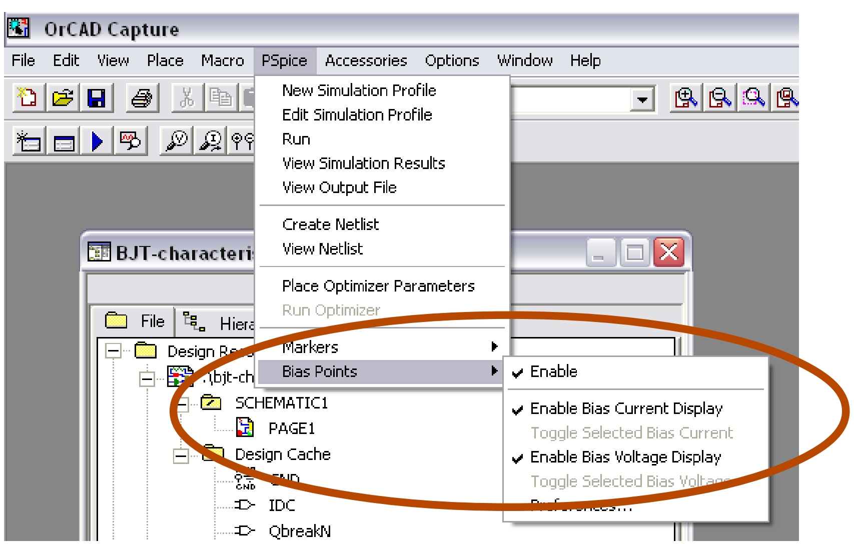


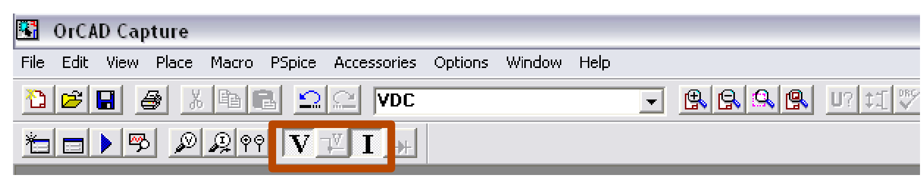
This is the circuit you have to deal with in the ***lab experience #8*** *!*

Bias point analysis



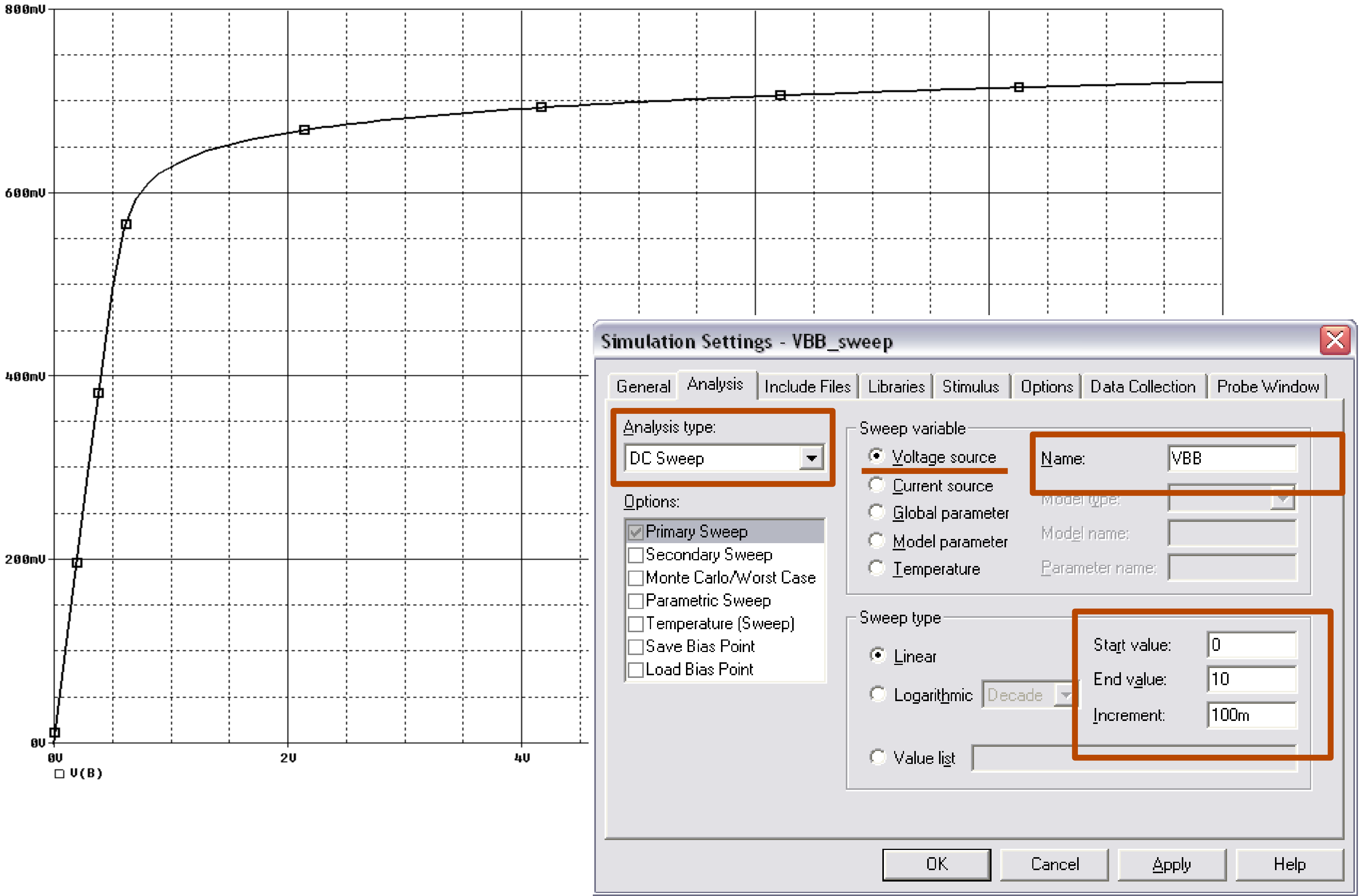






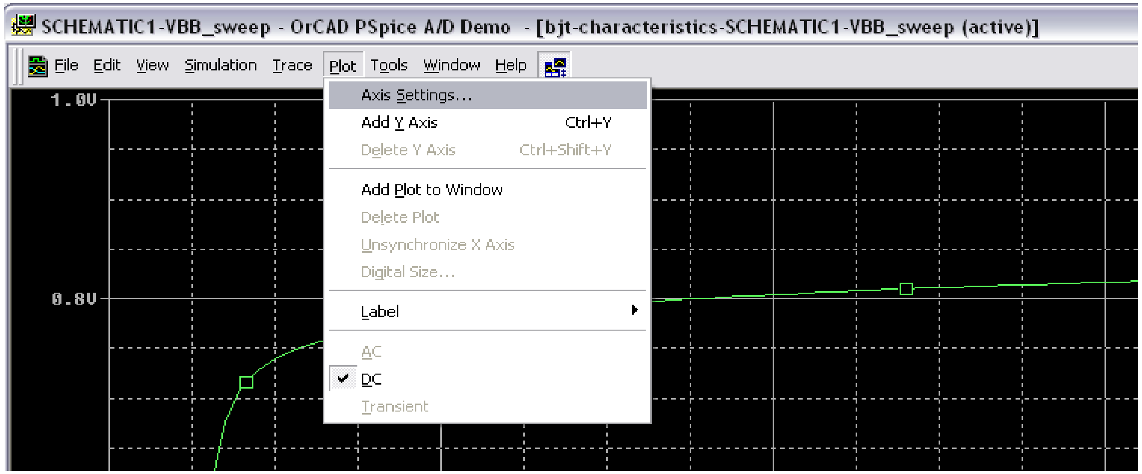
PSpice → Bias Points → Enable Bias Current (Voltage) Display

Vbe vs Vbb (Vce = const)

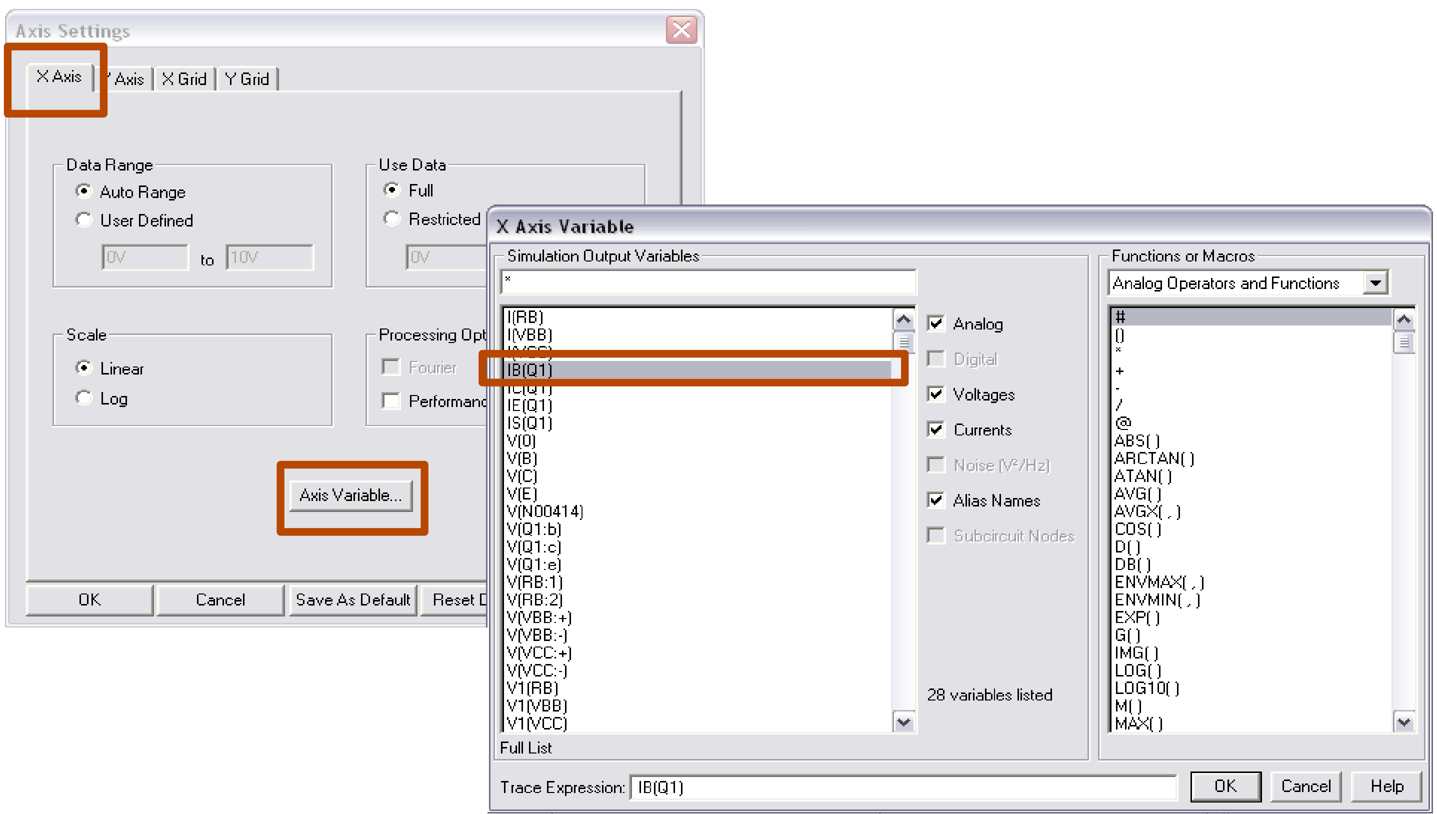


Vbe vs Ib (Vce = const)

Plot → Axis Settings...



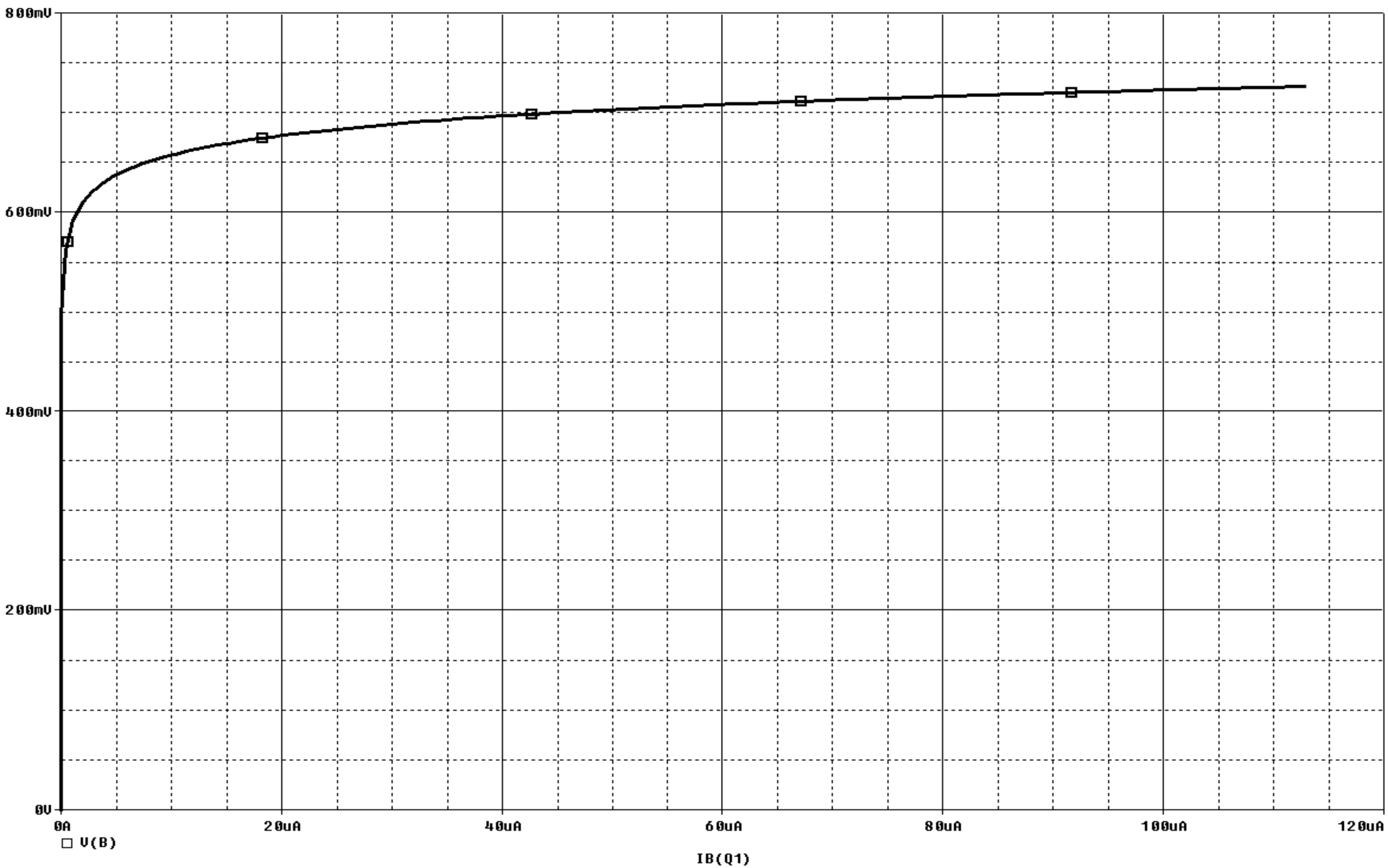
Change the x-axis variable in order to plot Vbe versus the base-current again



- **IB(Q1)**
- **IC(Q1)**
- **IE(Q1)**

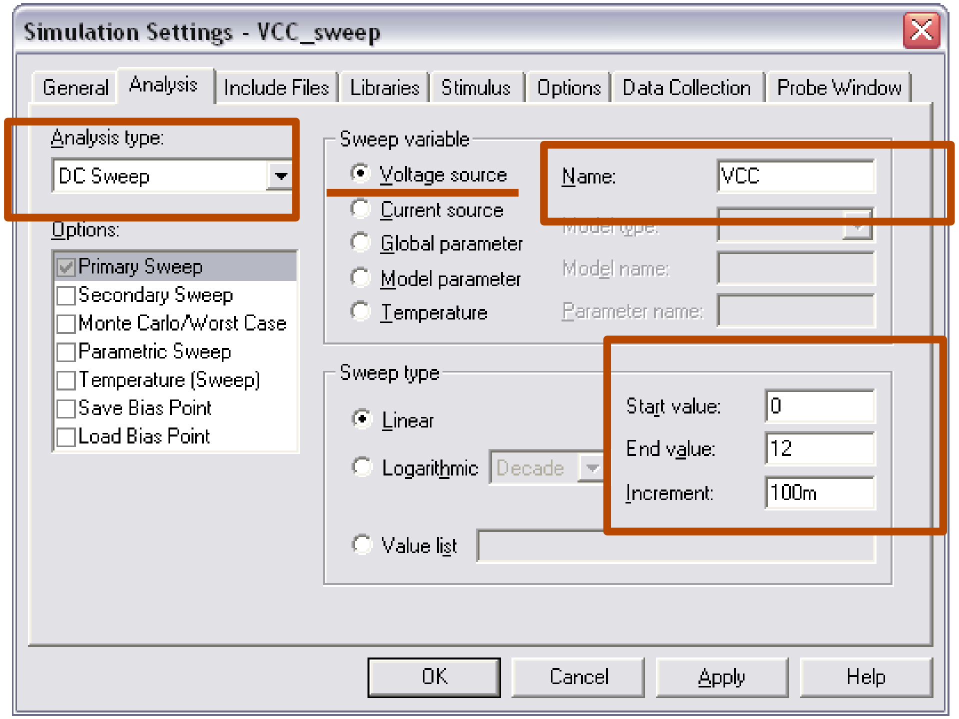
Vbe vs Ib (Vce = const)

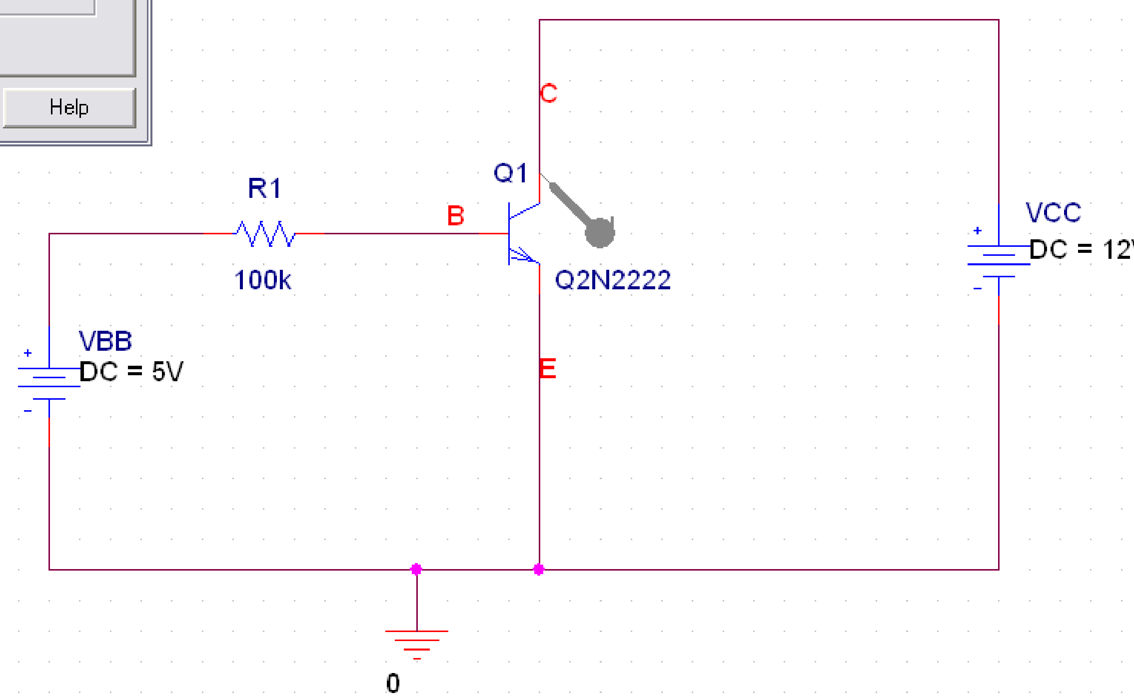
Input characteristic, $Vbe = f(Ib)$



When Vbe reaches ~700 mV the transistor is in full conduction regime !

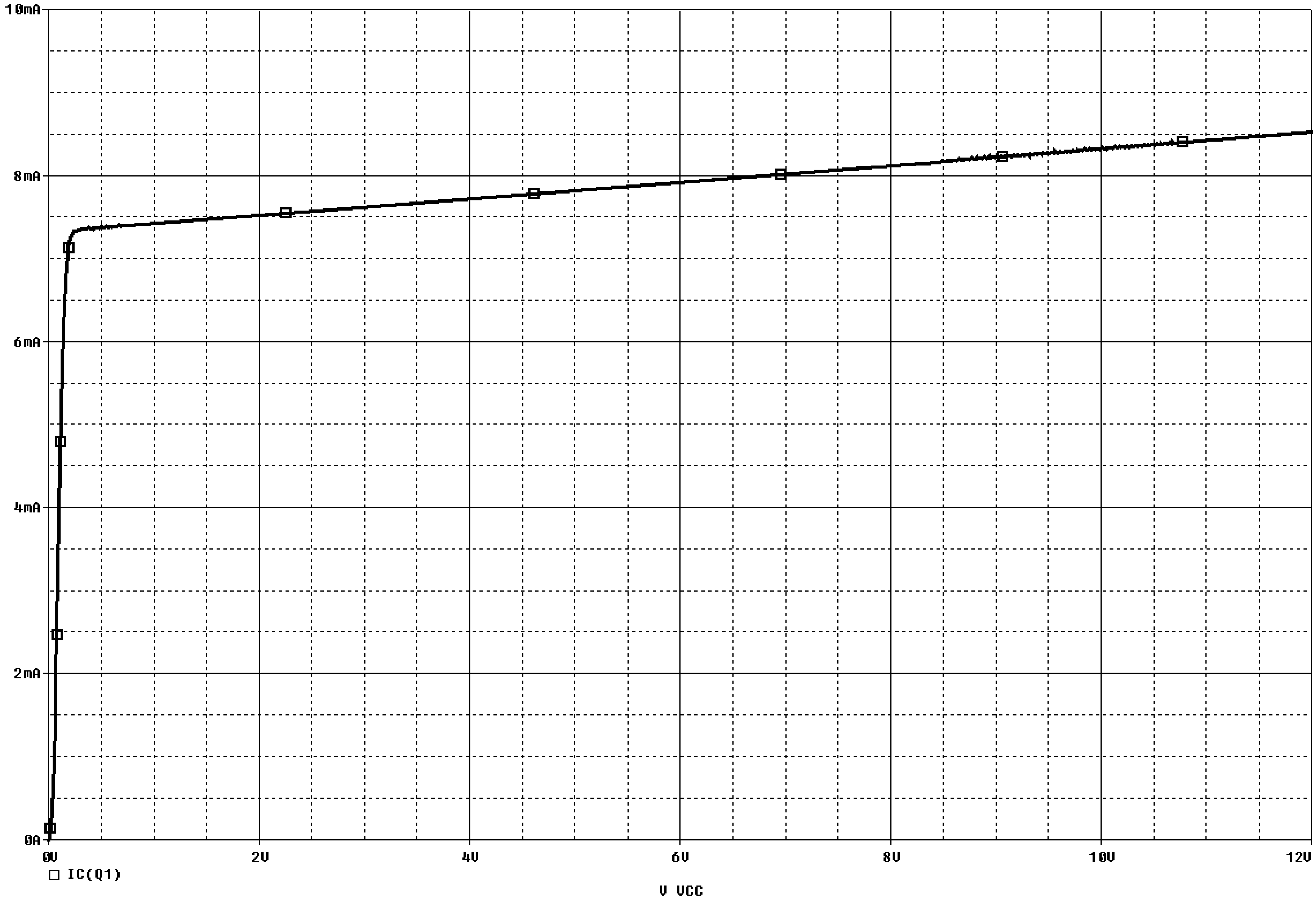
Ic vs Vce (Vbe = const)



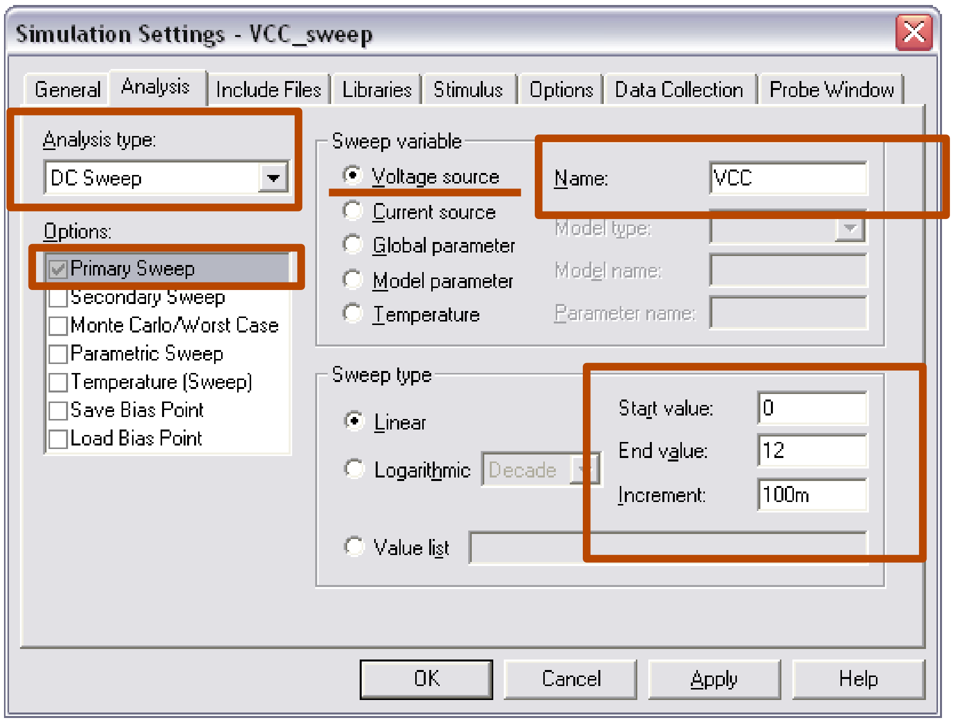


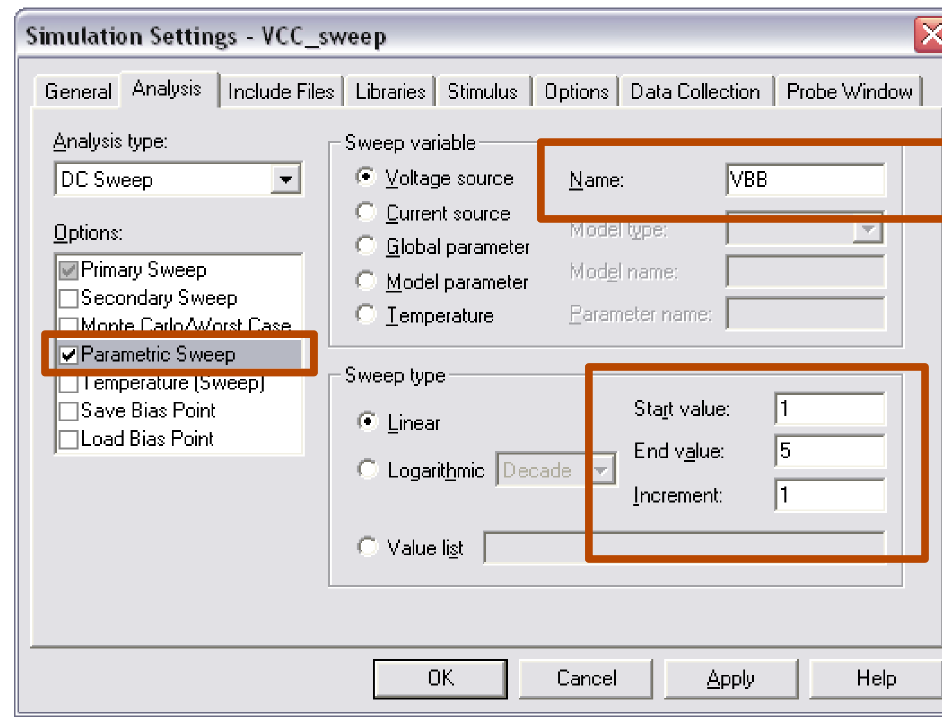
Ic vs Vce (Vbe = const)

Output characteristic, $Ic = f(Vce)$



Nested DC sweeps



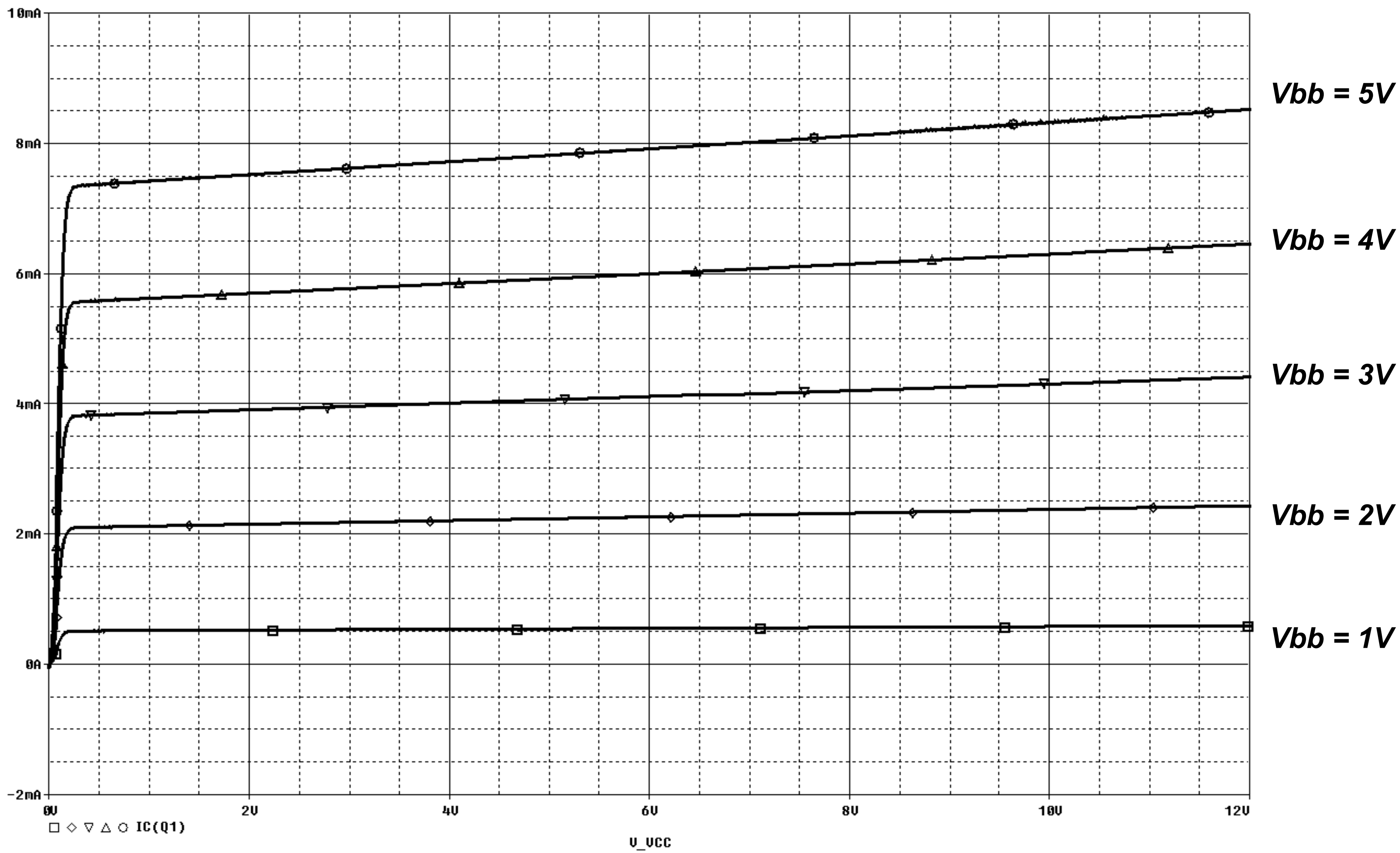


***Primary sweep** by varying Vcc for a fixed value of Vbb (and therefore of Vbe)*

*The primary sweep **is repeated** for different values of Vbb (and therefore of Vbe)*

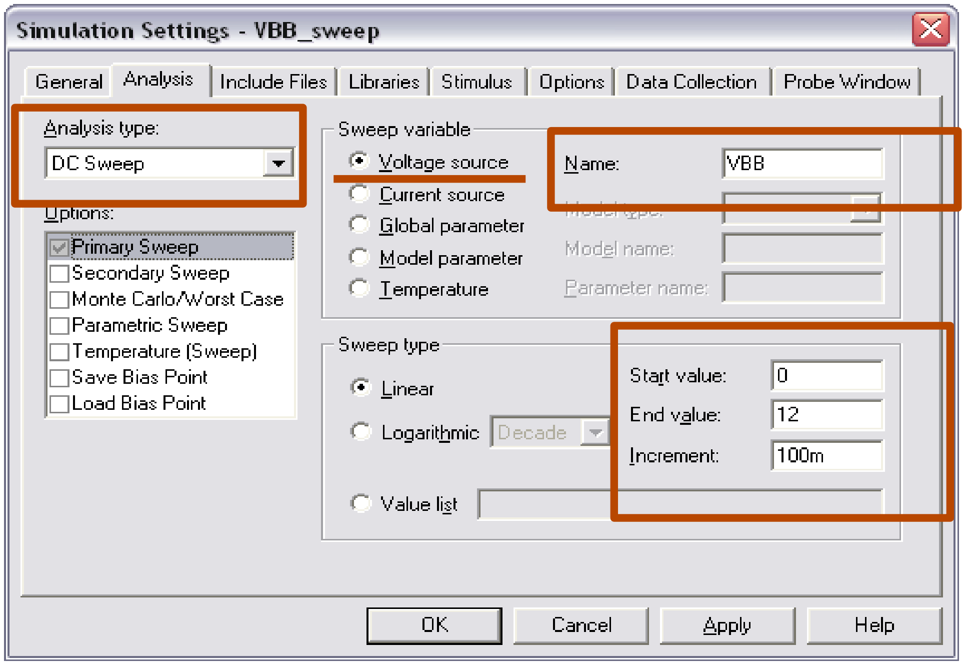
N.B.
The simulation profile is just one, don't create two different profiles !

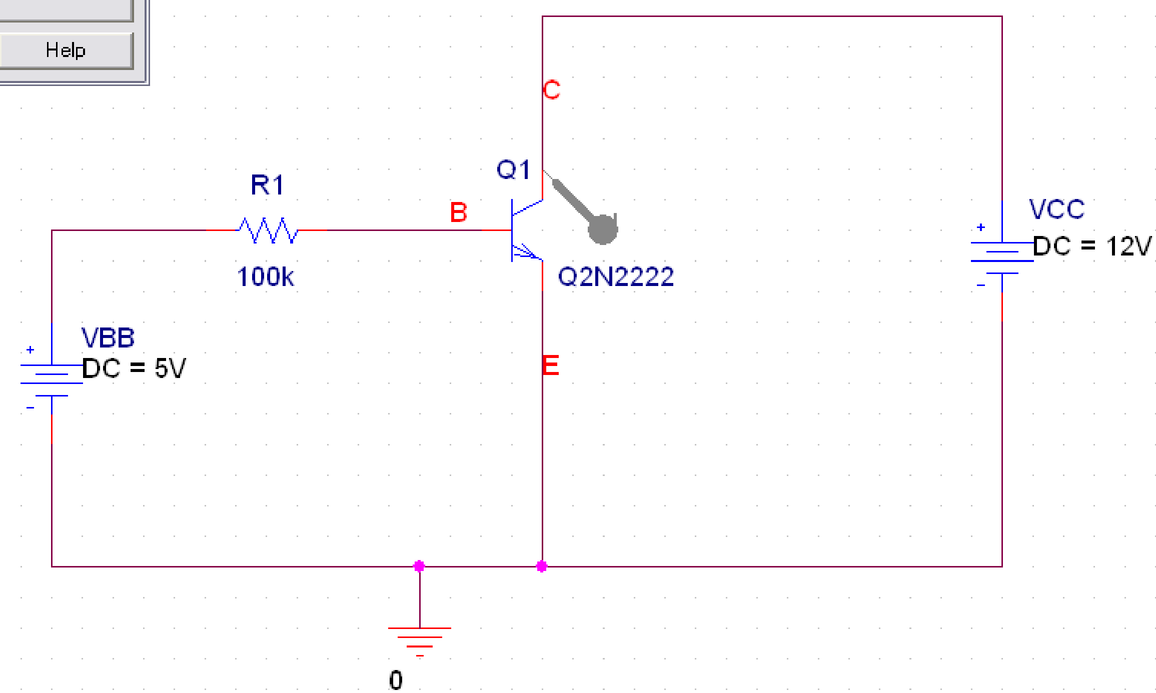
Ic vs Vce for different Vbb



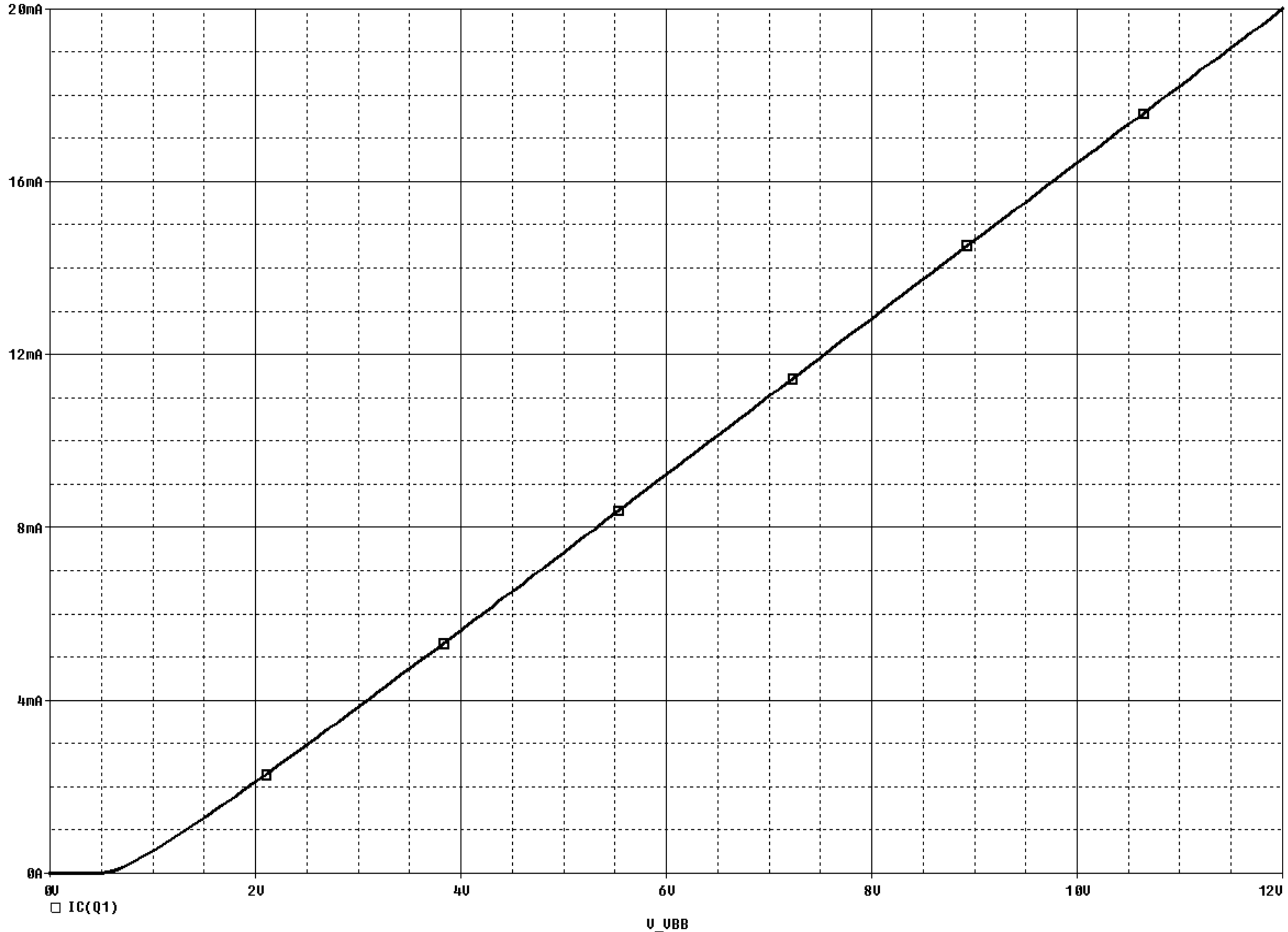
What's happening?

Ic vs Vbb (Vce = const)



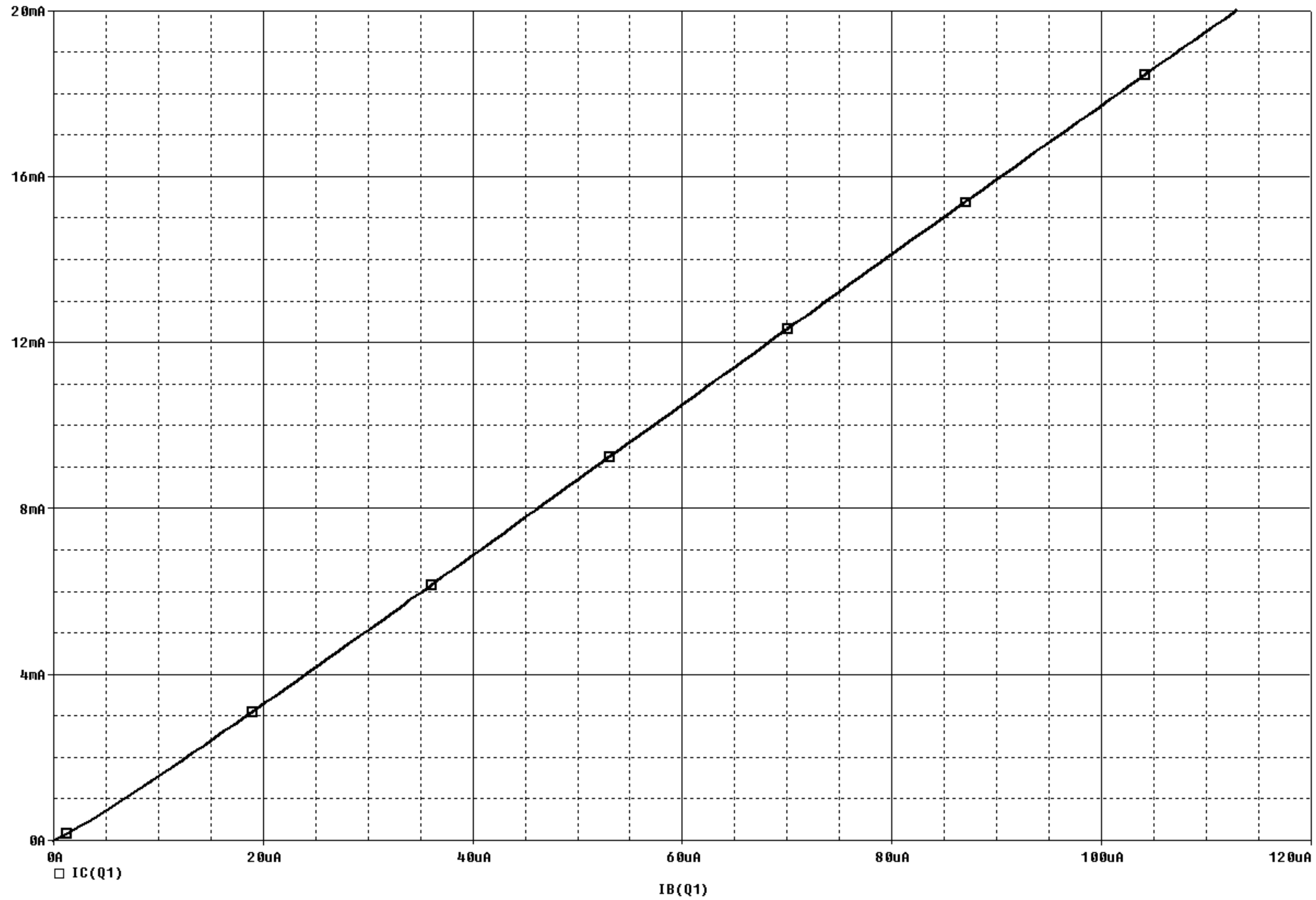


Ic vs Vbb (Vce = const)



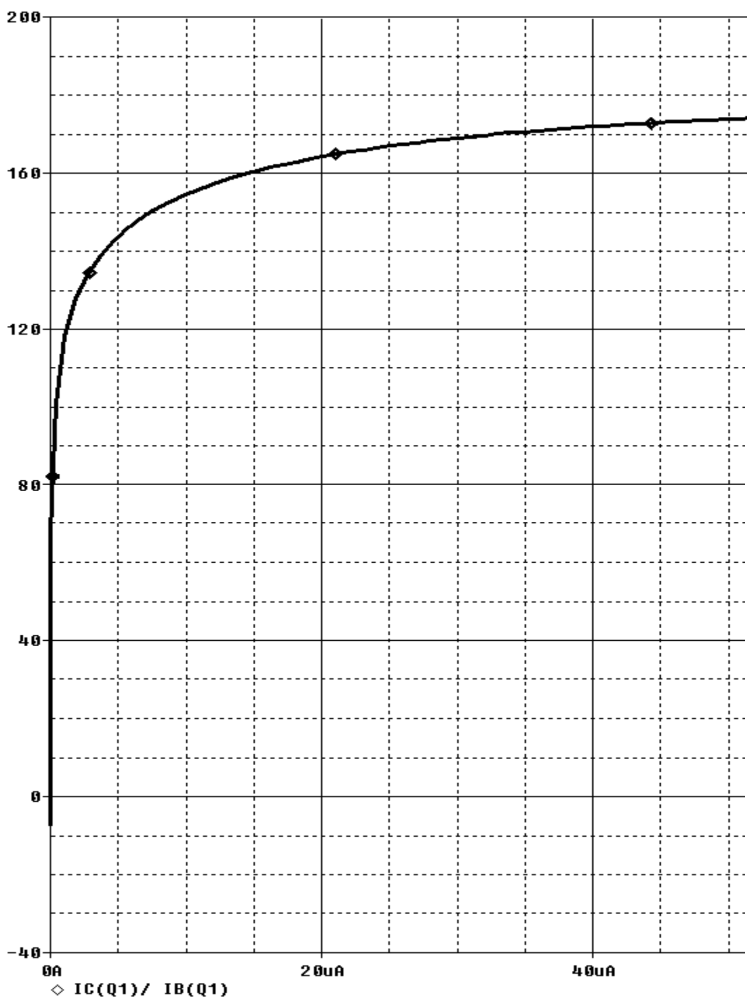
Ic vs Ib (Vce = const)

Change the x-axis variable....



*The relationship between Ic and Ib is **linear** !*

h_{FE} = Ic/Ib vs Ib (Vce = const)



Philips Semiconductors | Product specification

NPN switching transistors — 2N2222; 2N2222A

FEATURES

- High current (max. 800 mA)
- Low voltage (max. 40 V).

APPLICATIONS

- Linear amplification and switching.

DESCRIPTION

NPN switching transistor in a TO-18 metal package.
PNP complement: 2N2907A.

PINNING

PIN	DESCRIPTION
1	emitter
2	base
3	collector, connected to case

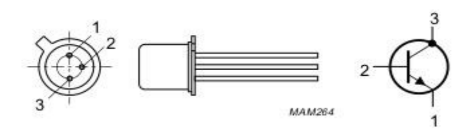
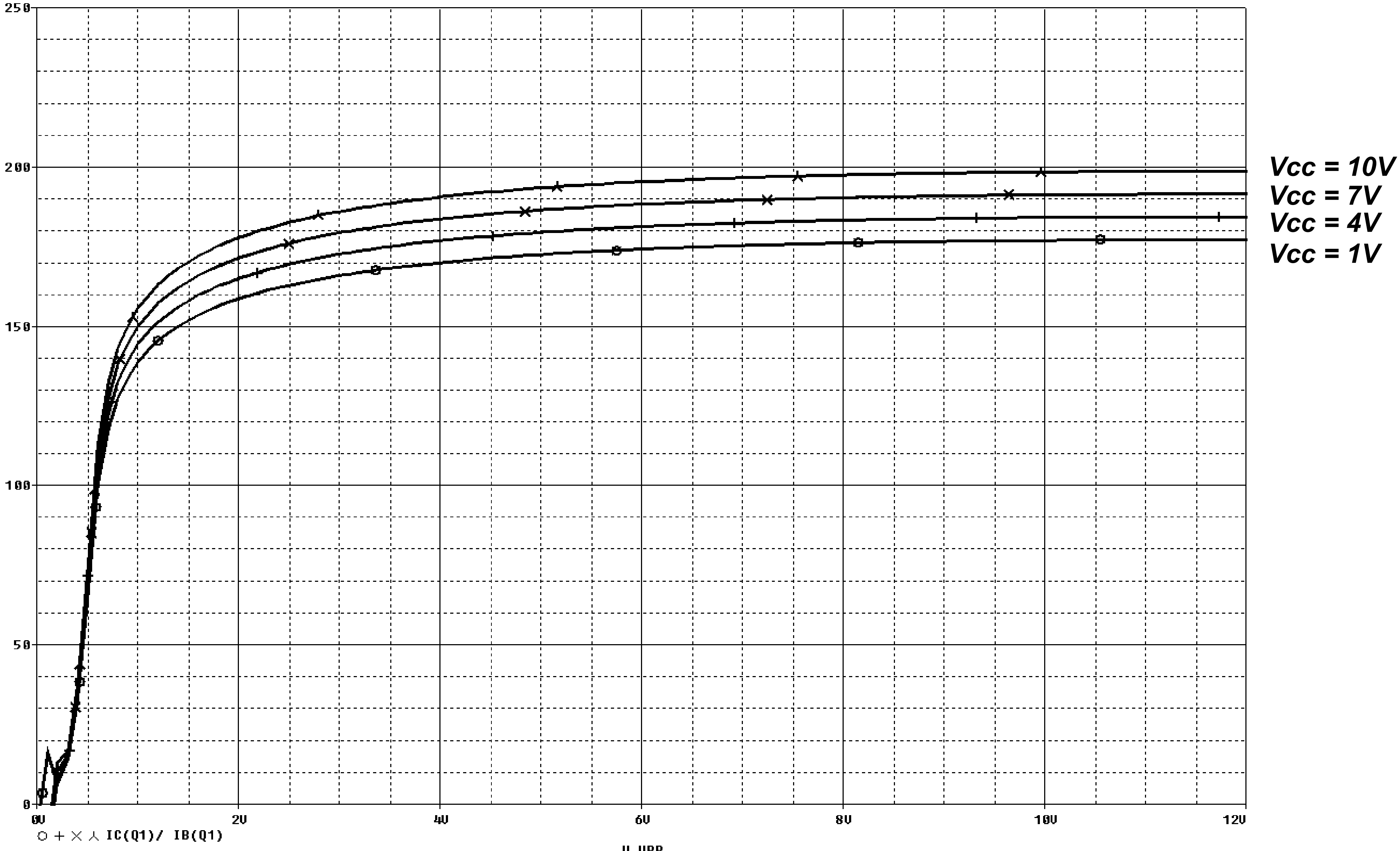


Fig.1 Simplified outline (TO-18) and symbol.

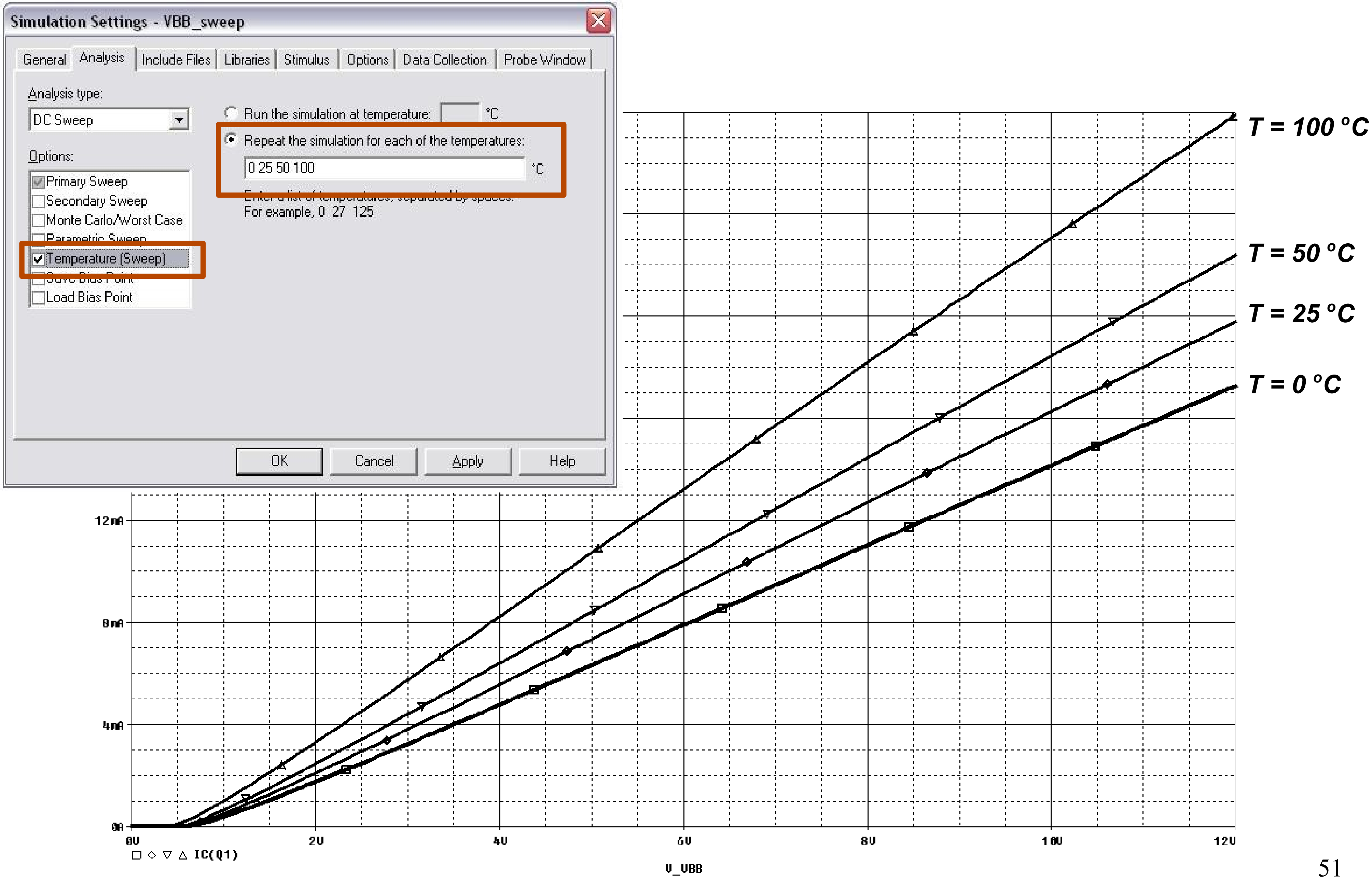
QUICK REFERENCE DATA

SYMBOL	PARAMETER	CONDITIONS	MIN.	MAX.	UNIT
V_{CBO}	collector-base voltage	open emitter			
	2N2222		–	60	V
	2N2222A		–	75	V
V_{CEO}	collector-emitter voltage	open base			
	2N2222		–	30	V
	2N2222A		–	40	V
I_C	collector current (DC)		–	800	mA
P_{tot}	total power dissipation	$T_{amb} \leq 25\,°C$	–	500	mW
h_{FE}	DC current gain	I_C = 10 mA; V_{CE} = 10 V	75	–	
f_T	transition frequency	I_C = 20 mA; V_{CE} = 20 V; f = 100 MHz			
	2N2222		250	–	MHz
	2N2222A		300	–	MHz
t_{off}	turn-off time	I_{Con} = 150 mA; I_{Bon} = 15 mA; I_{Boff} = –15 mA	–	250	ns

h_{FE} = Ic/Ib vs Vbb (Vce sweep)

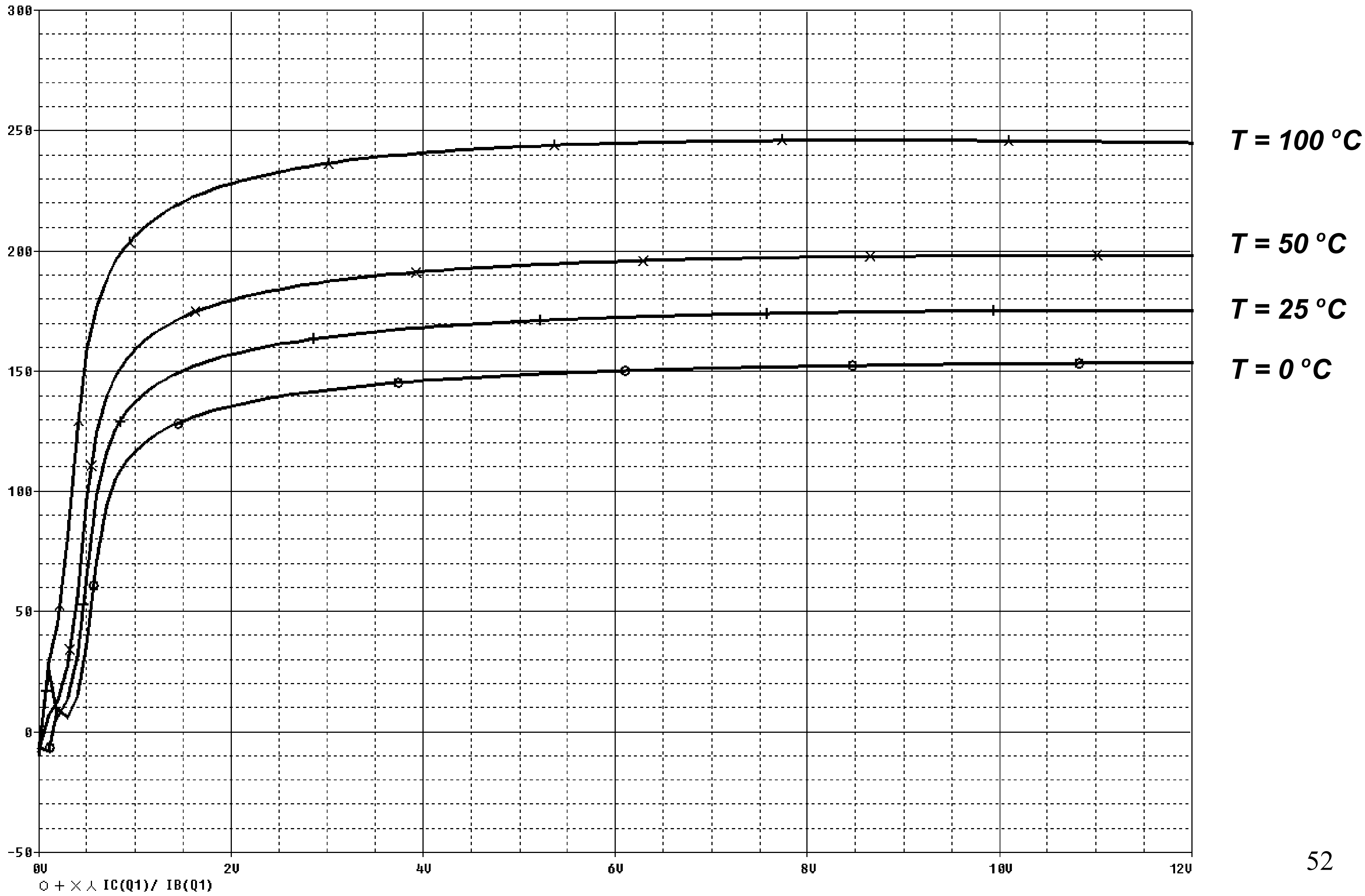


Temperature sweep



Temperature sweep

h_{FE} *increases with the temperature... does it make sense?*



Lab measurements

- ***input characteristics → Ib vs Vbe, Vce = const***

Vce = Vce1		Vce = Vce2	
Ib [μA]	**Vbe [mV]**	**Ib [μA]**	**Vbe [mV]**
…	…	…	…
…	…	…	…

…

…

- ***output characteristics → Ic vs Vce, Vbe = const***

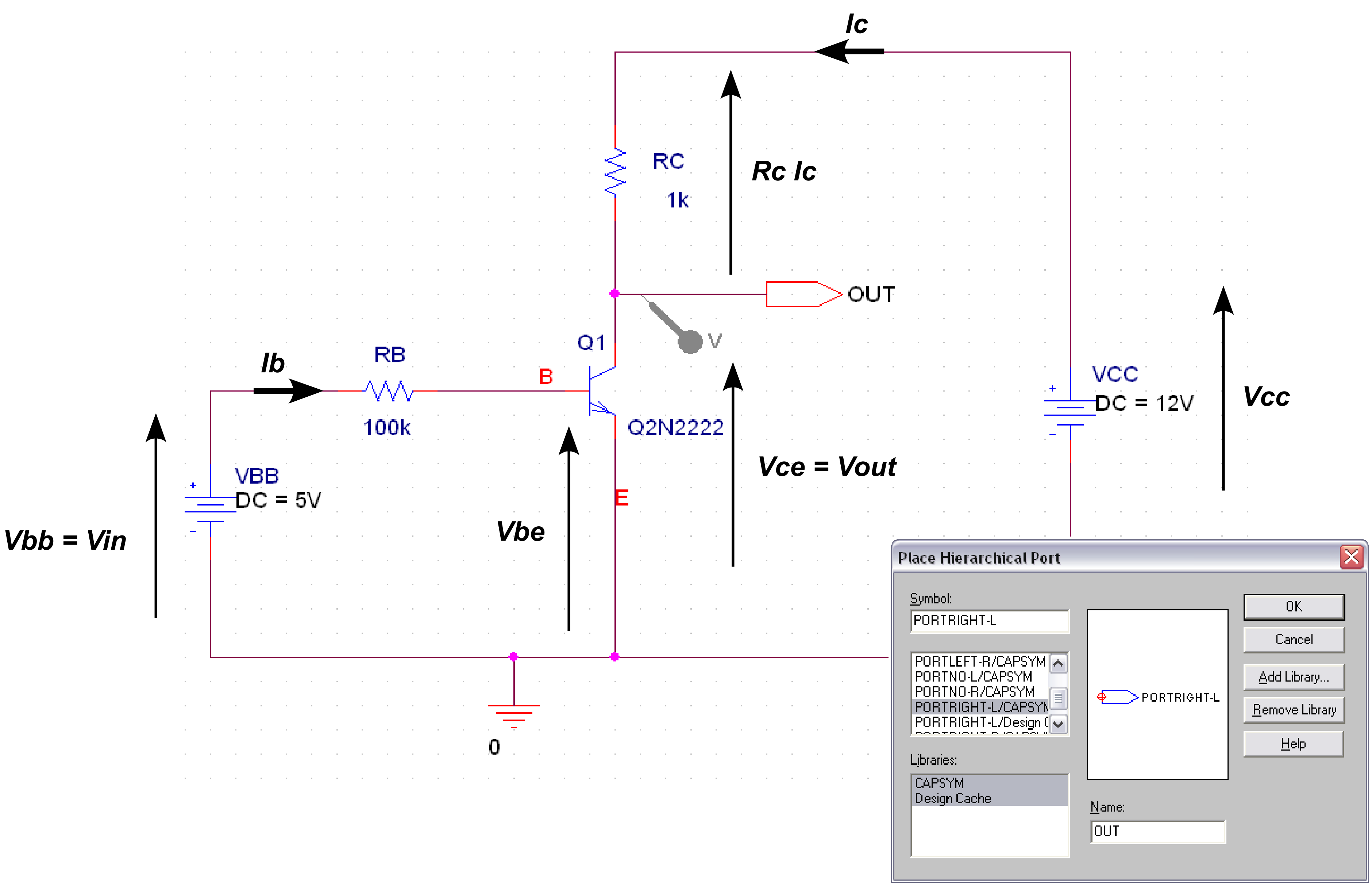
Ib = Ib1		Ib = Ib2	
Ic [μA]	**Vce [mV]**	**Ic [μA]**	**Vce [mV]**
…	…	…	…
…	…	…	…

…

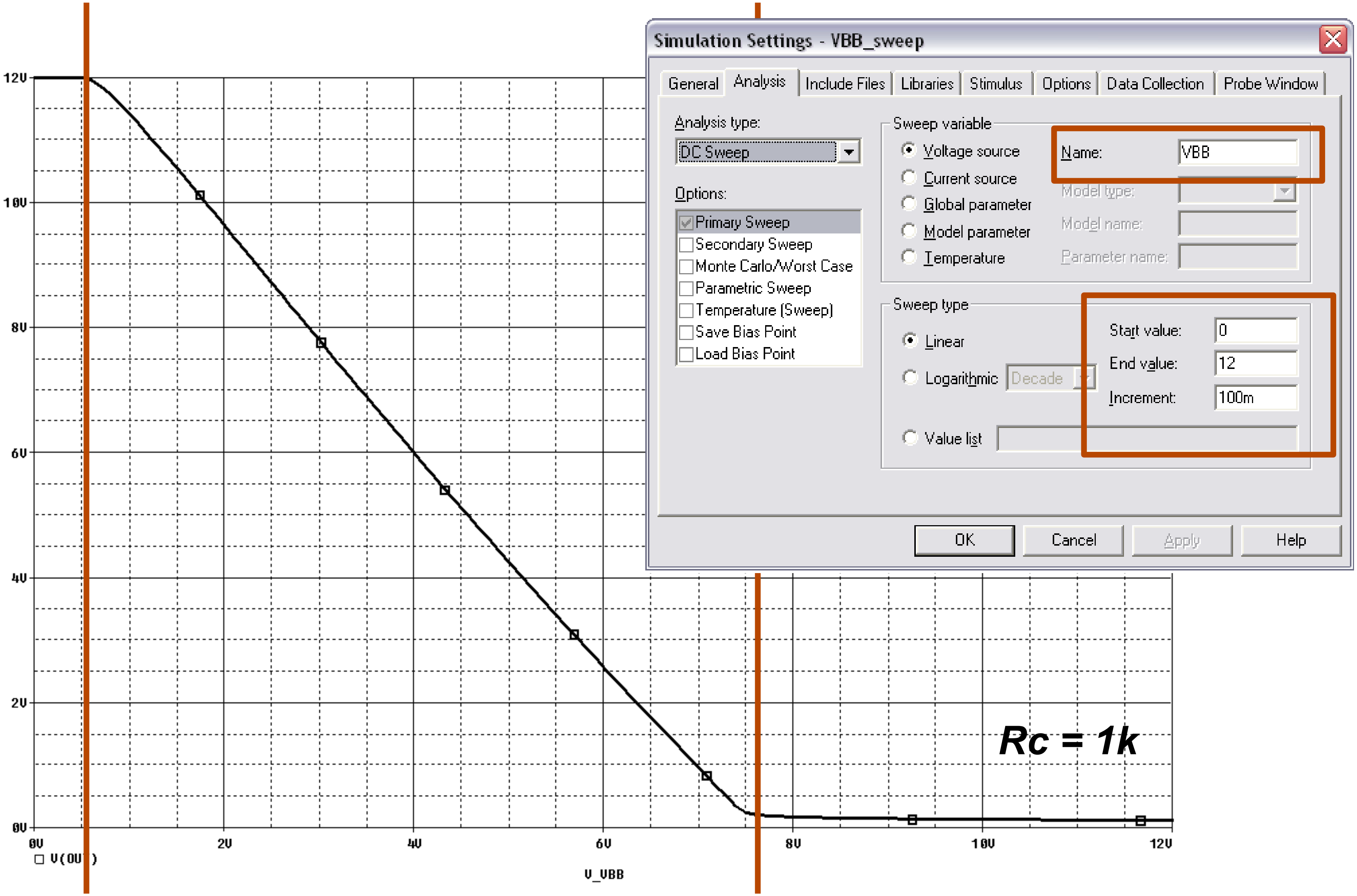
…

Part III
NPN common-emitter amplifier

NPN common-emitter amplifier

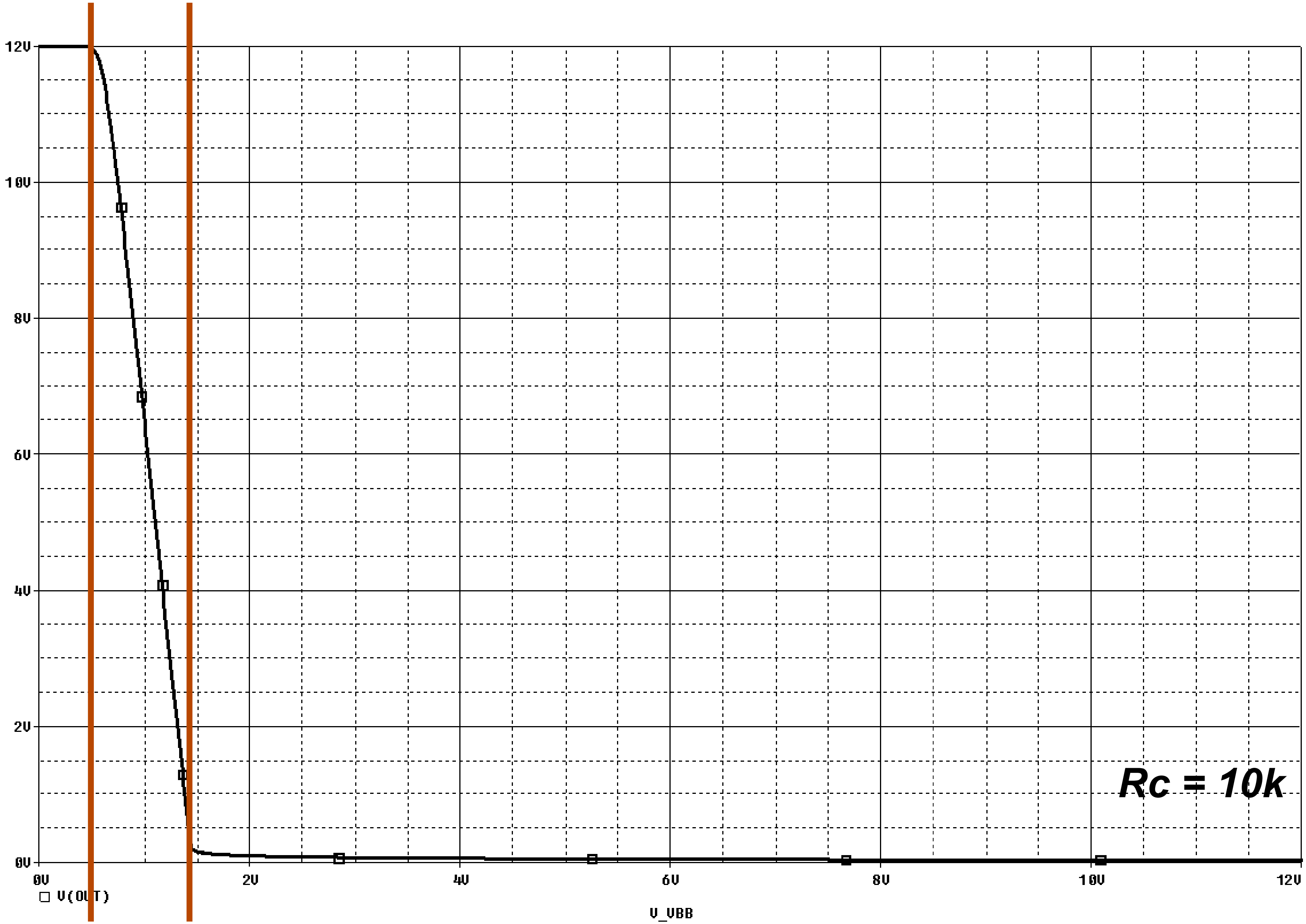


Voltage Transfer Characteristic (VTC)



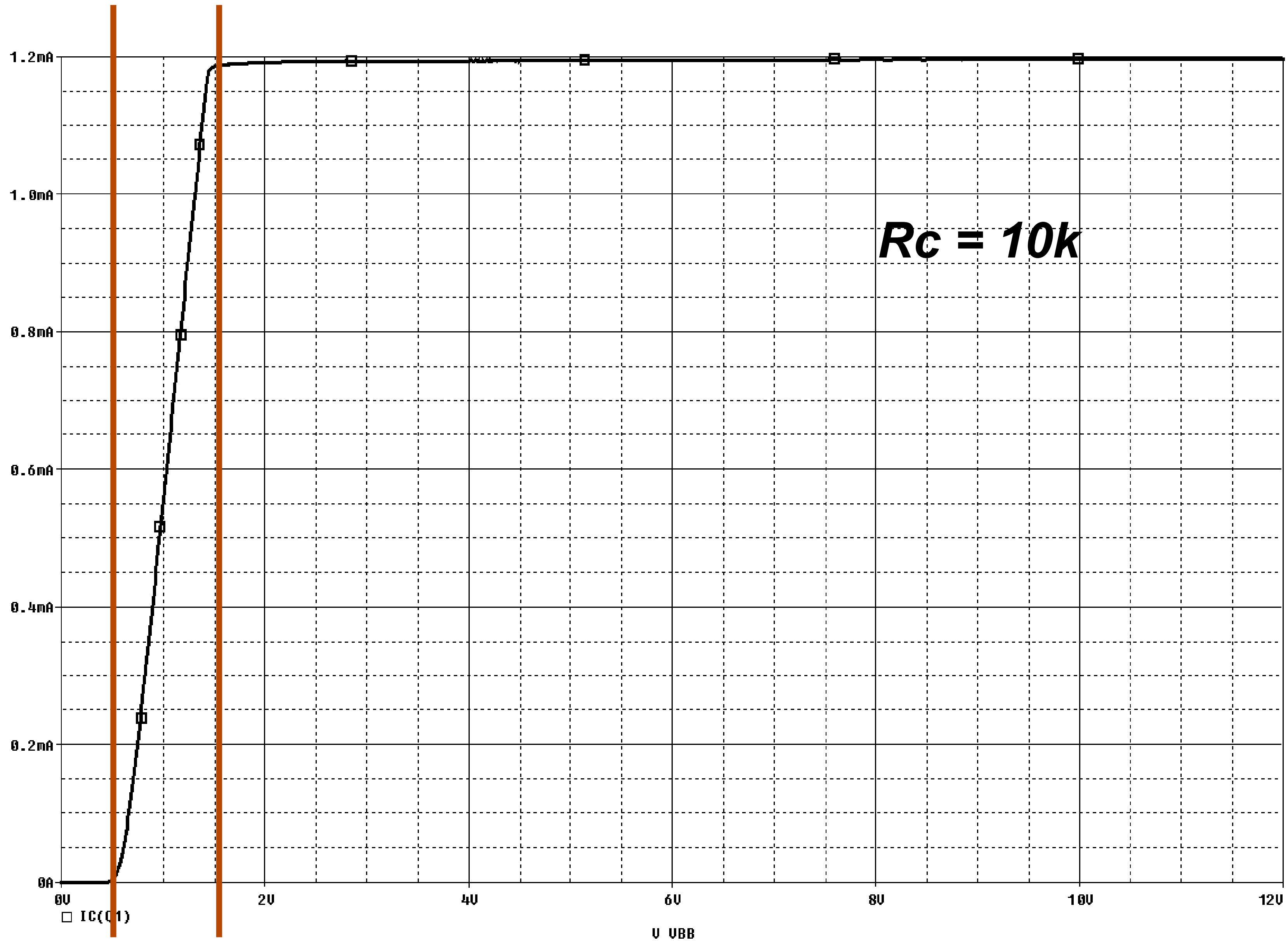
interdiction (cutoff) → active → saturation

Increasing the load



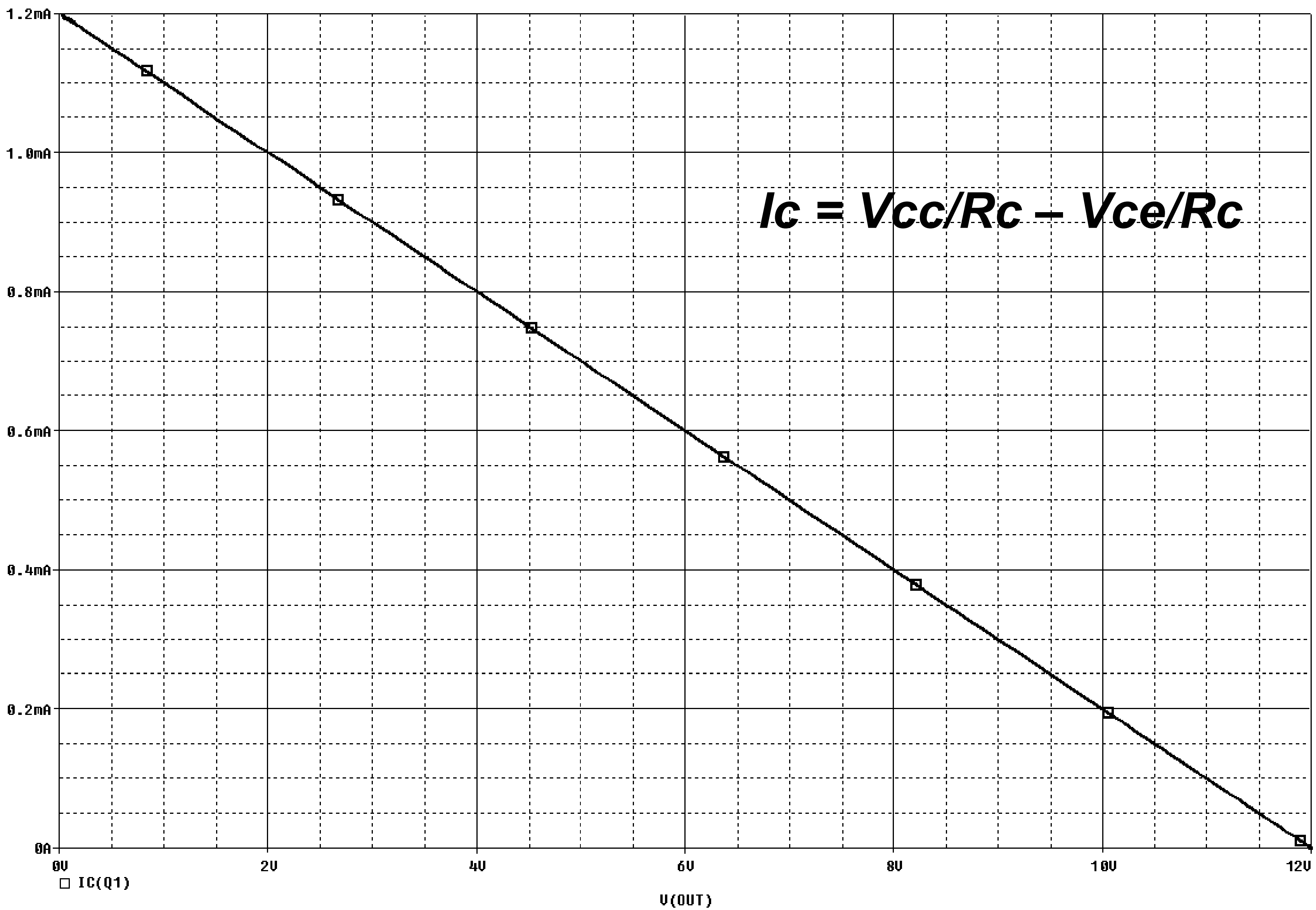
interdiction (cutoff) → active → saturation

Ic vs Vbb

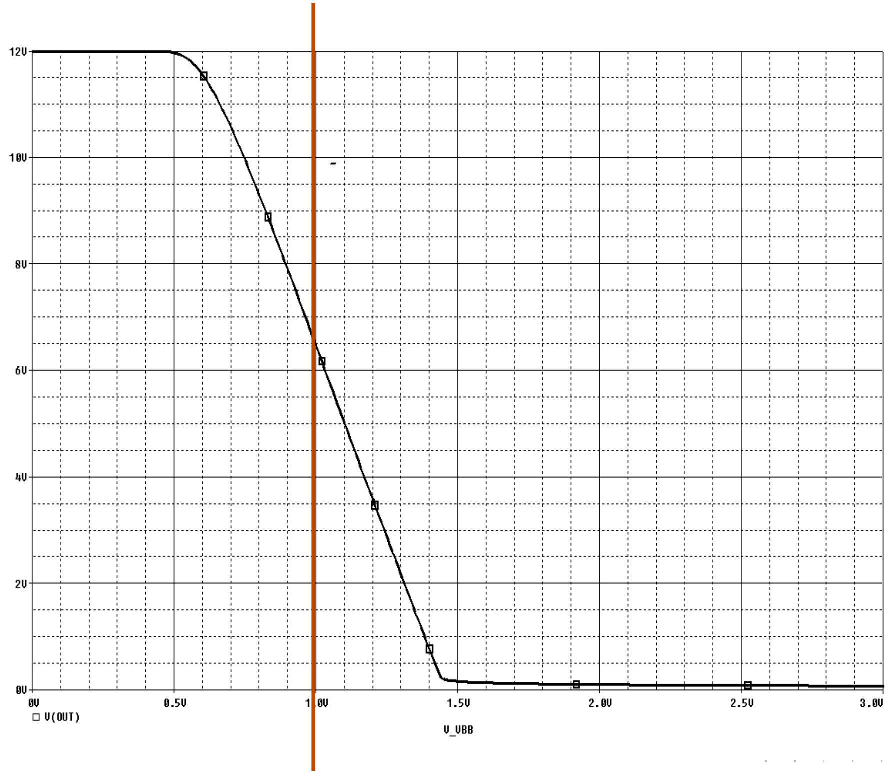


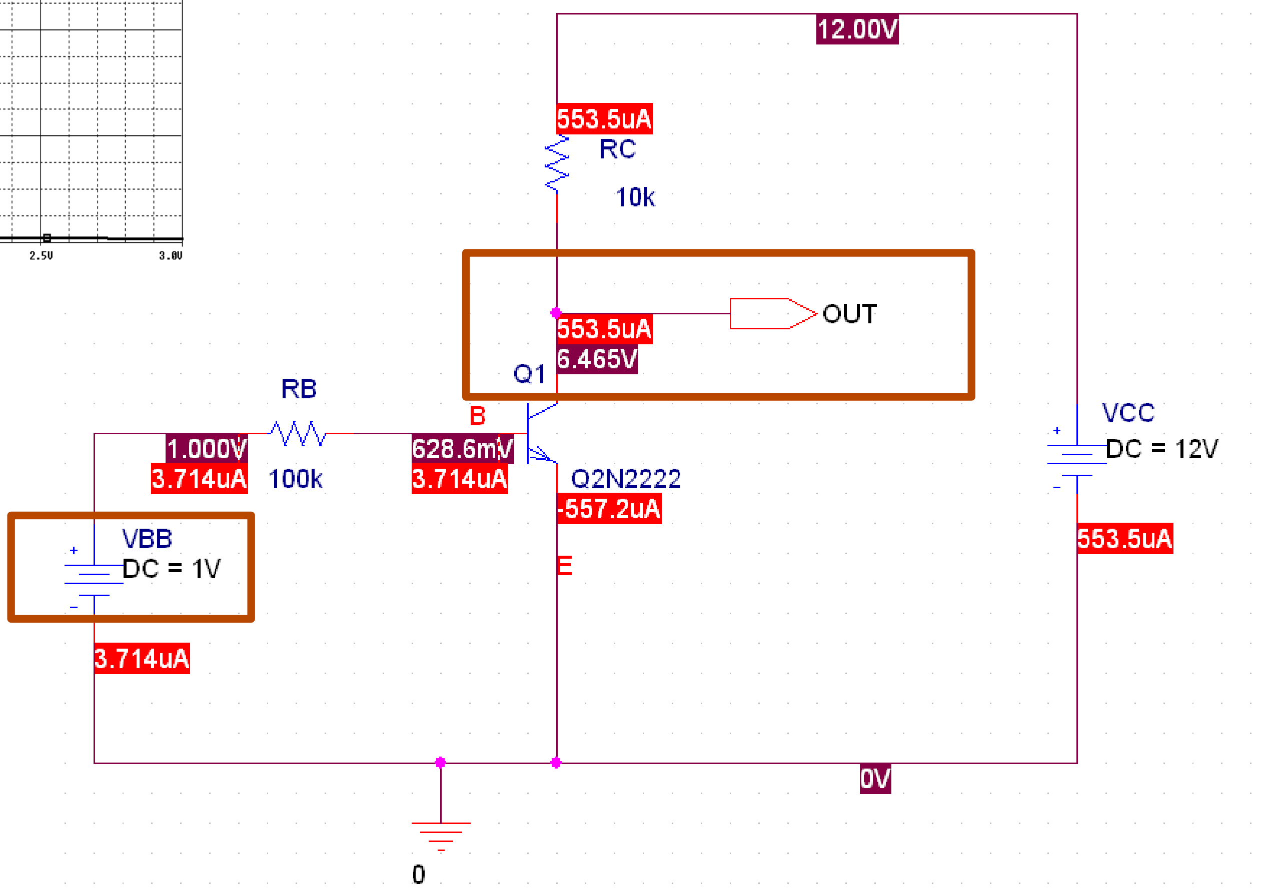
interdiction (cutoff) → active → saturation

Ic vs Vce... the load line !



DC operating point





Vin,DC ≈ 1V → Vout,DC = Vcc/2 = 6 V

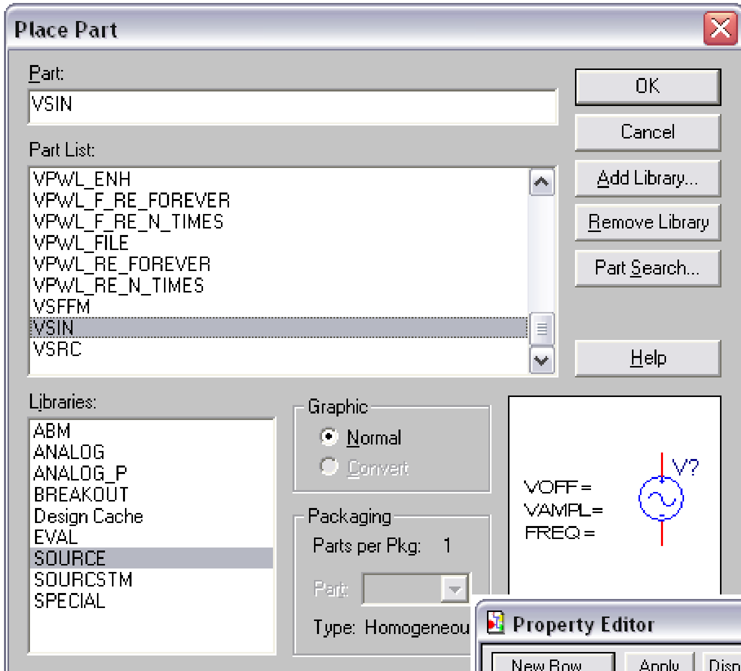
Transient analysis

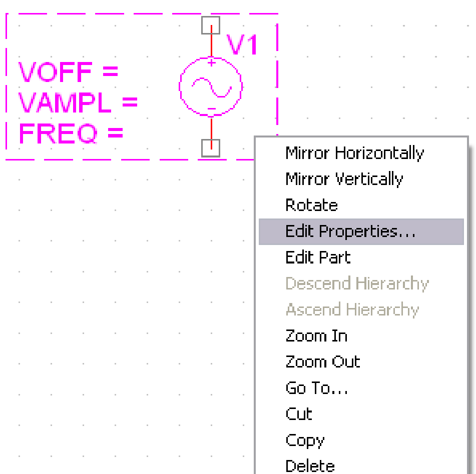
- ***large-signal*** response of the circuit to one or more ***time-dependent inputs***
  - ***numerical integration*** of a non linear differential equations system
  - a first DC analysis determines the ***initial circuit bias conditions***

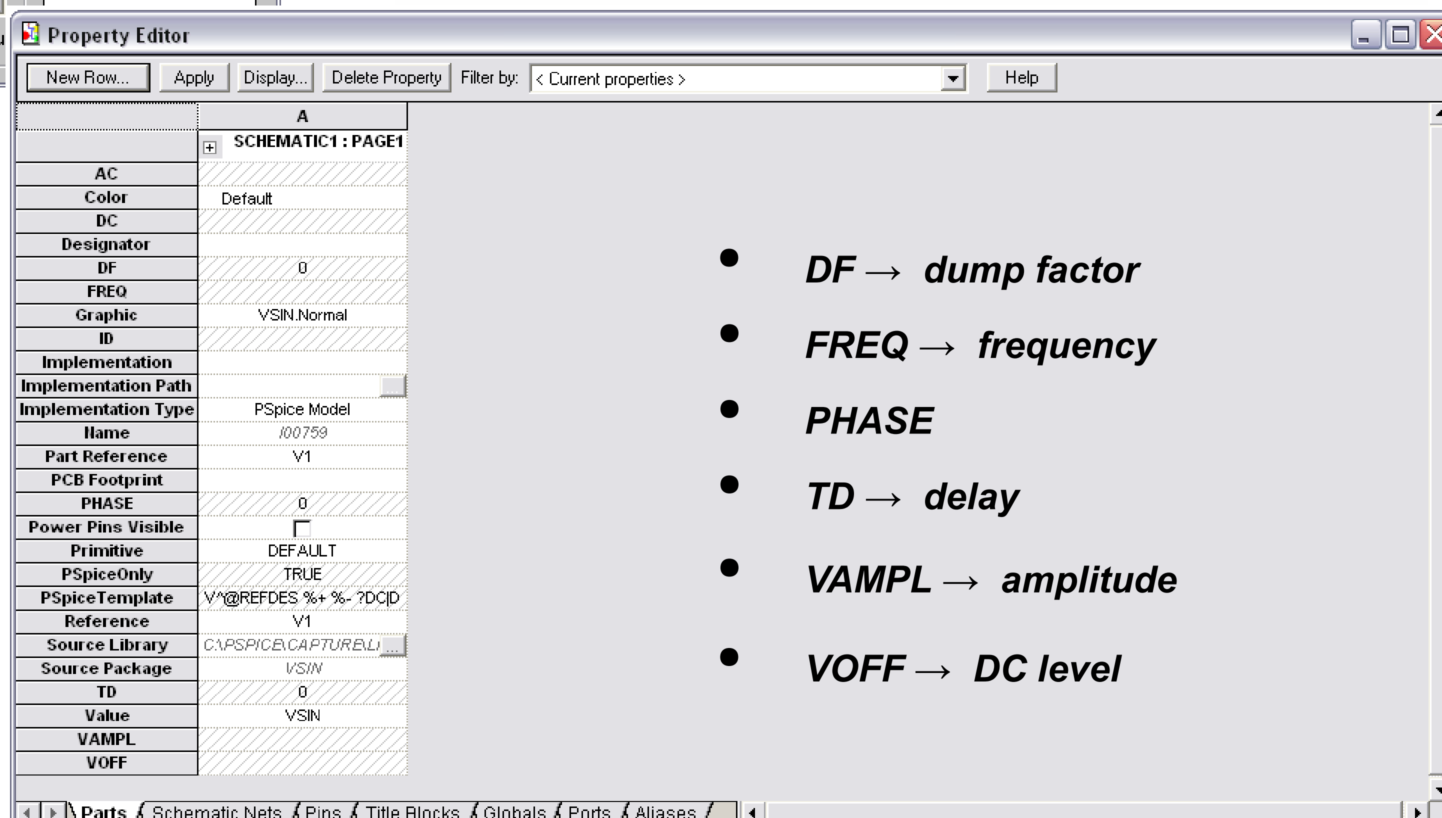
- voltages and currents are tracked over time
  - a smaller integration time step increases both the results accuracy and the simulation duration
  - sometimes convergence problems can occur

- you can specify in the simulation settings window the maximum step size in incrementing the time during transient analysis (***numerical integration time-step***)

VSIN source

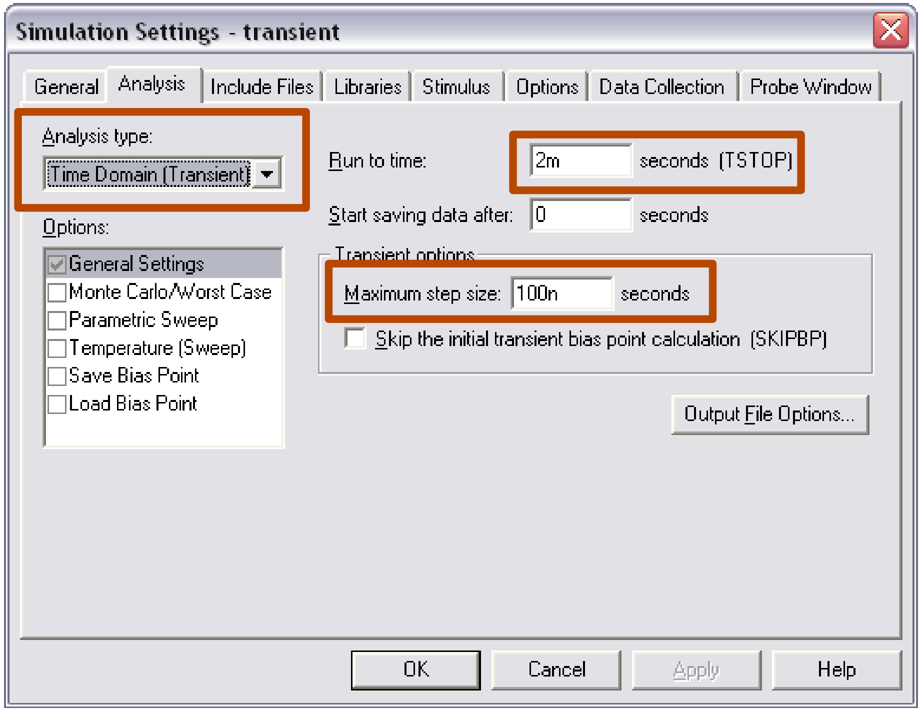




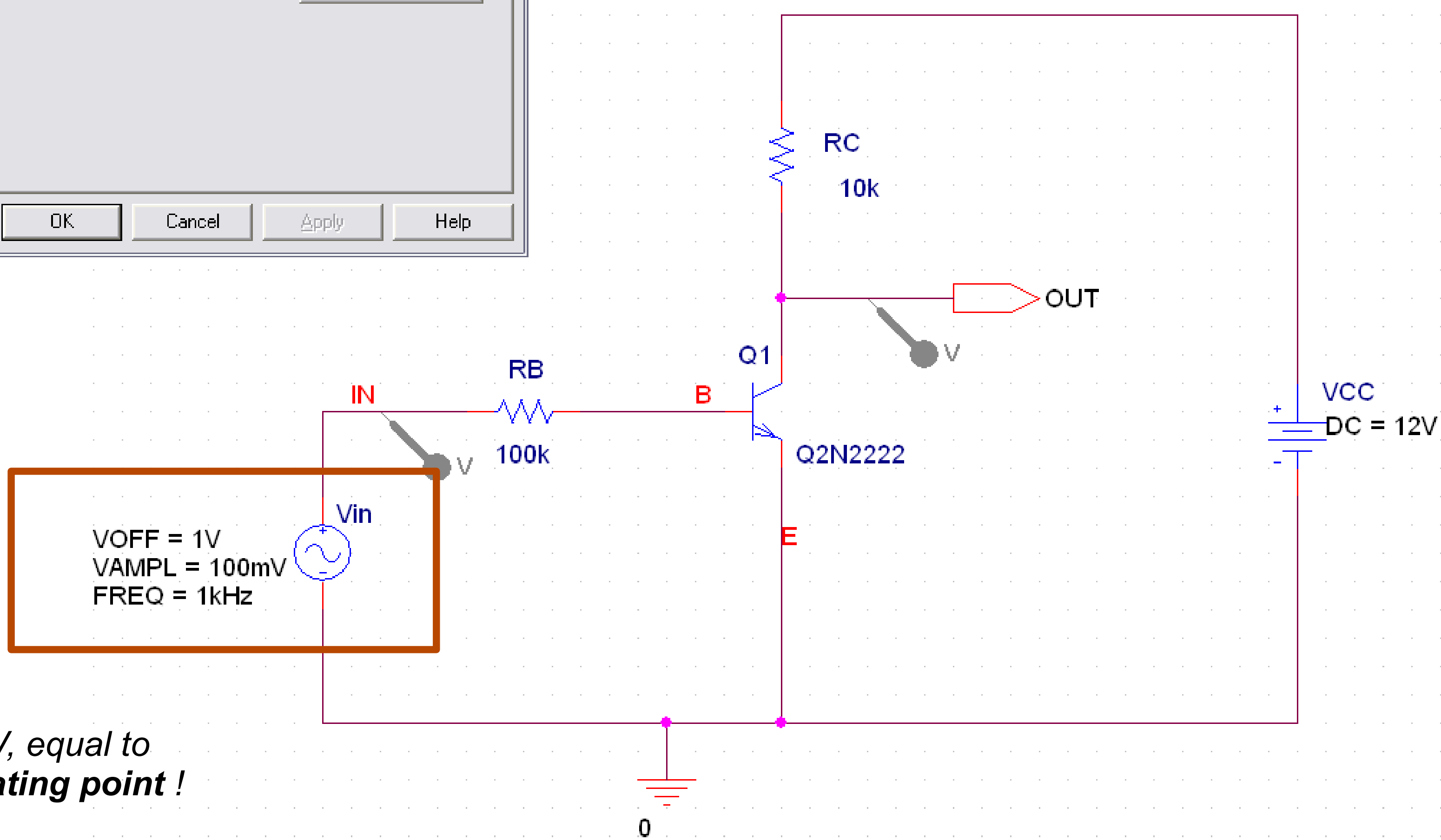


- ***DF → dump factor***
- ***FREQ → frequency***
- ***PHASE***
- ***TD → delay***
- ***VAMPL → amplitude***
- ***VOFF → DC level***

Transient analysis

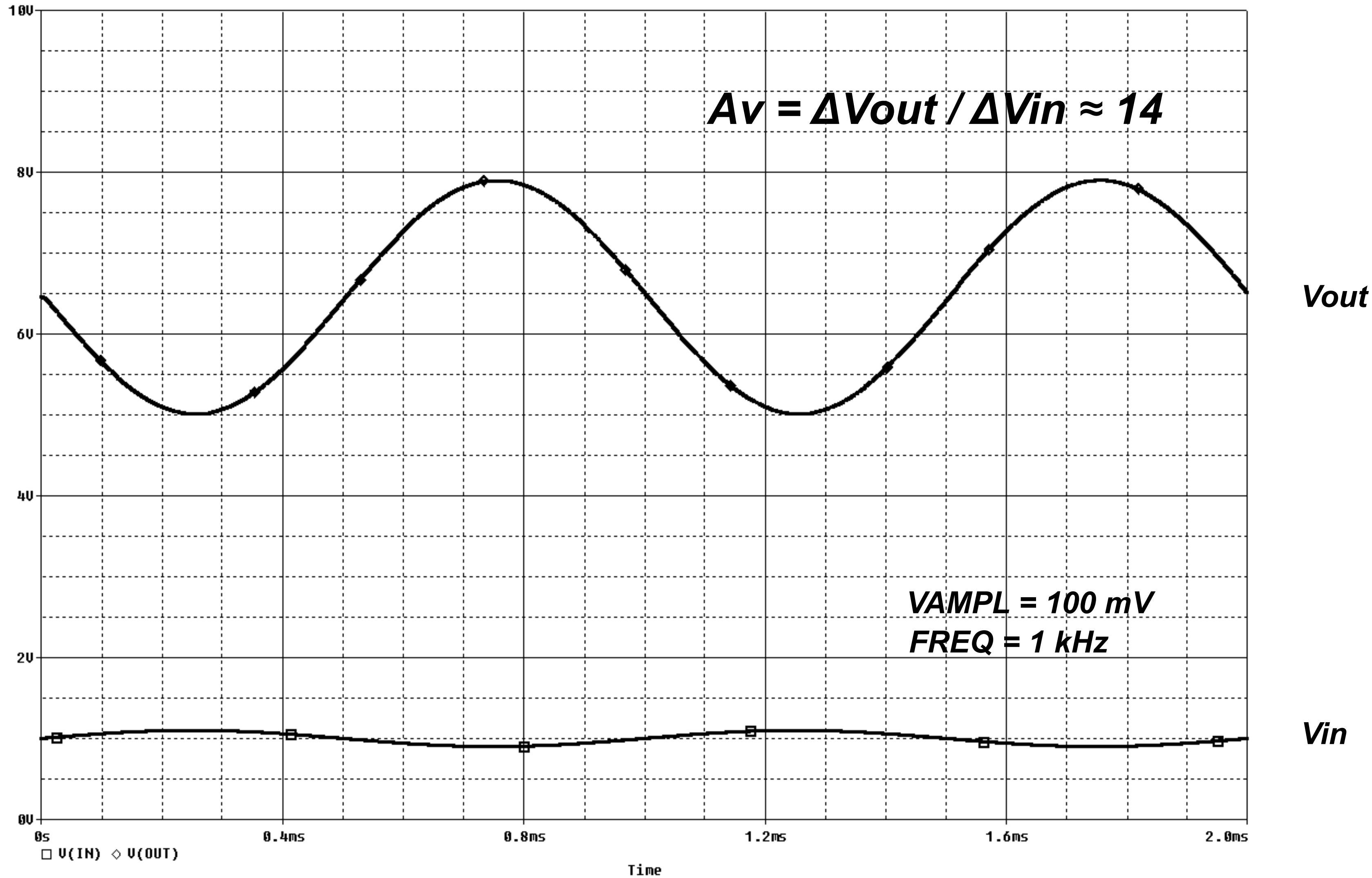


Choose a duration ~ 2/FREQ and an integration time-step of 10-100 ns



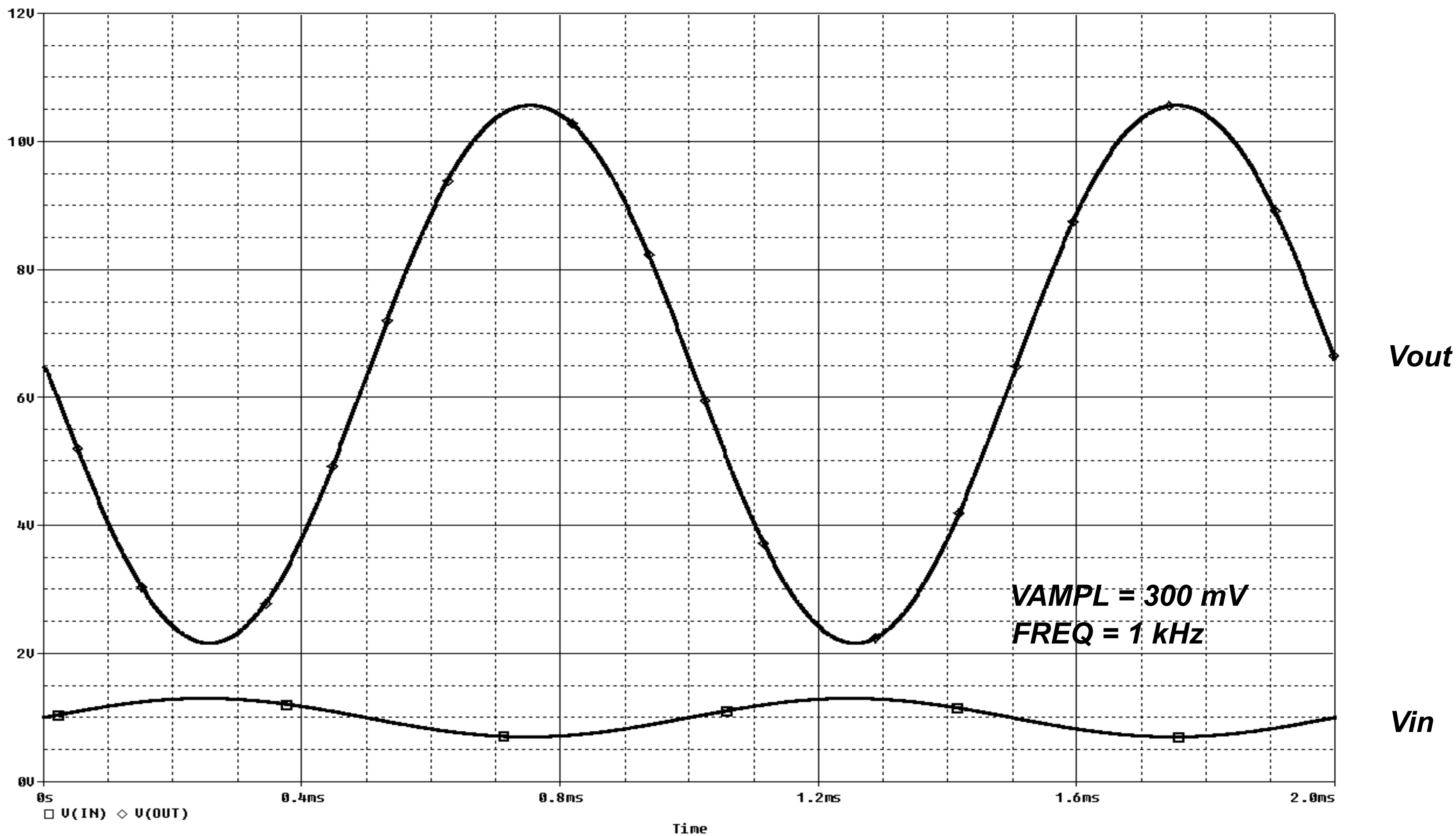
*VOFF = Vin,dc = 1V, equal to the **input DC operating point** !*

Transient analysis

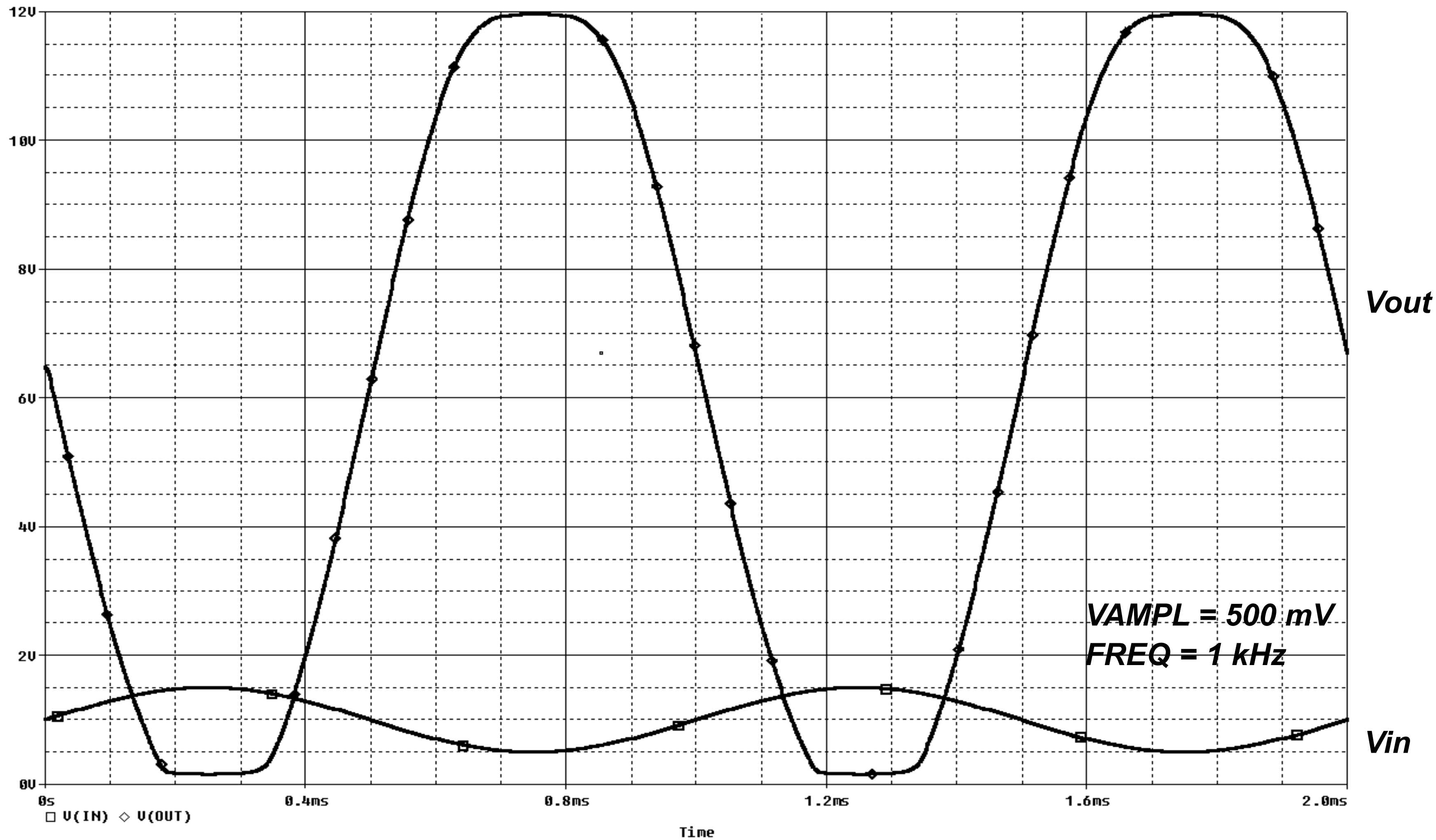


*The circuit is an **inverting amplifier** !*

Increasing the amplitude

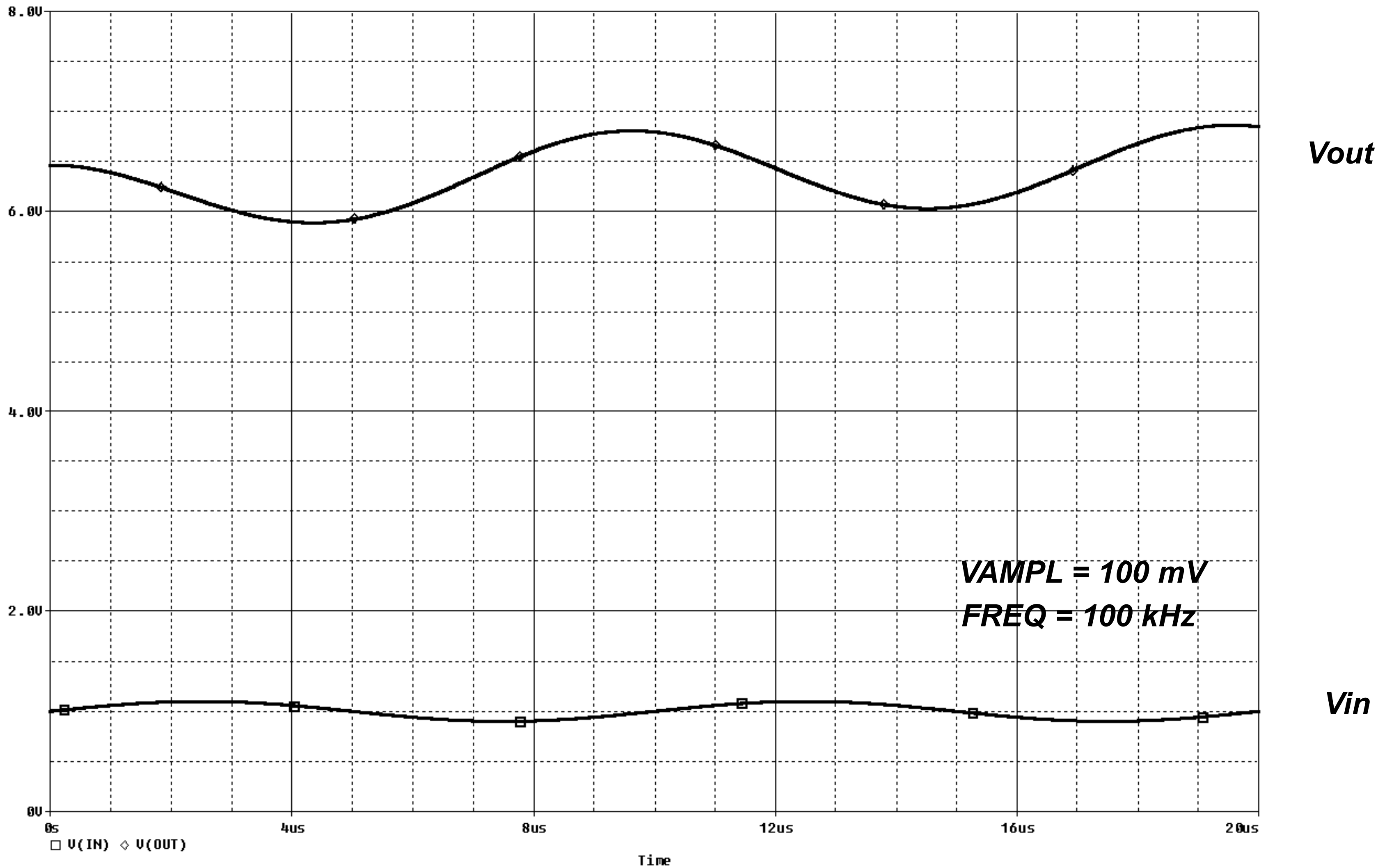


Output saturation



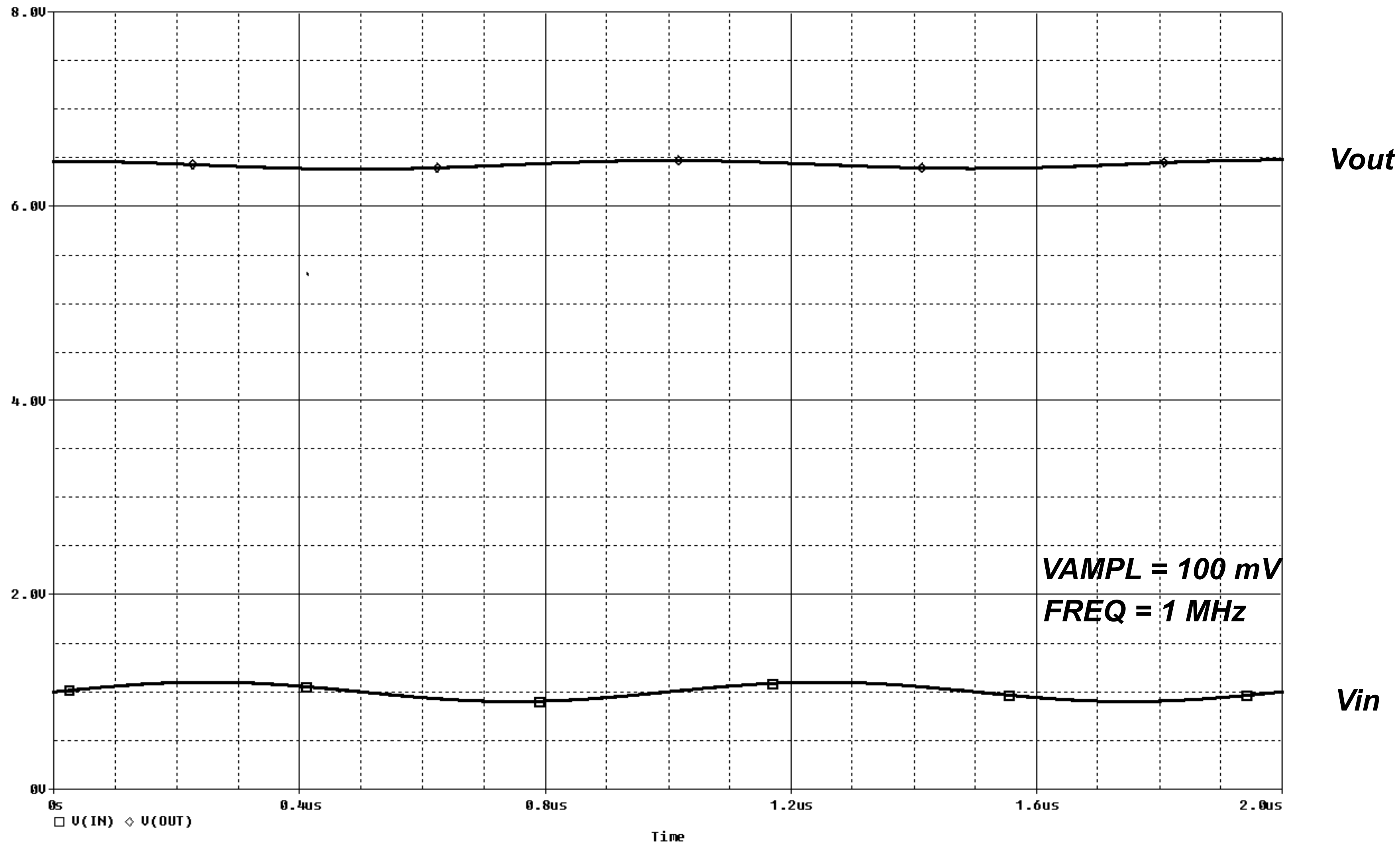
Are you surprised ?

Increasing the frequency



Note that a ***phase difference*** *exist !*

Increasing the frequency



AC analysis

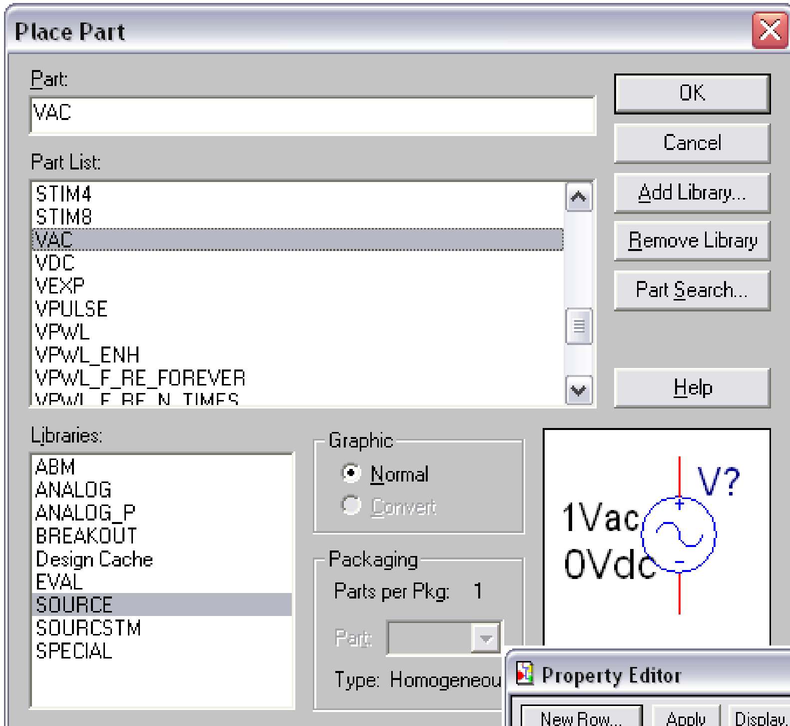
- ***small-signal*** frequency response of the circuit ***linearized*** around the bias point by sweeping one or more AC sources over a range of frequencies
  - ***non-linear devices*** (diodes, transistors etc.) are linearized to determine their AC ***small-signal models***
  - all independent voltage and current sources that have ***AC specifications*** are inputs to the circuit

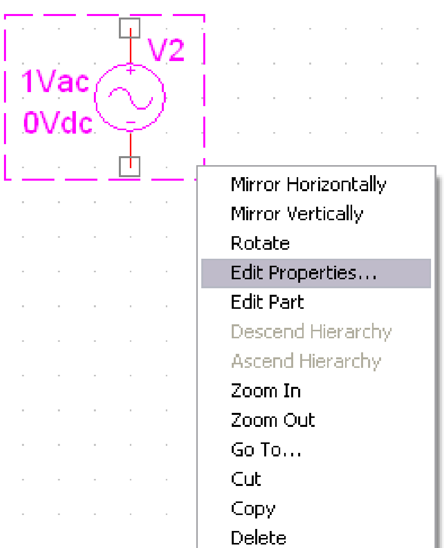
- outputs include voltages and currents with ***magnitude*** and ***phase***

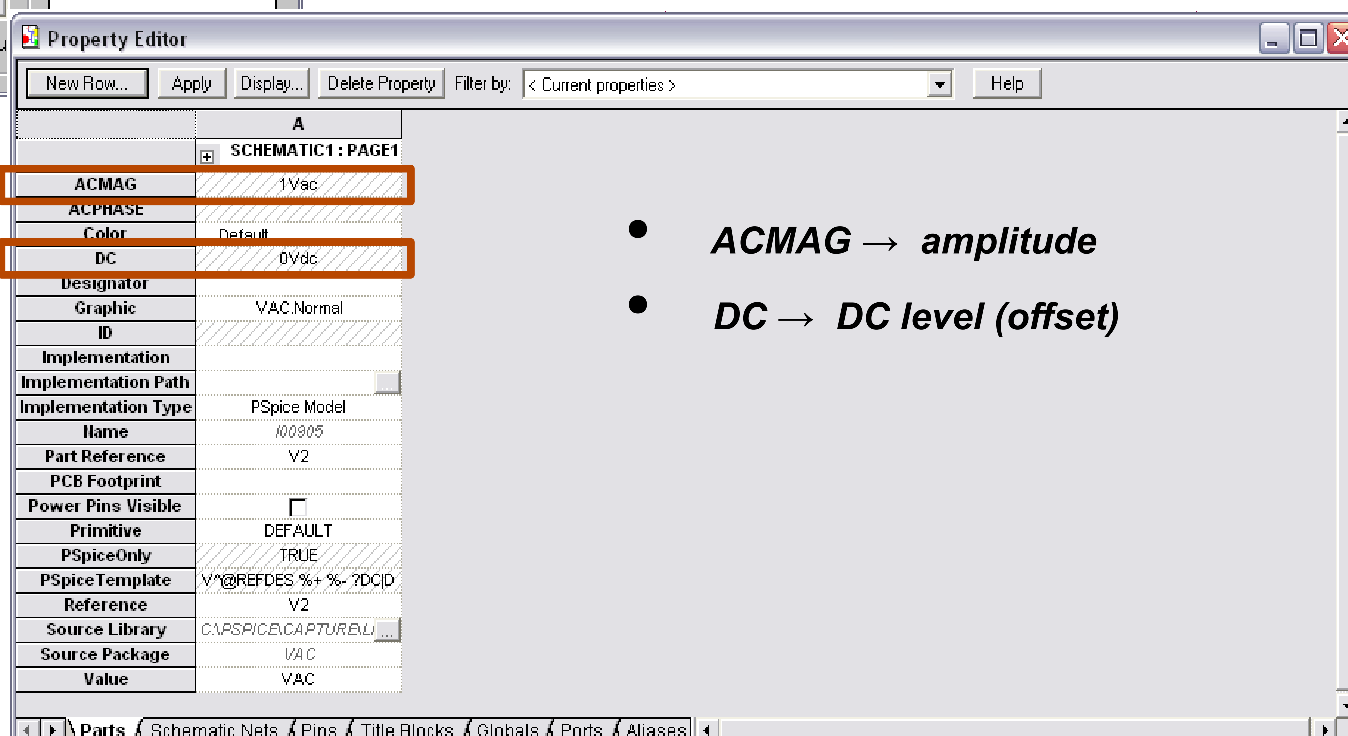
- the best way to use AC sweep analysis is ***to set the source magnitude to one*** (e.g. ACMAG = 1V, AC=1A) in this way ***the measured output equals the voltage/current gain*** (relative to the input source) at that output

- the sweep can be linear, logarithmically by decades or logarithmically by octaves

VAC source

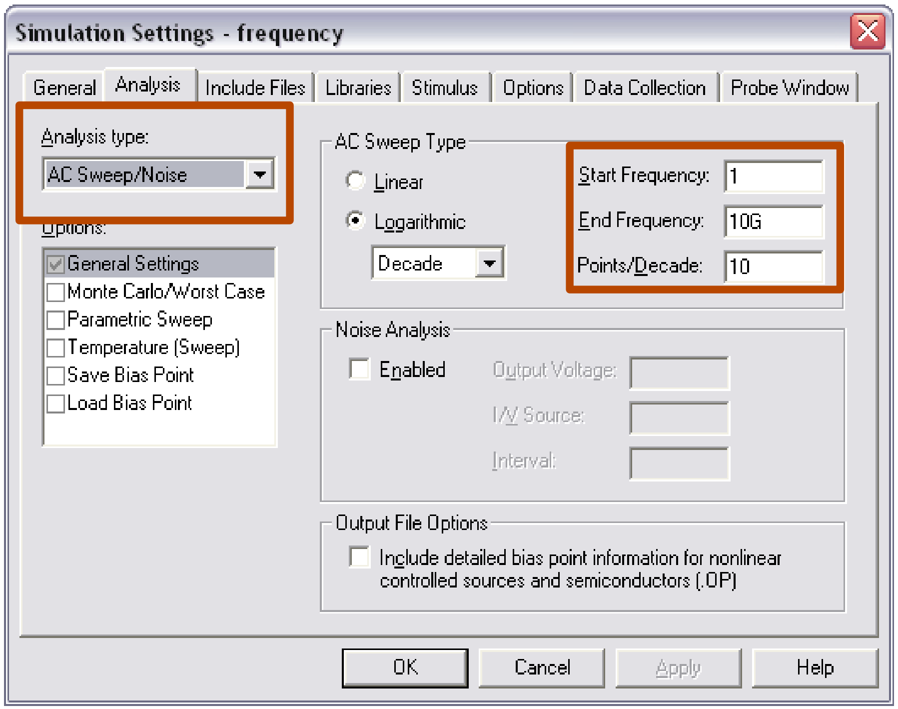


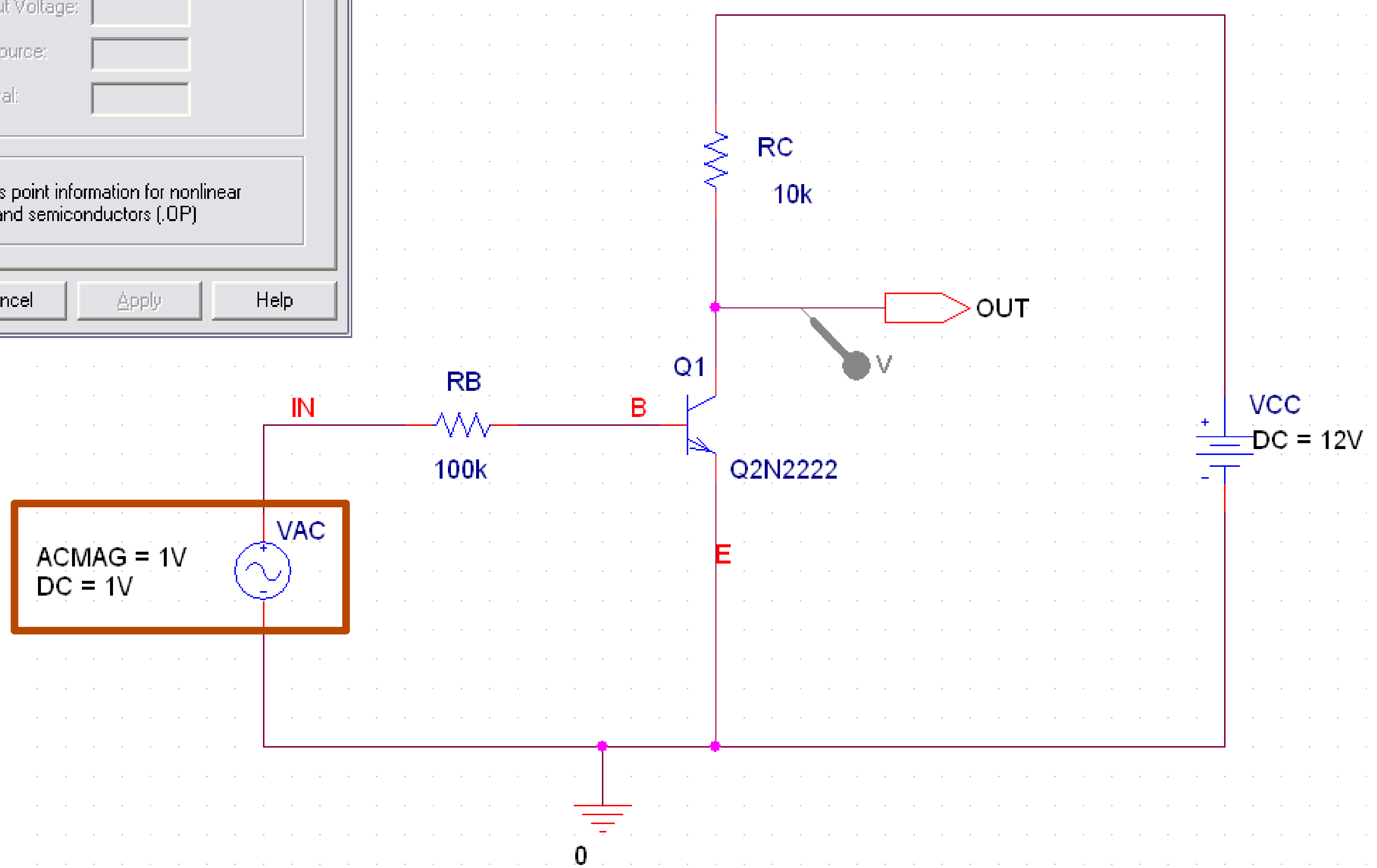




- *ACMAG* → *amplitude*
- *DC* → *DC level (offset)*

AC analysis

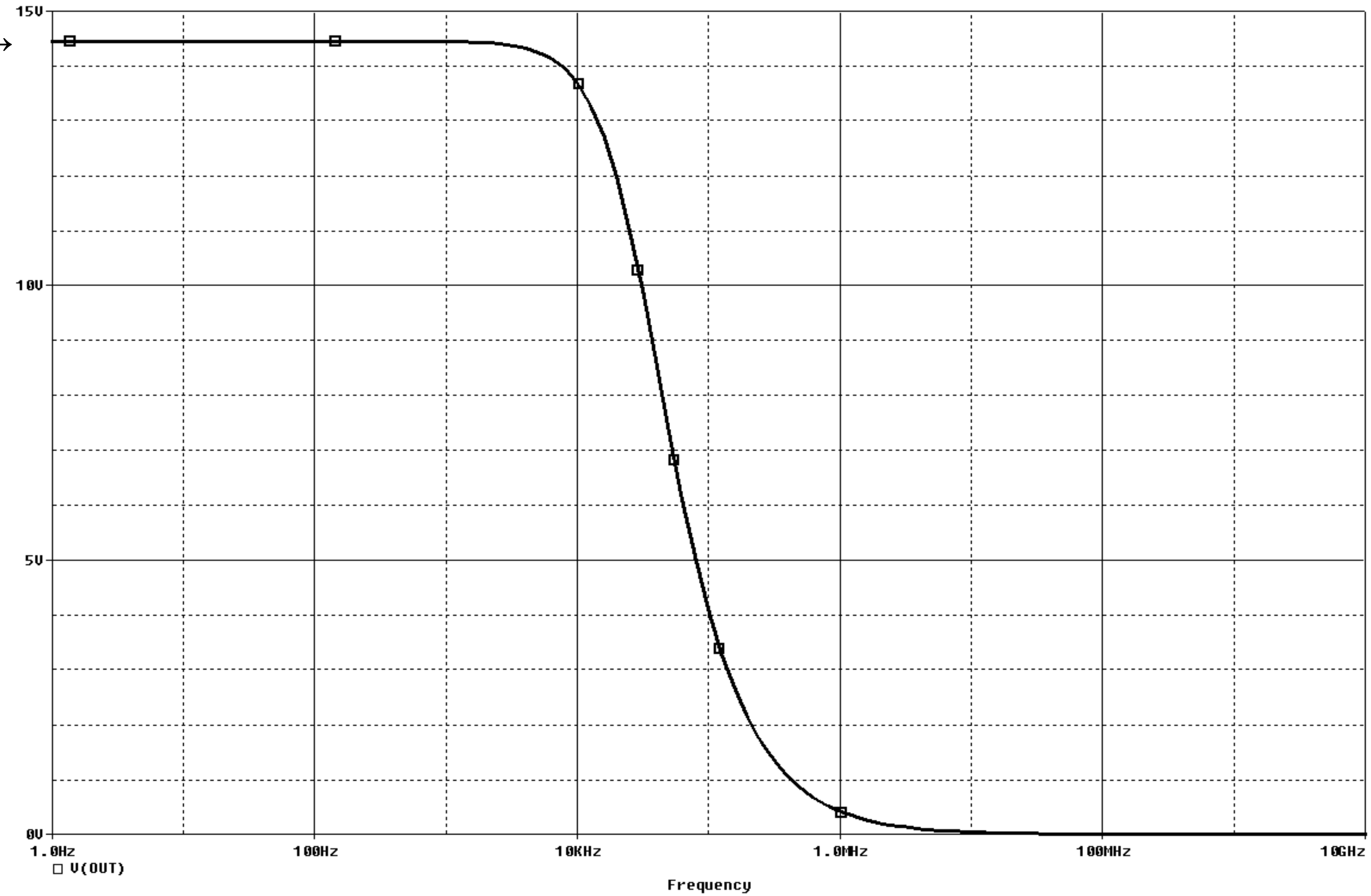




Remind !
*The AC analysis is a **small signal** analysis! The circuit is **linearized** around the DC operating point, hence you can use any ACMAG value! By using ACMAG=1V you immediately obtain **the voltage gain plot** with Vout(f)= Vout(f) /1V*

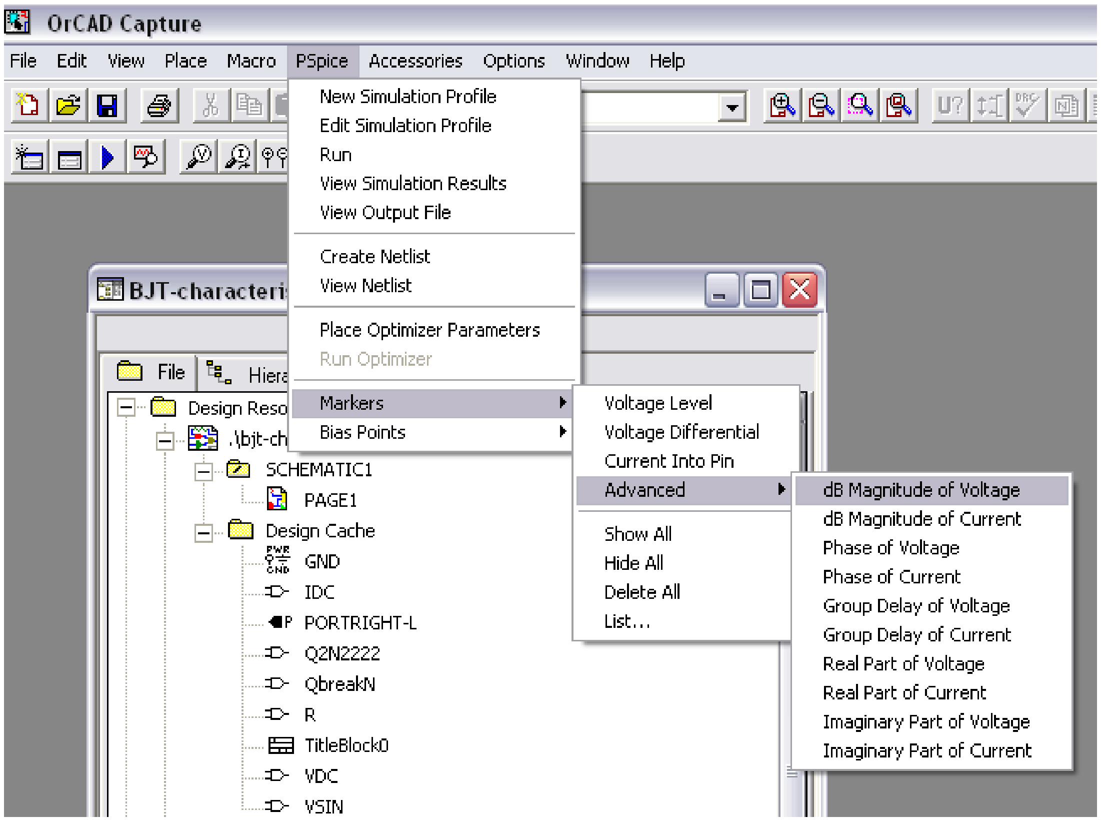
Vout/Vin frequency response

Av ≈ 14 →



The voltage gain decreases by increasing the signal frequency → ***low pass filter*** *!*

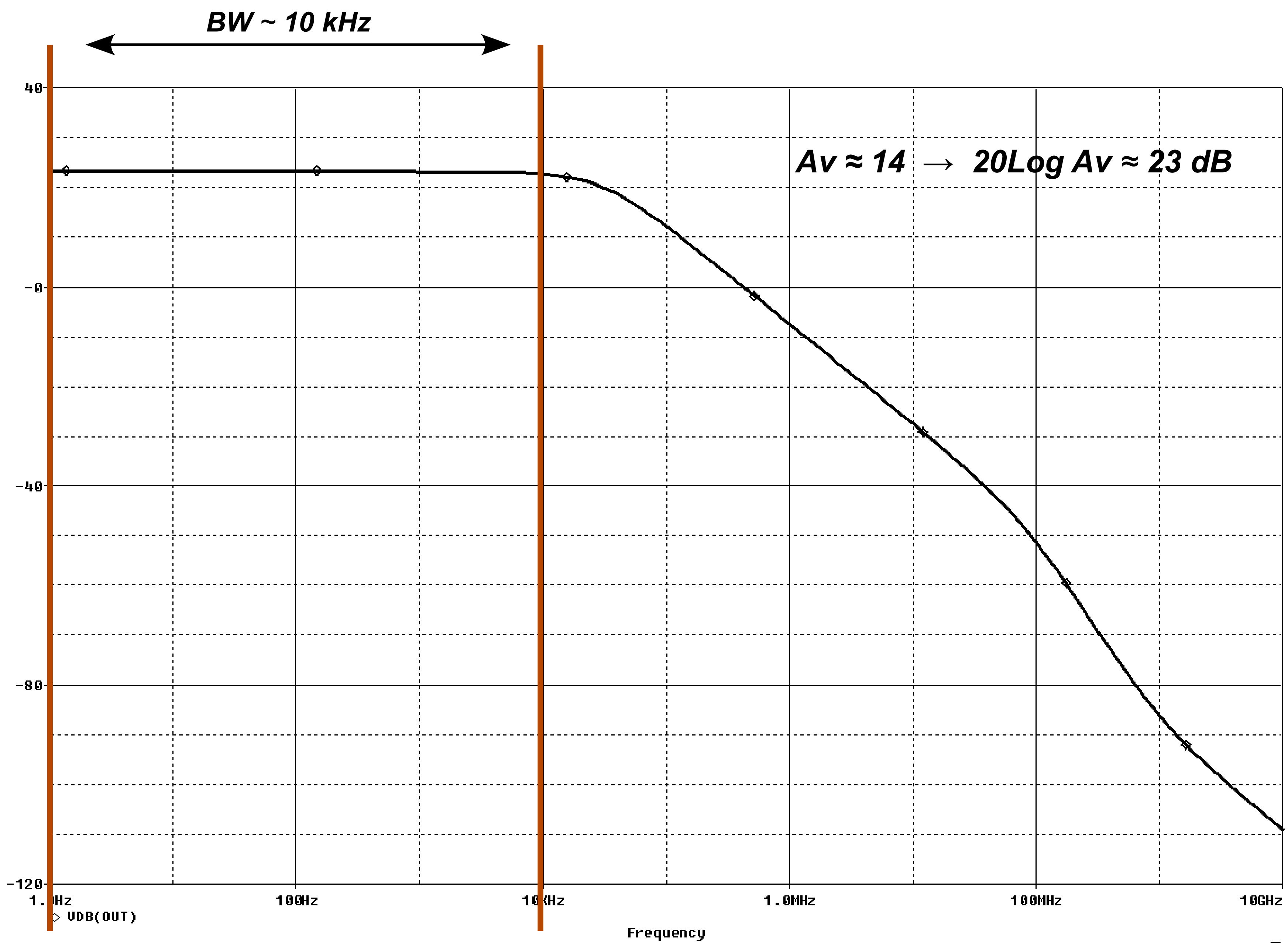
dB voltage gain



*You can plot **dB voltages and currents** and **phases** using special markers*

PSpice → Markers → Advanced → dB Magnitude of Voltage

dB voltage gain



Backup

Installation details

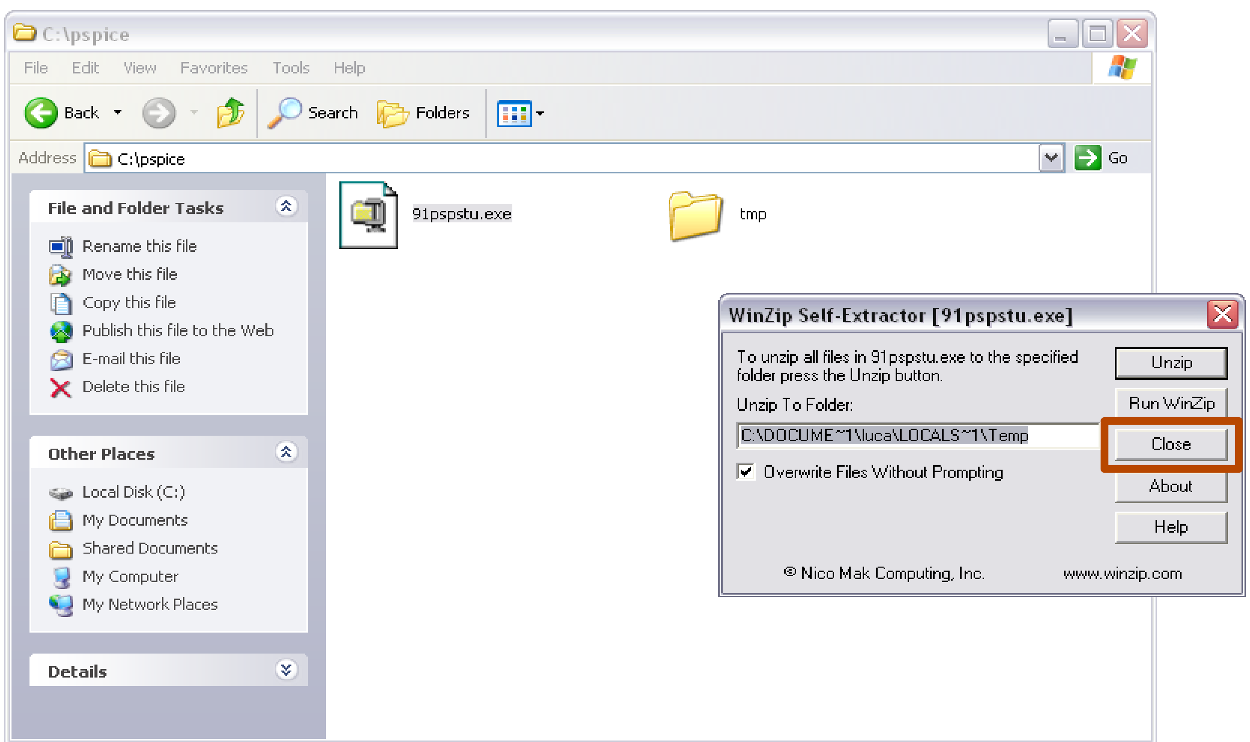
- create a main ***installation directory*** `C:\pspice`

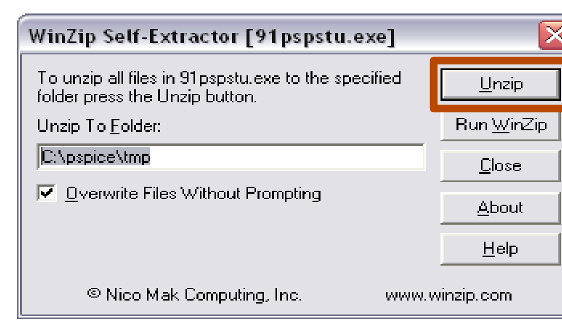
- download the Windows program executable ***91pspstu.exe*** (~27 MB) and put it in `C:\pspice`

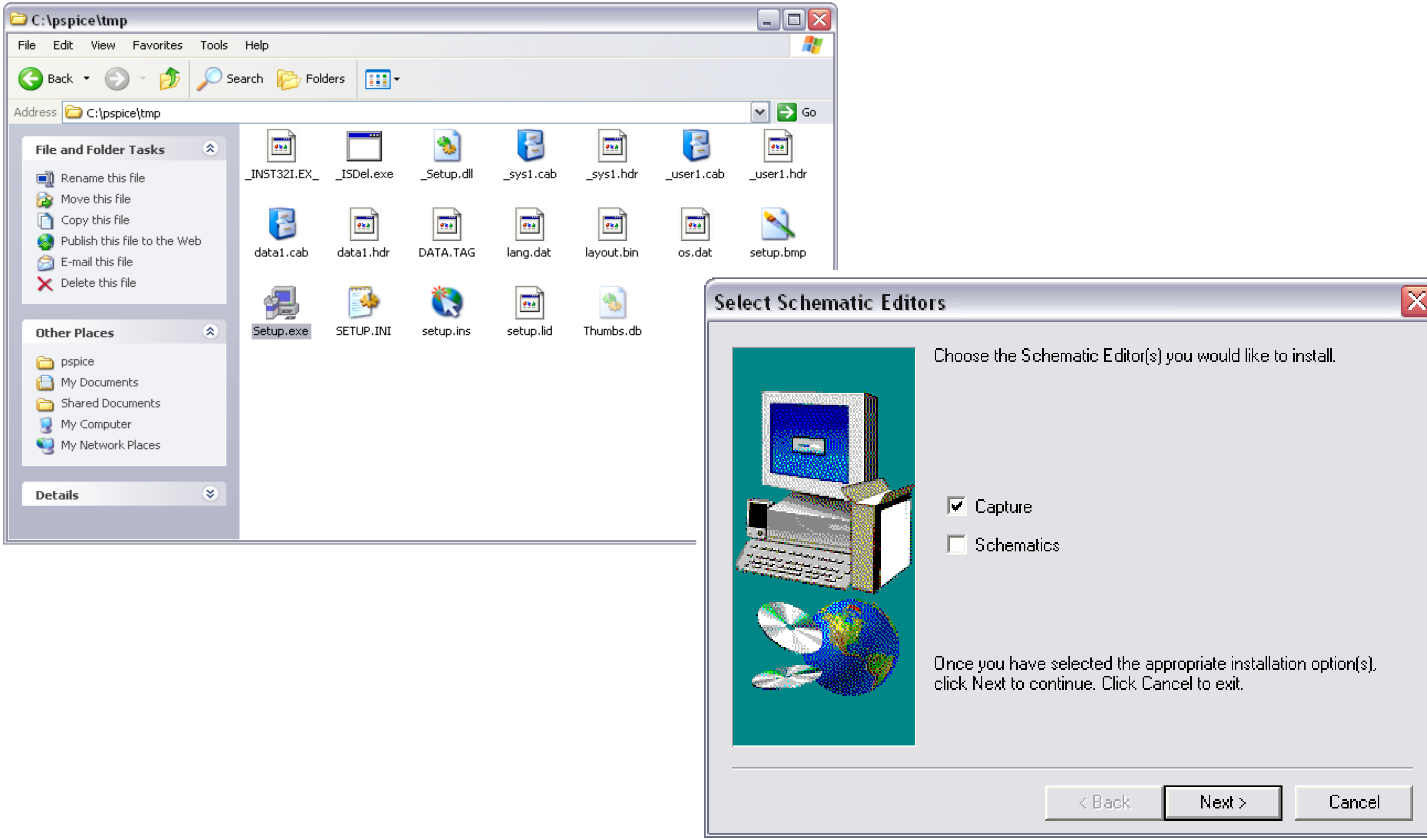
  http://personalpages.to.infn.it/~pacher/pspice/91pspstu.exe

  www.electronics-lab.com/downloads/schematic/013/index.html

- the executable is a self-extractor compressed file, create a `C:\pspice\tmp` temporary directory for the extracted installation files and launch the 91pspstu.exe executable

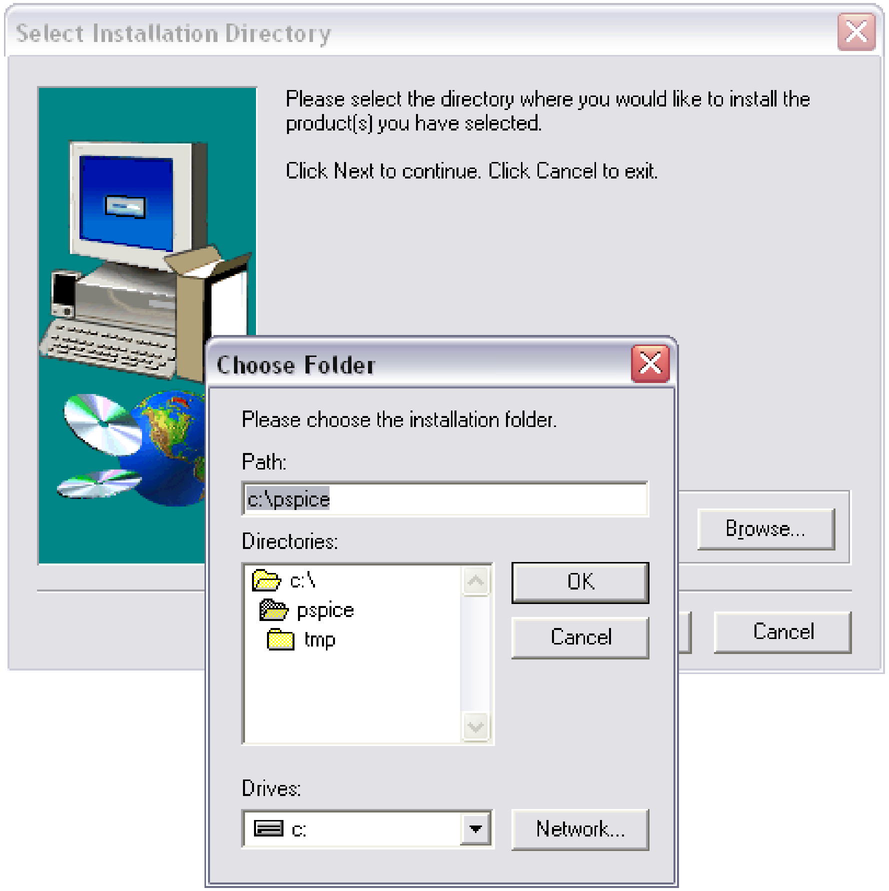


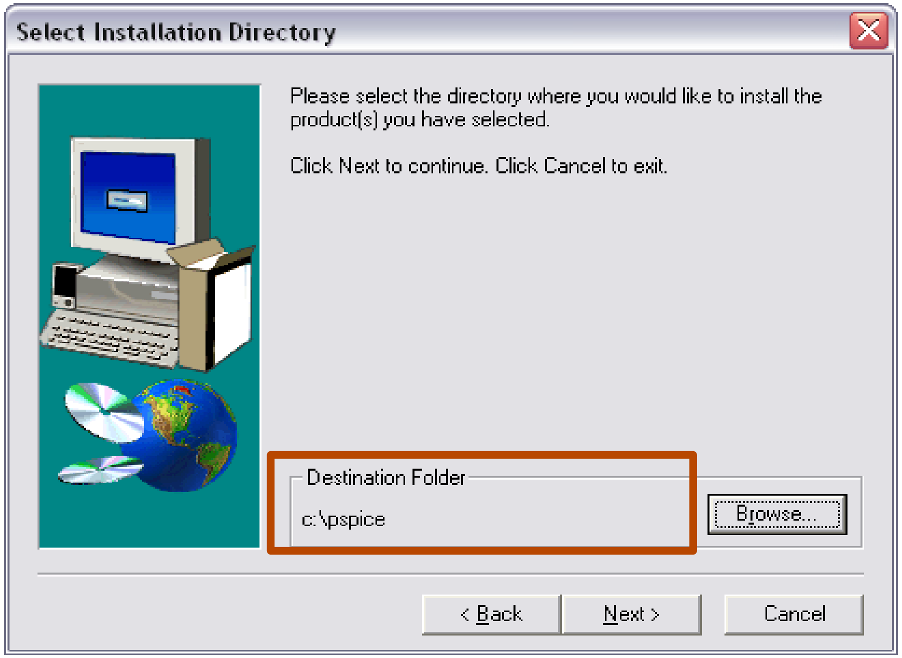


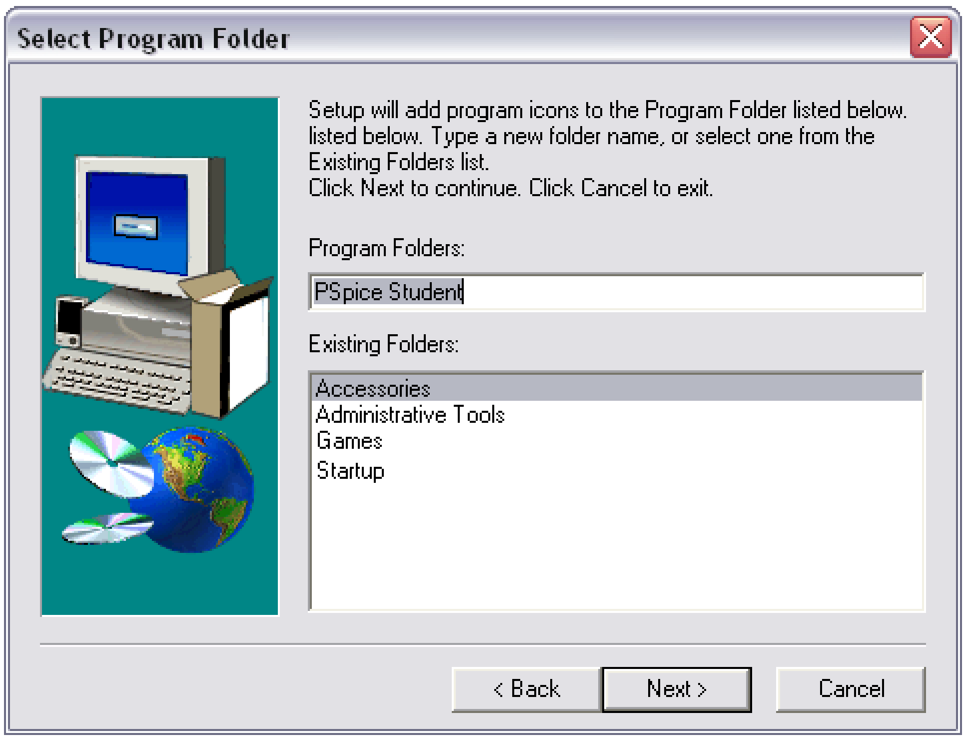


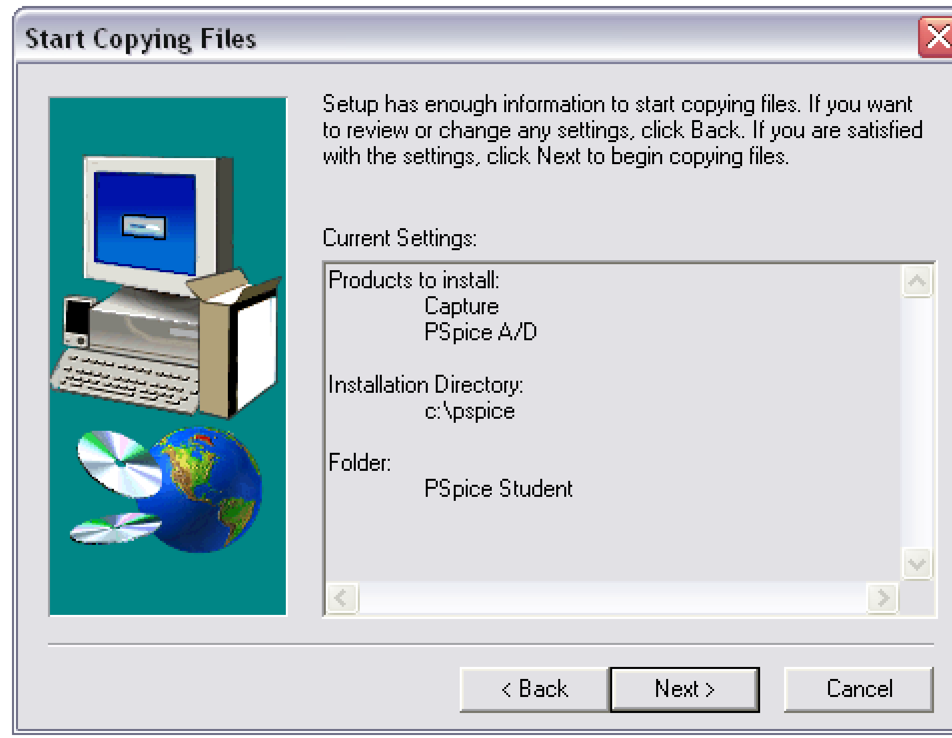
- when the extraction process has finished launch the ***Setup.exe*** installation program in the temporary directory
- choose ***Capture*** as schematic entry tool

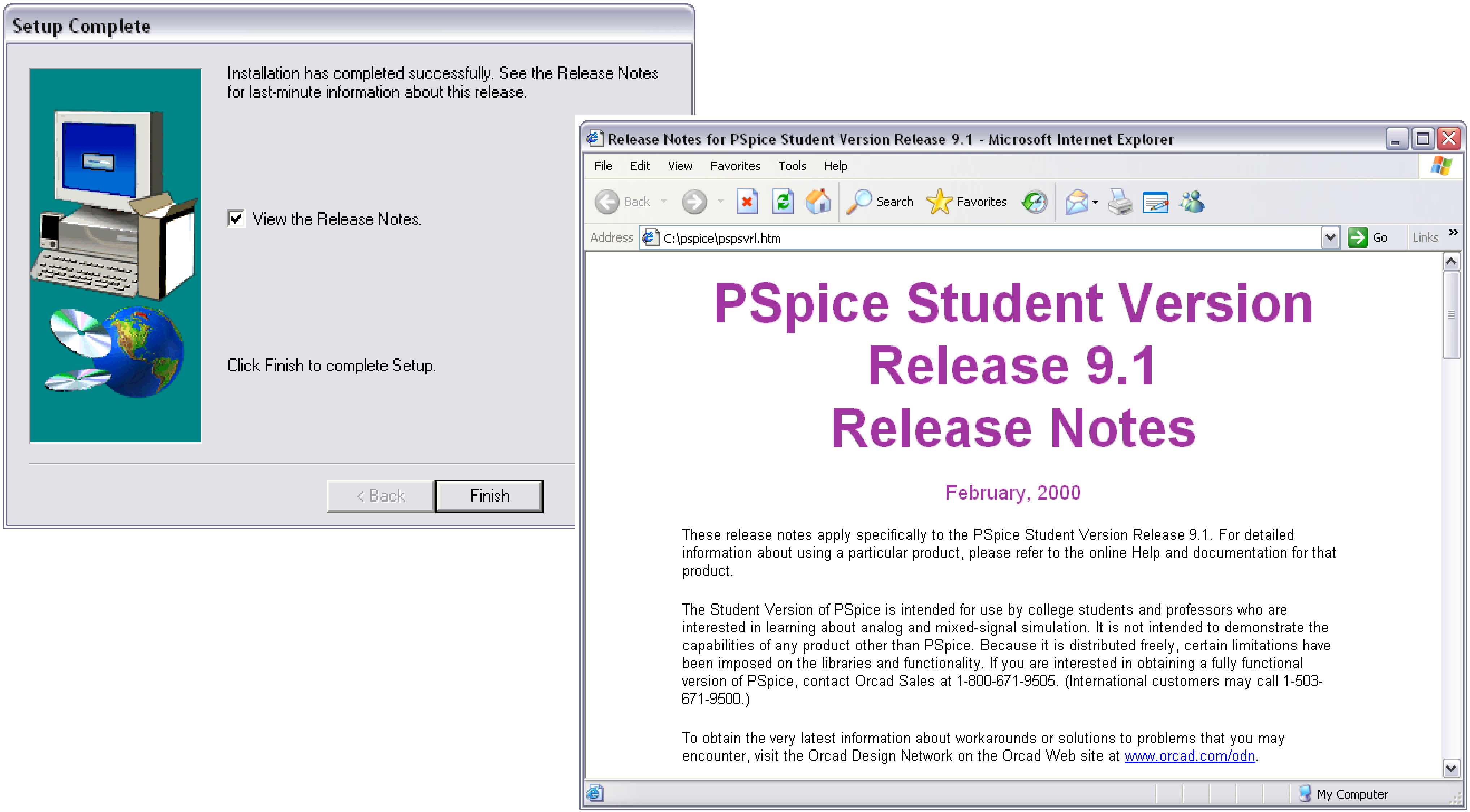
- change the default installation directory path into `C:\pspice`
- this is a safer choice because there are ***no blanks*** in the installation path !



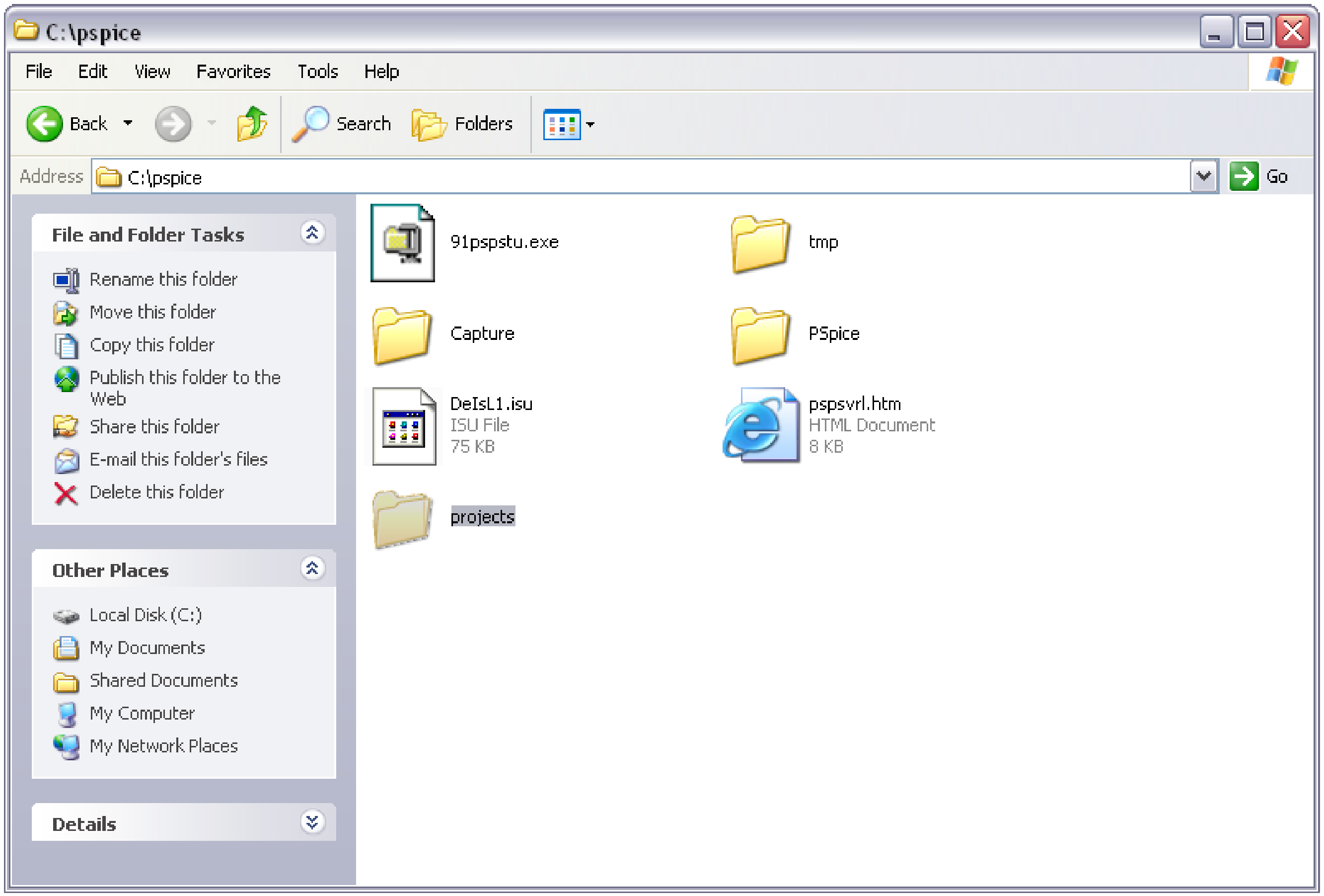



Select Program Folder
Setup will add program icons to the Program Folder listed below. listed below. Type a new folder name, or select one from the Existing Folders list.
Click Next to continue. Click Cancel to exit.
Program Folders:
PSpice Student
Existing Folders:
Accessories
Administrative Tools
Games
Startup
< Back
Next >
Cancel


Start Copying Files
Setup has enough information to start copying files. If you want to review or change any settings, click Back. If you are satisfied with the settings, click Next to begin copying files.
Current Settings:
Products to install:
Capture
PSpice A/D
Installation Directory:
c:\pspice
Folder:
PSpice Student
< Back
Next >
Cancel



If you want to learn more about this student version of PSpice (e.g. limits in the number of components) read the ***Release Notes***



- at the end of the installation process delete the 91pspstu.exe file and the temporary directory
- create a further `C:\pspice\projects` directory that will contain all your PSpice projects