A Capture / PSPICE Primer for ELEC2210 Fall 2010 Class -

Note Title 1/22/2010

Guofu Niu
Electrical and Computer Engineering Department
Auburn University

Aug 17, 2010

Please read this tutorial I have written up specifically for the Fall 2010 class using a simple resistor circuit as example. You need to reproduce the simulations. The lab PCs in Broun have Cadence's Pspice (Capture) installed. See Mr John Newton for PC related problems.

If you wish to use your own laptop, you can obtain a disc from me.

Installation:

turn off firewall, antivirus, anti malware etc

install from the DVD

turn on everything you turned off

Start Pspice AD first if running windows 7 / vista
Start Capture

Use "Apply" in simulation profile, not "OK", keep those windows open

there are some "tricks" like this depending on OS you use.
We will deal with them as we go along.

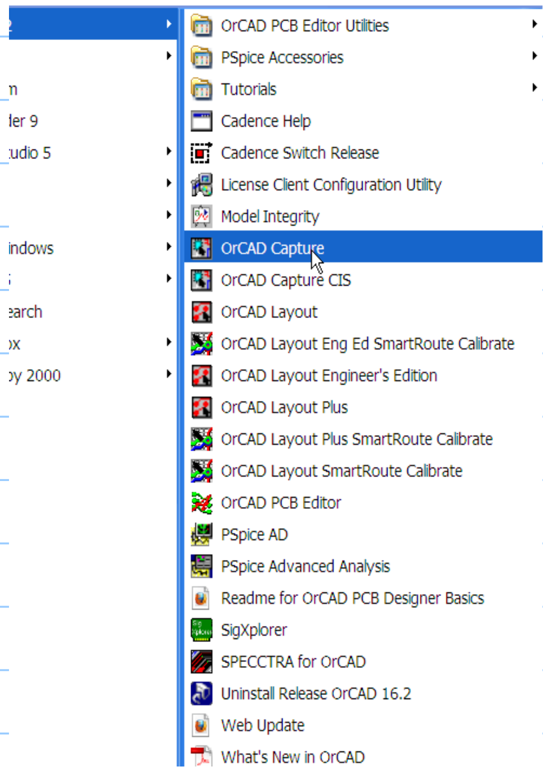
On some machines, it may take a while to start the program because it keeps on finding license, when it is not available. **So turn off wifi, then it will start immediately without looking for a license.**

This is machine specific, **if you have this problem, just turn off your wifi or unplug your network cable**,

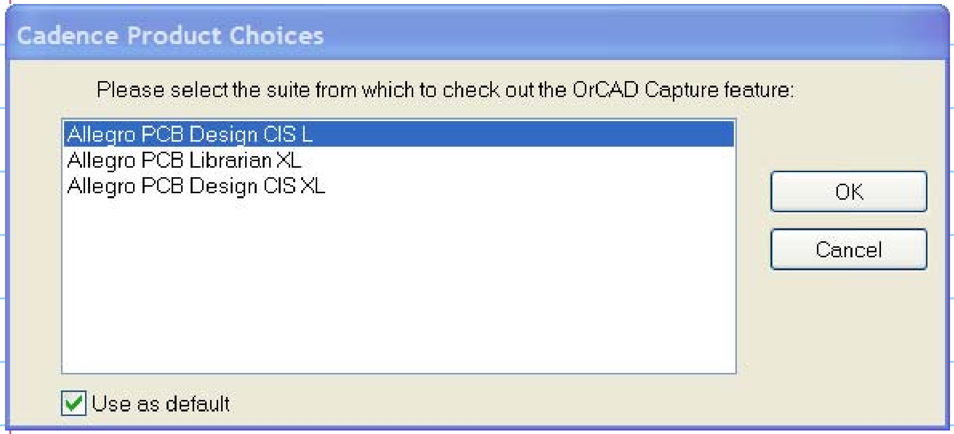
Once Pspice is started, you can turn it back on

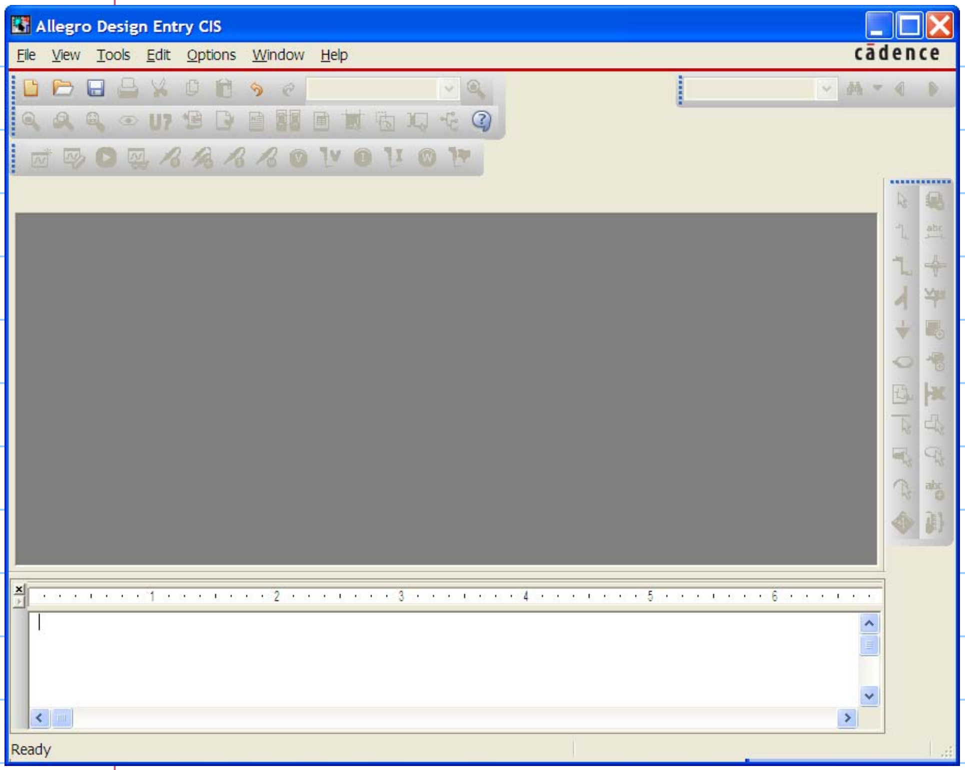
In Windows 7, use the Windows + X key combination to do this.

To open Orcad Capture - go to programs and select "Orcad



select "Allegro PCB Design CIS L"
Check "Use as default",
click "OK"

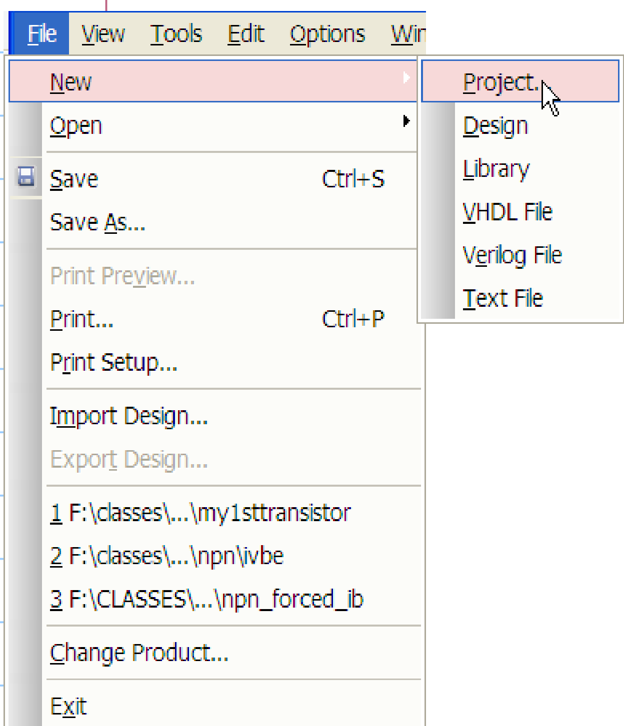


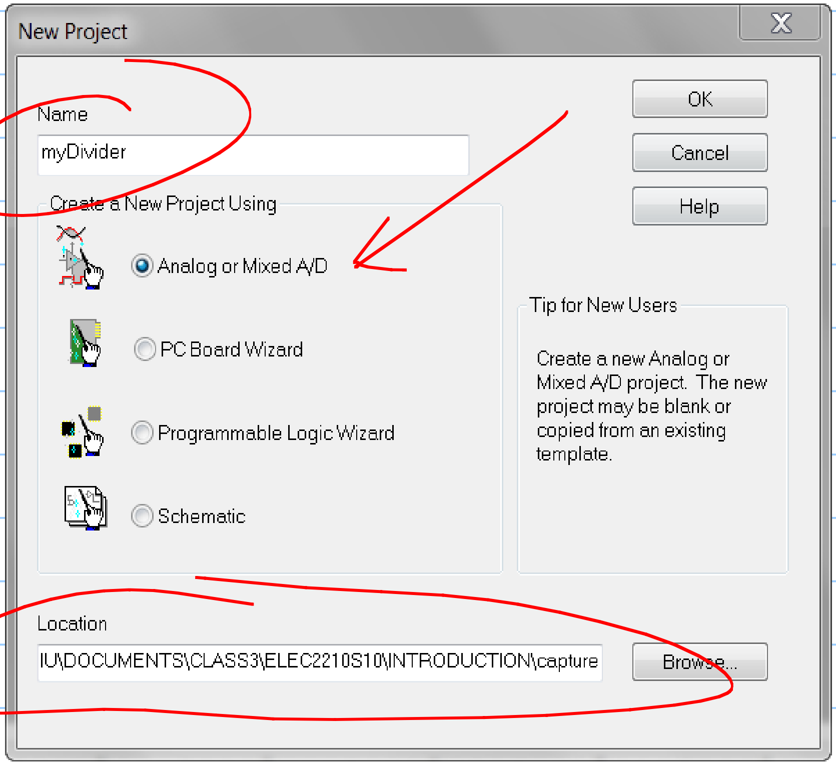

Allegro Design Entry CIS
File View Tools Edit Options Window Help
cadence
Ready

Select File/New/Project :

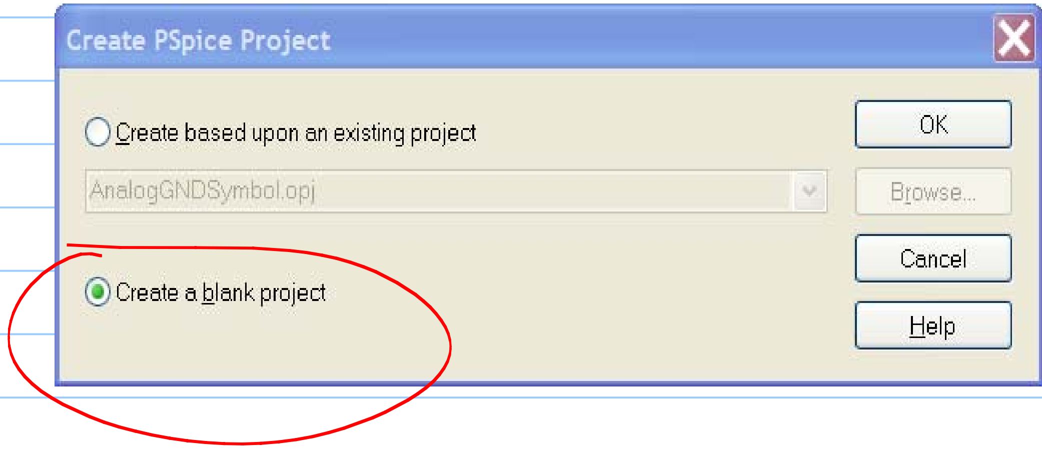
Type project name and location, Check "Analog and Mixed A/D"

Note Title 2/17/2008

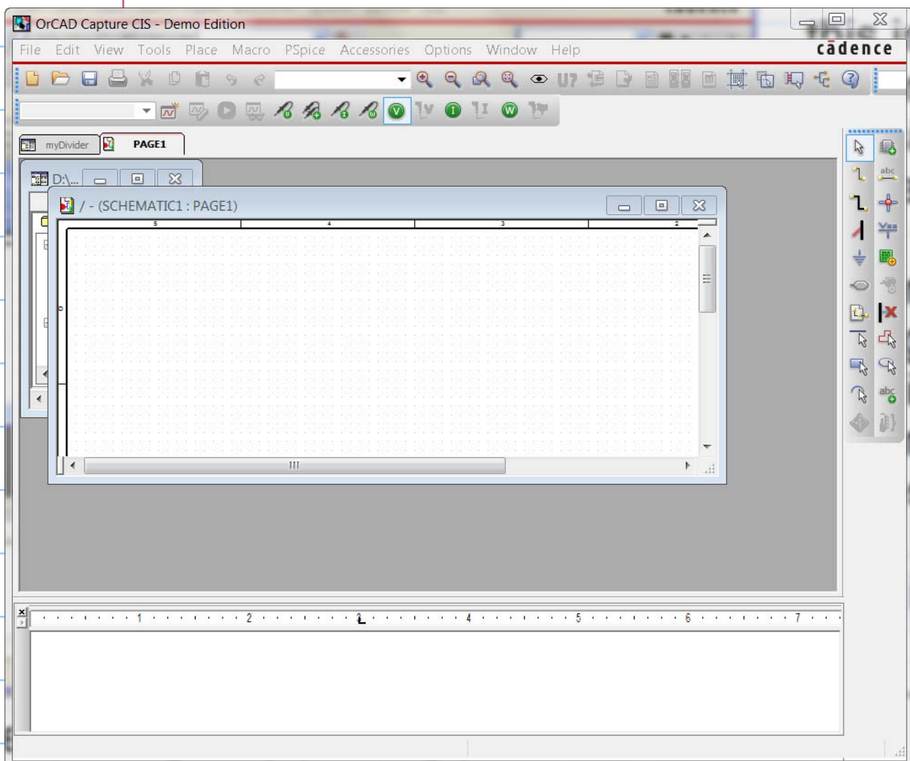




Select Create a blank project



now Capture is lauched for circuit schematic entry

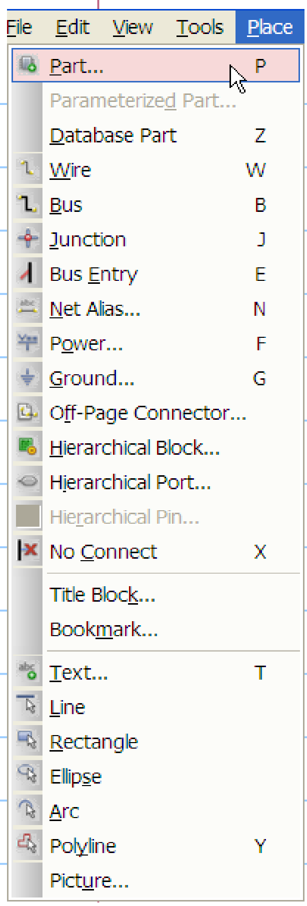


this involves:
placing parts, such as resistors, transistors, capacitors, inductors, diodes, opamp, power supplies,

placing ground ("0" node)
wiring
labeling nodes
placing current / voltage markers at nodes of interest

click in the Schematic page 1 window

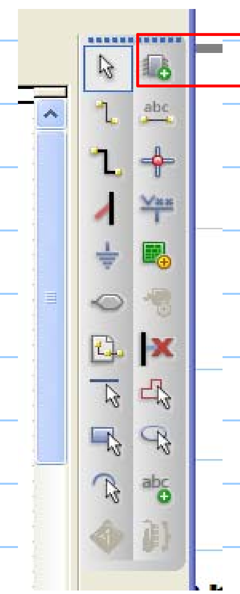
Let us begin by placing part



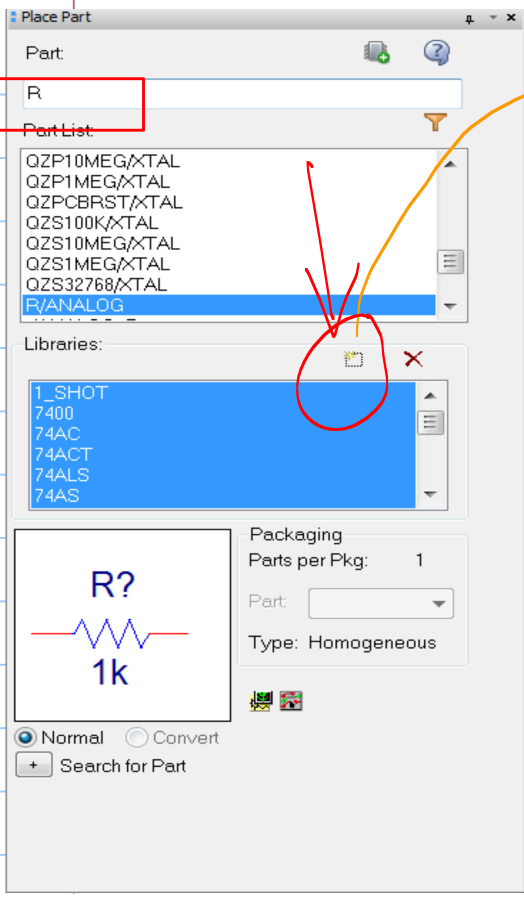
To place part :

Select Place/Part or

click on the place part



Part Browers window pops up.



In the Part box, type name of the component e.g.
R
Click OK icon.

if This is your first time, you need to add library by clicking the first icon (see picture on left)

it takes you to your installation directory to find the libraries

Just use "Ctrl +A" to select all of them, and add them.

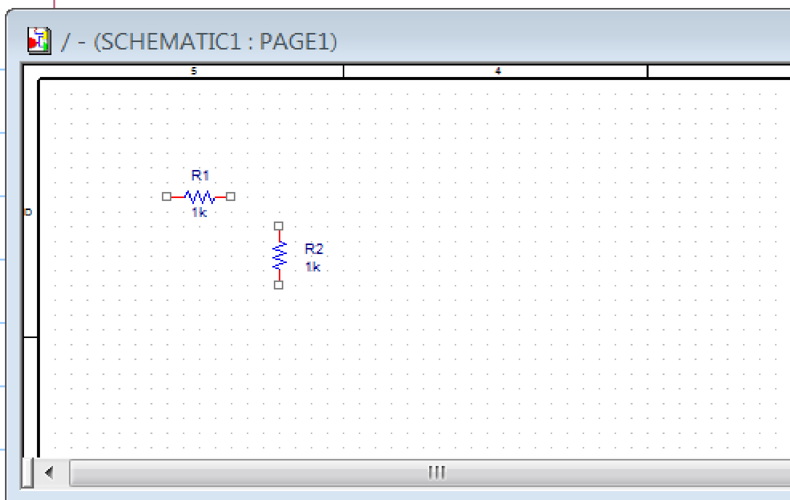
I cannot create this screen as I already did this long ago.

the first letter R indicates this is a resistor, the table next page shows the commonly used part name (first letter) in red

as we go more advanced, we can define our own part name

In future, if we need to use transistors for class / homework, the Breakout library is what we use in my ELEC2210 class for exercise and homeworks, we can edit models and create our own library

Part Name	Description
B	GaAsFET
C	Capacitor
D	Diode
E	Voltage-controlled voltage source
F	Current-controlled current source
G	Voltage-controlled current source
H	Current-controlled voltage source
I	Simple AC current source; DC
J	JFET
K	Non-linear, magnetic core
L	Inductor
M	MOSFET
Q	BJT bipolar transistor
R	Resistor
S	Voltage-controlled switch breakout device
T	Transmission Line
V	Voltage source
W	Current-controlled switch breakout device



Place Part:

Double click to select R

Move the

component (here a resistor) to the

desired position on

the schematic and

left-click.You can place more of them, to rotate, simply press "r"

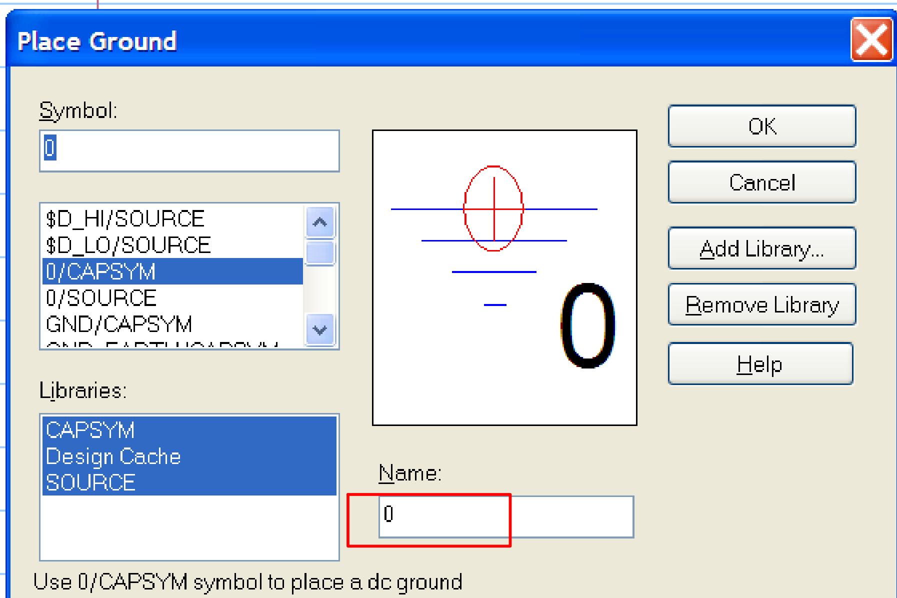
then

Cancel Place Part by:

press ESC or

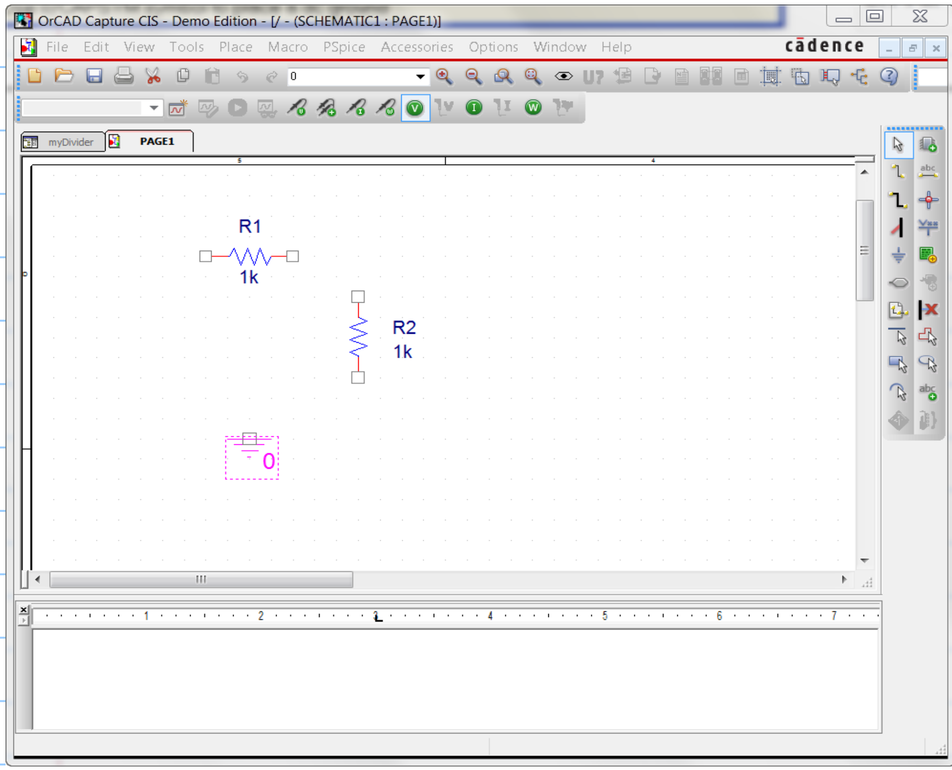
Ground Your Circuit:

Go to Place/Ground or click on the ground icon on the right hand side tool list



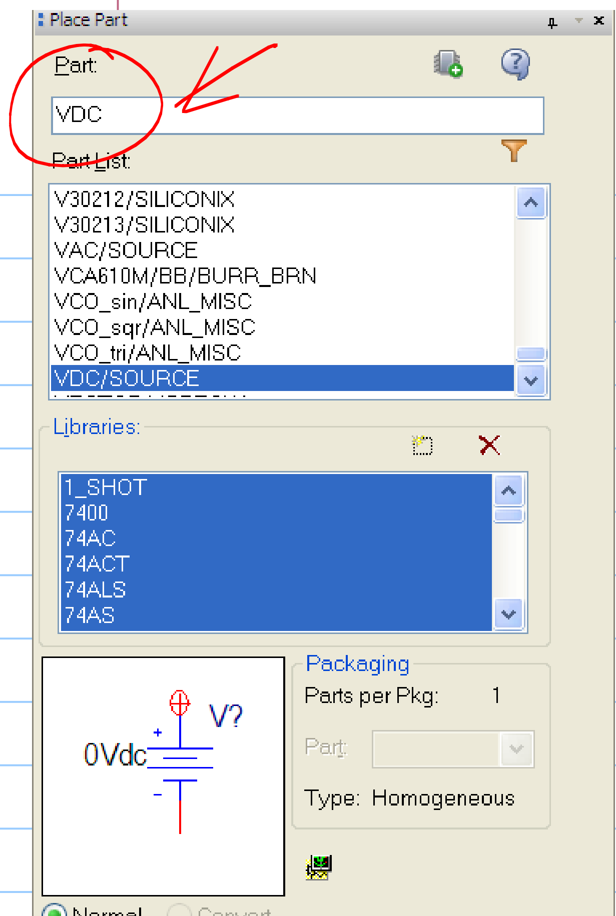
select 0/CAPSYM

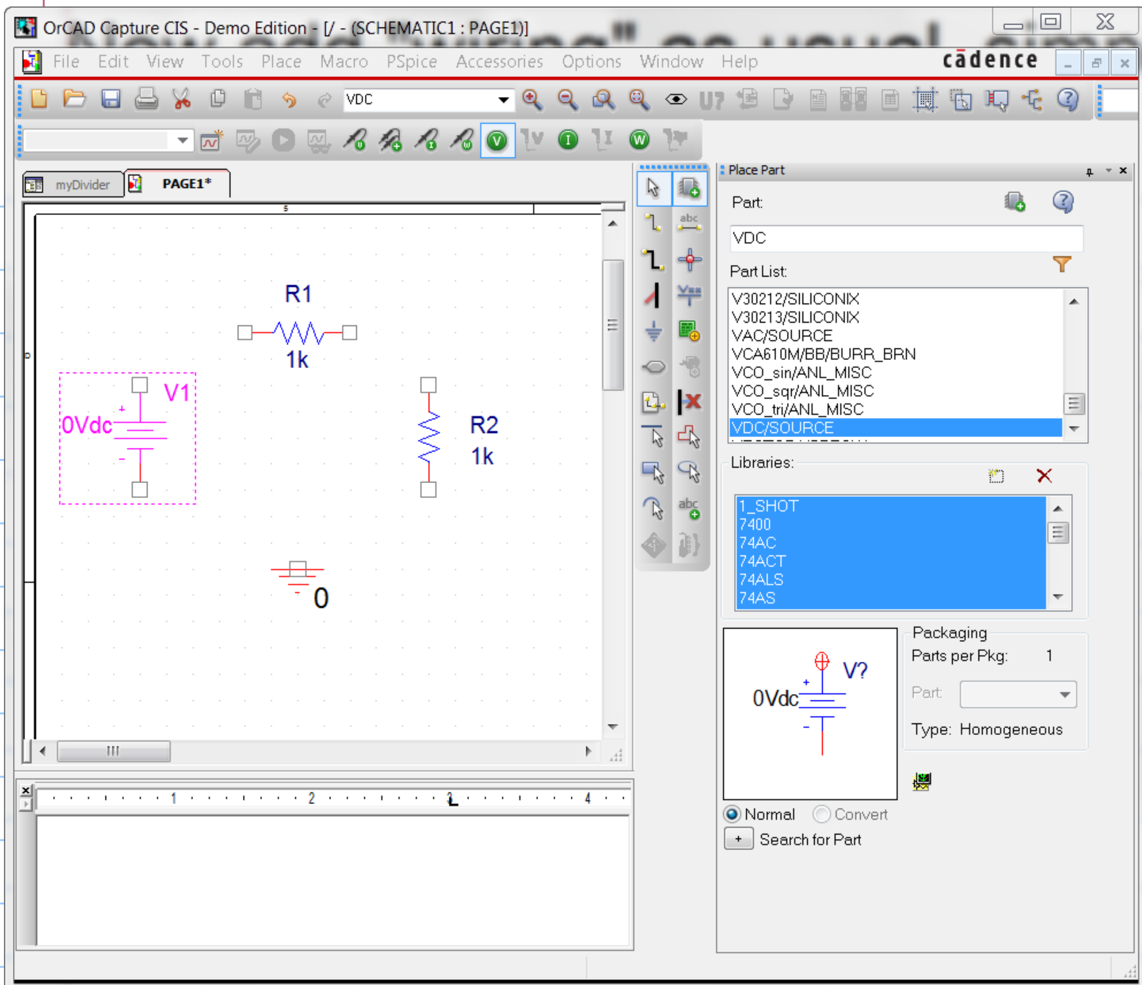
make sure you select the "0/CAPSYM" ground- the global ground for SPICE simulation
the ground can then be copied and placed elsewhere


OrCAD Capture CIS - Demo Edition - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Tools Place Macro PSpice Accessories Options Window Help
cadence
myDivider
PAGE1
R1
1k
R2
1k
0

next, try placing one dc voltage source (VDC)
yourself so we can set up bias for our voltage divider

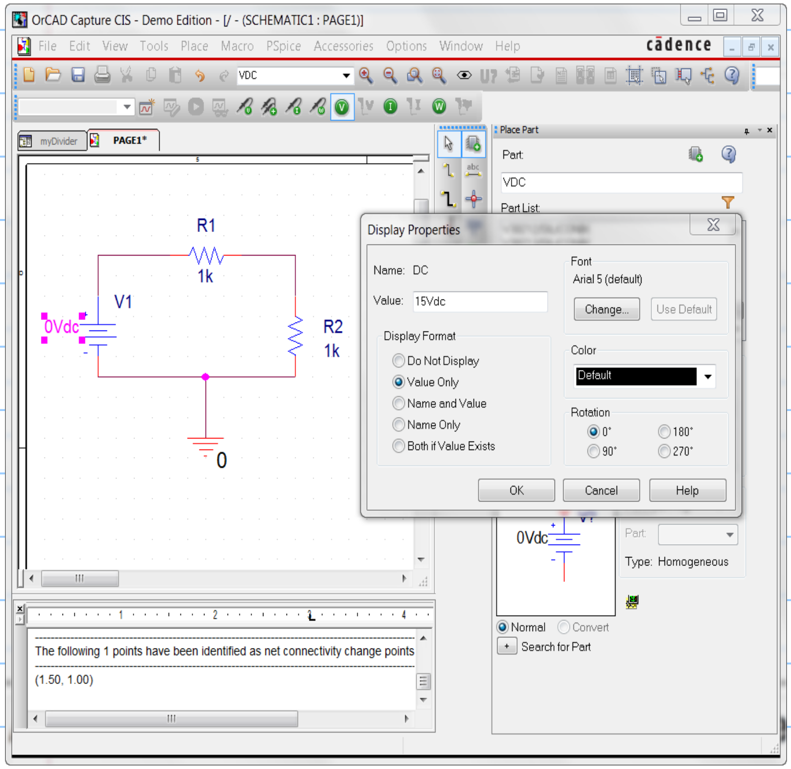
If needed, you can change the part reference name
of the voltage sources to other names, e.g. "Vs" (for
V1) by double clicking on the part reference name
You can
change the value by double clicking on "0Vdc"

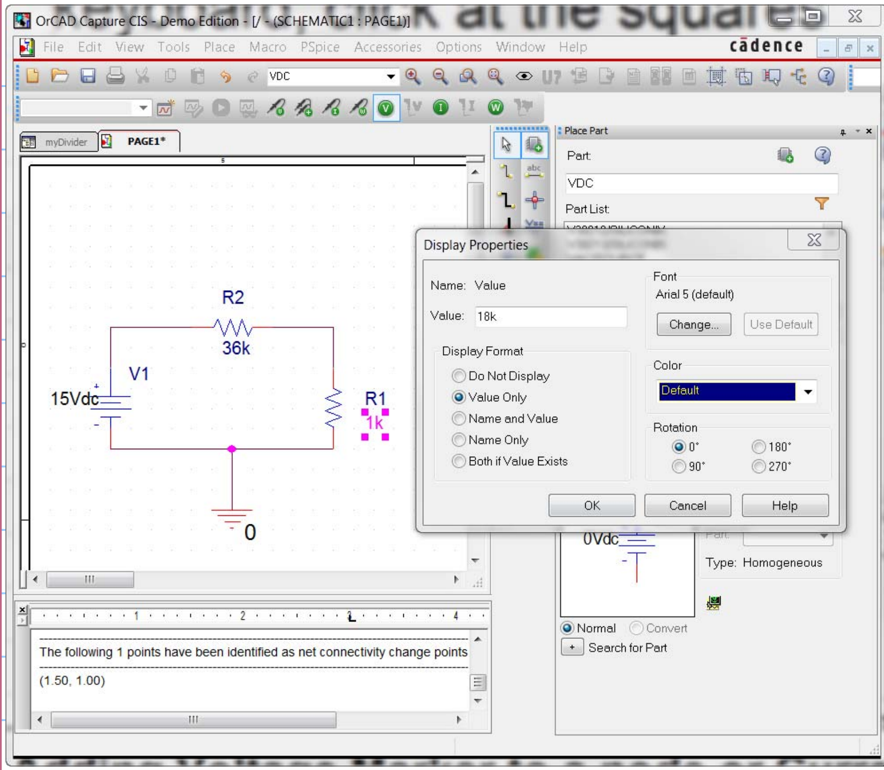

Place Part
Part:
VDC
Part List:
V30212/SILICONIX
V30213/SILICONIX
VAC/SOURCE
VCA610M/BB/BURR_BRN
VCO_sin/ANL_MISC
VCO_sqr/ANL_MISC
VCO_tri/ANL_MISC
VDC/SOURCE
Libraries:
1_SHOT
7400
74AC
74ACT
74ALS
74AS
0Vdc
V?
Packaging
Parts per Pkg: 1
Part:
Type: Homogeneous
Normal
Convert



OrCAD Capture CIS - Demo Edition - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Tools Place Macro PSpice Accessories Options Window Help
cādence
VDC
myDivider
PAGE1*
R1
1k
V1
0Vdc
R2
1k
0
Place Part
Part:
VDC
Part List:
V30212/SILICONIX
V30213/SILICONIX
VAC/SOURCE
VCA610M/BB/BURR_BRN
VCO_sin/ANL_MISC
VCO_sqr/ANL_MISC
VCO_tri/ANL_MISC
VDC/SOURCE
Libraries:
1_SHOT
7400
74AC
74ACT
74ALS
74AS
Packaging
Parts per Pkg: 1
Part:
Type: Homogeneous
V?
0Vdc
Normal
Convert
Search for Part

Now add "wiring" as usual, simply press "W" on keyboard, click at the squares



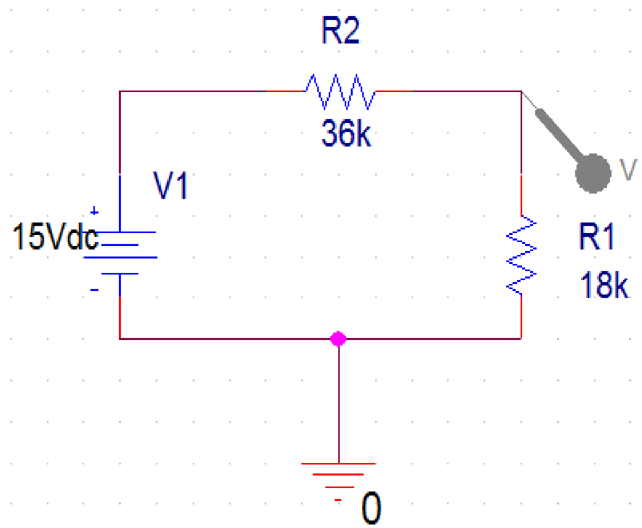


change reference name R1->R2, R2->R1 so they are the same as Fig. 1.23 in our textbook.

change values of V1, R1 and R2 (double clicking)

Adding Voltage Marker to a node or Current Marker to terminal of a part

click on the voltage marker icon, then place it on R1, we want to



Voltage and current markers are used to probe the voltage or current.

After simulation, PSPICE automatically plots

in Pspice, a current marker has to be added to a terminal (e.g. collector terminal of a bipolar transistor, drain node of a MOSFET)

You can also place various markers from the menu as follows:

Voltage marker

Select PSpice/Markers/Voltage level or

click voltage marker icon to place the voltage marker.

Current marker

Select PSpice/Markers/Current into pin or

click current marker to place the current marker.

Voltage Differential marker

Select PSpice/Markers/Voltage Differential or

click voltage differential marker. Place the two

marker + and - between the circuit.

Advanced marker

Enable Bias voltage display and Bias current display Icon

this is for dc bias only, simply click on the V and I display icons to enable / disable this feature

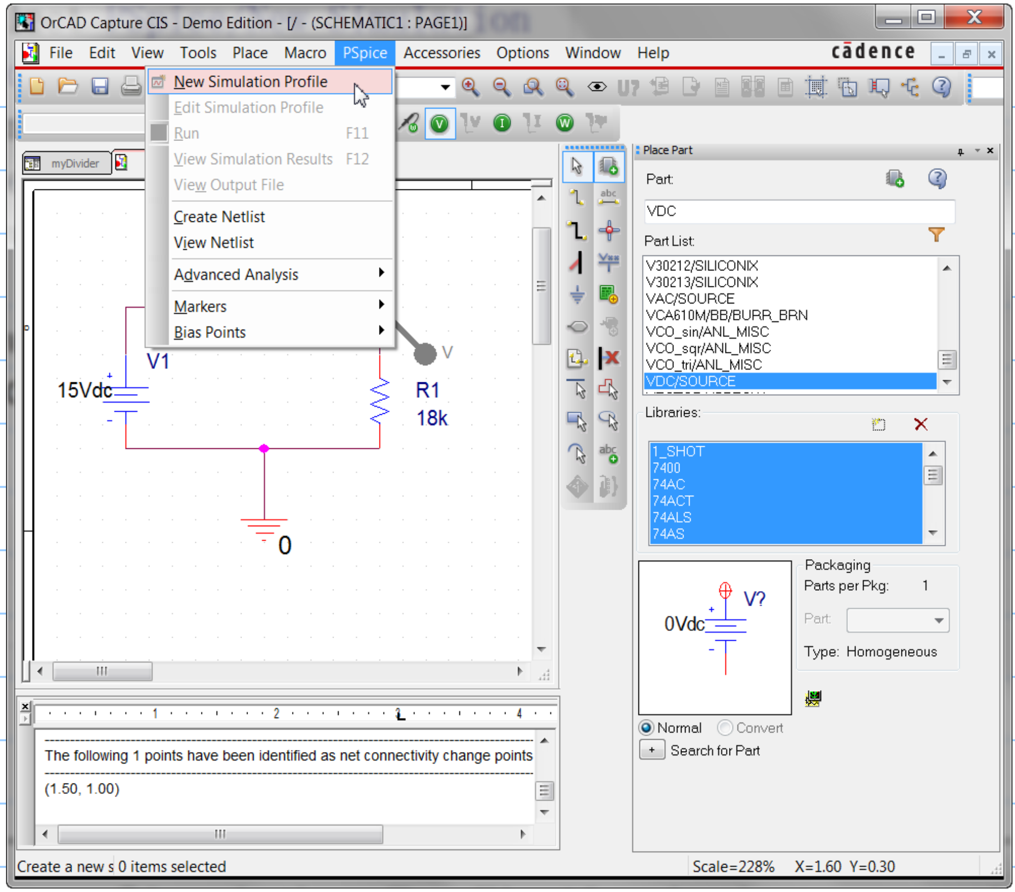
useful for looking at dc bias points

we will delay this discussion to bias examples

Set up Simulation Profile (Analysis type and details)

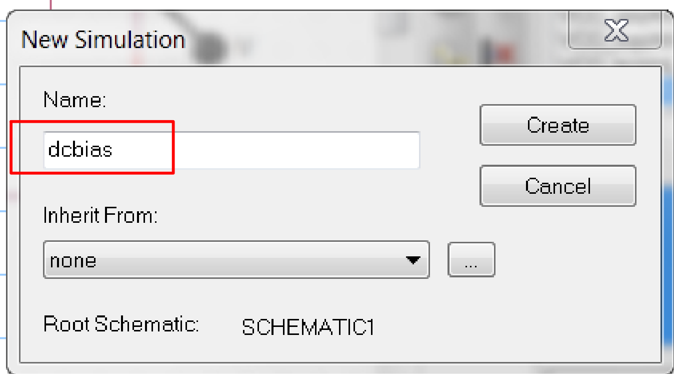
Select PSpice/New Simulation Profile or click on the new simulation profile icon to create new simulation.

New Simulation window will show up



Set up Simulation Profile (Analysis type and details)

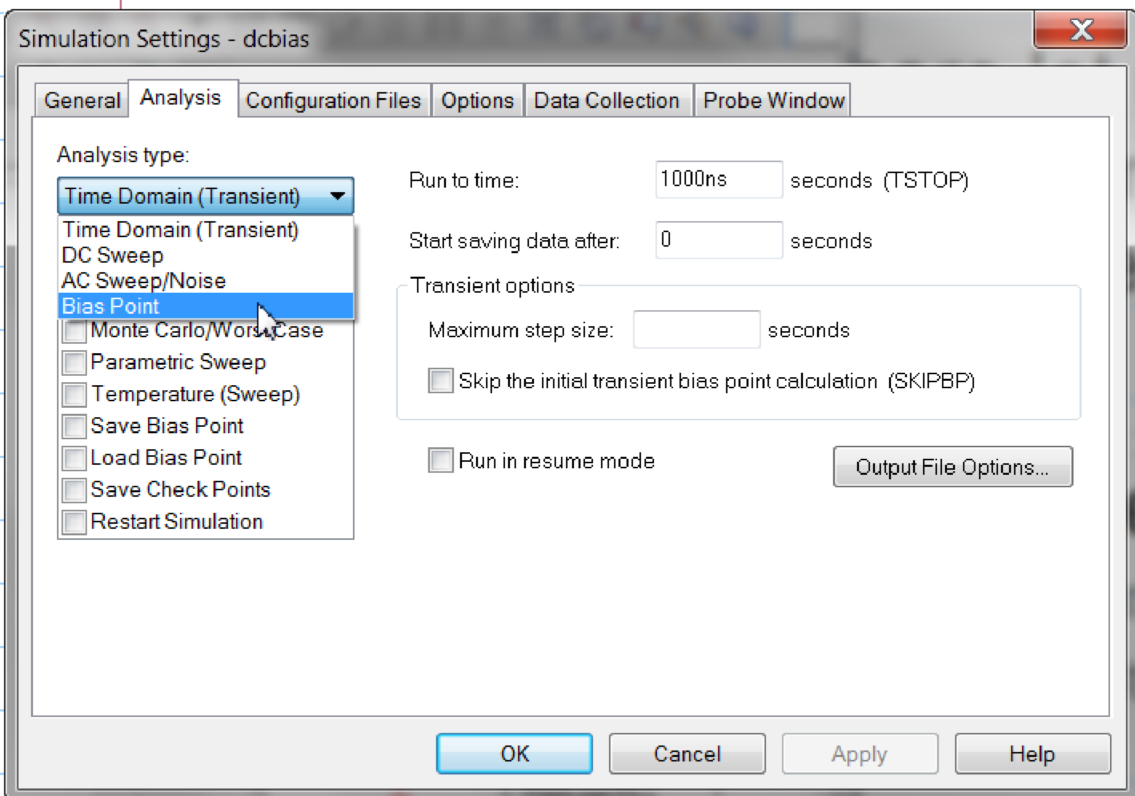
type in simulation profile name



Later: You can always come back and edit the simulation profile by:

Select PSpice/Edit Simulation Profile or click on the simulation

Select Analysis type

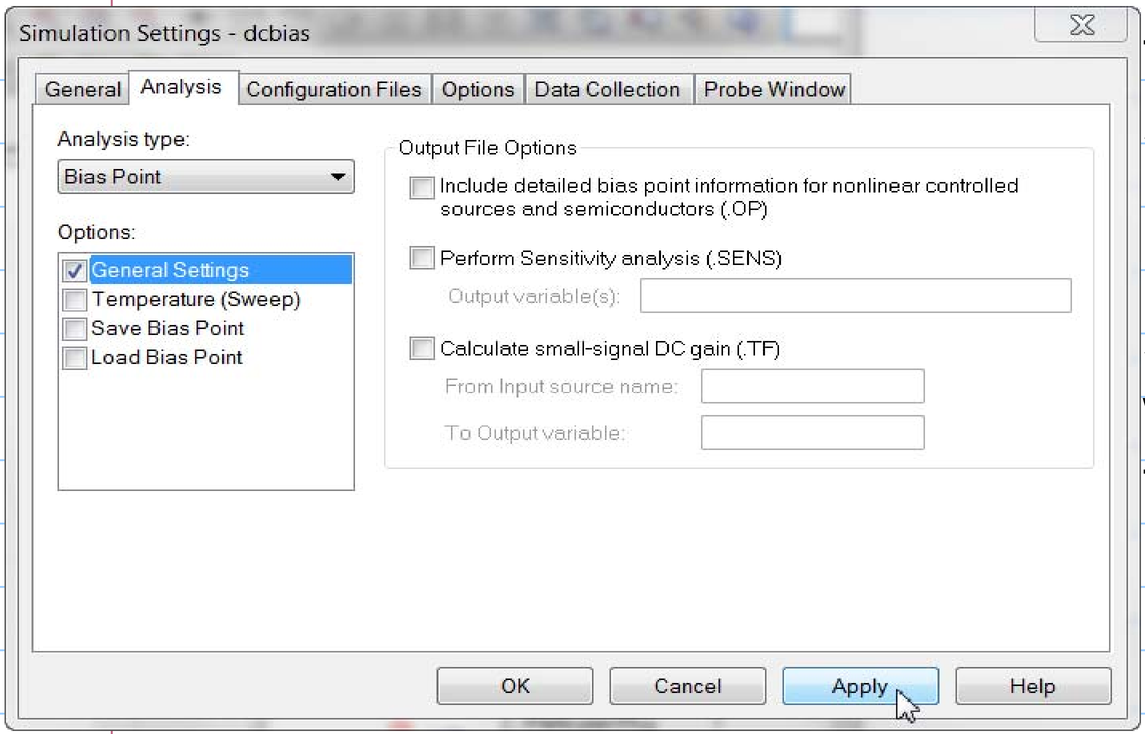


here let us choose Bias Point, and later we can do sweep

DC Sweep:

Sweep one or more DC voltage source at the same time.

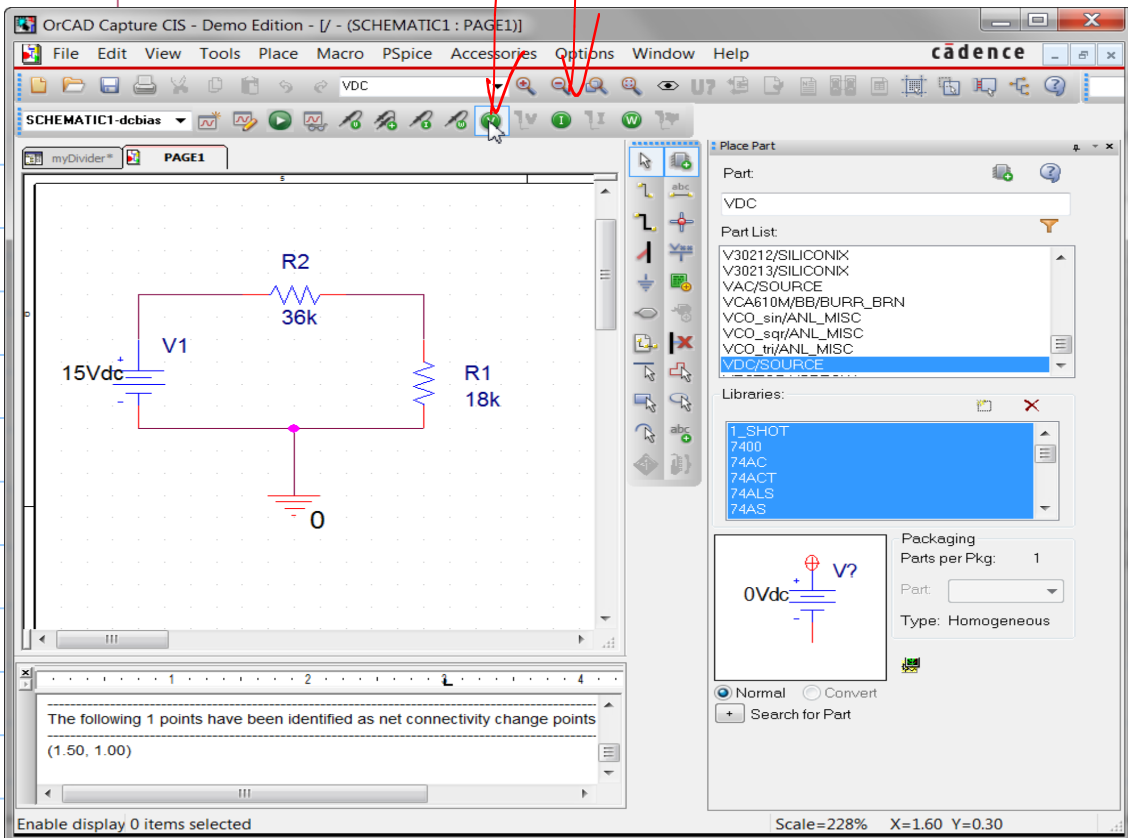
The DC output varies as the source is swept.



click on "Apply"

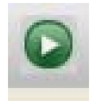
then go back to the schematic window

no need to close this simulation settings windows, just leave it open for convenience - also closing it might crash the program on some OS



enable voltage display (see the cursor)

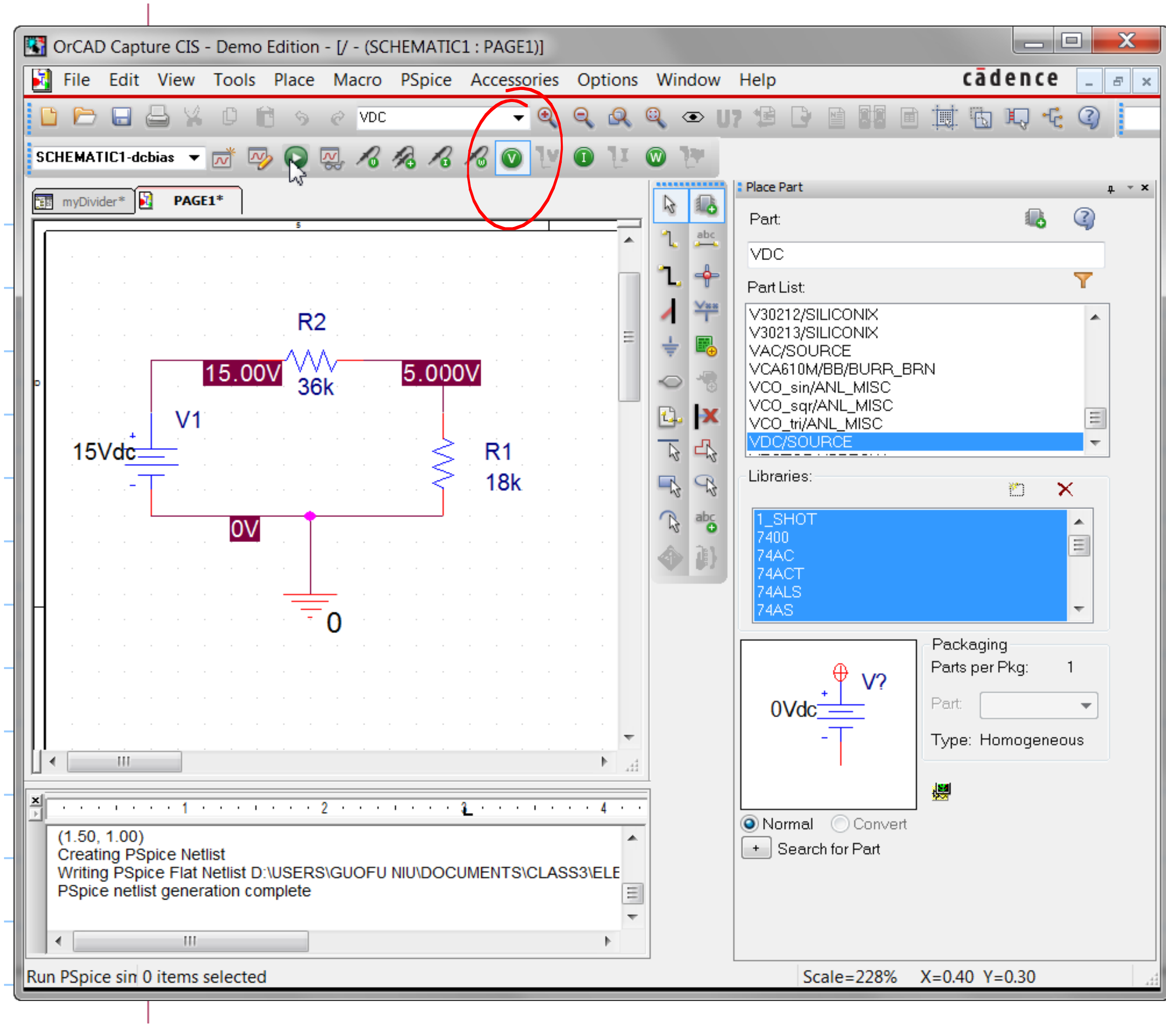
Run SPICE Simulation:



Select PSpice/Run [F11]
or click on the simulate
icon to run simulation.

or click "Run" icon

when using DC bias point analysis, after simulation, the DC values of voltage and current can be

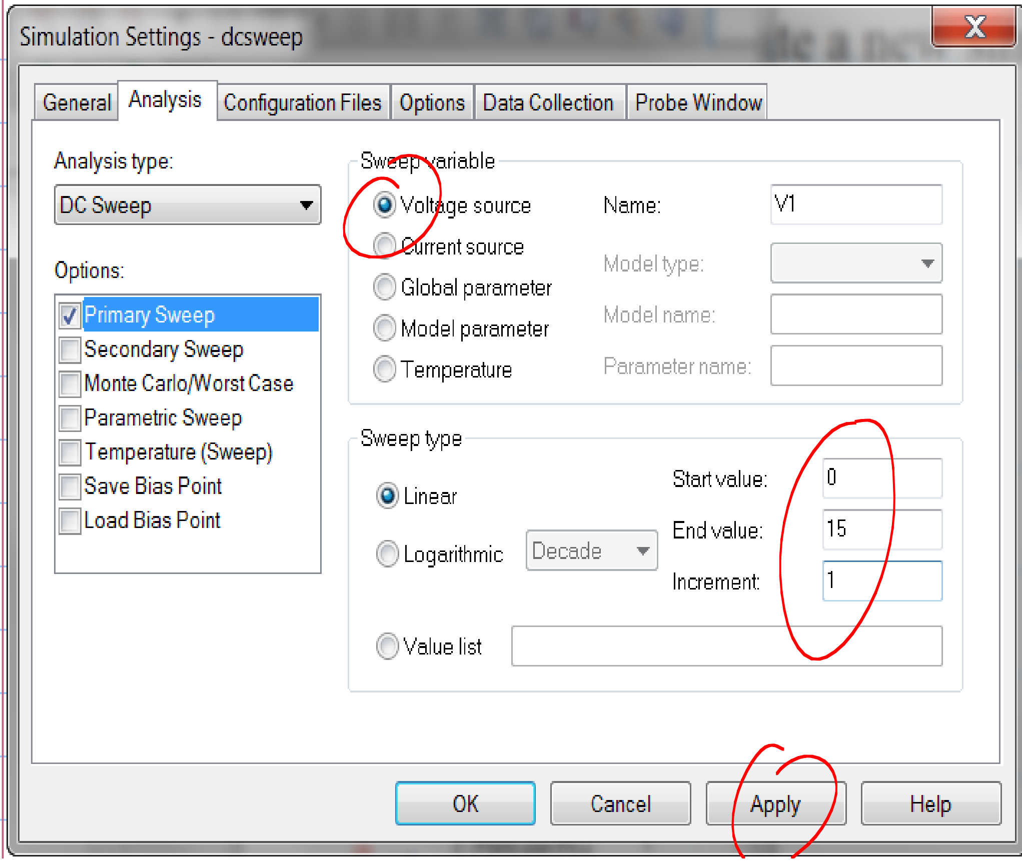


now you see the voltage displayed

V(R1) is 5V.

DC Sweep

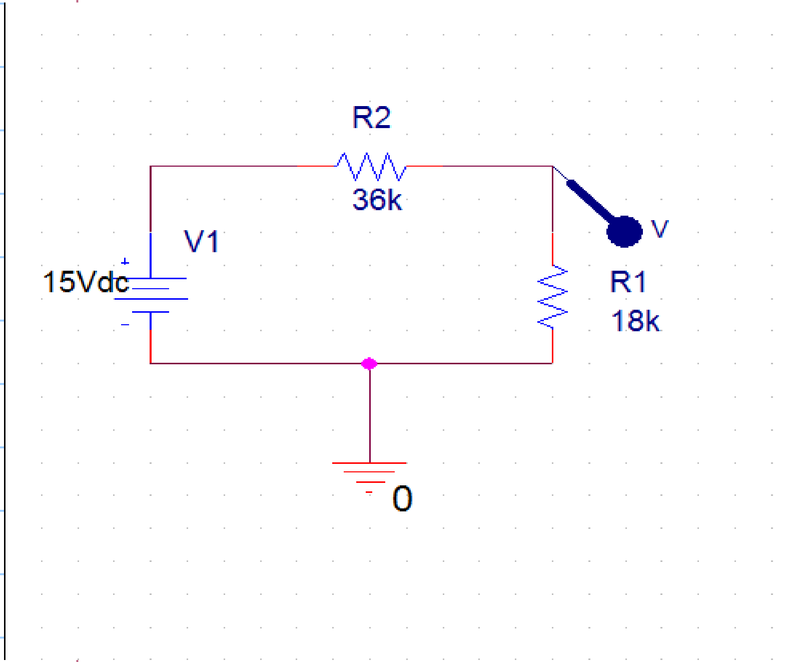
Say we want to sweep V1 and see how V(R1) responds. Simply create a new simulation profile - dc sweep

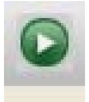


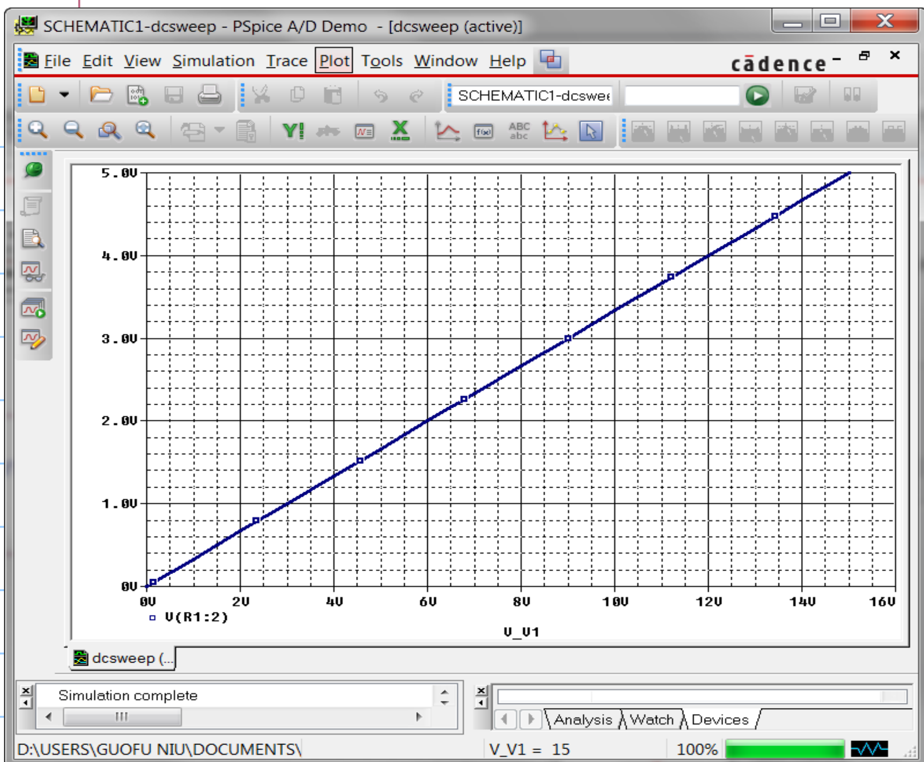
choose V1 to sweep, from 0 to 15V, in step of 1V,

apply, then go back to main window,

Add a voltage marker to R1, our interest, then Run







The PSPICE A/D window will automatically open and all of the value of markers will appear on the plot.

We can add many other new plots in new window or existing window using the plotting tool.

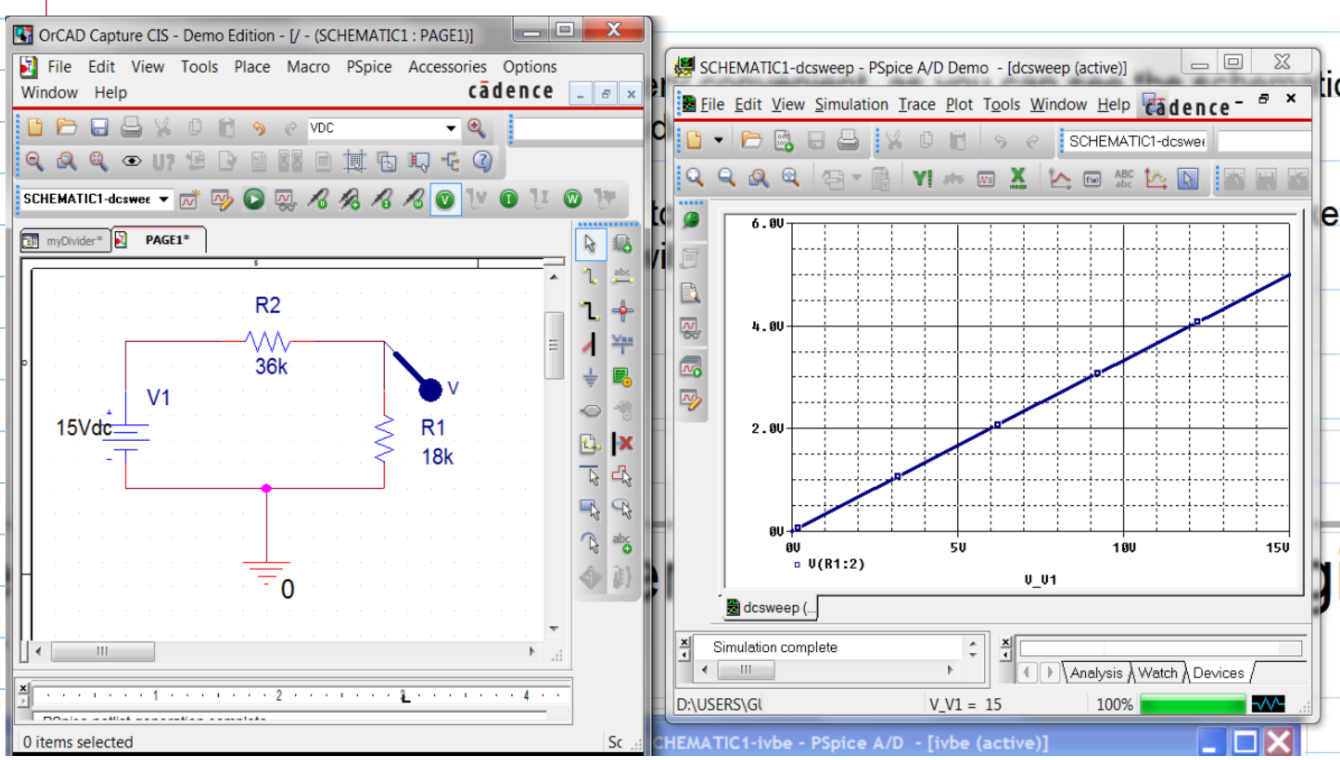
Viewing Results:

The PSPICE A/D program will display marker results

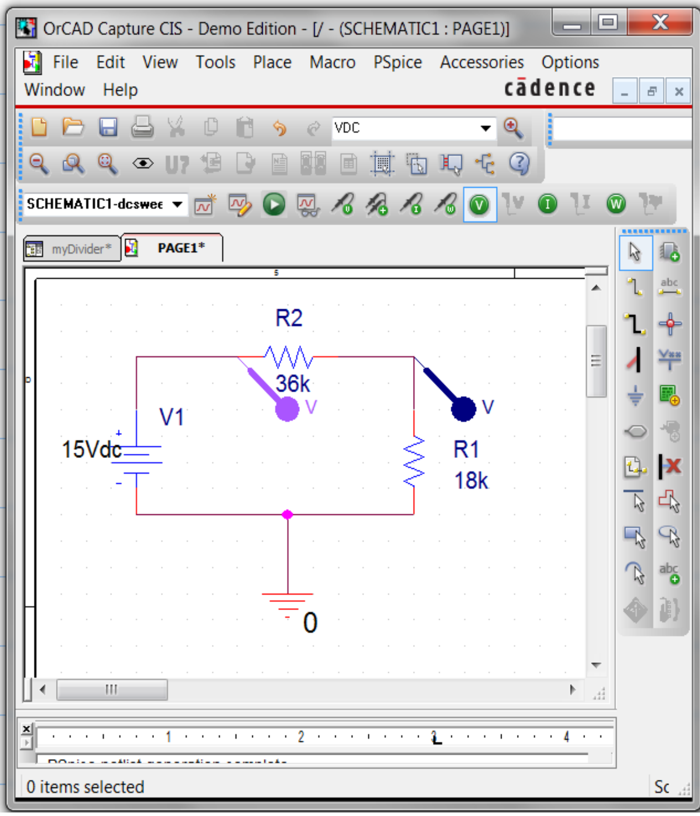
Here you can visually analyze and interactively manipulate the waveform data produced by circuit simulation.

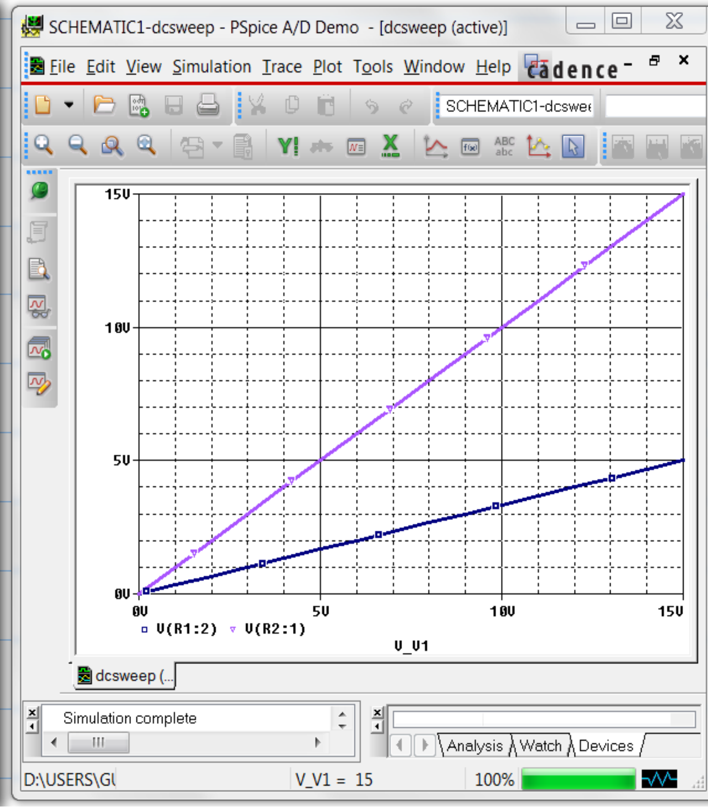
Next is a side by side display of Capture Window and PSPICE A/D window

This is very convenient, as you can see the schematic and results side-by-side.

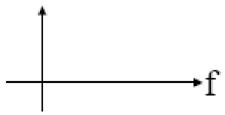


Probes are shown in different color, legends are given too under the plot





Monte Carlo and Worst Case Analysis

AC Sweep Analysis determines the output with 

Monte Carlo Analysis
During a Monte Carlo analysis, Pspice performs several runs of a DC, AC, or transient analysis, each time varying component values randomly within the tolerance range.

Worst Case Analysis
Worst Case Analysis does not use random variations in several runs of a DC, AC, or transient analysis, each time varying component values within the tolerance range.