

# A Capture / PSPICE Primer for ELEC2210 Fall 2010

Note: **Class -**

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.



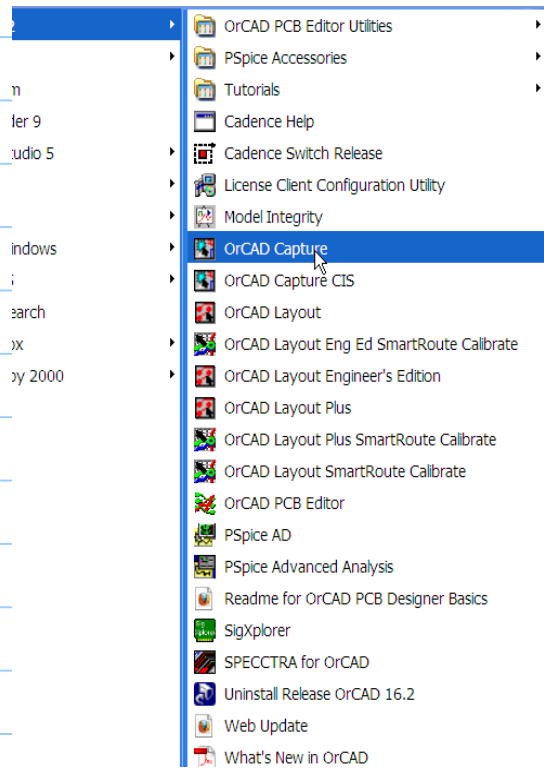
On some machines, it may take a while to start the program because it keeps on finding license, when it is not available. **So turn off wifi, then it will start immediately without looking for a license.**

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

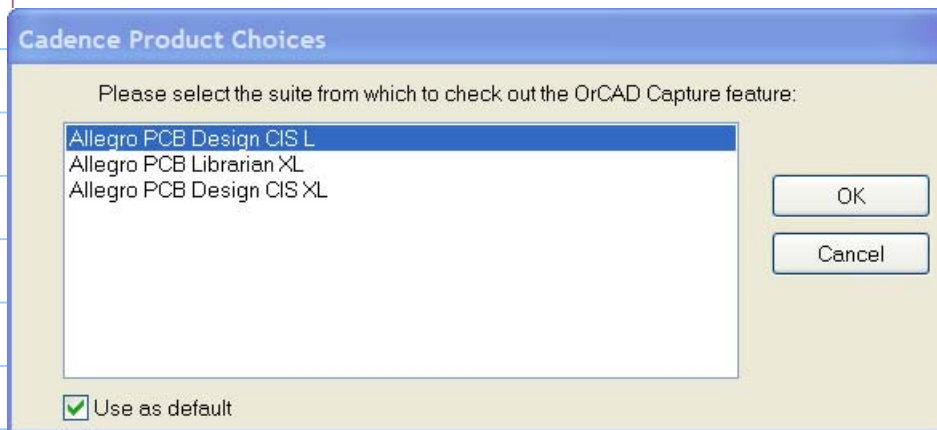
Once Pspice is started, you can turn it back on

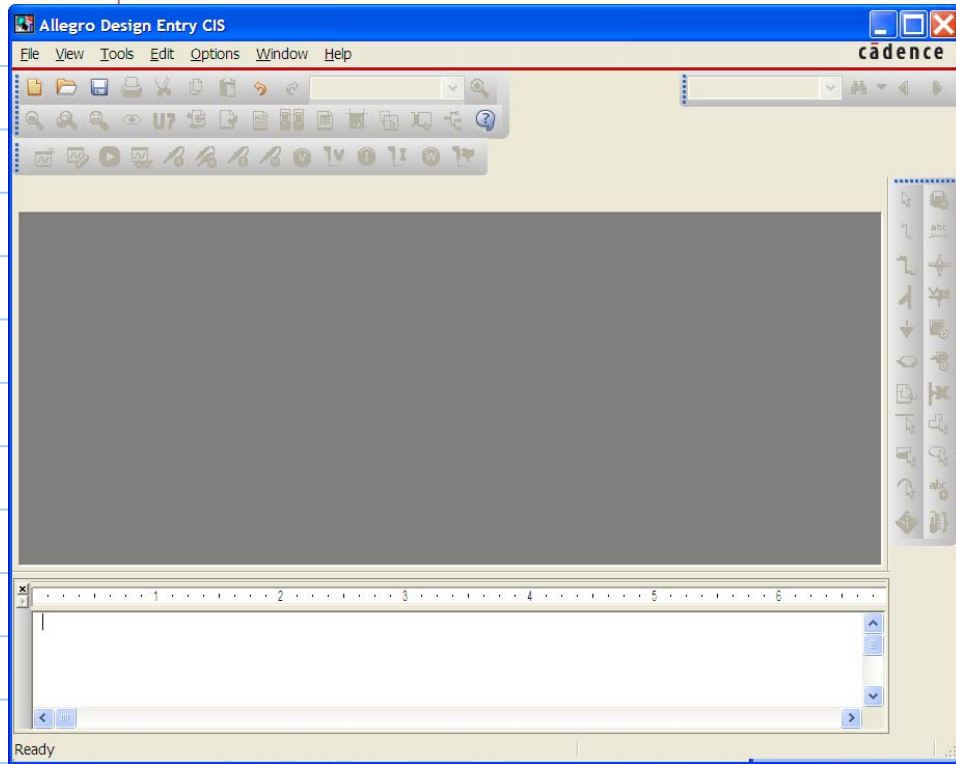
In Windows 7, use the Windows + X key combination to do this.

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



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



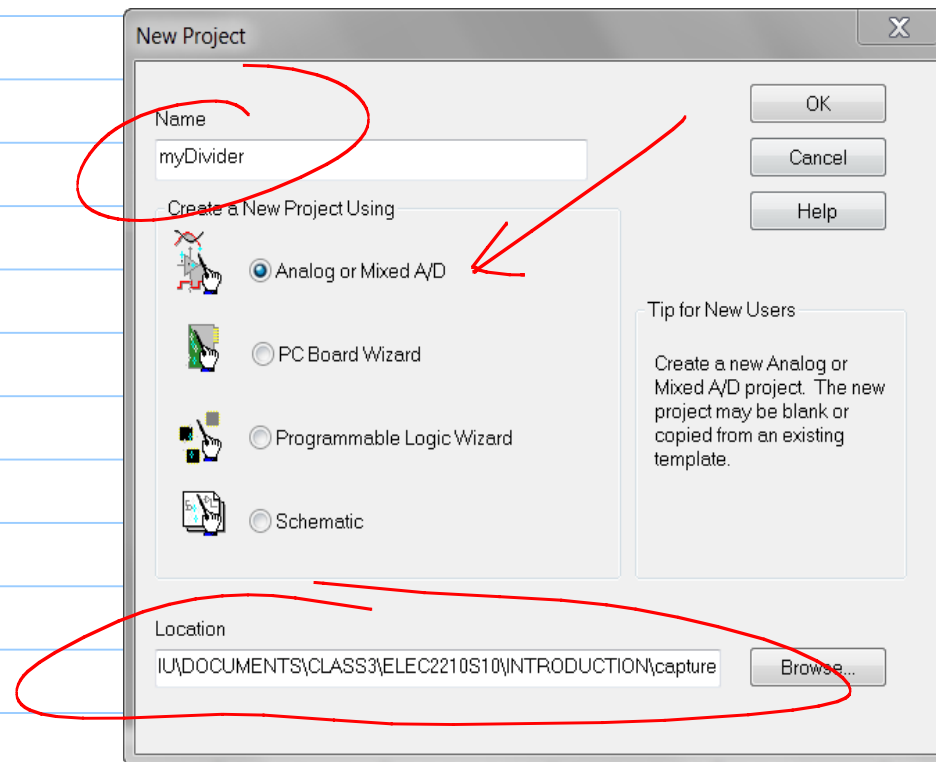
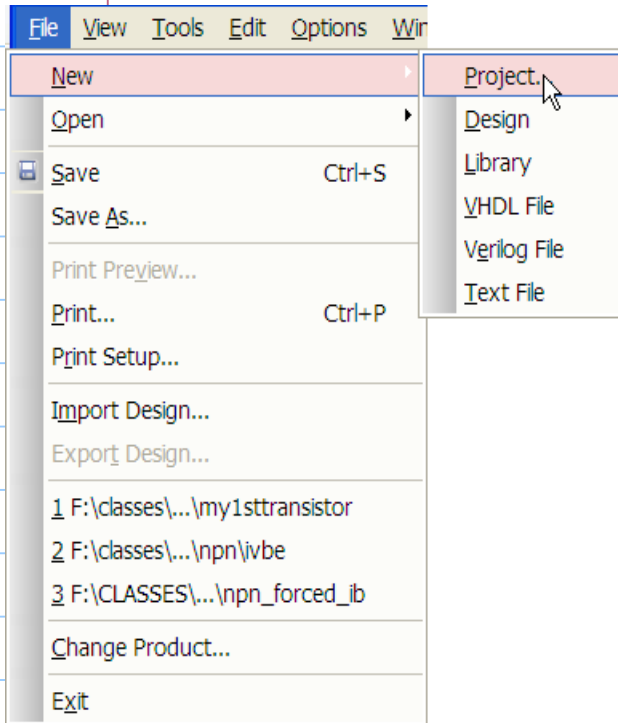


Select **File/New/Project** :

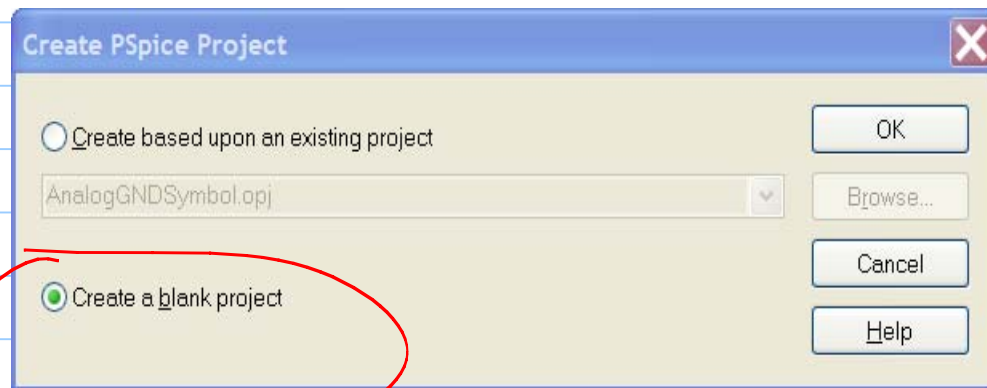
Type project name and location, Check "Analog and Mixed A/D"

Note Title

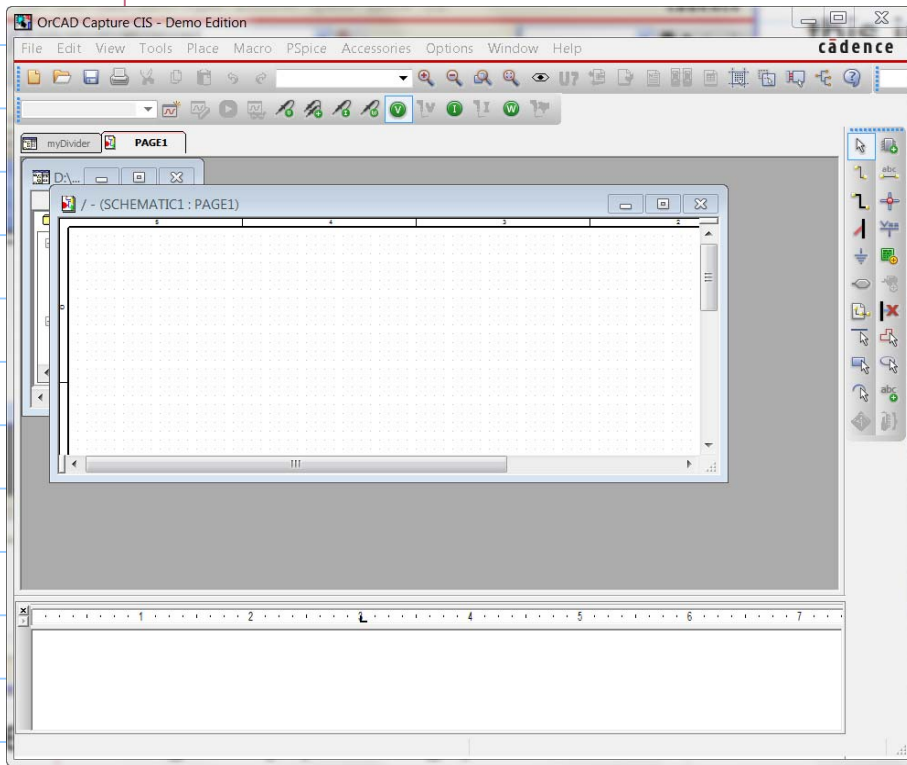
2/17/2008



## Select Create a blank project



now Capture is launched for circuit schematic entry



this involves:

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

placing ground ("0" node)

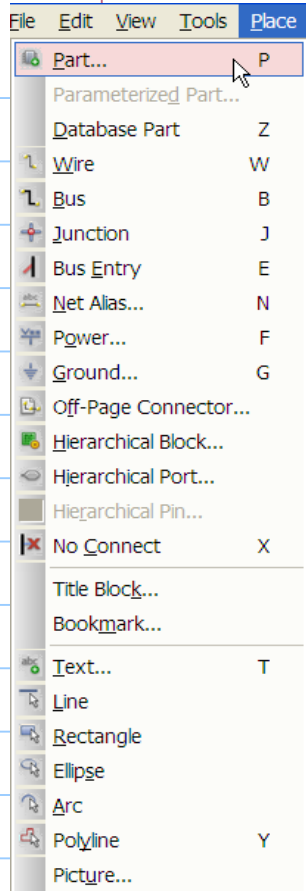
wiring

labeling nodes

placing current / voltage markers at  
nodes of interest

click in the Schematic page 1 window

Let us begin by placing part



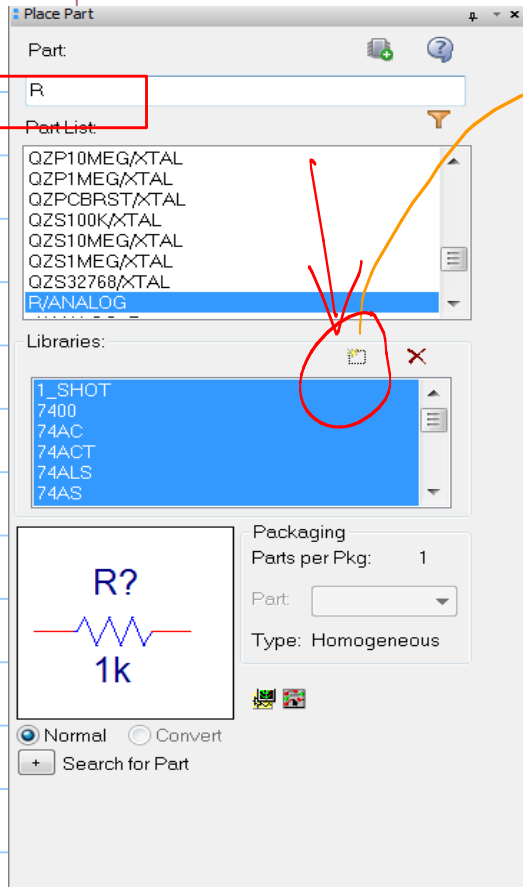
To place part :

Select Place/Part or

click on the place part



Part Browsers 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.

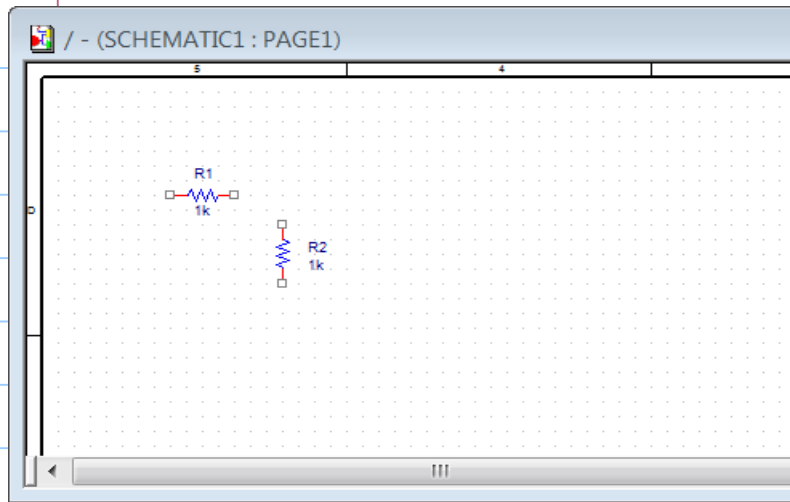
I cannot create this screen as I  
already did this long ago.

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"

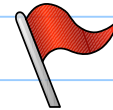
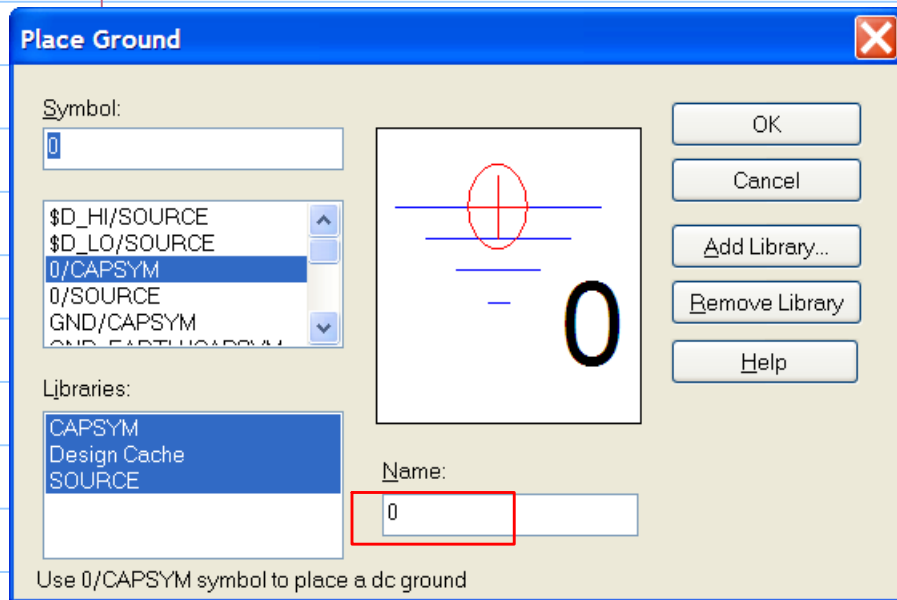
then

**Cancel Place Part by:**

press **ESC** or

## Ground Your Circuit:

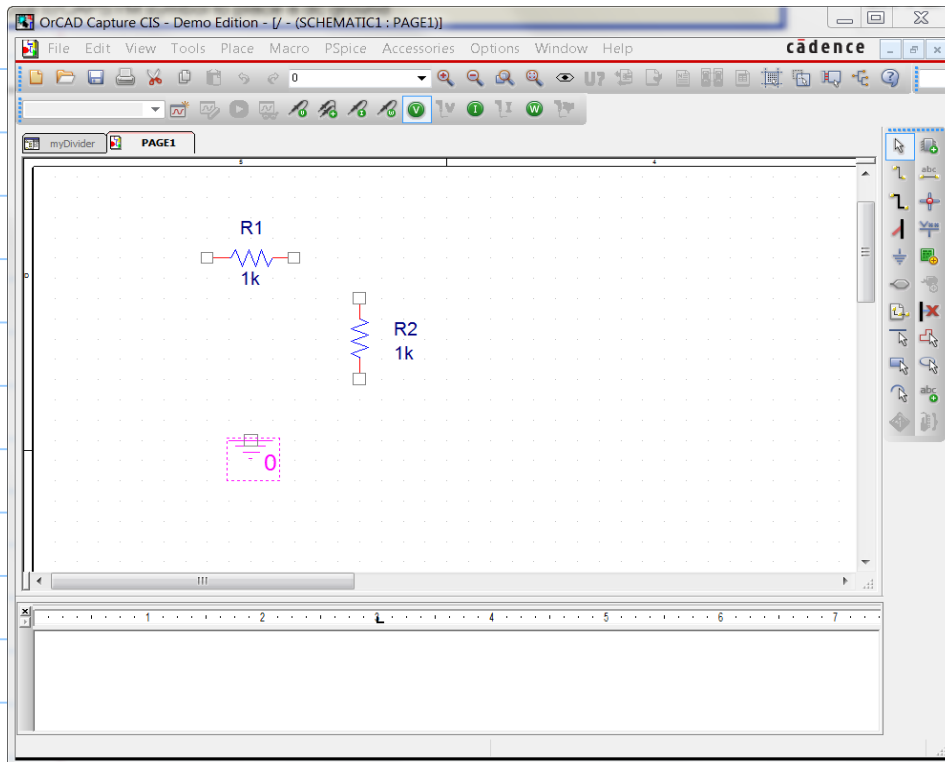
Go to **Place/Ground** or click on the ground icon on the right hand side tool list



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



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

Packaging

Parts per Pkg: 1

Part:

Type: Homogeneous

Normal

OrCAD Capture CIS - Demo Edition - [/ - (SCHEMATIC1 : PAGE1)]

File Edit View Tools Place Macro PSpice Accessories Options Window Help

cadence

vdc

myDivider PAGE1\*

R1  
1k

R2  
1k

0Vdc V1

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

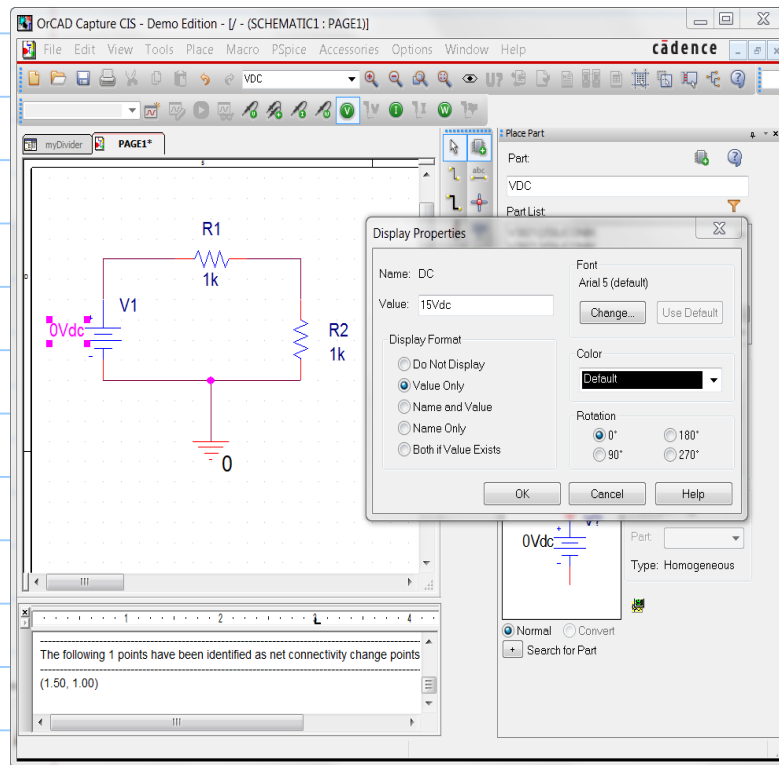
Packaging  
Parts per Pkg: 1

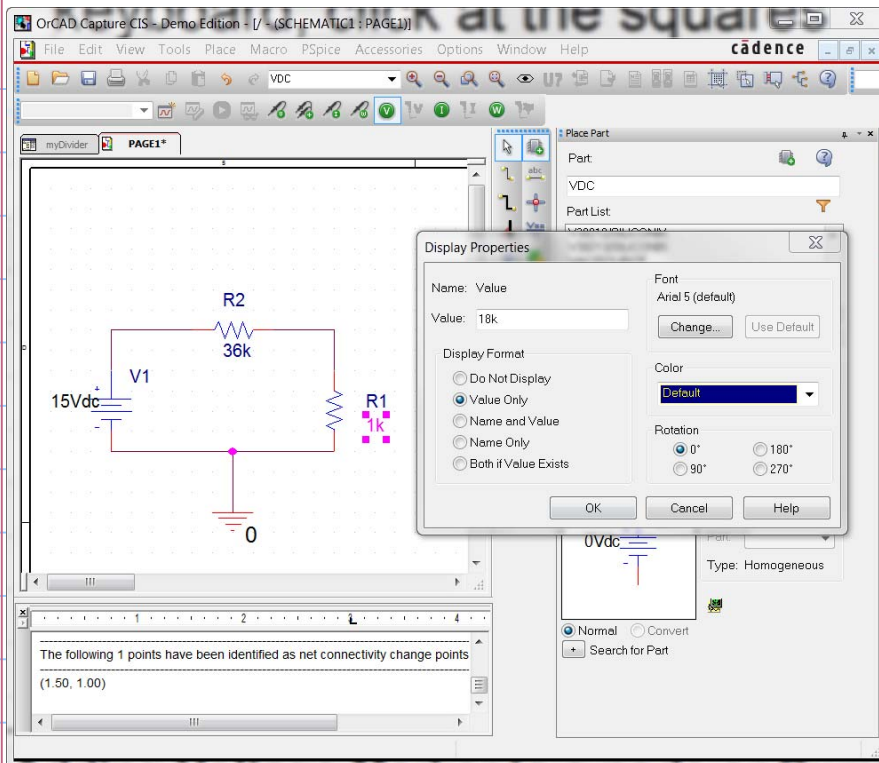
Part:

Type: Homogeneous

Normal  Convert

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





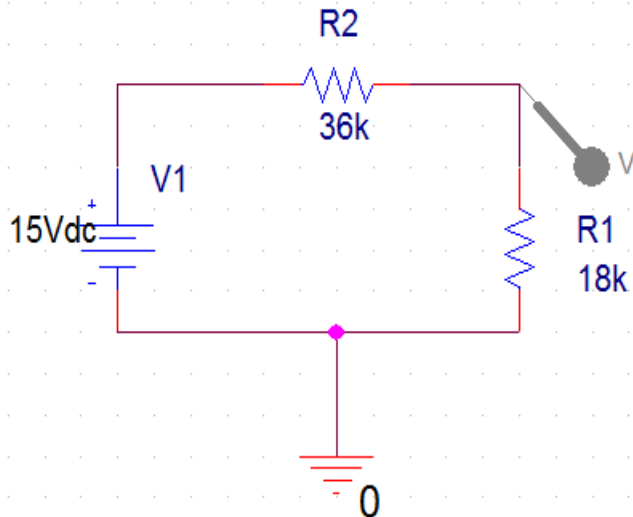
change reference name  
R1->R2, R2->R1 so they are  
the same as Fig. 1.23 in our  
textbook.

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

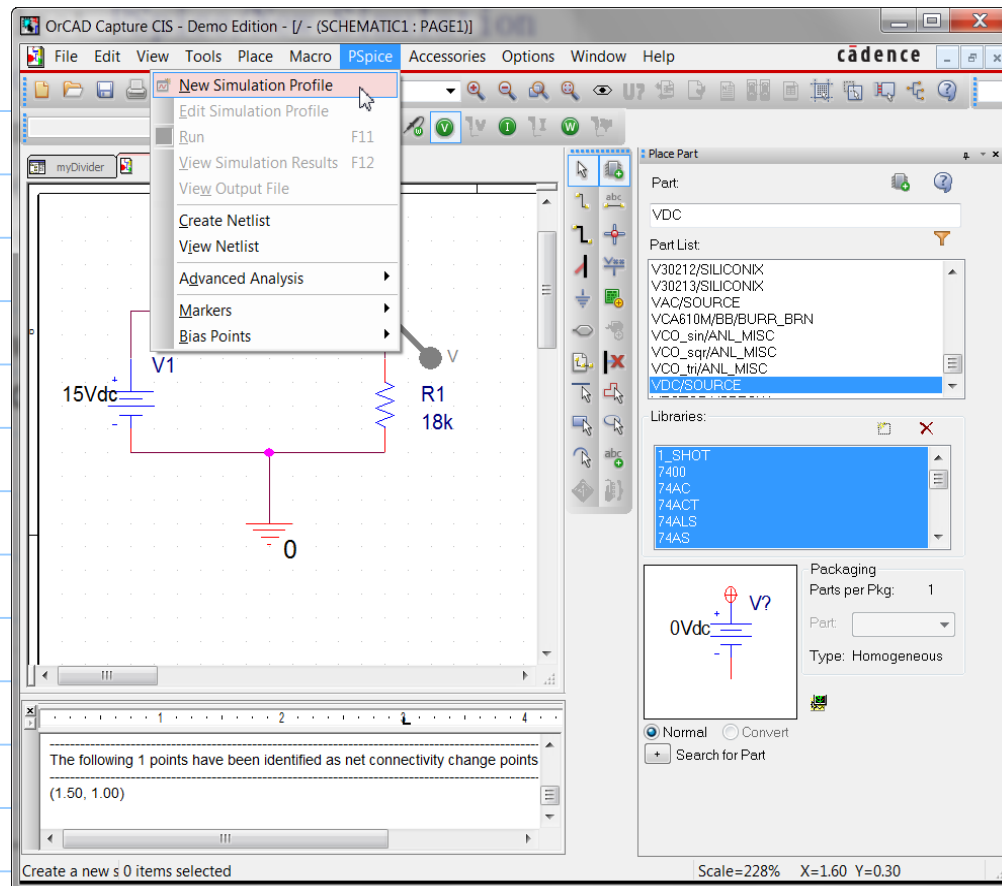
we will delay this discussion to bias  
examples

## Set up Simulation Profile (Analysis type and details)

Select PSpice/New Simulation

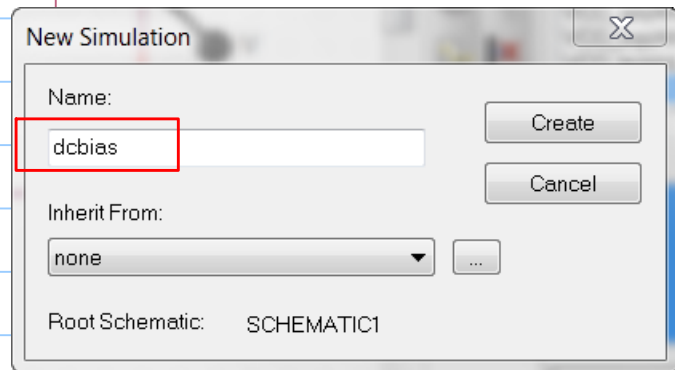
Profile or click on the new simulation profile icon to create new simulation.

New Simulation window will show up



## Set up Simulation Profile (Analysis type and details)

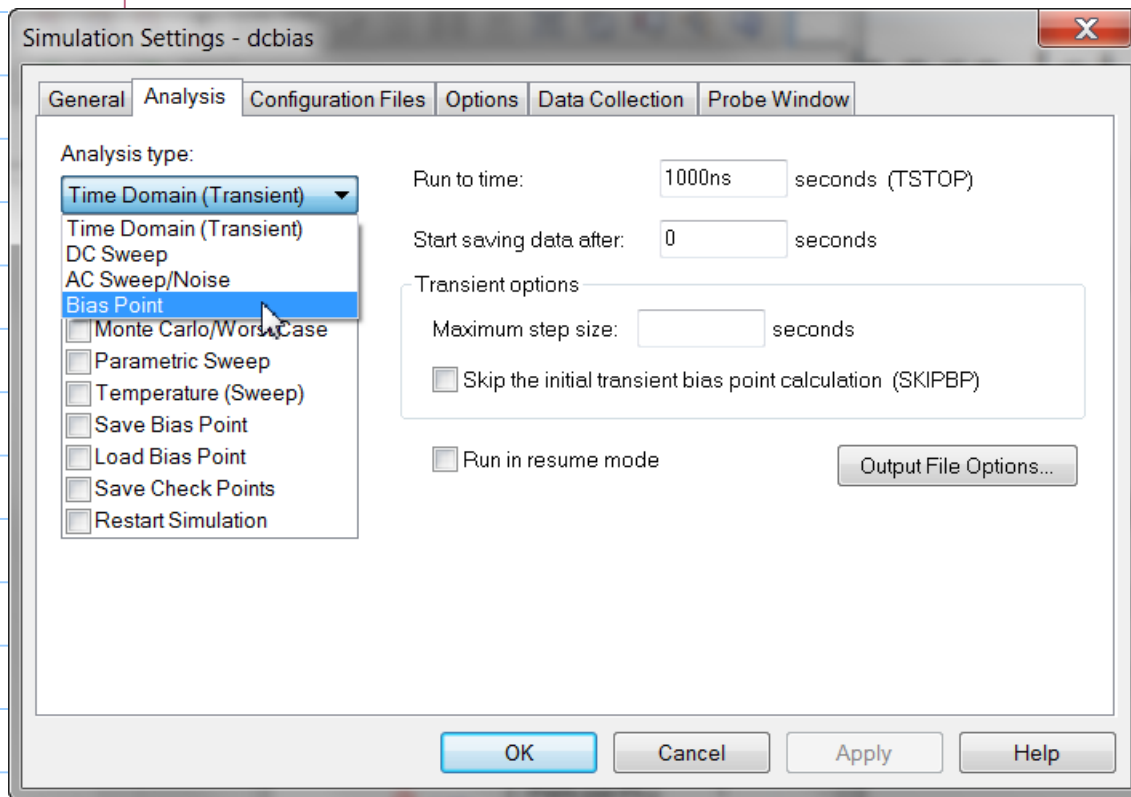
type in simulation profile name



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:**

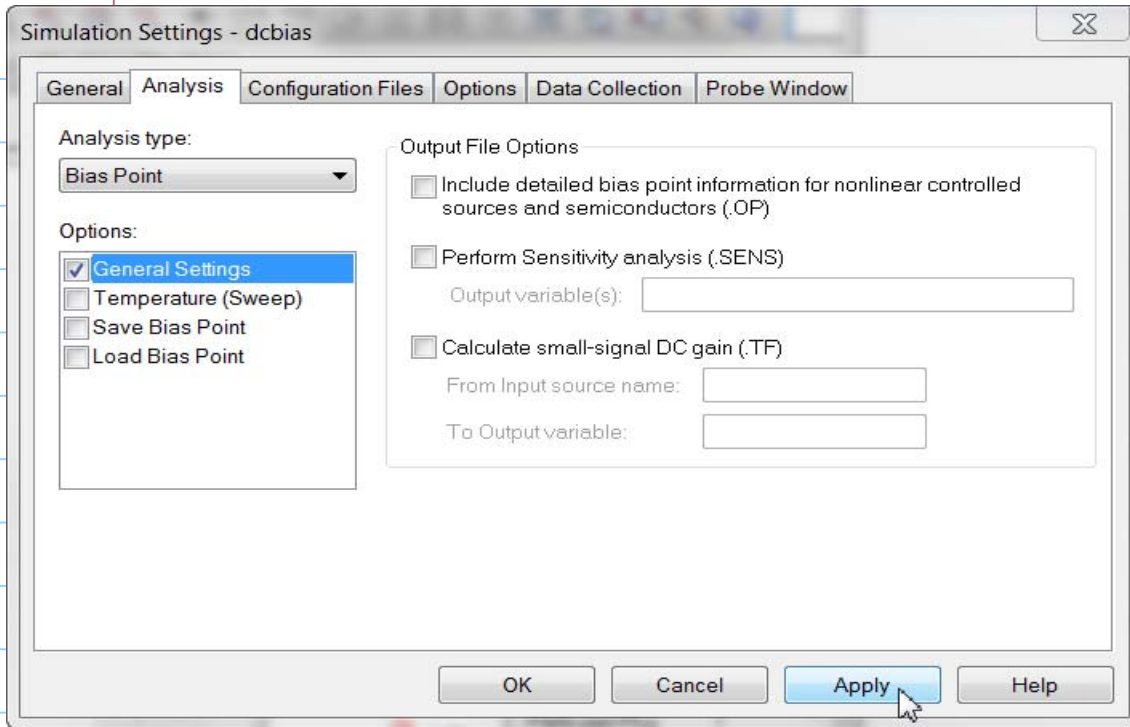
Sweep one or more DC voltage source at the same time.

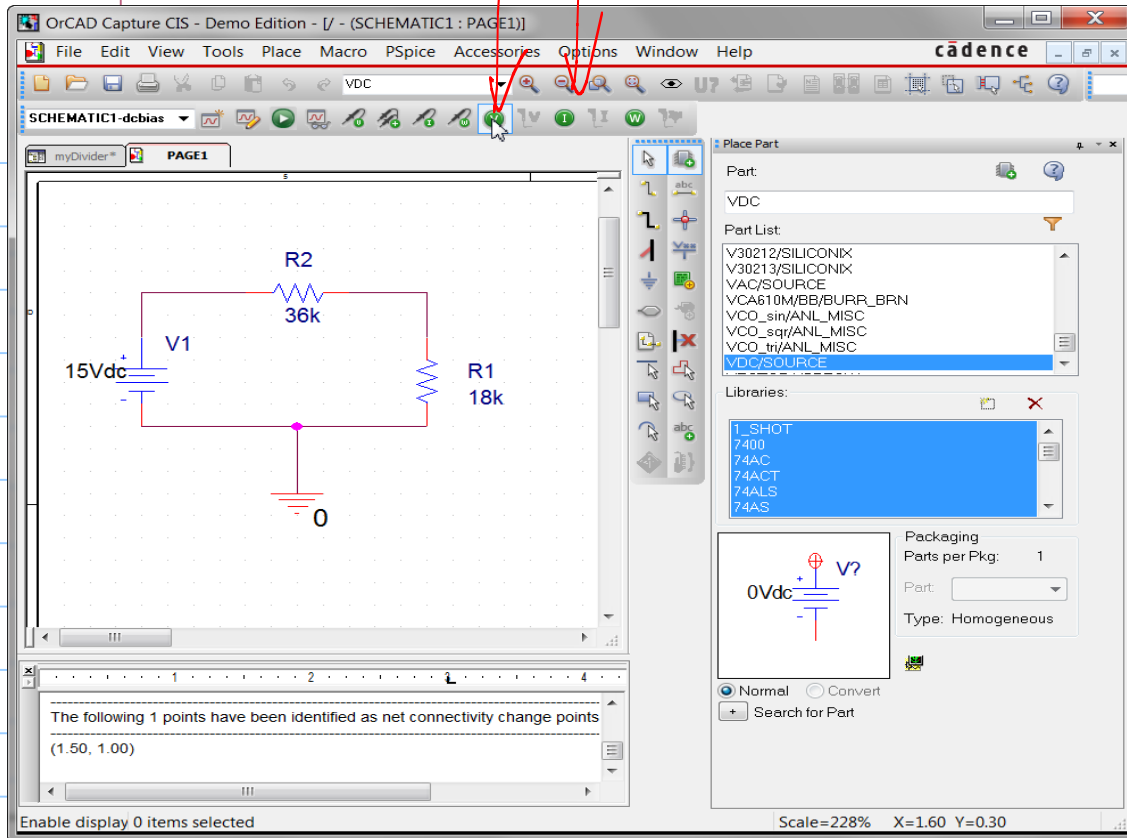
The DC output varies as the source is swept.

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:



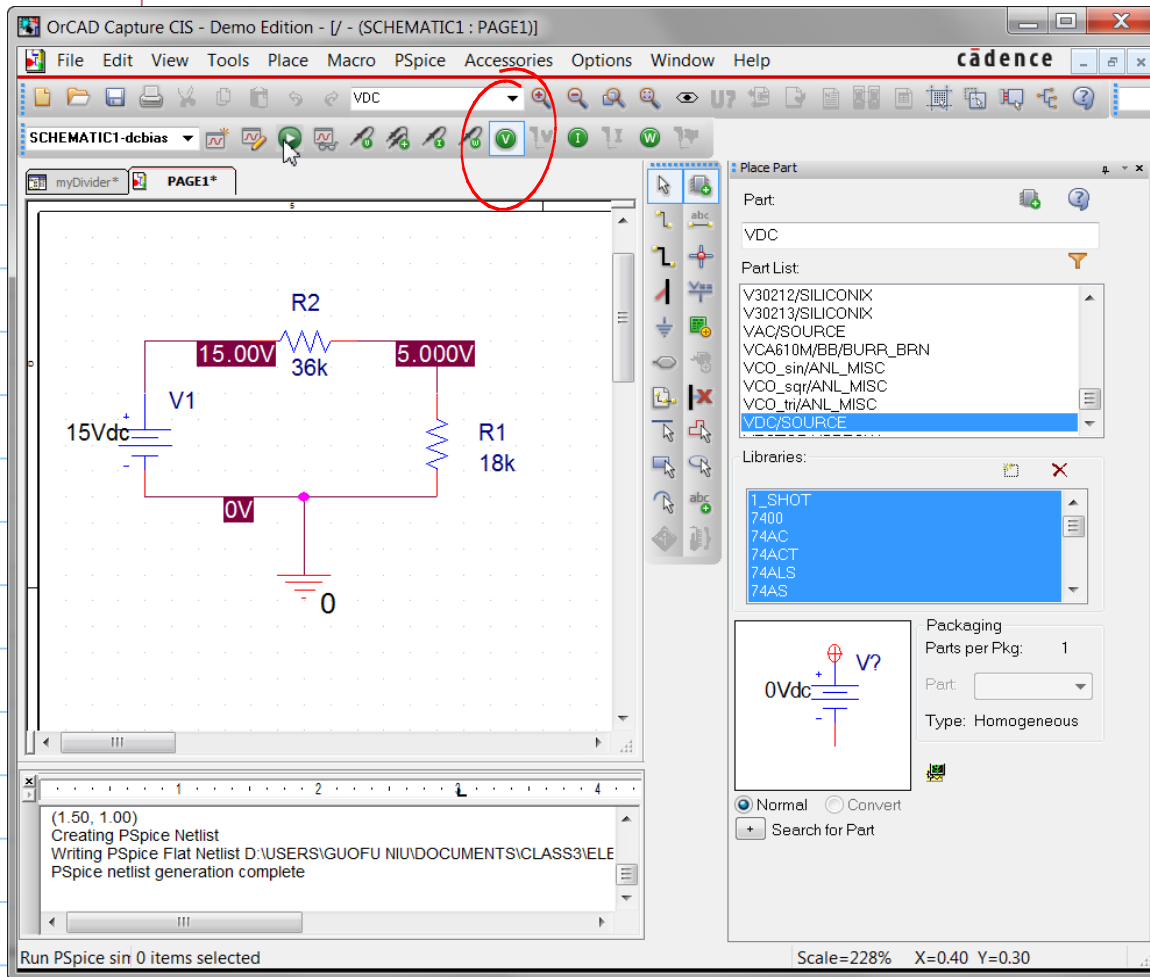
Select PSpice/Run [F11]

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

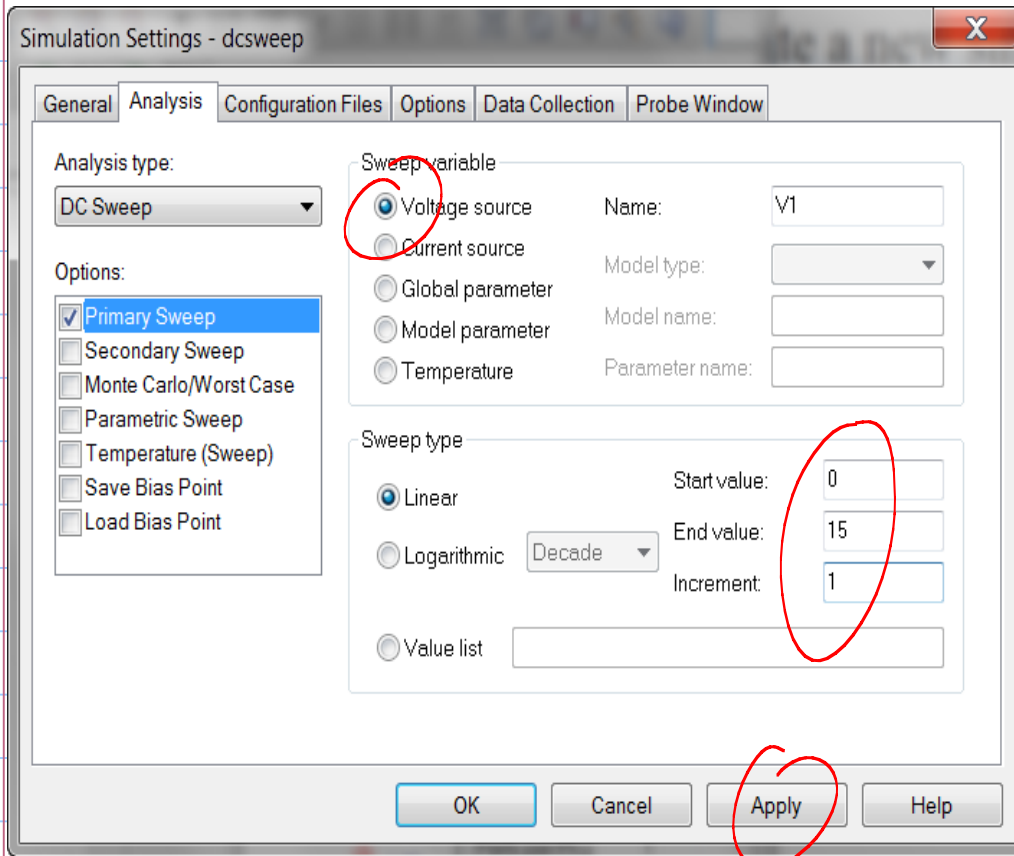


now you see the  
voltage displayed

$V(R1)$  is 5V.

## DC Sweep

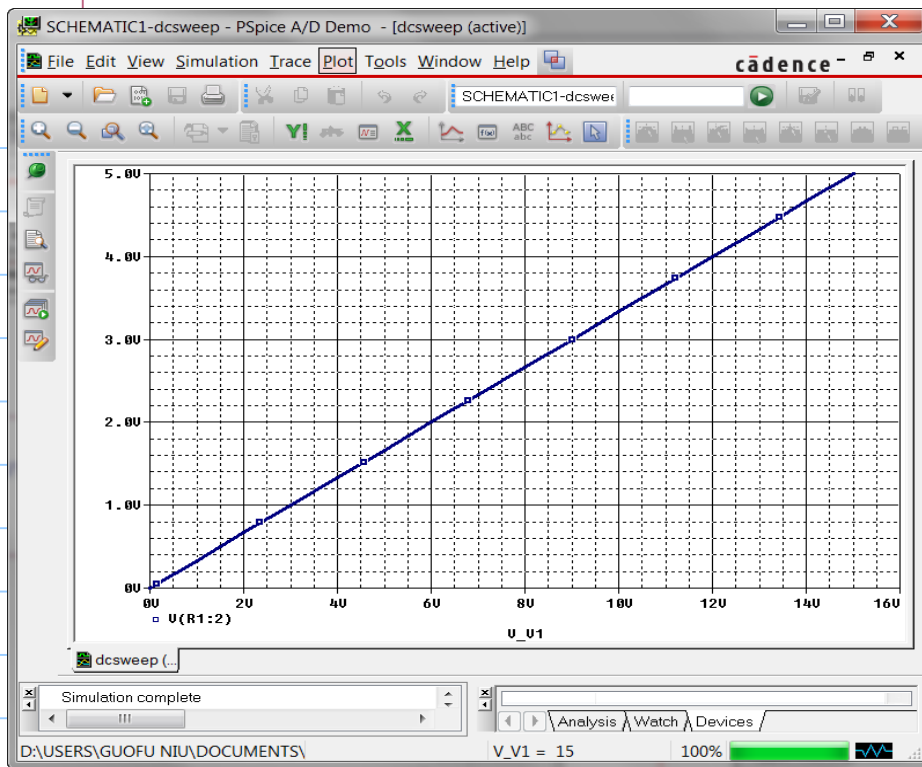
Say we want to sweep V1 and see how V(R1) responds. Simply create a new simulation profile - dc sweep



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

apply, then go back to main window,





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:

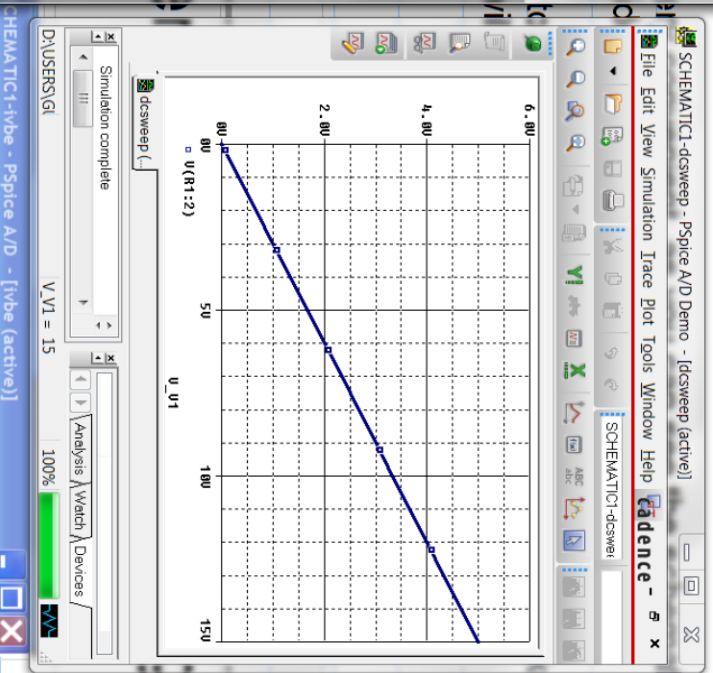
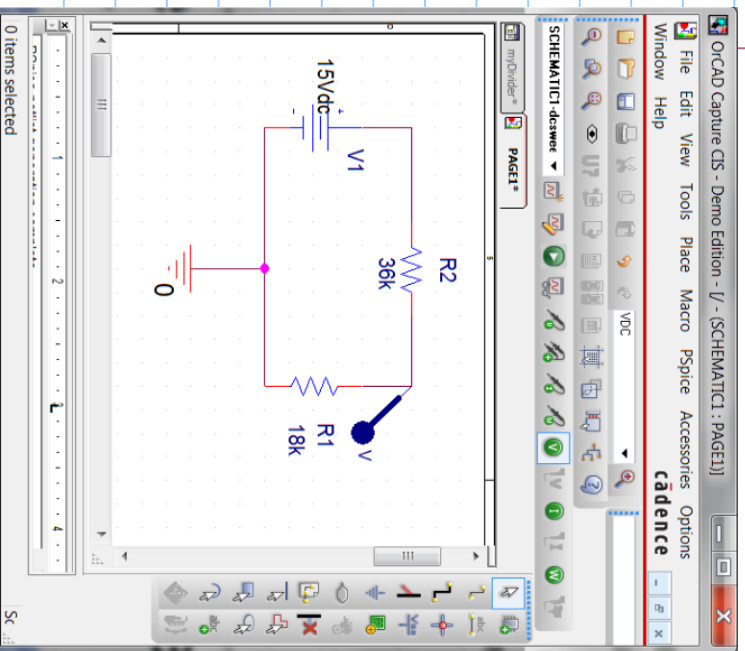
The PSPICE A/D program will display marker results

Here you can visually analyze and **interactively**

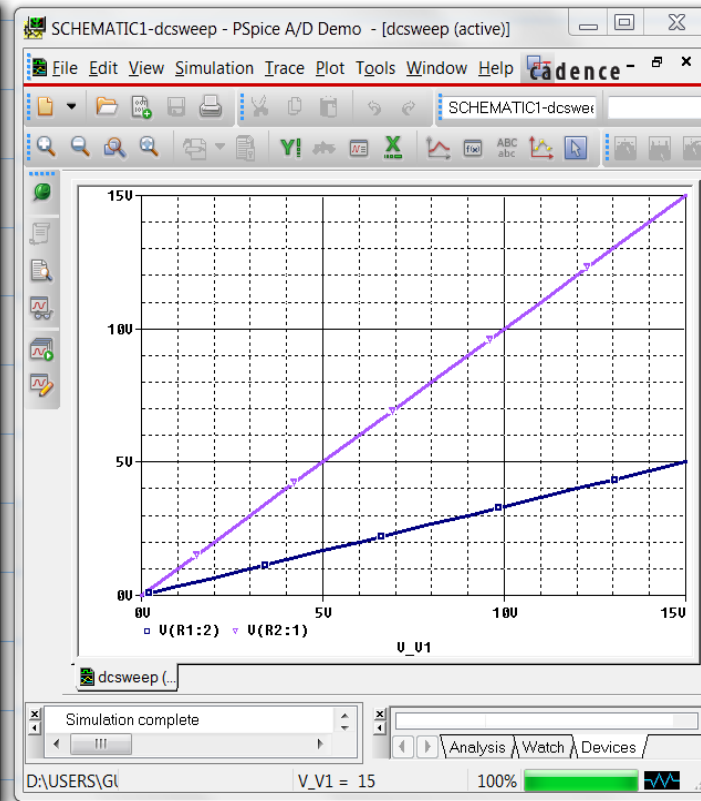
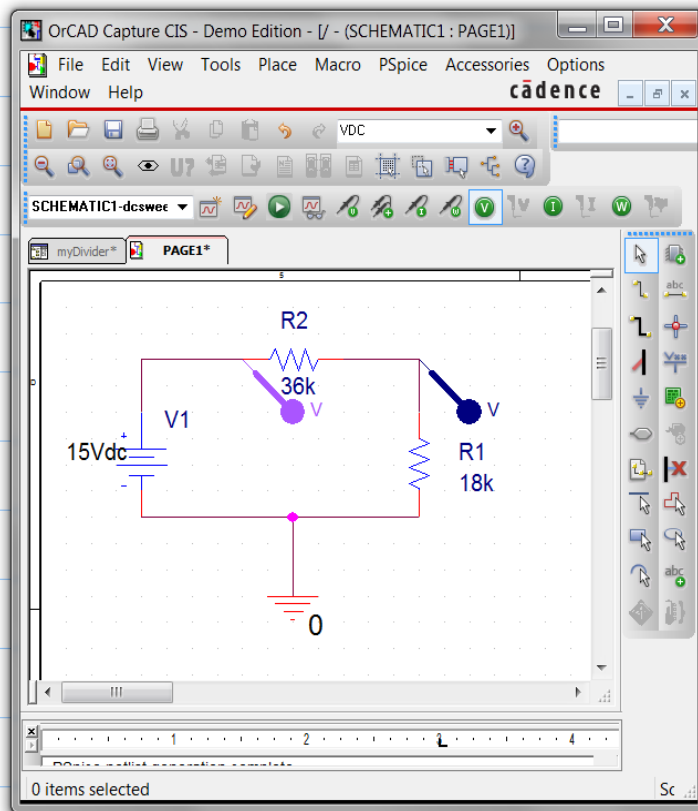
manipulate the waveform data produced by circuit simulation.

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.

