Esperimentazioni II

Traccia delle esercitazioni PSpice

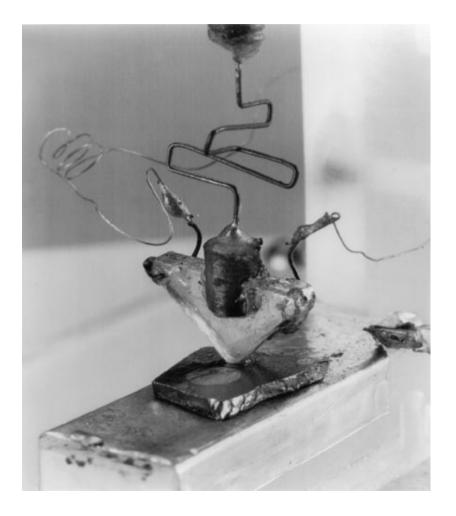
L. Pacher

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

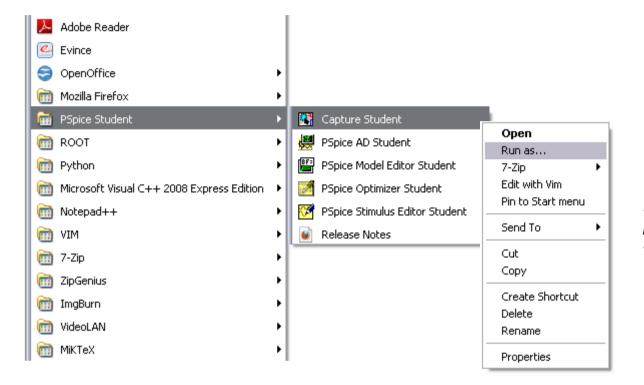
1956 Nobel prize in Physics

"for their researches on semiconductors and their discovery of the transistor effect"

SPICE and PSpice

- Simulation Program with Integrated Circuits Emphasis, 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
- Personal 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

🕼 OrCAD Capture		
<u>Eile Vi</u> ew Edit <u>O</u> ptions <u>W</u> indow <u>H</u> elp		
≥≥ # % # E <u>≥</u> ⊆		
E Session. 🖻 🗆 🔀		
Session Log		
	··· 2 · · · · · · · · · · · · · · · · ·	
Ready		Session Log
Noduy		pession Log

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

$File \rightarrow New \rightarrow Project...$

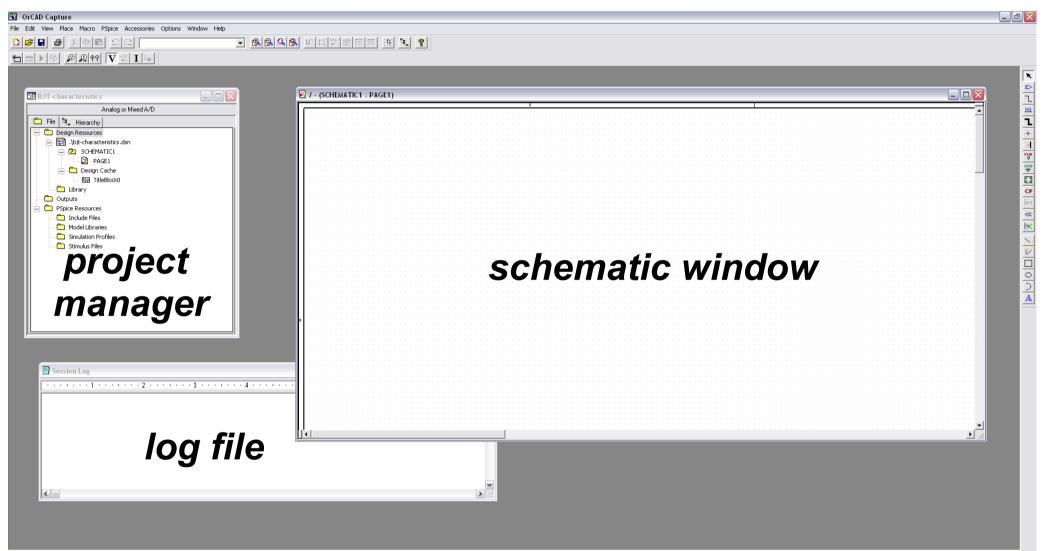
👫 OrCAD	Capture					
File View	Edit Options	Window Help	_			you
New Open Save	Ctrl+S	Project Design Library VHDL File				• sp
Save As.		Text File				na
Print Pres	Ctrl+P					● sin Ar
Import D Export D	-					An
Exit	New Proje	ect				
	Name BJT-chara	cteristics		_		OK Cancel
	Create a	New Project Using	<u>,</u>			Help
	×.	 Analog or Mixe 	ed A/D		- Tip for New	v Users
		PC Board Wize	ard		Mixed A/D	ew Analog or project. The t may be blank:
		Programmable	Logic Wizard			rom an existing
		Schematic				
	Location	.projects\BJT-char	racteristics			Browse

Create a 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

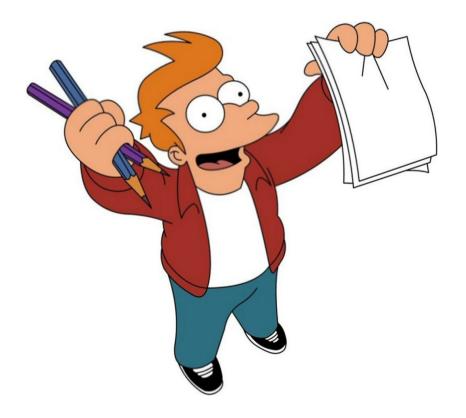
Create PSpice Project	X
C Create based upon an existing project	ОК
	Browse
Create a blank project	Cancel
	Help

The graphical interface



0 items selected Scale=100% X=2.10 Y=0.80

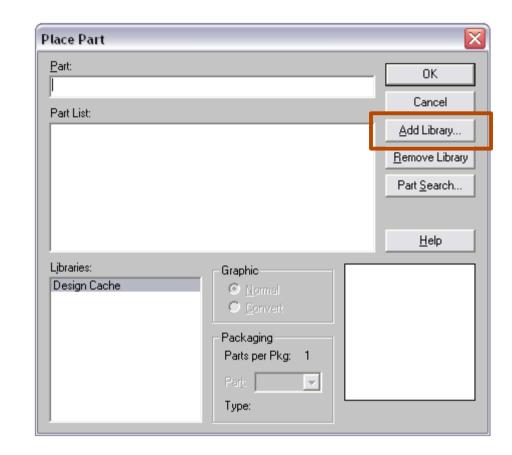
Part I Schematic entry



Place components

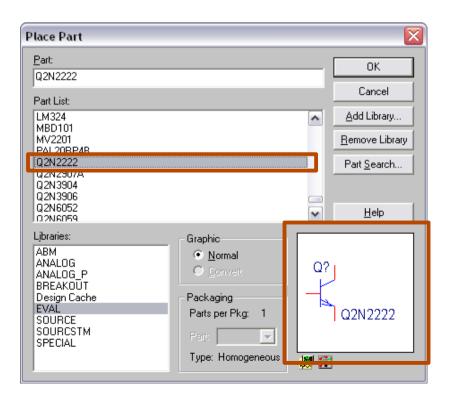
$\textit{Place} \rightarrow \textit{Part...}$

	OrCA	D Cap	ture				
File	Edit	View	Place	Macro	PSpice	Accessories	Options
<u>ک</u>	B		Par	t		Shift+P	
_			Wir	е		Shift+W	
Ť			Bus	i		Shift+B	
			Jun	ction		Shift+J	
			Bus	Entry		Shift+E	
			Net	Alias		Shift+N	
	100	BJT-c	Pov	ver		Shift+F	
			Gro	und		Shift+G	_
			Off	-Page Co	onnector.		
		🗅 File	Hier	rarchical	Block		
		- 6	Hier	rarchical	Port		
			Hier	rarchical	Pin		
			No	Connect		Shift+X	
			Title	e Block			
			Boo	kmark			
			Tex	:t		Shift+T	
			Line	•			
			Rec	tangle:			
	l E	🛅	Ellip	se			
	ш.		Arc				
			Poly	/line		Shift+Y	1
			Pict	ure			1
			📛 Stii	mulus File	es		_
							-
							•



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

Place Part Eart: Part List:	OK Cancel Add Library Bemove Library Part Search
Libraries: Design Cache Packaging Parts per Pkg: 1 Part:	Browse File
Туре:	File name: ''abm.olb'' ''analog.olb'' ''analog_p.olb'' ''breako Open Files of type: Capture Library(".olb) Cancel Open as read-only



- **ANALOG** contains ideal R, L, C and controlled voltage/current sources
- BREAKOUT contains *ideal semiconductor devices* (transistors, diodes etc.)

Basic libraries

- **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

NPN switching transistors

Product specification

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

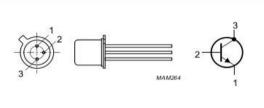
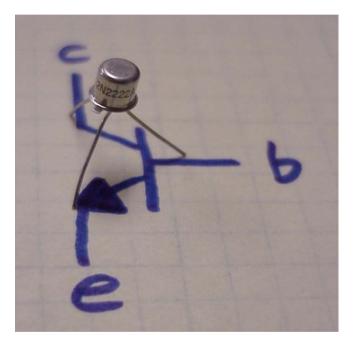


Fig.1 Simplified outline (TO-18) and symbol.

QUICK REFERENCE DATA

SYMBOL	PARAMETER	CONDITIONS	MIN.	MAX.	UNIT
V _{CBO}	collector-base voltage 2N2222	open emitter		60	v
	2N2222A		-	75	v
V _{CEO}	collector-emitter voltage	open base			
	2N2222			30	V
	2N2222A			40	V
I _C	collector current (DC)		T 2	800	mA
Ptot	total power dissipation	T _{amb} " 25 1C	-	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

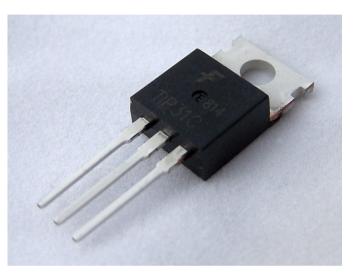
Check the datasheet again!

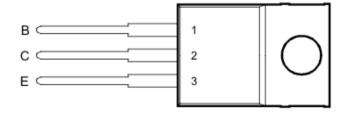
TIP31, TIP31A, TIP31B, TIP31C NPN SILICON POWER TRANSISTORS

JULY 1968 - REVISED MARCH 1997

electrical characteristics at 25°C case temperature

	PARAMETER		TEST CONDITI	ONS	MIN	TYP	MAX	UNIT
V _{(BR)CEO}	Collector-emitter breakdown voltage	l _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
ICEO	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
VBE	Base-emitter	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			



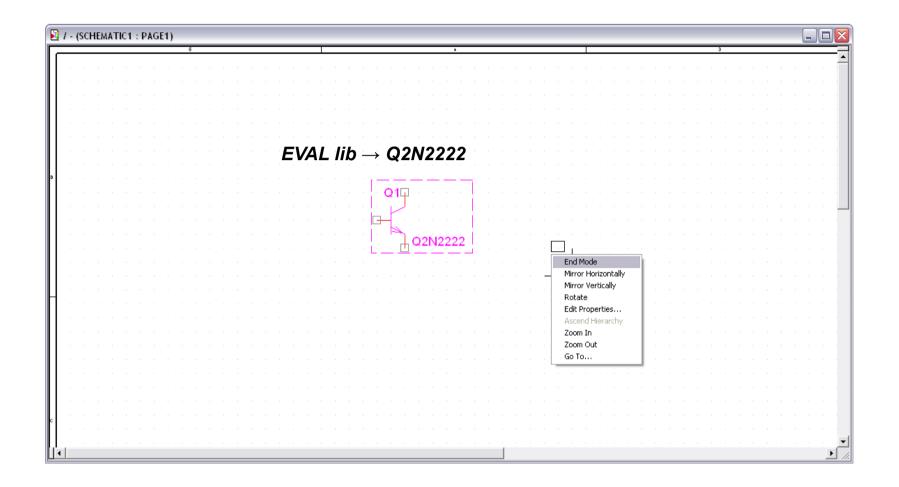


NOTES: 5. These parameters must be measured using pulse techniques, t_p = 300 µs, duty cycle \leq 2%.

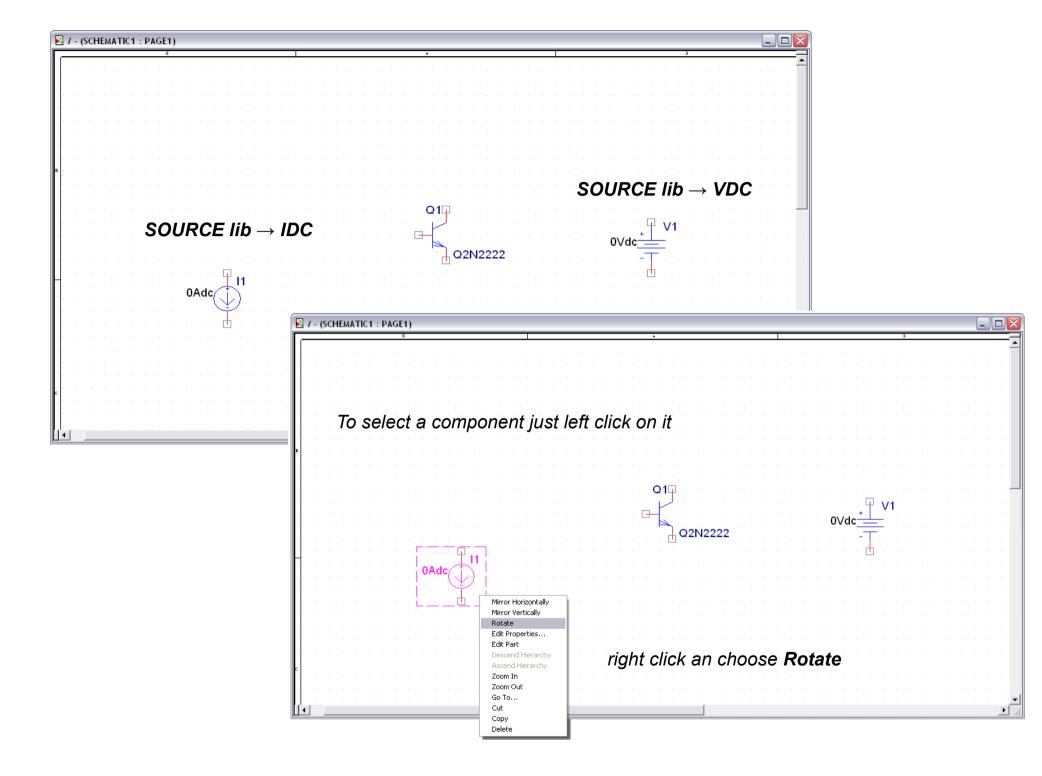
6. These parameters must be measured using voltage-sensing contacts, separate from the current carrying contacts.

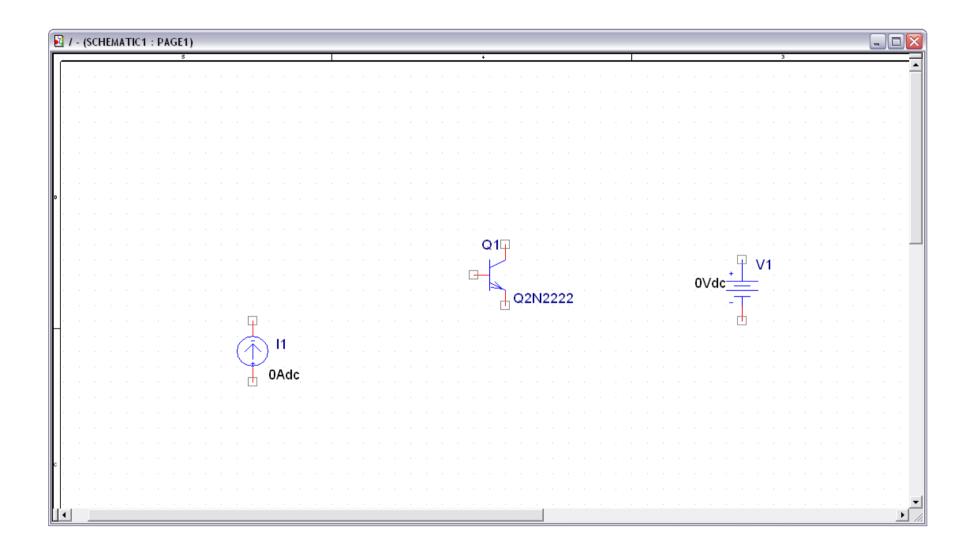
Another general-purpose commercial BJT

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** \rightarrow **End Mode** to come back to the select mode

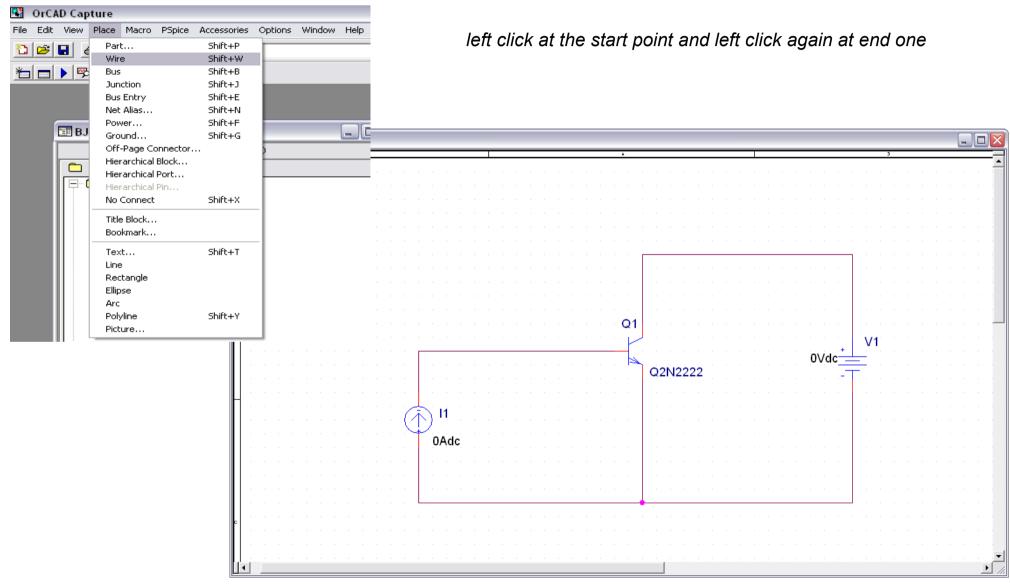




Components are on the table, now we have to make connections...

Make connections

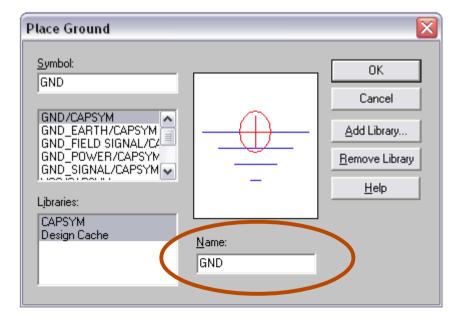
$\textit{Place} \rightarrow \textit{Wire...}$

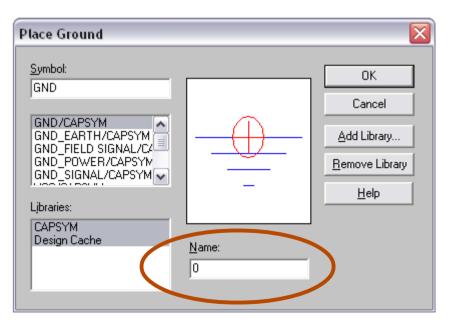


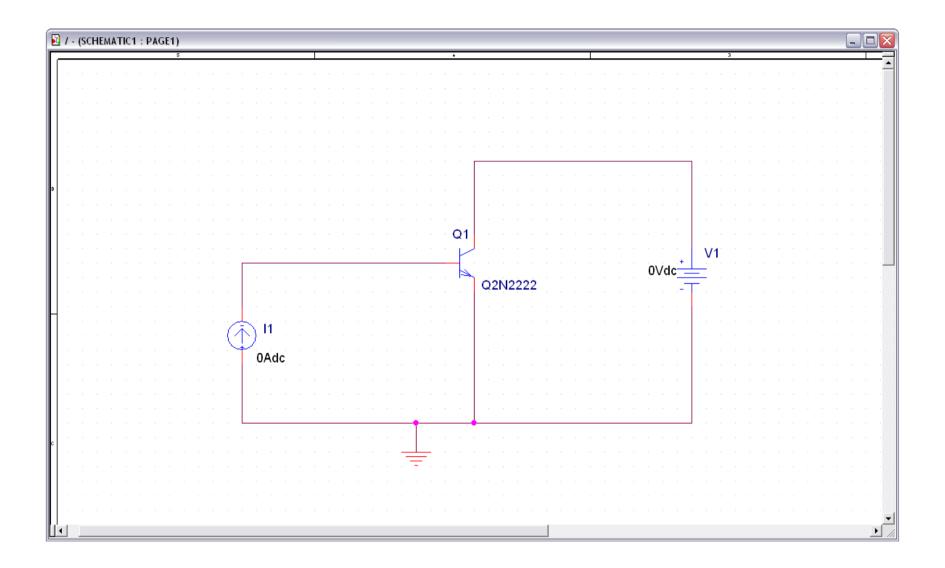
Place grounds

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









Edit instance properties

	Mirror Vertically	Property Editor New Column Apply Display Delete Property Filter by: < Current properties > Help
	Edit Properties	Color DC Designator Graphic ID Implementation Implementation Part Reference 1 + SCHEMATIC1:PAGE1:V1 Default 0Vdc VDC.Normal PSpice Model //0201 V1
· · · · ·		The schematic1: PAGE1: V1 Default 0Vdc VDC.Normal PSpice Model 00201 V1
	Accord Historychu	
	Zoom Out Go To	
	COPY	
	Delete	

- 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* \rightarrow *Edit Properties...*

New Column Apply Display Delete Property Filter by: < Current properties >			
Color DC Designator Graphic ID Implementation Implementation Path Implementation Type Name Part Reference PC			
SCHEMATIC1: PAGE1: V1 Default 0Vdc VDC.Normal PSpice Model 100201 V1			
Property Editor			
New Column Apply Display Delete Property Filter by: < Current properties > Image: Column and the properties >			
Color DC Designator Graphic ID Implementation Implementation Path Implementation Type			ce P(▲
Default 0Vdb VDC.Norman PSpice Wodel	100201	V1	
Edit Delete Property Display			
✓ Parts			• •

New Row Ap	ply Display Delete Property	Filter by: < Current properties >	✓ Help	
	A	,		
	+ SCHEMATIC1 : PAGE1			
Color	Default			
DC	0Vdc			
Designator				
Graphic	VDC.Normal			
ID				
Implementation				
nplementation Path				
nplementation Type	PSpice Model			
Name	100201			
Part Reference	V1			
PCB Footprint				
Power Pins Visible				
Primitive	DEFAULT			
PSpiceOnly	TRUE			
PSpiceTemplate	V^@REFDES %+ %- @DC			
Reference	V1			
Source Library	CAPSPICE/CAPTURELI			
Source Package	VDC			
Value	VDC			

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

New Row App	ly Display Delete Property	Filter by: < Current properties >
	A	
	+ SCHEMATIC1 : PAGE1	
Color	Default	
DC	12V	
Designator		
Graphic	VDC.Normal	
ID		
Implementation		
plementation Path		
plementation Type	PSpice Model	Don't forget to left click on Apply
Name	100005	
Part Reference	VCC	to make changes effective
PCB Footprint		-
ower Pins Visible		
Primitive	DEFAULT	
PSpiceOnly	TRUE	
	VARREEDES WAW / MDC	
Reference	VCC	
-	UN-SHIELUARTONELLI	
Source Package	VDC	
Value	VDC	

Property Editor					
New Row Ap	ply Display Delete Property Filter by: < Current	properties >	✓ Help		
Color DC	A SCHEMATIC1: PAGE1 Default 12V	Display Properties		<u> </u>	
Designator Graphic ID	VDC.Normal	Name: DC	Font Arial 7 (default)		
Implementation Implementation Path Implementation Type	PSpice Model	Value: 12V	Change Use Default		
Name Part Reference PCB Footprint	100005 VCC	Display Format Do Not Display	Color-		DC = 12V
Power Pins Visible Primitive PSpiceOnly		Value Only Name and Value Name Only	Rotation		$ \begin{array}{c} \underline{} \\ \underline{} $
PSpiceTemplate Reference	V*@REFDES %+ %- @DC VCC	© Both if Value Exists	© 0° © 180° © 90° © 270°		
Source Library Source Package Value	CAPSPICE.CAPTURELD VDC VDC	ОК	Cancel Help		I
▲► \Parts (Sche	matic Nets 〈Pins 〈 Title Blocks 〈 Globals 〈 Port	s (Aliases /			22

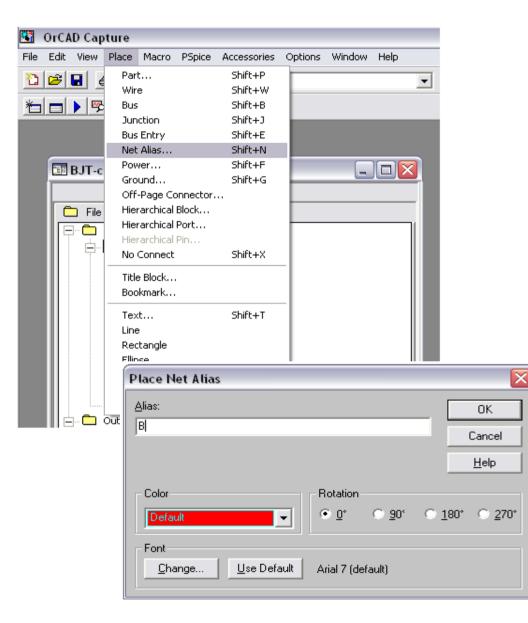
SPICE SI units and prefixes

name	SI	SPICE	C/C++ style
tera	Т	T, t	1e12, 1E12
giga	G	G, g	1e9, 1E9
mega	Μ	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	р	Р, р	1e-12, 1E-12
femto	f	F, f	1e-15, 1E-15

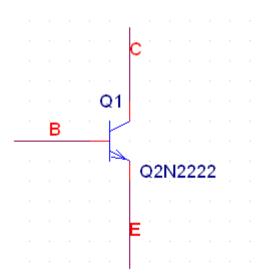
- SPICE is *not case-sensitive*, upper case and lower case letters are equivalent
- be careful not to use M for mega! 15Mohm are 15 milliohm for SPICE
- *the unit name can be neglected*, hence 10 and 10V are equivalent for SPICE
- numerical values and prefixes must be typed without spaces, e.g. 10uF,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

Naming nets

$Place \rightarrow Net Alias...$



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



Back to the ground...

-				
		Π		
	1			
		Mirror Horizontally		
		Mirror Vertically	Ľ.,	
		Rotate	1	
		Edit Properties	÷ .	
		Ascend Hierarchy	÷ .	
		Zoom In		
		Zoom Out		
		Go To		
		Cut	1	
		Сору	1	
		Delete	· -	

Property Editor			
New Row Apply Display Delete Property Filter by: < Current prope	erties >	▼ Help	
A SCHEMATIC1 : PAGE1 : 0 Itame O SDTSourceLibName //C:VdU#DEVICE.LIB/// Source Library Source Symbol	Display Properties		<u> </u>
	Name: Name Value: 0 Display Format O Da Nat Display Value Only O Name and Value O Name Only O Both if Value Exists OK	Font Arial 7 Change Use Default Color Default Rotation © 0° © 180° © 90° © 270° Cancel Help	*
▼ ▶ \ Parts & Schematic Nets & Pins & Title Blocks & Globals & Ports & A	Aliases /		>

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

Complete schematic

/ - (SCHEMATIC1 : PAGE1)	_ 🗆 🖂
5 4 3	
	· · · · ·
	· · · · ·
	· · · · ·
••••••••••••••••••••••••••••••••••••••	
$\begin{array}{c} ID \\ \hline \\ \hline \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ $	
\square	
Q2N2222 - Vcc =	Vco
	VCE
IBB DC = 1uA	
DC = 1uA	
· · · · · · · · · · · · · · · · · · ·	
	· · · · · •
	► //

Useful shortcuts

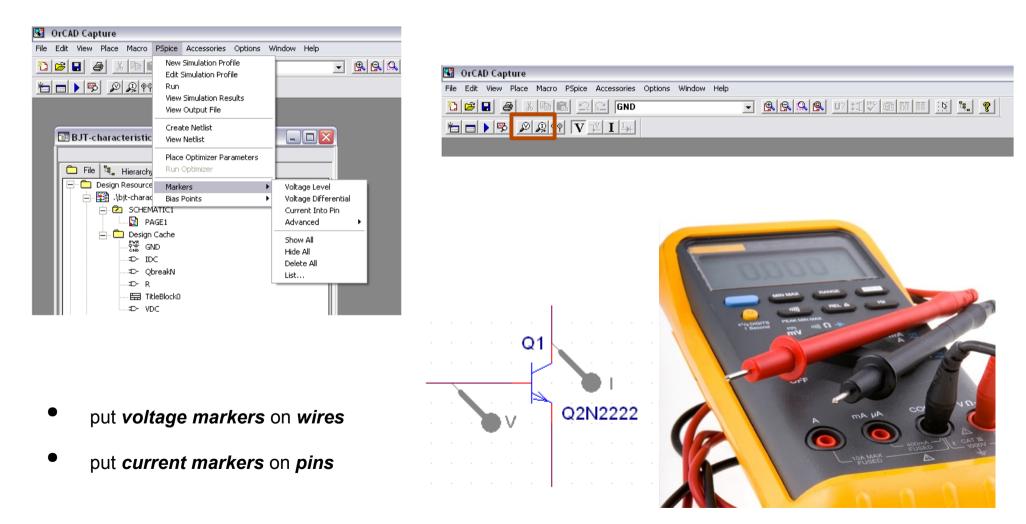
Capture shortcut	description
Р	place part
Ctrl + A	add library
G	place ground
F	place power
Ctrl + E	edit component properties
W	place wire
Ν	place net alias
J	place junction
ESC	end mode
R	rotate component
H / V	mirror horizontally/vertically
Т	place text
Ι /Ο	zoom in/out
Ctrl + X / Ctrl + V	cut/paste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

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



Voltage and current markers

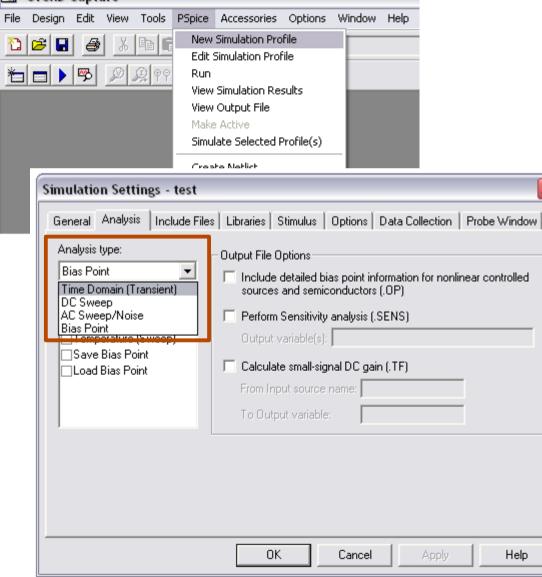
$\textbf{PSpice} \rightarrow \textbf{Markers} \rightarrow \textbf{Voltage Level / Current Into Pin}$



Simulation profiles

Great Capture

PSpice → New Simulation Profile



New Simulation	×
Name: test	Create
Inherit From:	Cancel
none	
Root Schematic: SCHEMATIC1	

- transient analysis
- DC sweeps

X

- frequency (AC) analysis
- bias point

Bias point

Iarge signal DC solution for a particular input voltage/current condition

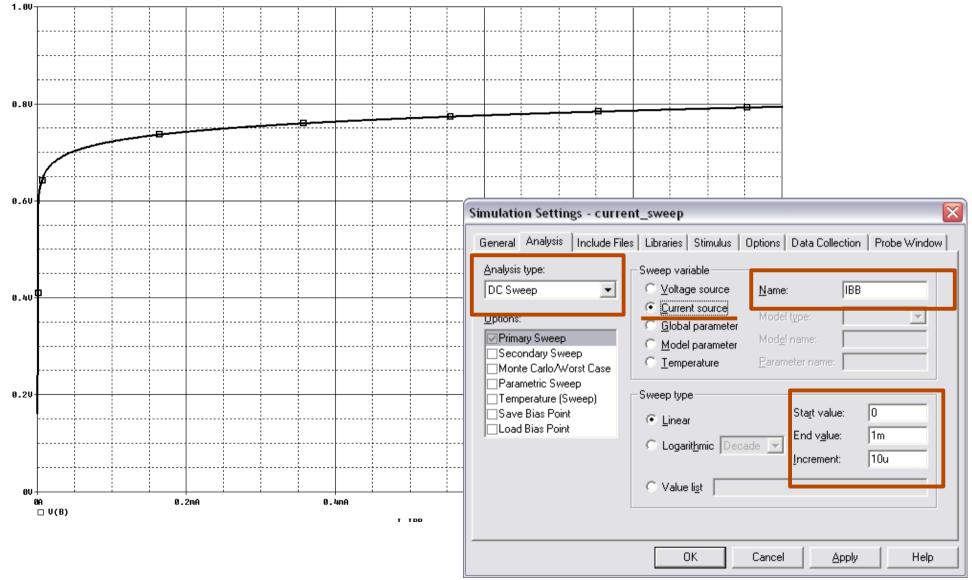
• the time is removed from the circuit

- 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

- Iarge 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

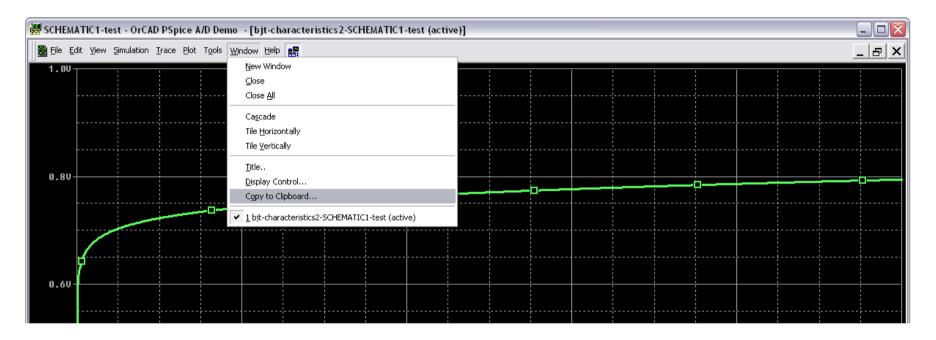
Vbe vs lbb (Vce = const)

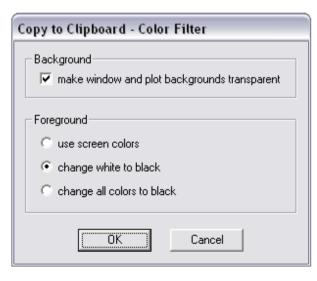


Is it in agreement with our expectations?

Export the plot image

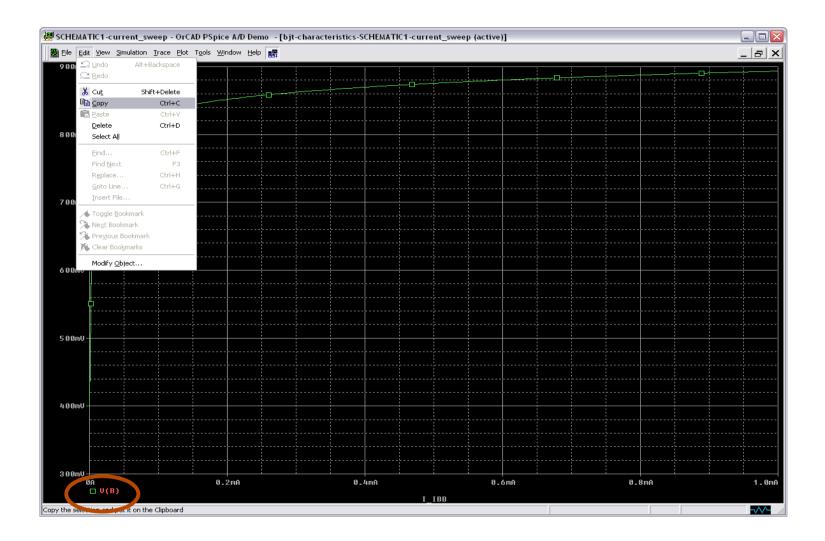
Window \rightarrow Copy to Clipboard...





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

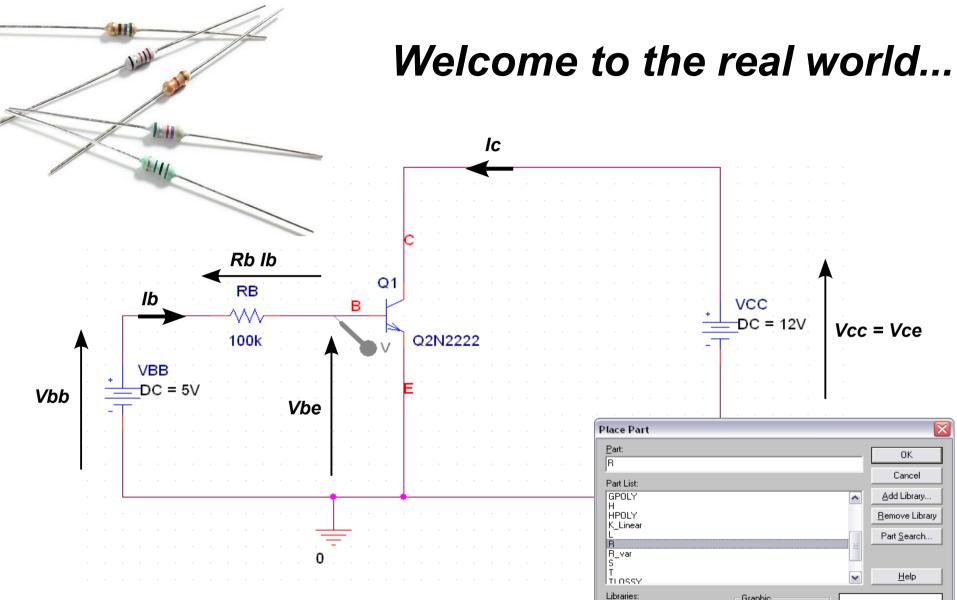
Export numerical values



- 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 $Edit \rightarrow Copy$

🕞 Untitled - Notepad		
File Edit Format View Help I Undo Ctrl+Z Ctrl+Z Cut Ctrl+X Copy Ctrl+C Paste Ctrl+V Delete Del		
Find Ctrl+F	🕟 Untitled - Notepad	_ 🗆 🔀
Find Next F3	Eile Edit Format View Help	
Replace Ctrl+H Go To Ctrl+G	I_IBB V(B) 0 0.399951636791229	<u>^</u>
Select All Ctrl+A Time/Date F5	$ \begin{array}{c} 0 & 0. 399951636791229 \\ 5e-006 & 0.756280660629272 \\ 1e-005 & 0.784695386886597 \\ 2e-005 & 0.792136669158936 \\ 2.5e-005 & 0.792132661444 \\ 3e-005 & 0.80661004781723 \\ 4e-005 & 0.80661004781723 \\ 4e-005 & 0.810063719749451 \\ 4.5e-005 & 0.813110113143921 \\ 5e-005 & 0.813830306797028 \\ 6e-005 & 0.822620987892151 \\ 7e-005 & 0.822620987892151 \\ 7e-005 & 0.824537754058388 \\ 7.5e-005 & 0.824537754058388 \\ 7.5e-005 & 0.824537754058388 \\ 7.5e-005 & 0.824532795445381165 \\ 9e-005 & 0.8237594455991058 \\ 8.5e-005 & 0.823559445381165 \\ 9e-005 & 0.823559445381165 \\ 9e-005 & 0.8235956604 \\ 0.00011 & 0.833762884140015 \\ 0.00011 & 0.83372958604 \\ 0.00011 & 0.83622795343399 \\ 0.000115 & 0.837377667427063 \\ 0.000125 & 0.8395342864326 \\ 0.000125 & 0.8395342864326 \\ 0.000135 & 0.841524839401245 \\ 0.00014 & 0.842465460300446 \\ 0.00015 & 0.842465460300446 \\ 0.00015 & 0.842465460300446 \\ 0.00015 & 0.842465460300446 \\ 0.00015 & 0.842465460300446 \\ 0.00015 & 0.842465460300446 \\ 0.00015 & 0.842465460300446 \\ 0.00015 & 0.842465460300446 \\ 0.00015 & 0.842465460300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.84265466300446 \\ 0.00015 & 0.8426546632491 \\ 0.00015 & 0.8426548841757 \\ 0.00018 & 0.848265488524871 \\ 0.00018 & 0.848965466022491 \\ 0.00018 & 0.848965466022491 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.00018 & 0.849674165248871 \\ 0.0001$	

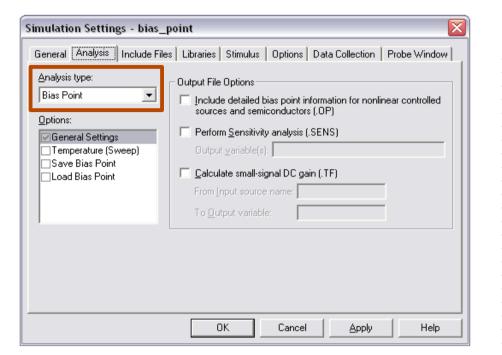
- open a text editor (*Notepad* works fine)
- simply do a 'paste' inside the text editor
- save the file with the *.csv* extension, it can be opened with Excel, Mathematica, ROOT etc.

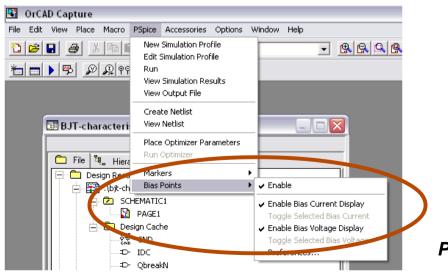


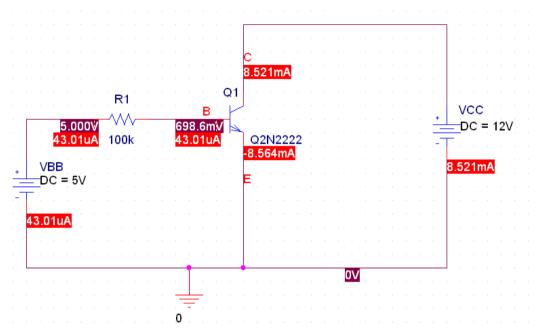
This is the circuit you have to deal with in the **lab experience #8**!

H HPOLY K_Linear L R_var S T LIOSSY Libraries: ABM ANALOG ANALOG ANALOG ANALOG Part Search... Breakour Design Cache EVAL SOURCE SOURCE SOURCESTM SPECIAL Part Search... Graphic Corrvett Packaging Parts per Pkg: 1 Parts Processon Type: Homogeneous I Mormal Corvett The Source Structure Type: Homogeneous

Bias point analysis



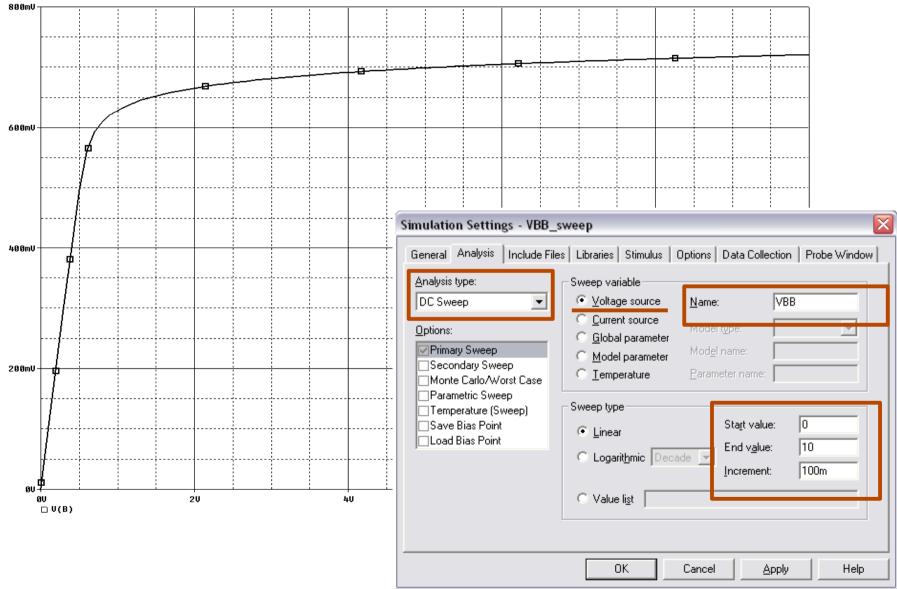




🕼 OrCAD Capture	
File Edit View Place Macro PSpice Accessories Options Window Help	

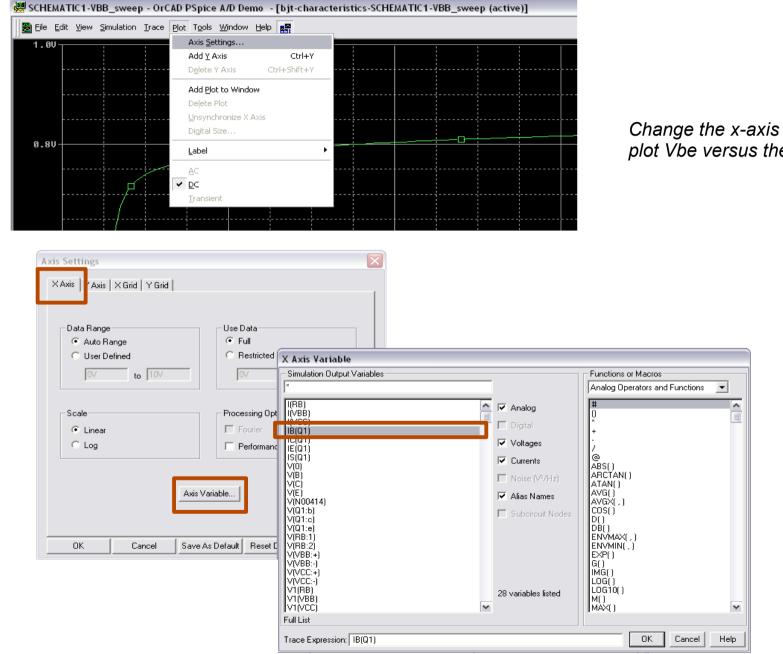
 $\textbf{PSpice} \rightarrow \textbf{Bias Points} \rightarrow \textbf{Enable Bias Current (Voltage) Display}$

Vbe vs Vbb (Vce = const)

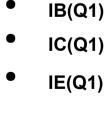


Vbe vs lb (Vce = const)

Plot \rightarrow **Axis Settings**...

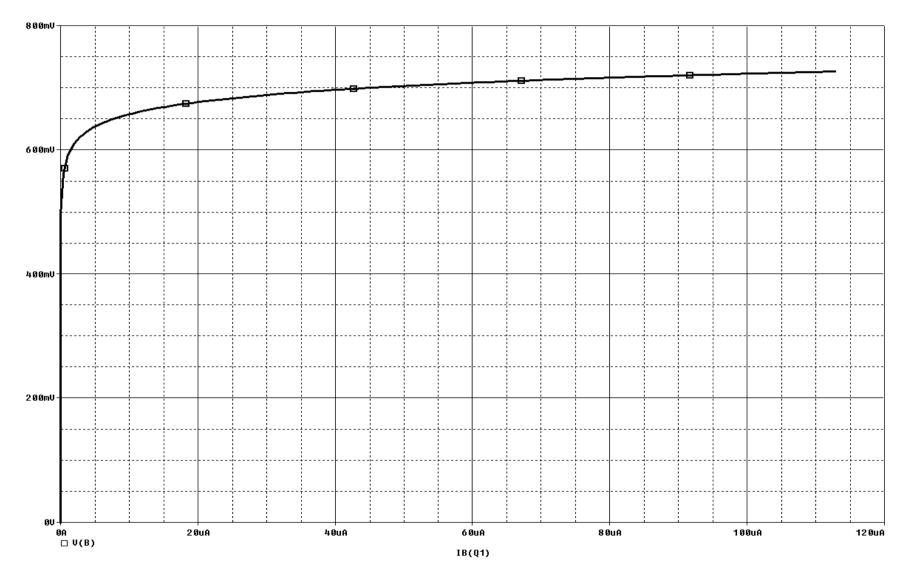


Change the x-axis variable in order to plot Vbe versus the base-current again



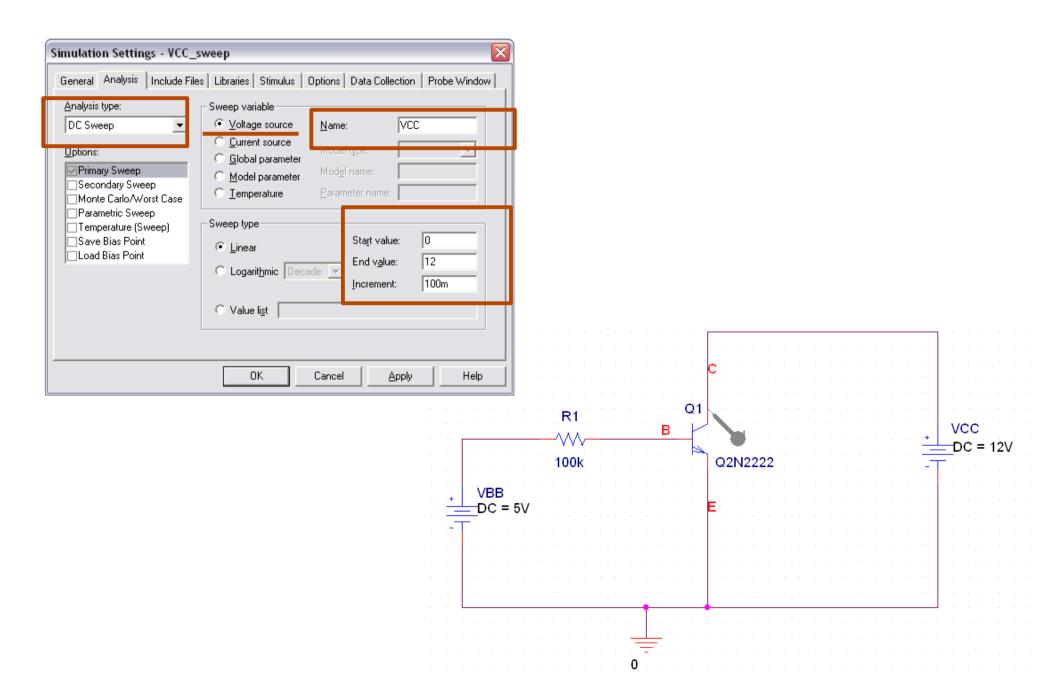
Vbe vs lb (Vce = const)

Input characteristic, Vbe = f(Ib)



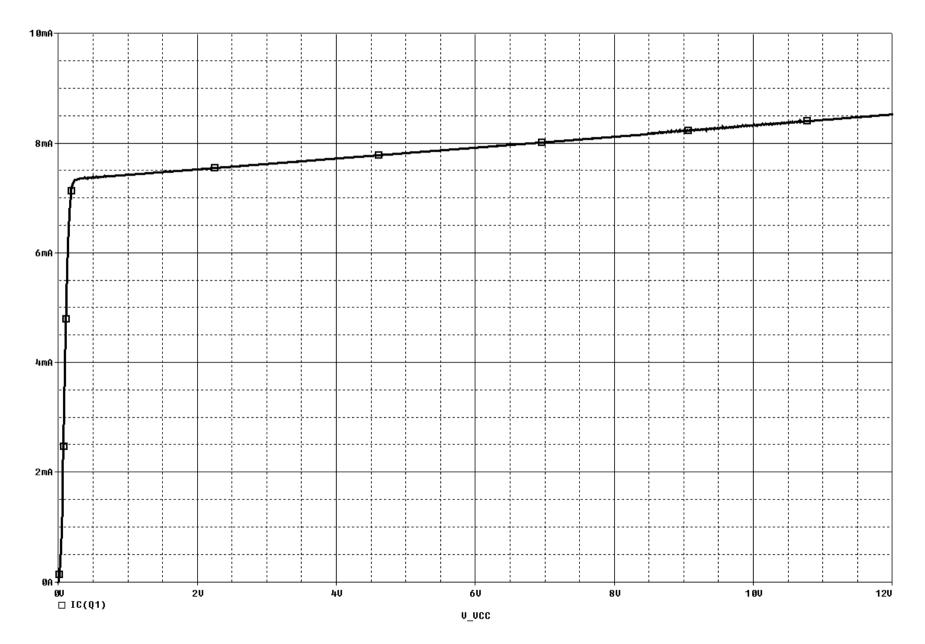
When Vbe reaches ~700 mV the transistor is in full conduction regime !

Ic vs Vce (Vbe = const)



Ic vs Vce (Vbe = const)

Output characteristic, Ic = f(Vce)



Nested DC sweeps

Simulation Settings - VCC_sweep		
General Analysis Include File	es Libraries Stimulus Options Data Collection Probe Window	
Analysis type: DC Sweep	Sweep variable • Voltage source • Current source • Model type:	
Primary Sweep Secondary Sweep Monte Carlo/Worst Case	C Global parameter C Model parameter Model parameter Model name: C Temperature	
Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	Sweep type Start value: 0 Image: End value: 12	
	C Logarithmic Decade Increment: 100m	
	OK Cancel <u>Apply</u> Help	

Analysis type:	Sweep variable © <u>V</u> oltage source	Name: VBB
Deptions: Primary Sweep Secondary Sweep Monte Carlo/Worst Case	C <u>C</u> urrent source <u>G</u> lobal parameter <u>M</u> odel parameter <u>I</u> emperature	Model type:
Parametric Sweep I emperature (Sweep) Save Bias Point Load Bias Point	Sweep type C Linear C Logarit <u>h</u> mic Deca	de v Sta <u>r</u> t value: 1 End v <u>a</u> lue: 5 Increment: 1

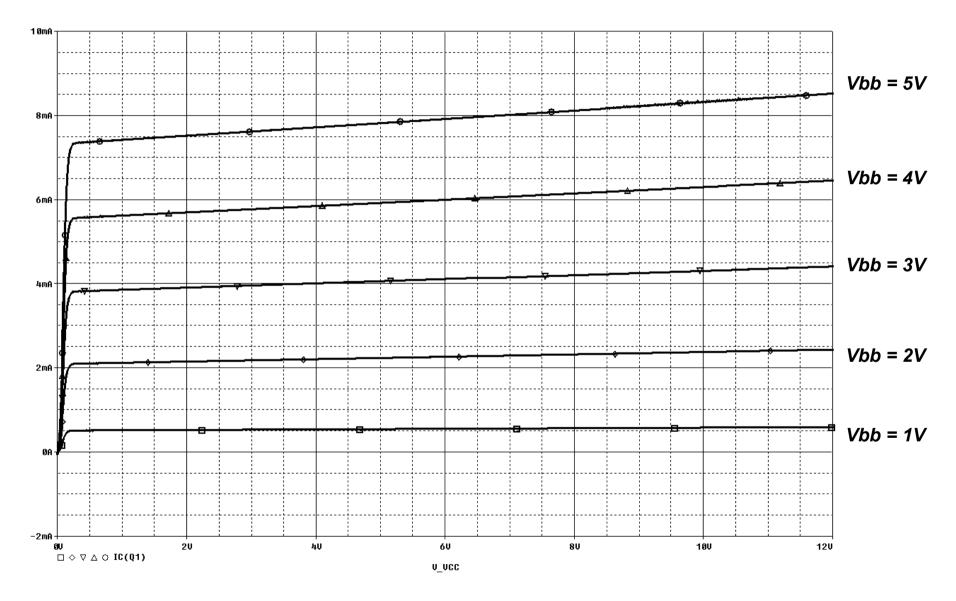
Primary sweep by varying Vcc for a fixed value of Vbb (and therefore of Vbe)

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

N.B.

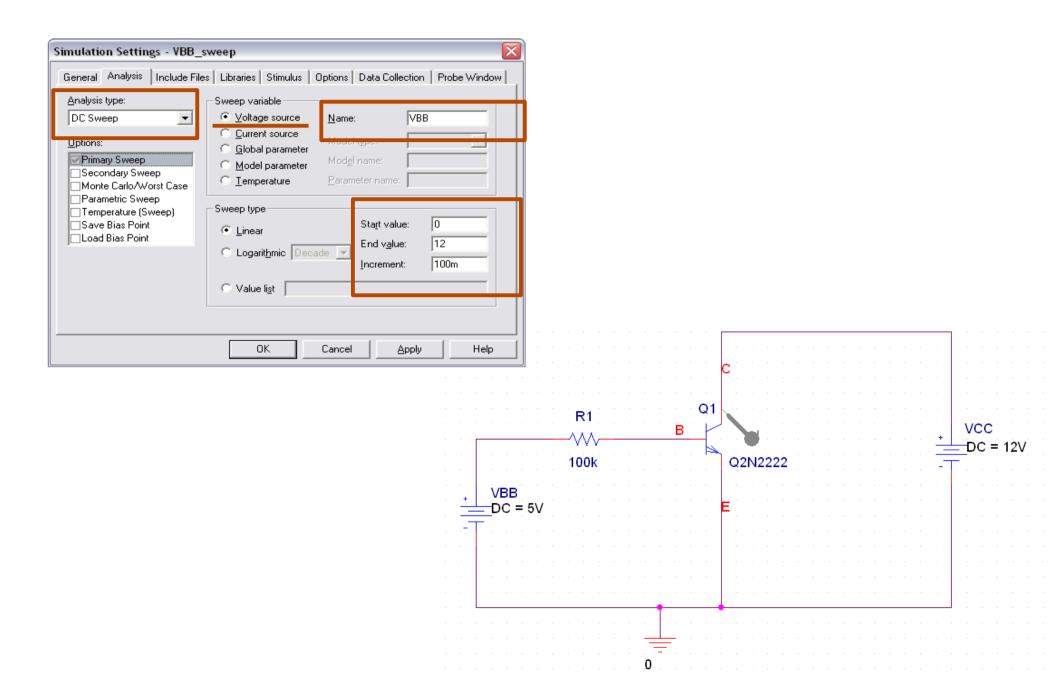
The simulation profile is just one, don't create two different profiles !

Ic vs Vce for different Vbb

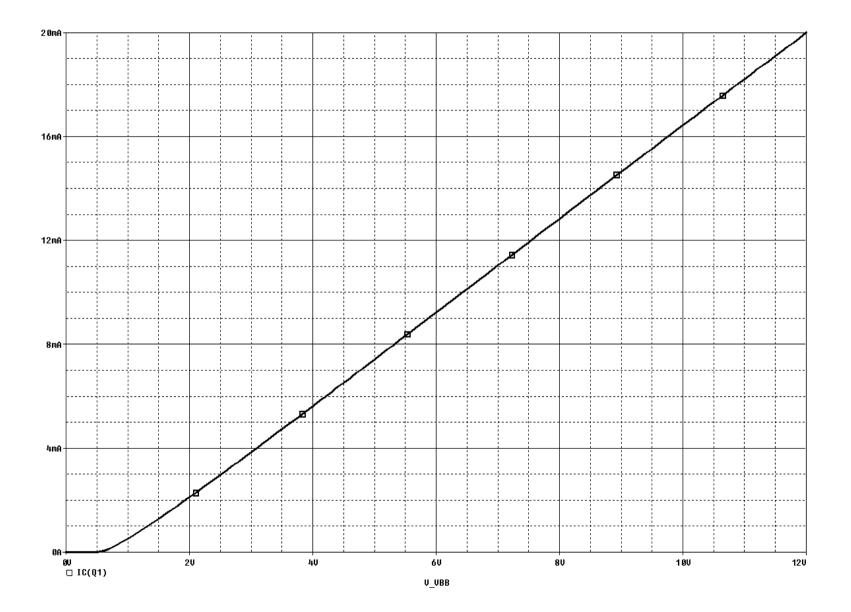


What's happening?

Ic vs Vbb (Vce = const)

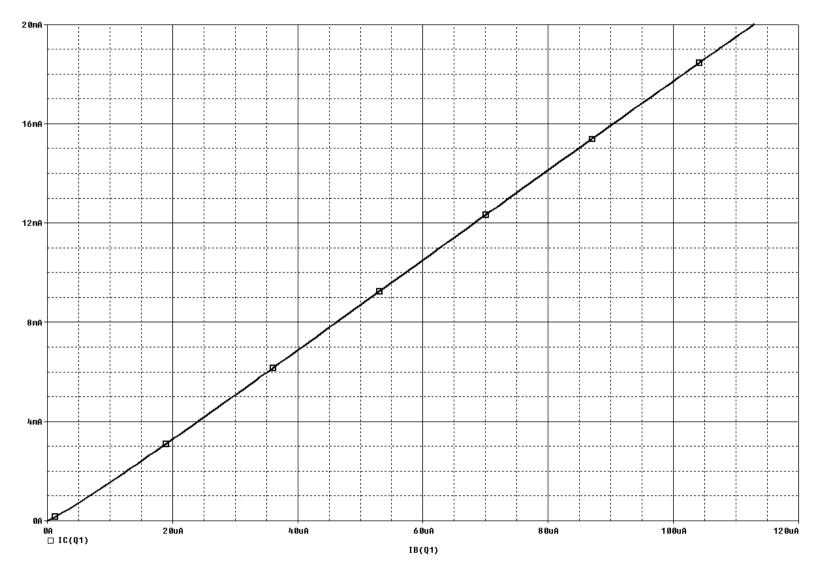


Ic vs Vbb (Vce = const)



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)

I_C = 10 mA; V_{CE} = 10 V

20 MA, VCE - 20 V, I - 100 MIL

I_{Con} = 150 mA; I_{Bon} = 15 mA; I_{Boff} = -15 mA

75

250

300

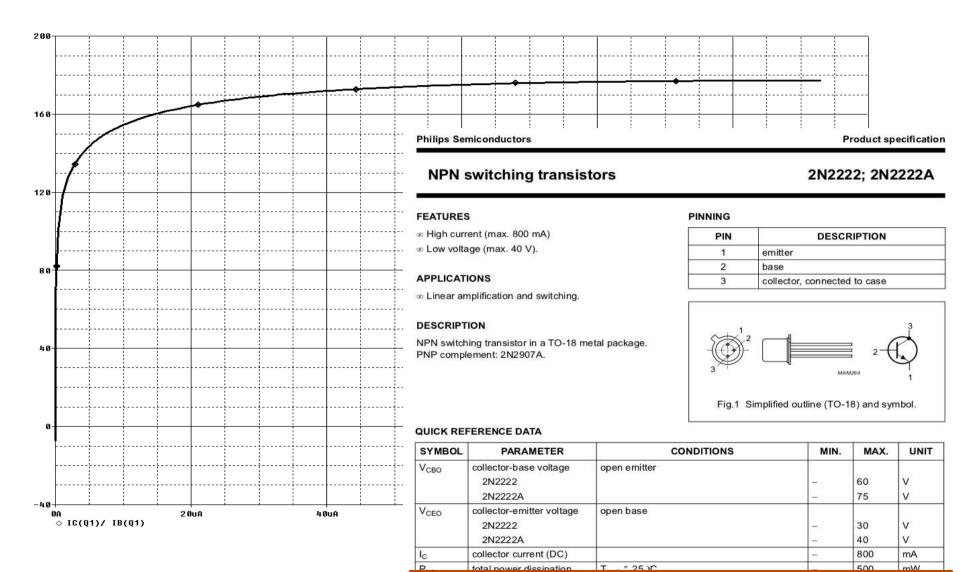
_

250

MHz

MHz

ns



DC current gain

2N2222

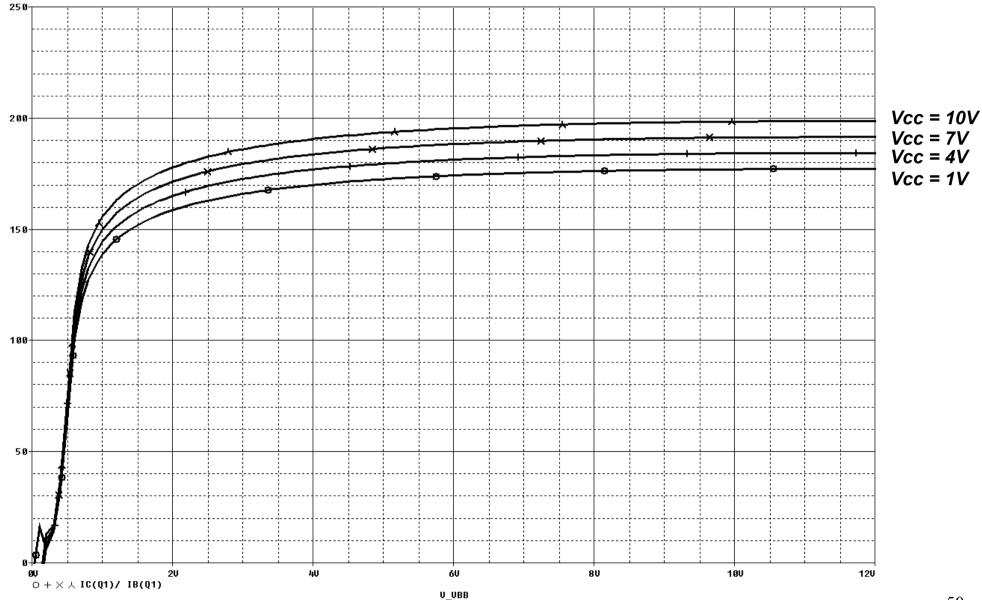
2N2222A turn-off time

answon nequency

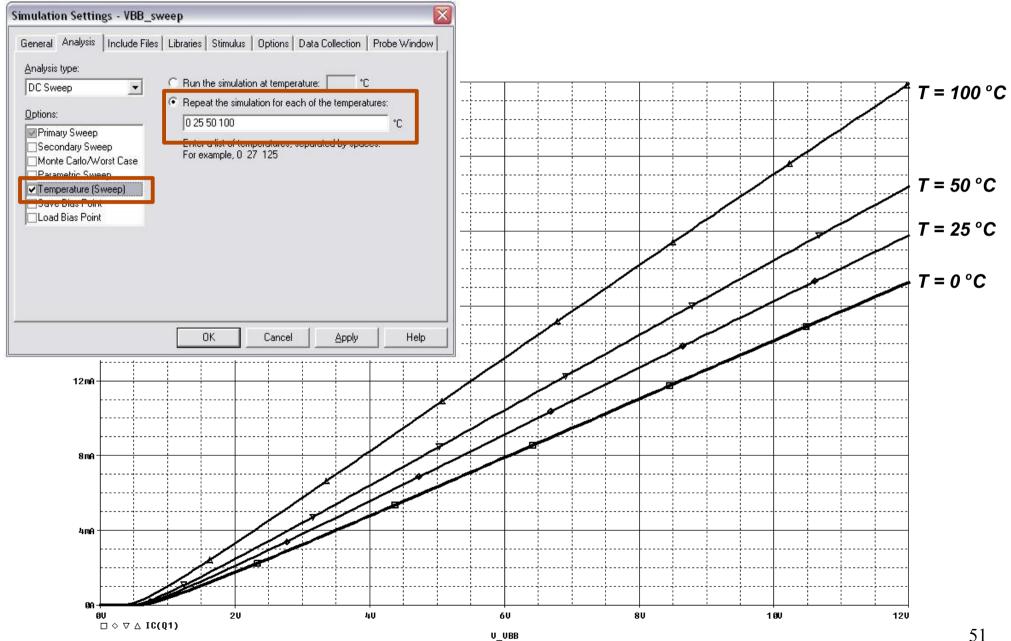
h_{FE}

toff

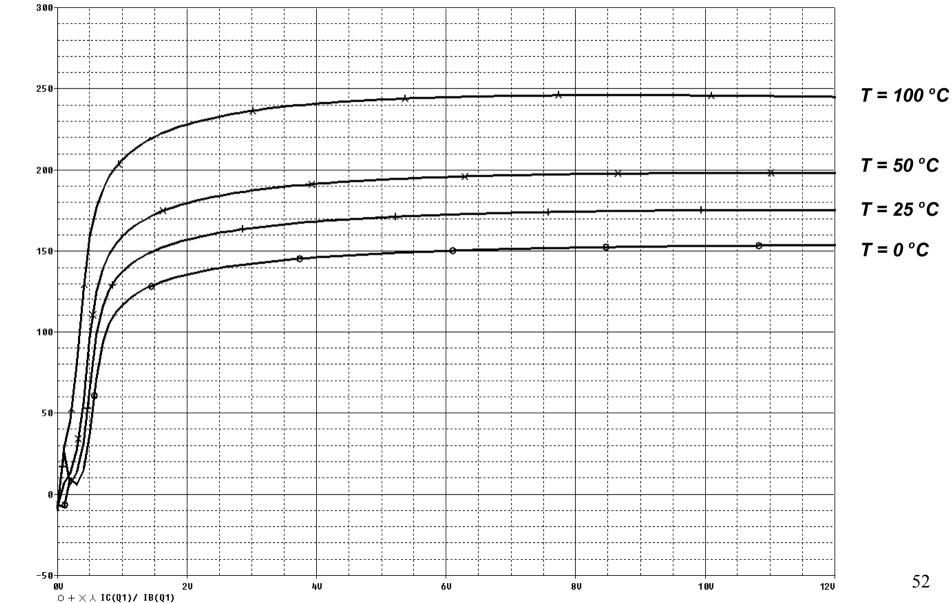
*h*_{FE} = *Ic/Ib* vs *Vbb* (*Vce sweep*)



Temperature sweep



Temperature sweep



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

Lab measurements

...

...

• input characteristics \rightarrow Ib vs Vbe, Vce = const

Vce = Vce1		Vce = Vce2	
lb [µA]	Vbe [mV]	lb [µA]	Vbe [mV]

• output characteristics \rightarrow Ic vs Vce, Vbe = const

lb = lb1		lb = lb2	
lc [µA]	Vce [mV]	lc [μA]	Vce [mV]

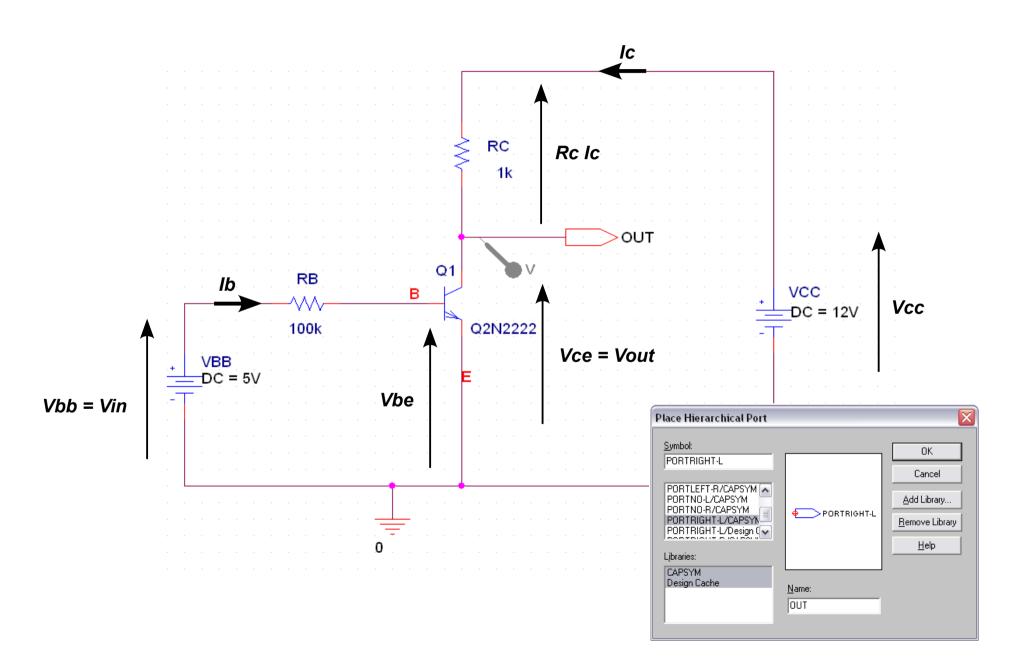
...

...

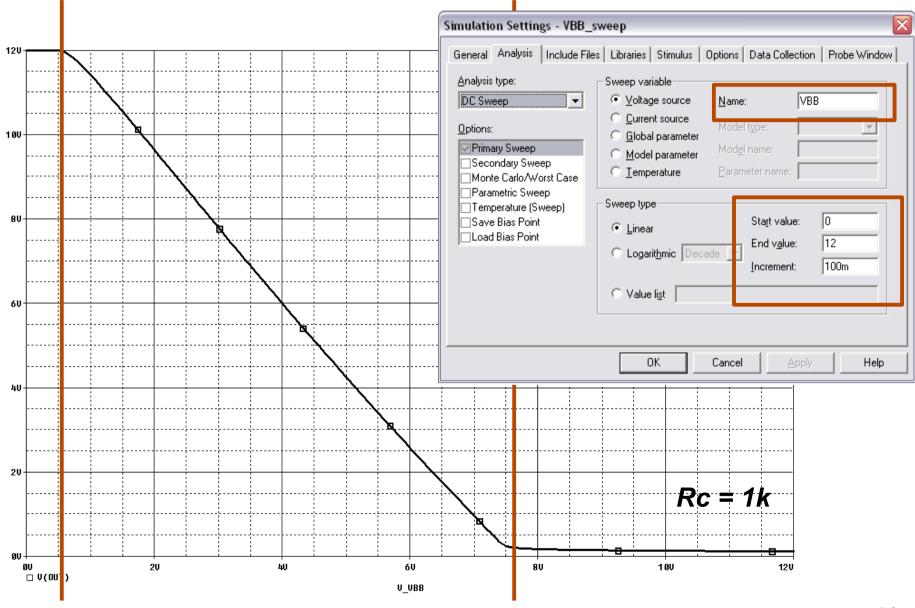
Part III NPN common-emitter amplifier



NPN common-emitter amplifier

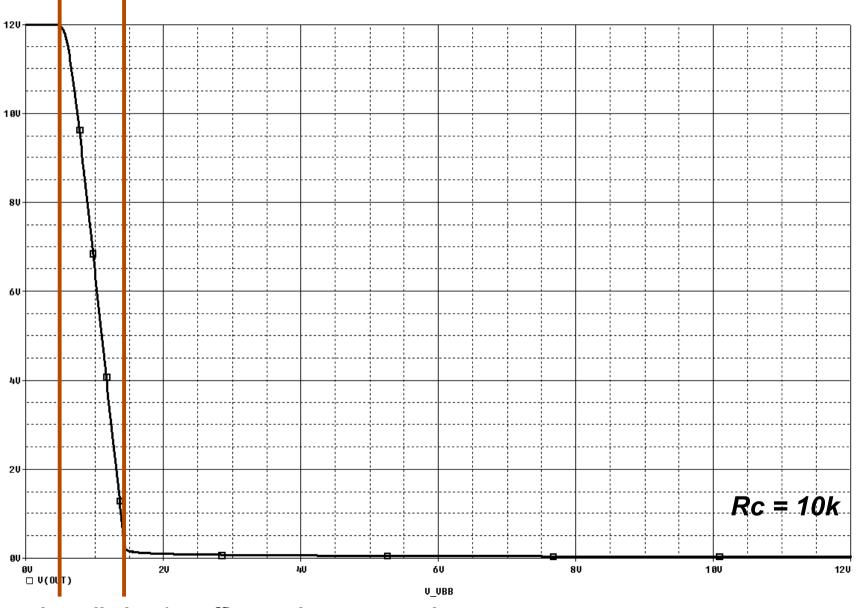


Voltage Transfer Characteristic (VTC)



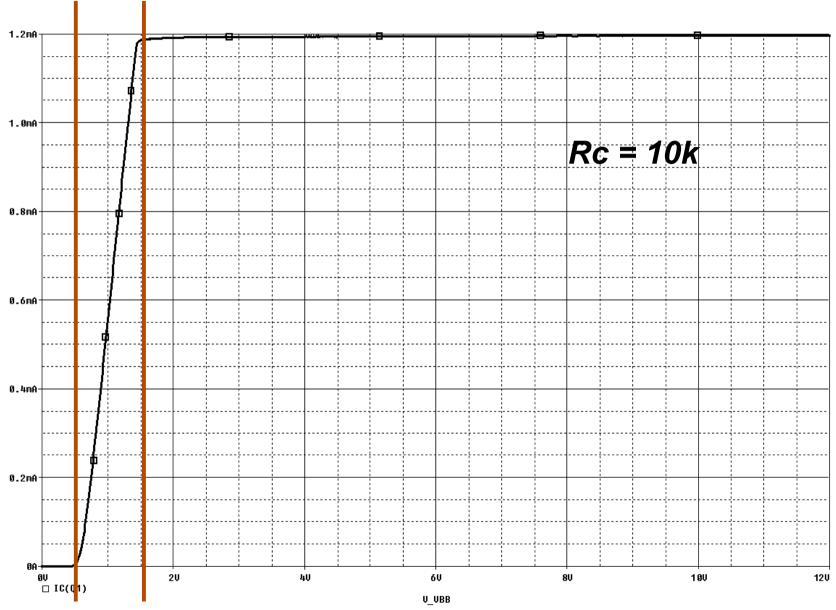
interdiction (cutoff) \rightarrow active \rightarrow saturation

Increasing the load



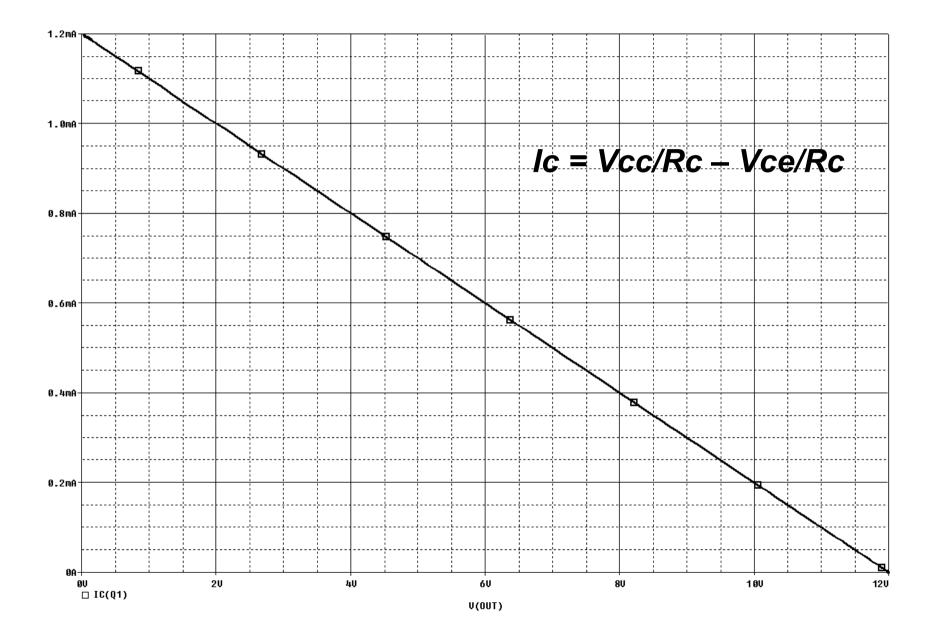
interdiction (cutoff) \rightarrow active \rightarrow saturation

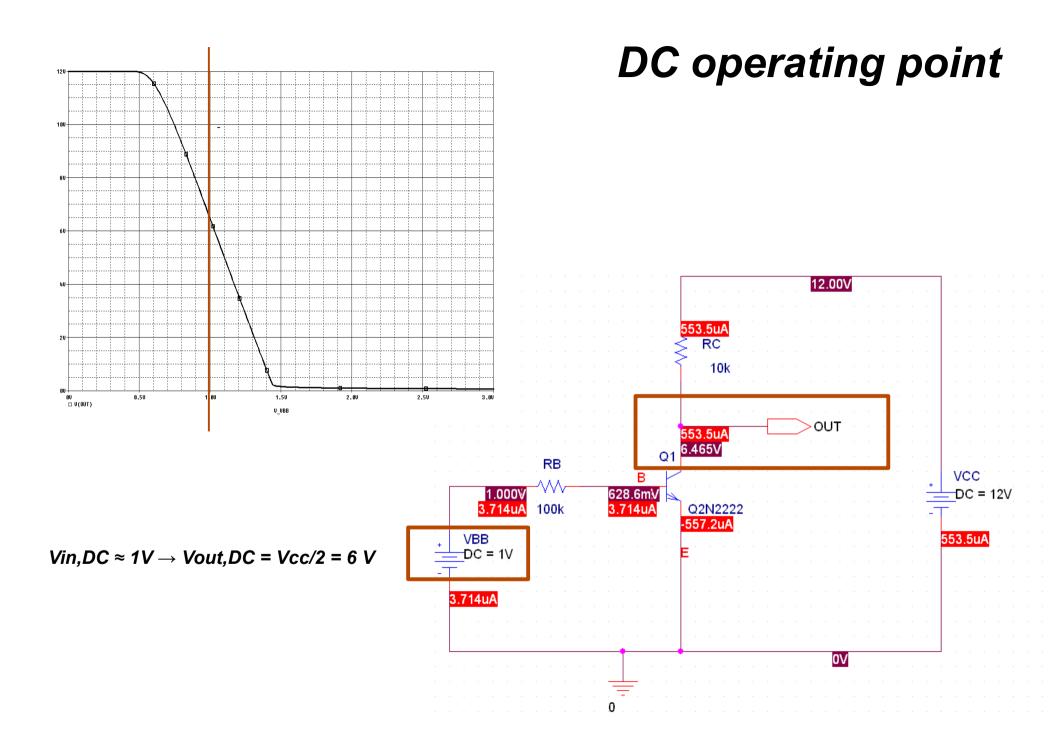
Ic vs Vbb



interdiction (cutoff) \rightarrow active \rightarrow saturation

Ic vs Vce... the load line !

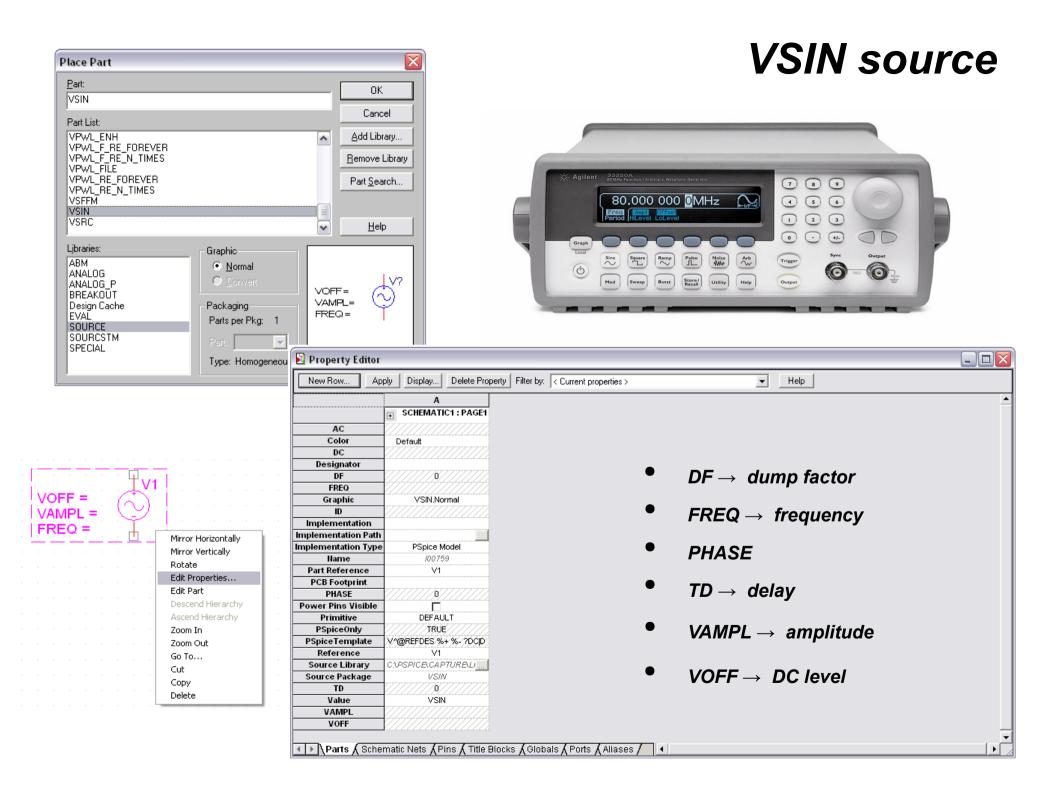


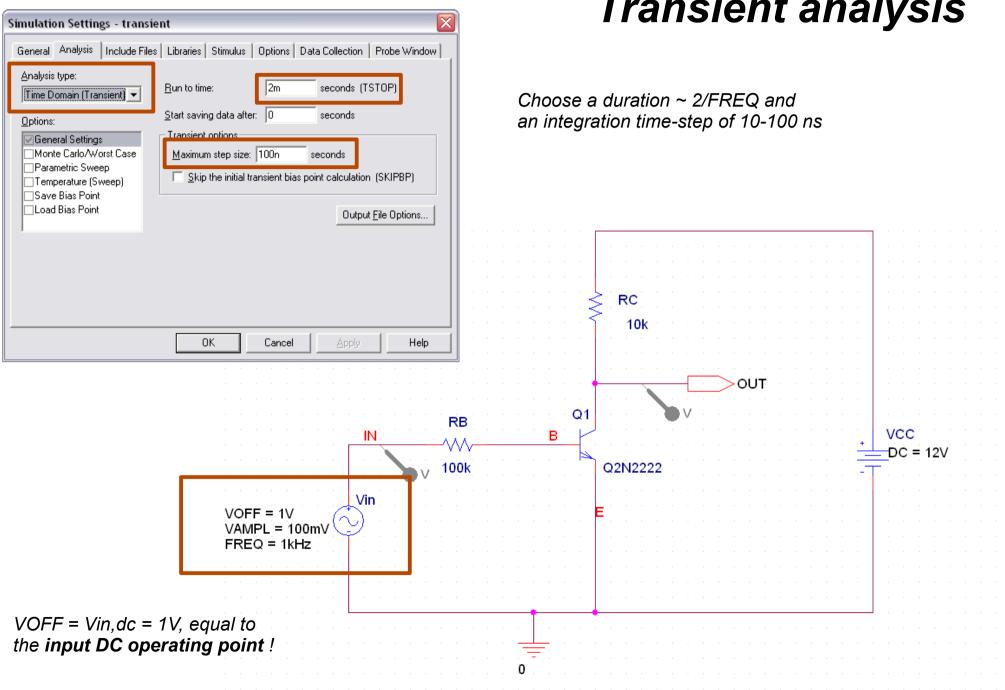


Transient analysis

- Iarge-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*

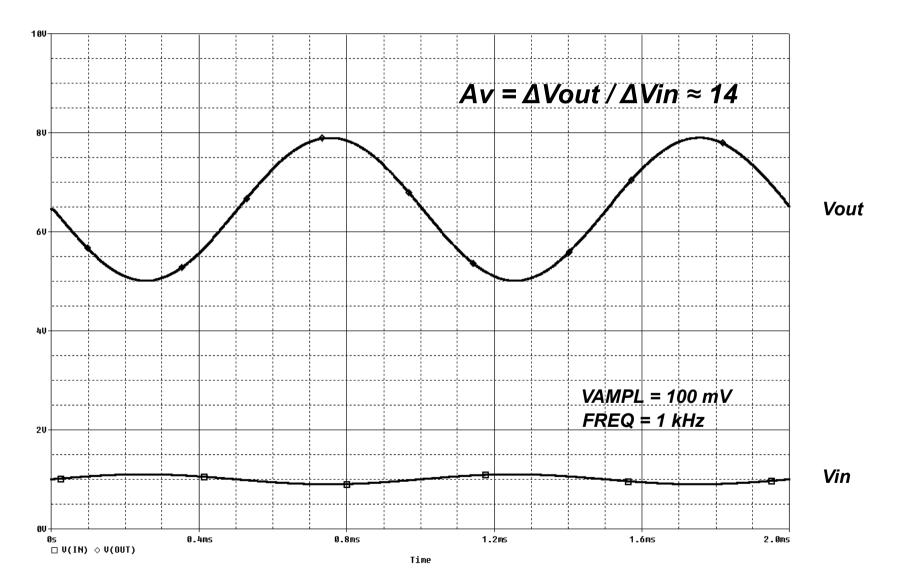
- voltages and currents are tracked over time
 - a smaller integration time step increases both the results accuracy and the simulation duration
 - sometimes convergence problems can occur
- you can specify in the simulation settings window the maximum step size in incrementing the time during transient analysis (*numerical integration time-step*)





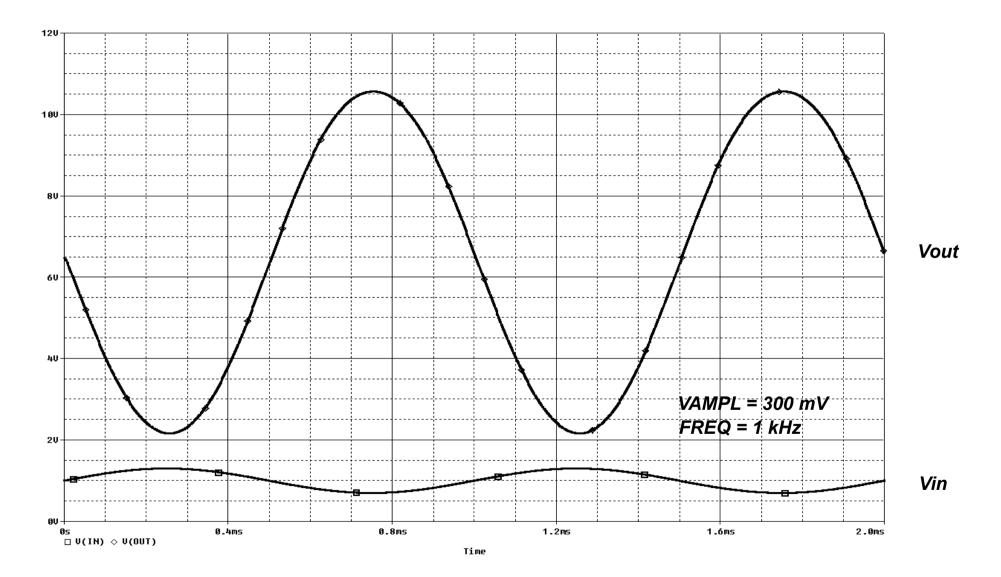
Transient analysis

Transient analysis

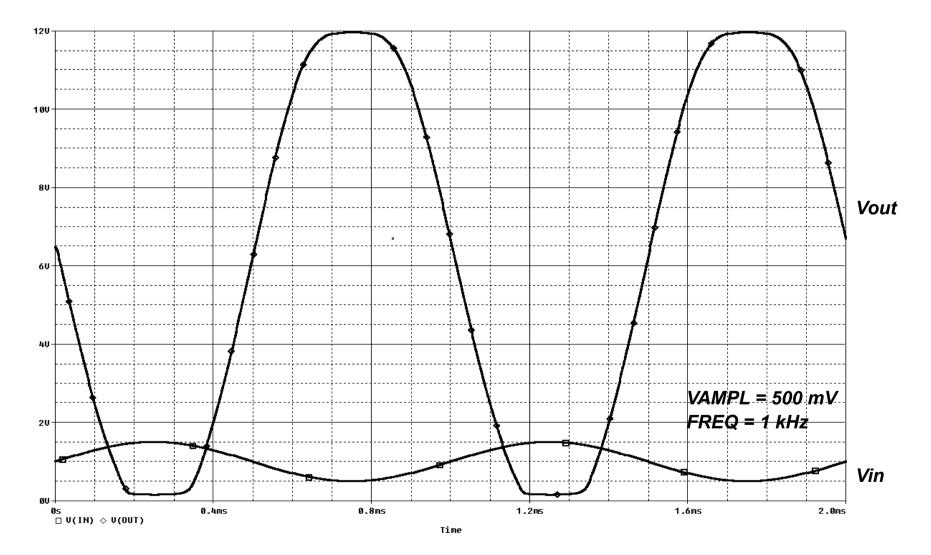


The circuit is an inverting amplifier !

Increasing the amplitude

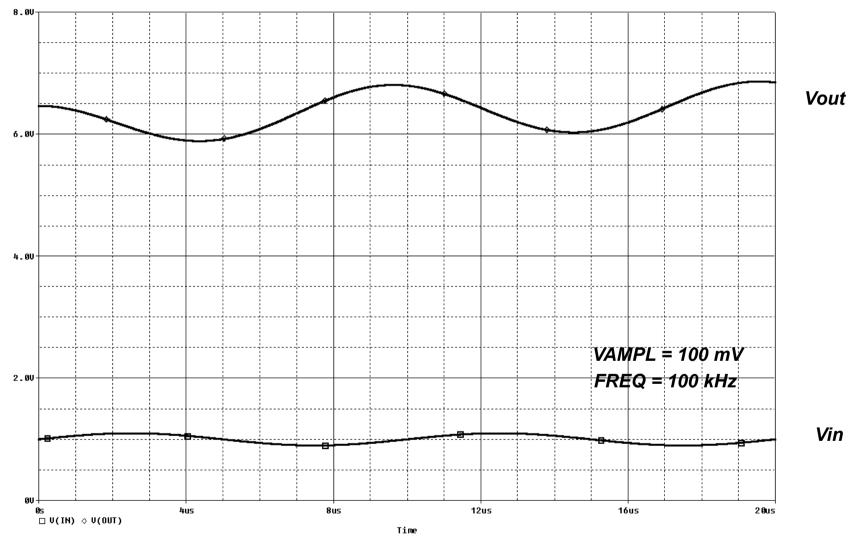


Output saturation



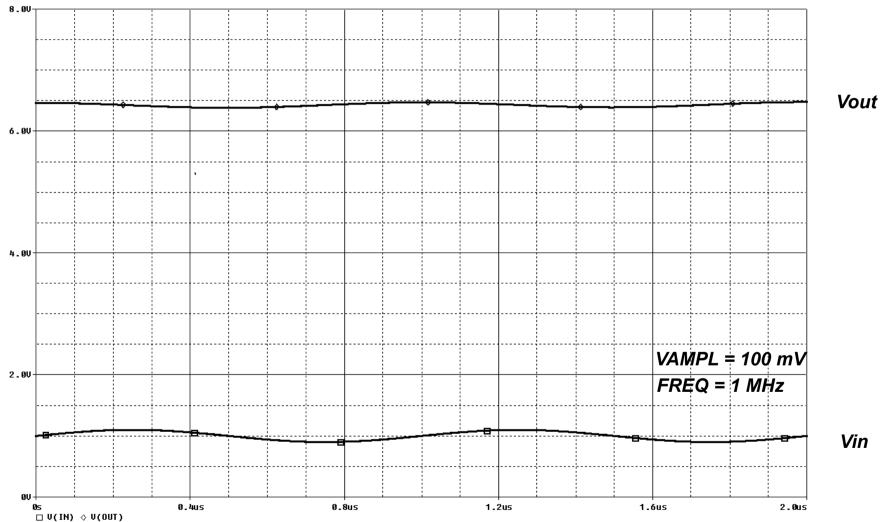
Are you surprised ?

Increasing the frequency



Note that a **phase difference** exist !

Increasing the frequency

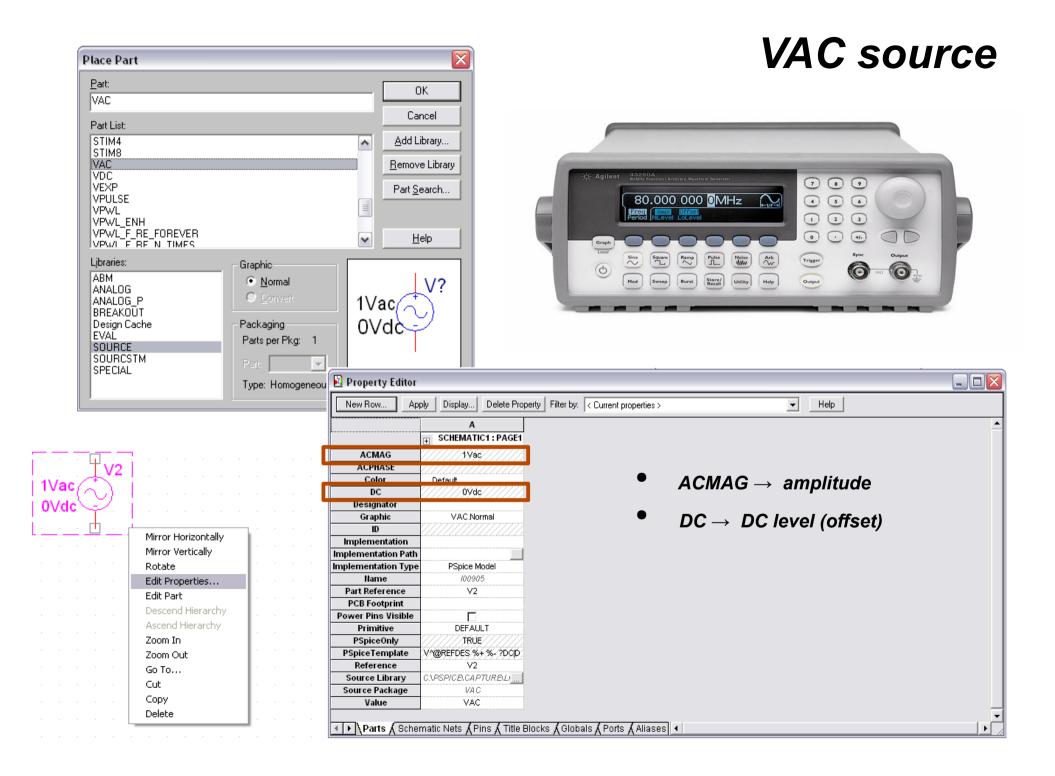


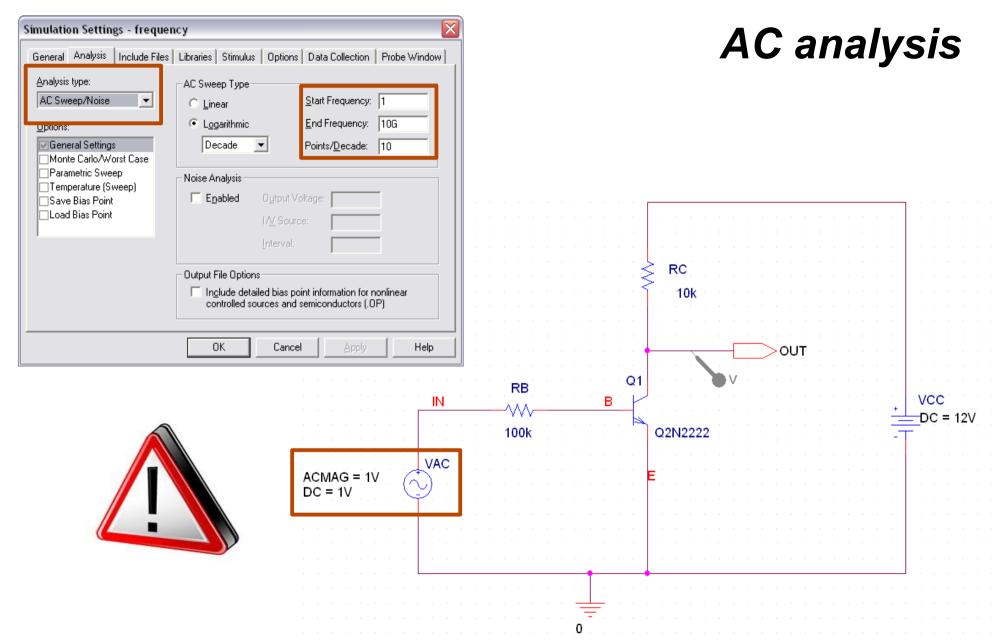
Time

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

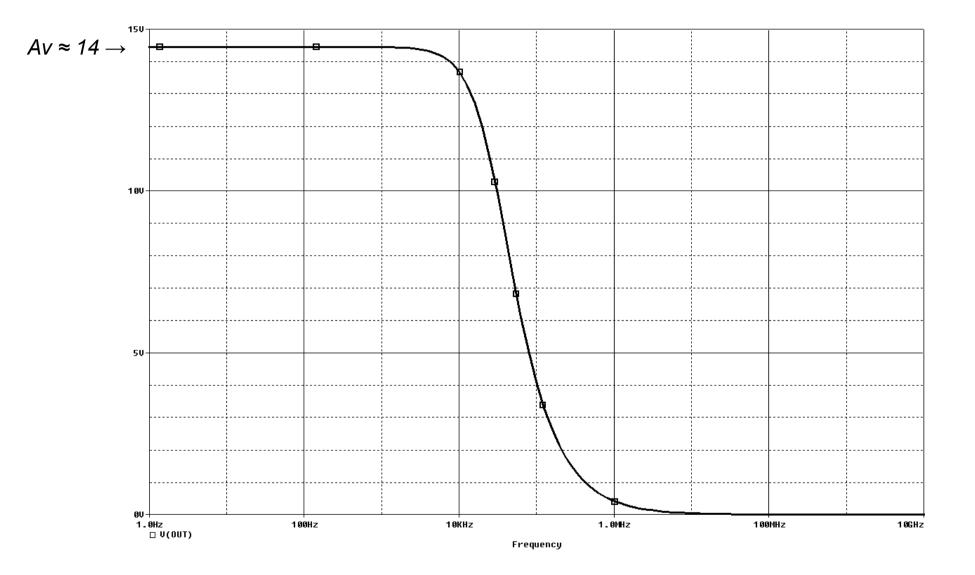




Remind !

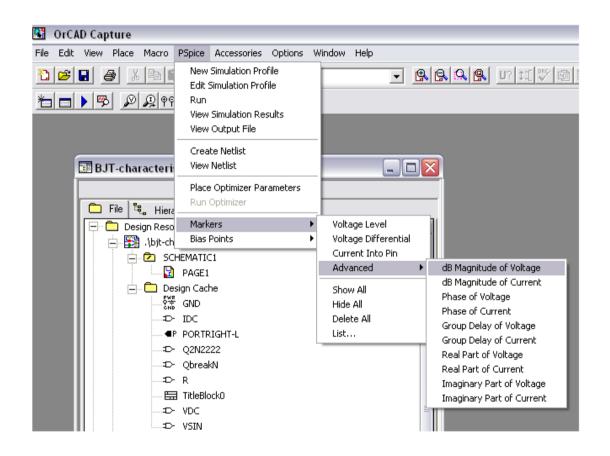
The AC analysis is a **small signal** analysis! The circuit is **linearized** around the DC operating point, hence you can use any ACMAG value! By using ACMAG=1V you immediately obtain **the voltage gain plot** with Vout(f) = Vout(f) /1^{*}M

Vout/Vin frequency response



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

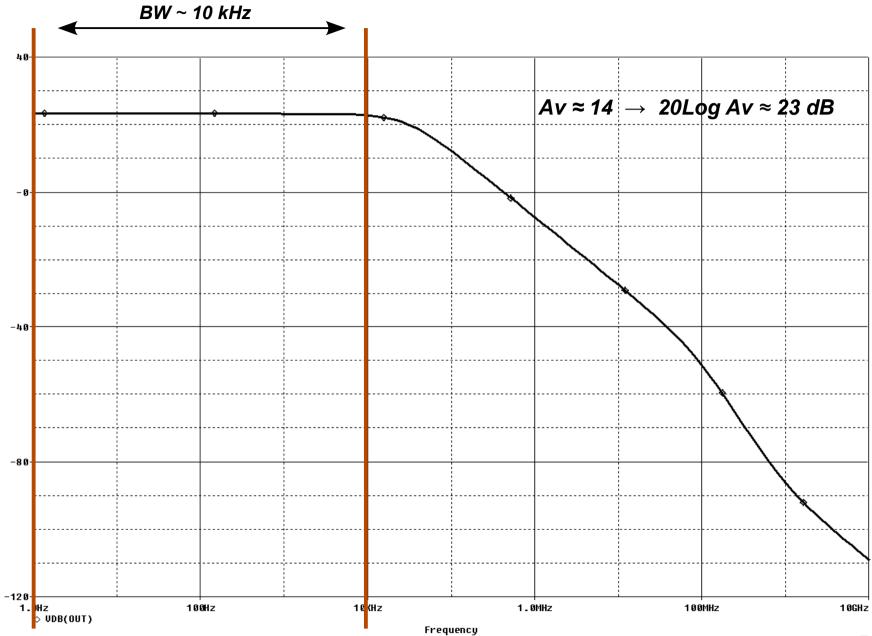
dB voltage gain



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

 $PSpice \rightarrow Markers \rightarrow Advanced \rightarrow dB Magnitude of Voltage$

dB voltage gain



Backup

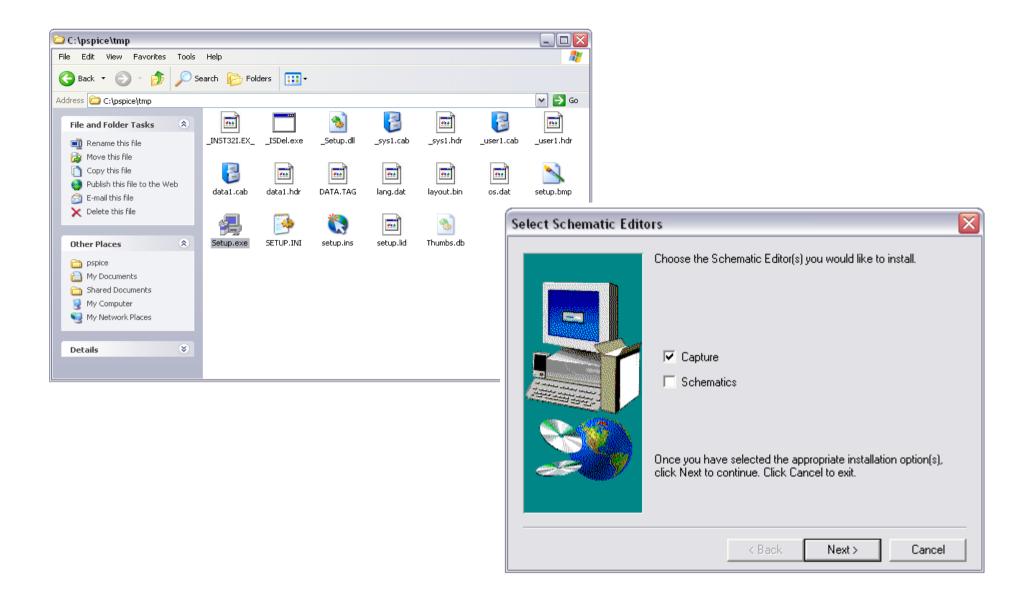
Installation details

- 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

C:\pspice					
File Edit View Favorites Tools	Help		A		
🌀 Back 🝷 🕥 🕤 🏂 🔎 S	earch 😥 Folders 🔢 🕶				
Address 🛅 C:\pspice		Image: Second	Go		
File and Folder Tasks 💲	91pspstu.exe	tmp			
🗐 Rename this file					
Copy this file		WinZip Self-Extractor [91pspstu.exe]		WinZip Self-Extractor [91pspstu.exe]	
 Publish this file to the Web E-mail this file Delete this file 		To unzip all files in 91pspstu.exe to the specified folder press the Unzip button.	Unzip	To unzip all files in 91pspstu.exe to the specified folder press the Unzip button.	
		Unzip To Folder: C:\DOCUME~1\luca\LOCALS~1\Temp	Run WinZip	Unzip To Folder: C:\pspice\tmp	Run <u>W</u> inZip
Other Places 🙁		Overwrite Files Without Prompting	Close	verwrite Files Without Prompting	<u>C</u> lose
🕪 Local Disk (C:)			About		About
My Documents Constant			Help		Help
🚽 My Computer		© Nico Mak Computing, Inc. www	v. winzip. com	© Nico Mak Computing, Inc. www.	winzip.com
🧐 My Network Places					
Details 🔹					76



- 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 !

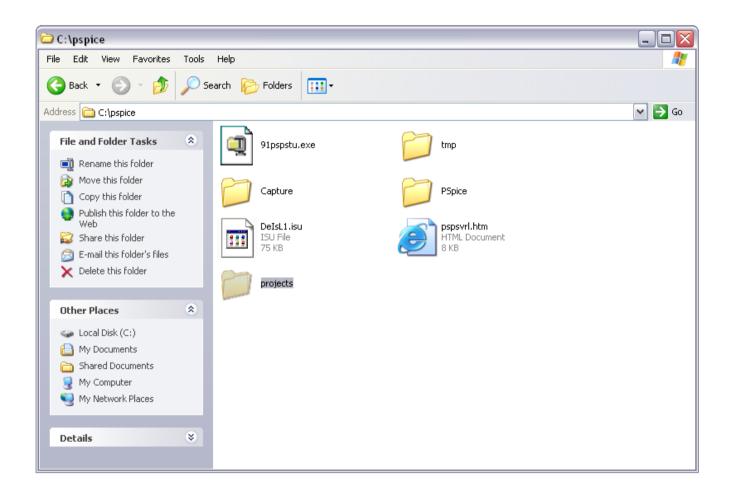
Select Installation	Select Installation Directory 🛛 🔀			
	Please select the directory where you would like to install the product(s) you have selected. Click Next to continue. Click Cancel to exit.			
	Please choose the installation folder. Path: C:\pspice Directories: Cancel Browse Cancel Cancel Drives: Circle			

Select Installation Directory 🛛 🔀		
	Please select the directory where you would like to install the product(s) you have selected. Click Next to continue. Click Cancel to exit.	
d5 9	C:\pspice	
	< <u>B</u> ack <u>N</u> ext > Cancel	

Select Program Folder	N 1997	St	art Copying Files		
	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			Setup has enough information to start copying files. If y to review or change any settings, click Back. If you are 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			< Back Next > 0	Cancel

Setup Complete		
	Installation has completed successfully. See the Re for last-minute information about this release.	lease Notes
		🖉 Release Notes for PSpice Student Version Release 9.1 - Microsoft Internet Explorer
		File Edit View Favorites Tools Help
	View the Release Notes.	Back Image: Search Image: Search
		PSpice Student Version
	Click Finish to complete Setup.	Release 9.1
	Click Finish to complete Setup.	
		Release Notes
	< Back Finish	February, 2000
		These release notes apply specifically to the PSpice Student ∀ersion Release 9.1. For detailed information about using a particular product, please refer to the online Help and documentation for that product.
		The Student Version of PSpice is intended for use by college students and professors who are interested in learning about analog and mixed-signal simulation. It is not intended to demonstrate the capabilities of any product other than PSpice. Because it is distributed freely, certain limitations have been imposed on the libraries and functionality. If you are interested in obtaining a fully functional version of PSpice, contact Orcad Sales at 1-800-671-9505. (International customers may call 1-503- 671-9500.)
		To obtain the very latest information about workarounds or solutions to problems that you may encounter, visit the Orcad Design Network on the Orcad Web site at www.orcad.com/odn
		🕘 🤤 🚽 🛃 My Computer

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