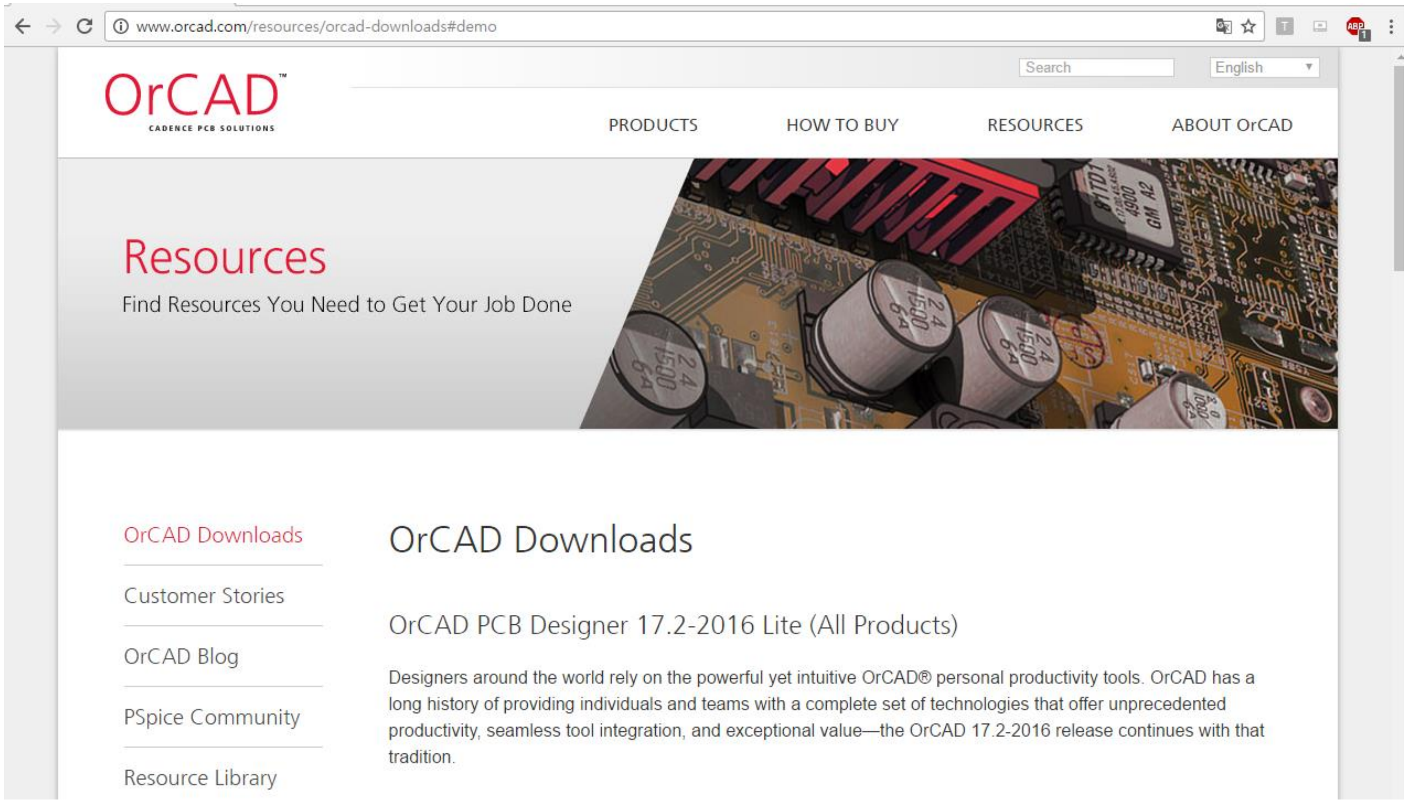
Lect. 4: PSpice Tutorial

◆ OrCAD PSPICE Installation

http://www.orcad.com/resources/orcad-downloads#demo



◆ OrCAD PSPICE Installation

Click Download FREE-OrCAD 17.2 Lite in OrCAD 17.2 Lite title(Capture & PSpice Only)

OrCAD PSpice Designer Lite (Capture & PSpice Only)

TThe OrCAD PCB Designer Lite (Capture & PSpice Only) will let you experience the features and functionality of the latest OrCAD software, with the limitations of design size and complexity, but no time limit.

The OrCAD PSpice Designer Lite includes the following tools: OrCAD Capture, OrCAD Capture CIS, PSpice A/D, PSpice Advanced Analysis.

Download FREE - OrCAD PSpice Designer Lite (Capture & PSpice)
(You must use the path/folder option in your zip tool when extracting this archive.)

Having issues with your download? Request a hard copy of the OrCAD PSpice Designer Lite DVD

◆ OrCAD PSPICE Installation

Fill out your information.

File will be sent through your e-mail address you filled out.

Check your e-mail, and download file from given URL link.

Download Lite Request

Please fill out all of the fields below to submit your request for OrCAD software. Once submitted, you will receive an email with links to your requested software, so be sure to use an email address you have access to. If you do not receive your email, don't forget to check your spam/junk folders.

First Name *

Last Name *

Email *

Company/University *

Job Title *

- Select -

Address *

Country *

- Select -

Phone *

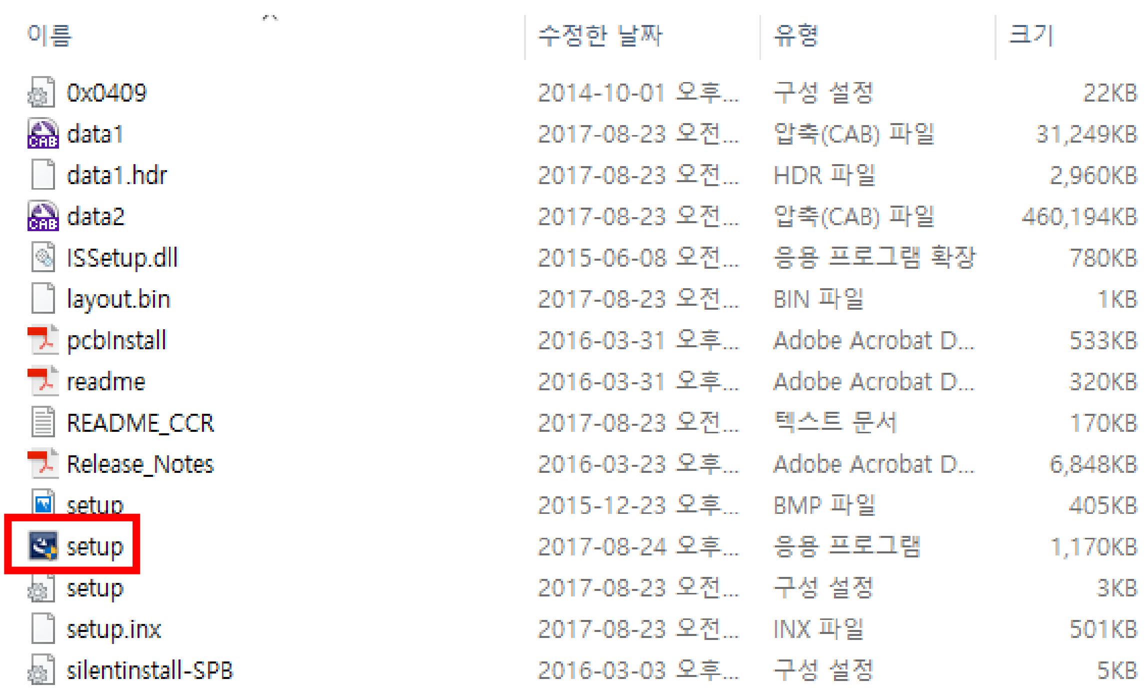
Software requested *

- ☐ OrCAD 17.2 PCB Designer Lite Software (All products, download)
- ☐ OrCAD 17.2 PSpice Designer Lite Software (Capture / PSpice only, download)
- ☐ OrCAD 17.2 PCB Designer Lite Software (All products, request DVD)

Submit

◆ OrCAD PSPICE Installation

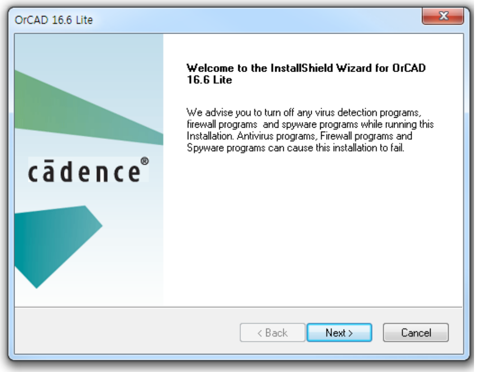
Unzip downloaded file and execute it



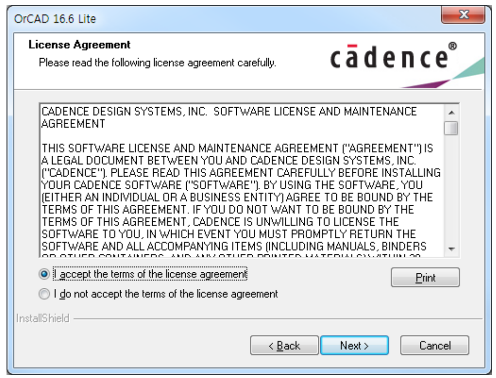
이름	수정한 날짜	유형	크기
0x0409	2014-10-01 오후...	구성 설정	22KB
data1	2017-08-23 오전...	압축(CAB) 파일	31,249KB
data1.hdr	2017-08-23 오전...	HDR 파일	2,960KB
data2	2017-08-23 오전...	압축(CAB) 파일	460,194KB
ISSetup.dll	2015-06-08 오전...	응용 프로그램 확장	780KB
layout.bin	2017-08-23 오전...	BIN 파일	1KB
pcbInstall	2016-03-31 오후...	Adobe Acrobat D...	533KB
readme	2016-03-31 오후...	Adobe Acrobat D...	320KB
README_CCR	2017-08-23 오전...	텍스트 문서	170KB
Release_Notes	2016-03-23 오후...	Adobe Acrobat D...	6,848KB
setup	2015-12-23 오후...	BMP 파일	405KB
setup	2017-08-24 오후...	응용 프로그램	1,170KB
setup	2017-08-23 오전...	구성 설정	3KB
setup.inx	2017-08-23 오전...	INX 파일	501KB
silentinstall-SPB	2016-03-03 오후...	구성 설정	5KB

◆ OrCAD PSPICE Installation

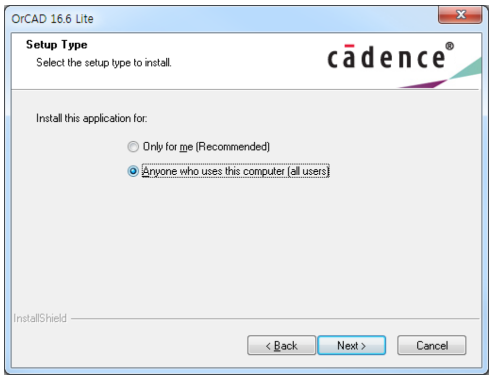
①



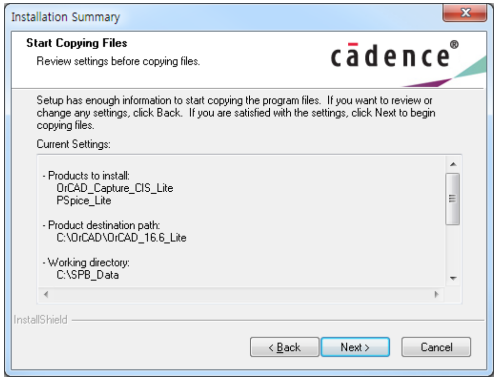
②



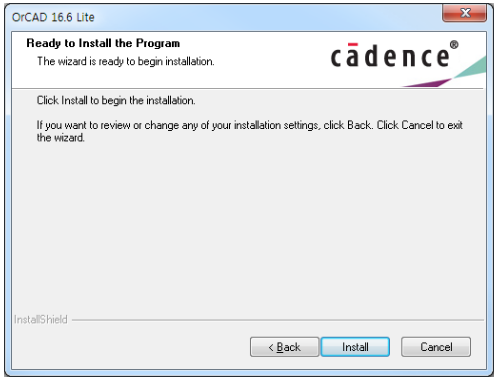
③



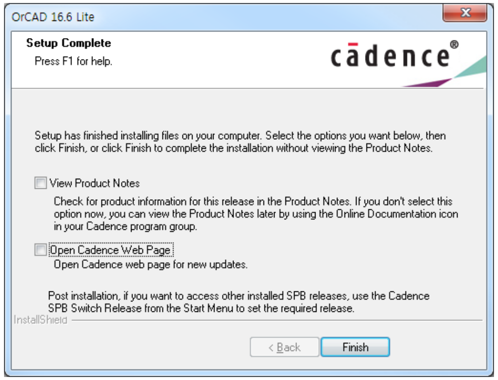
④



⑤



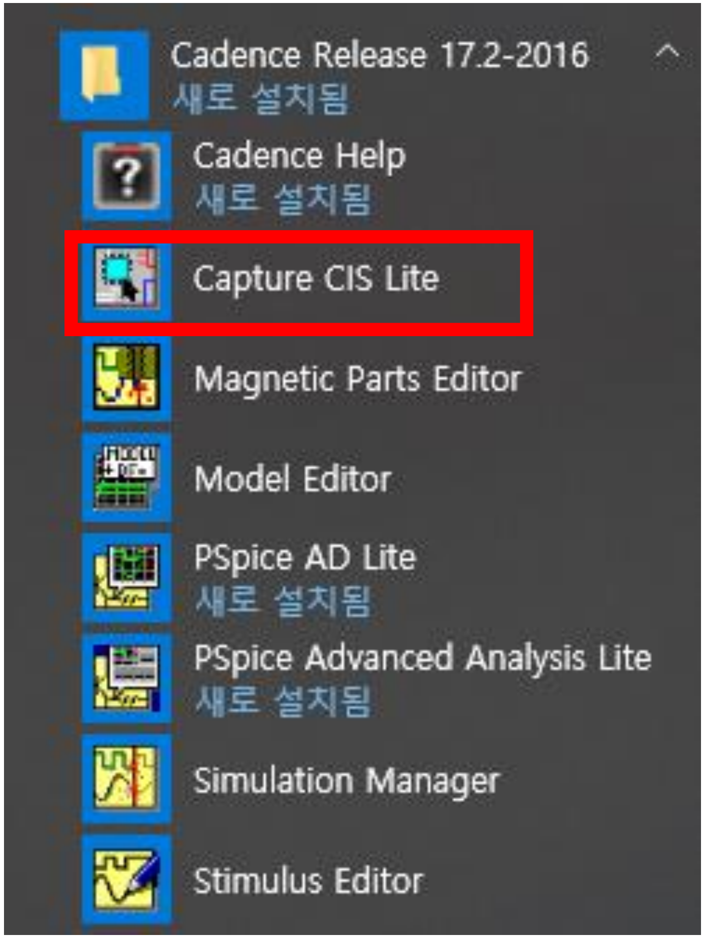
⑥



The path name should be in English!

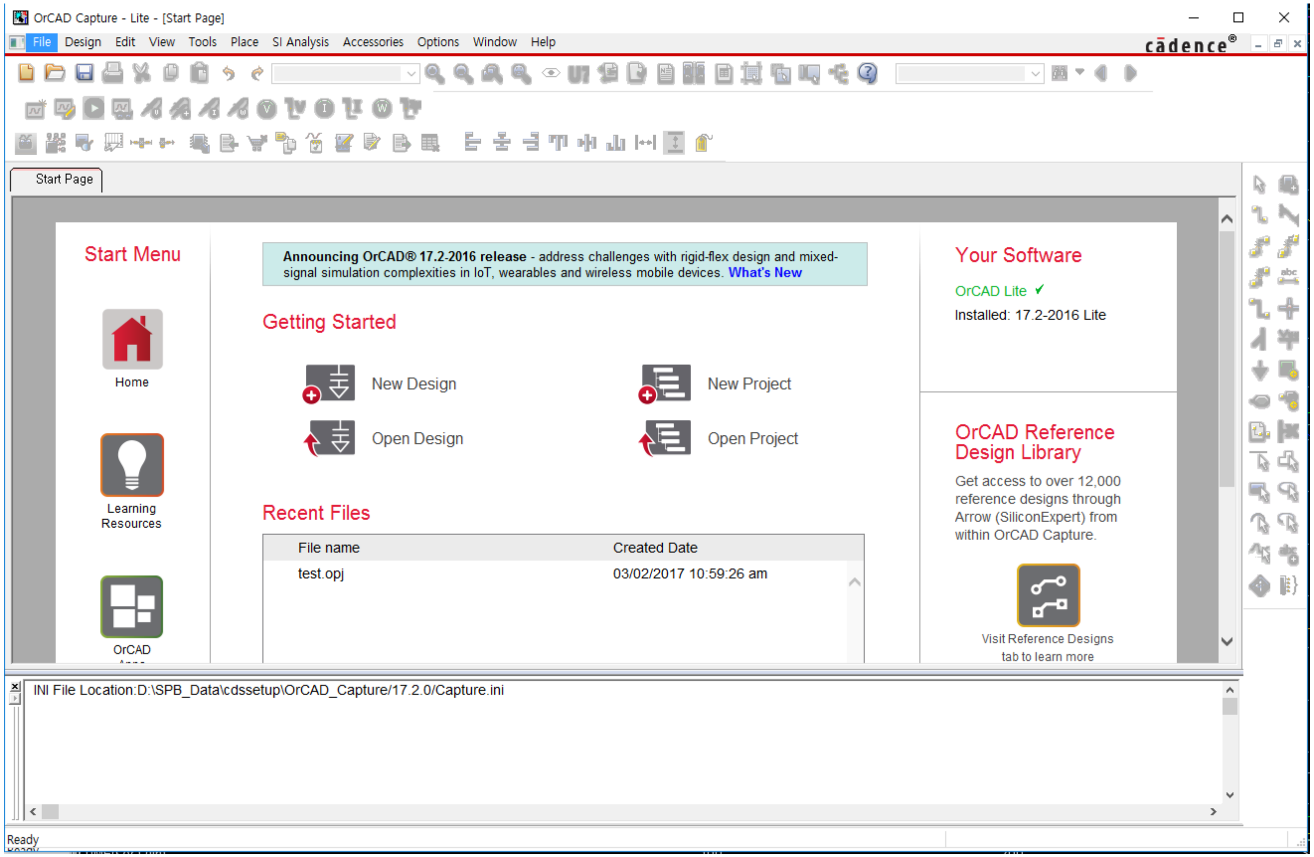
◆ OrCAD PSPICE Installation

Start OrCAD Capture CIS Lite



◆ OrCAD PSPICE Installation

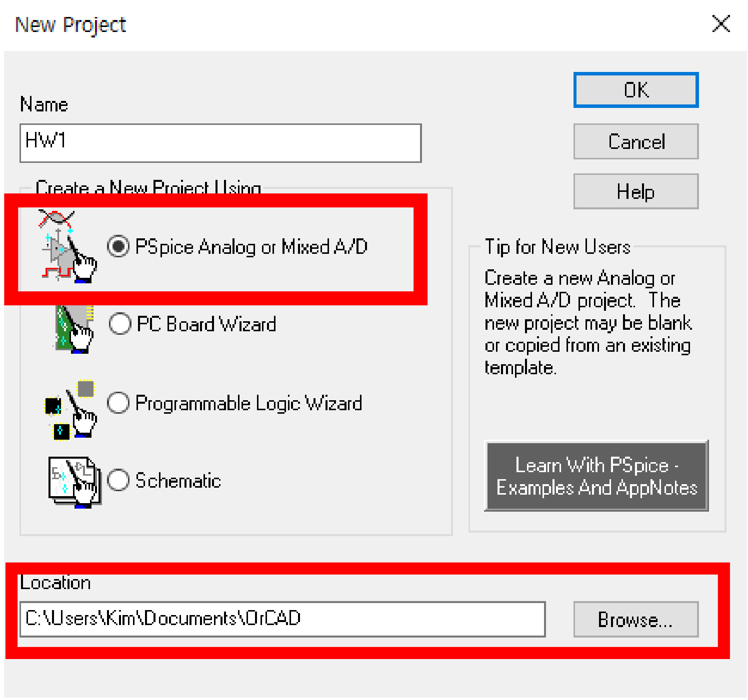
File -> New Project to open schematics



◆ OrCAD PSPICE Installation

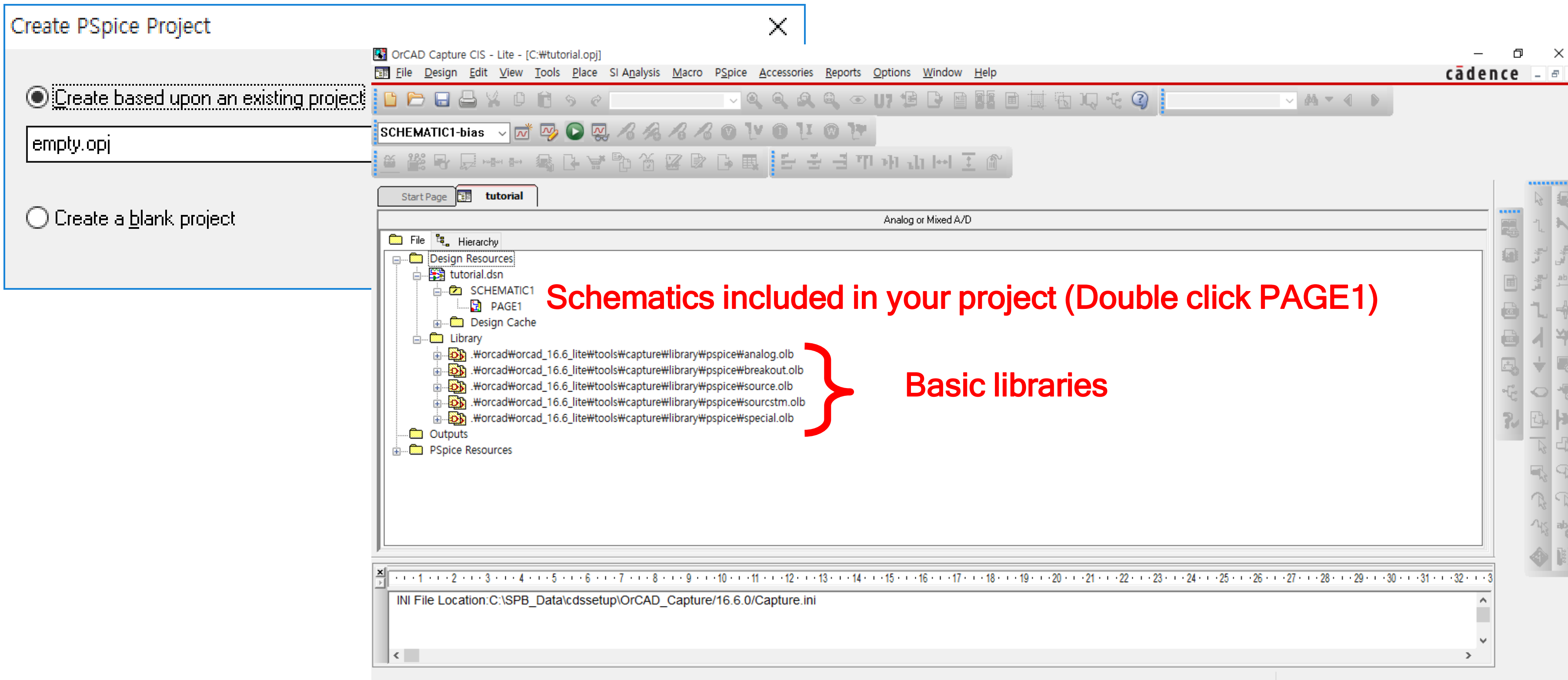
Choose PSpice Analog or Mixed A/D project

File path name should be in English!



◆ OrCAD PSPICE Installation

Choose empty.opj for the base project. If not, basic libraries will not be included in your project

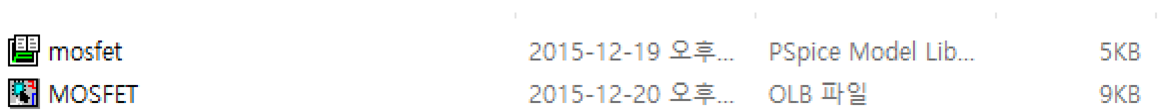


W.-Y. Choi

◆ MOSFET Library Setup

Download ‘MOSFET_OrCAD.zip’ file which is uploaded in YSCEC.

Check mosfet.lib & MOSFET.olb is in your unzipped folder.



◆ MOSFET Library Setup

1. Download 'MOSFET_OrCAD.zip' file uploaded in YSCEC.

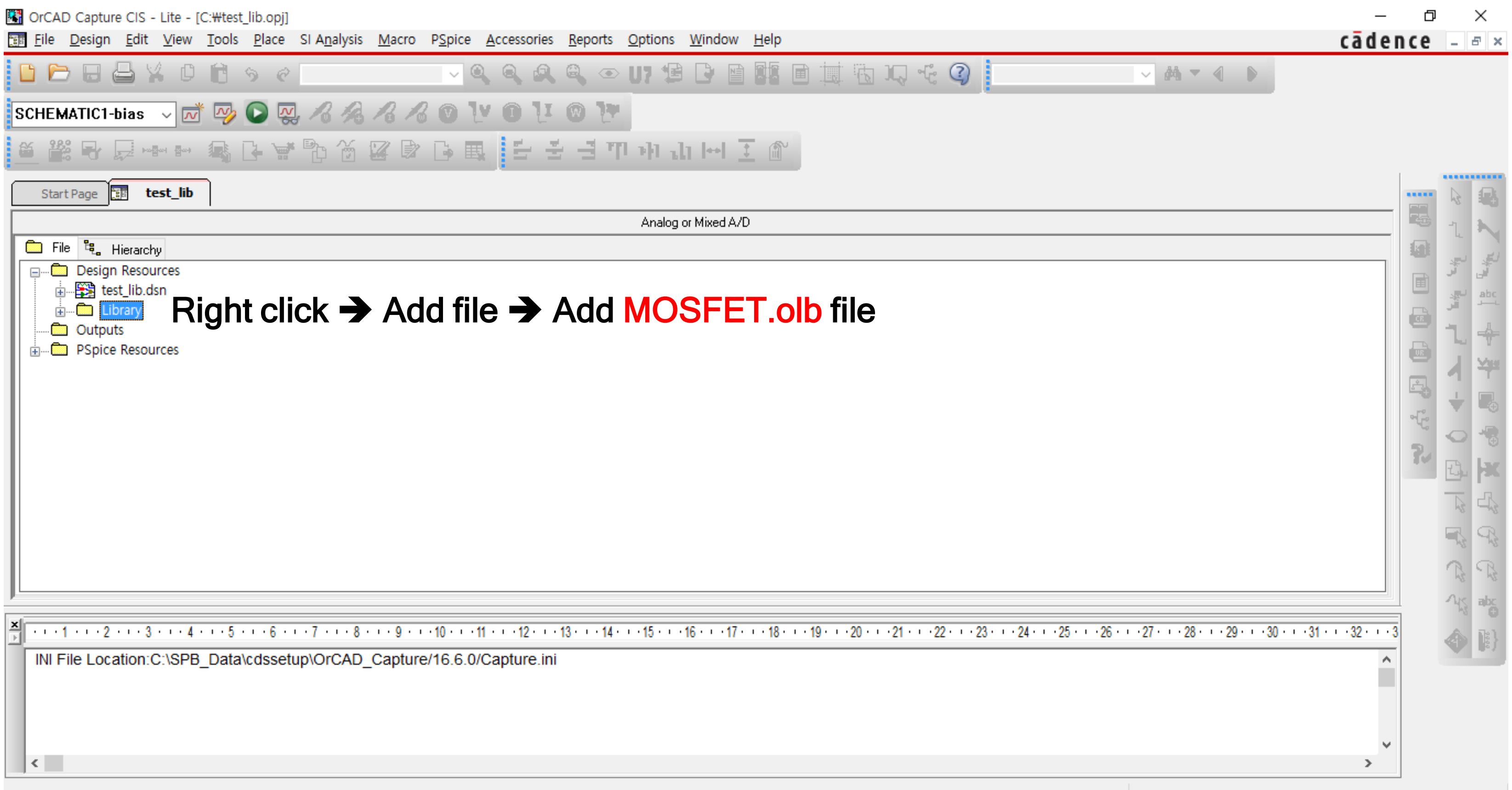
2. Check mosfet.lib & MOSFET.olb is in your unzipped folder.

3. File path of this library folder must not have the Korean. (Move this folder to 'C drive' or' My Document')

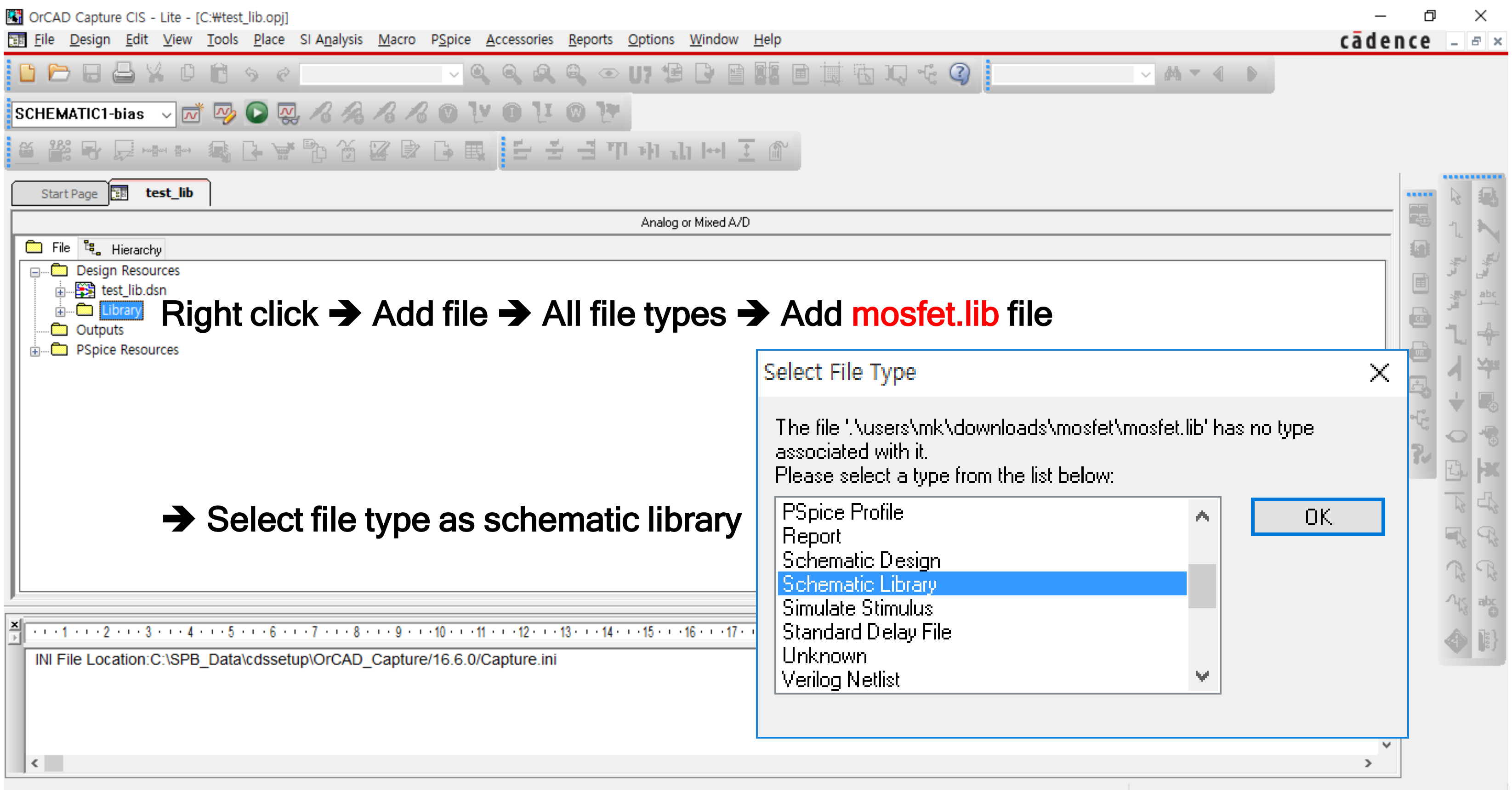
mosfet	2015-12-19 오후...	PSpice Model Lib...	5KB
MOSFET	2015-12-20 오후...	OLB 파일	9KB

W.-Y. Choi

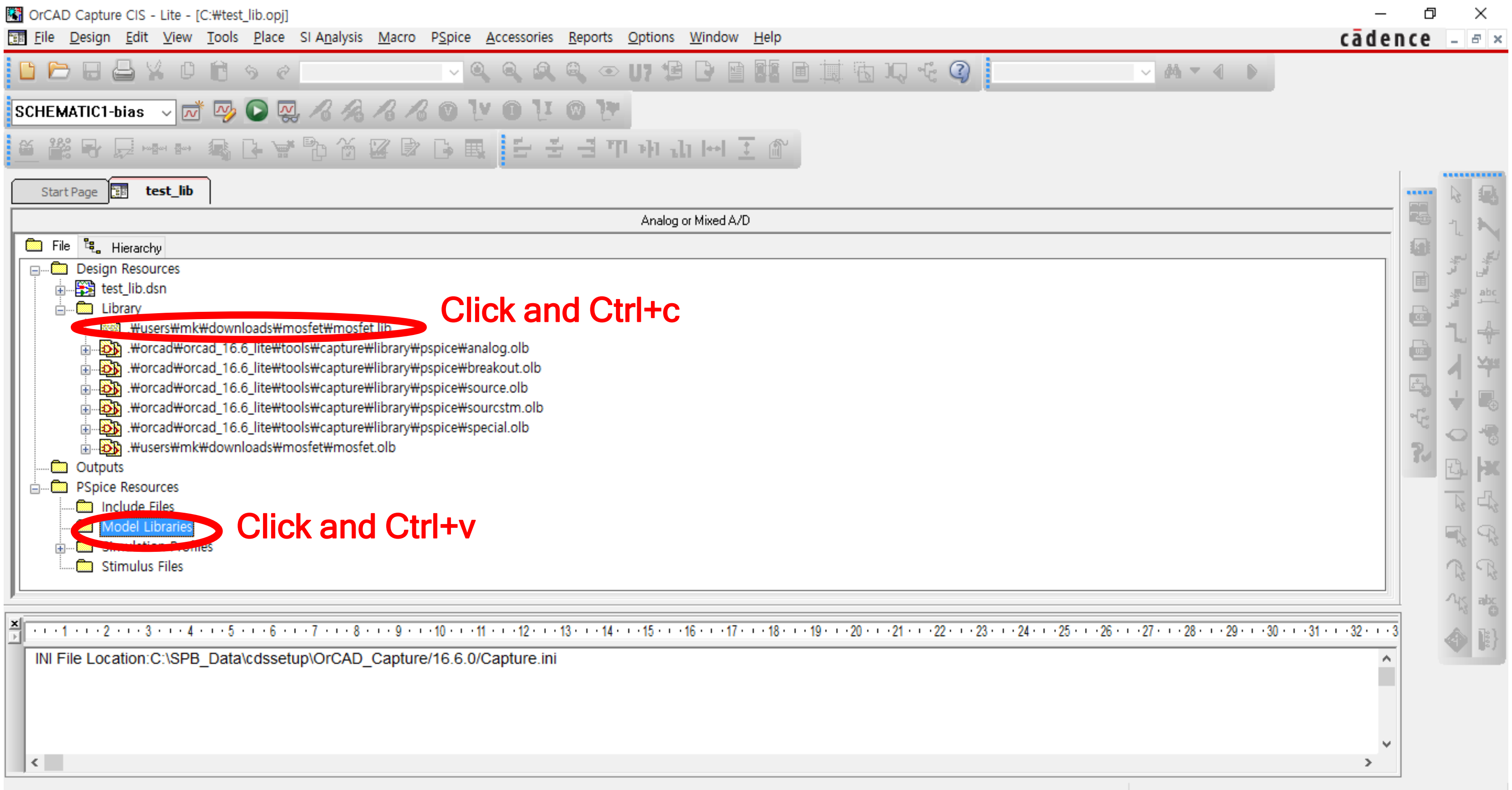
◆ MOSFET Library Setup



◆ MOSFET Library Setup

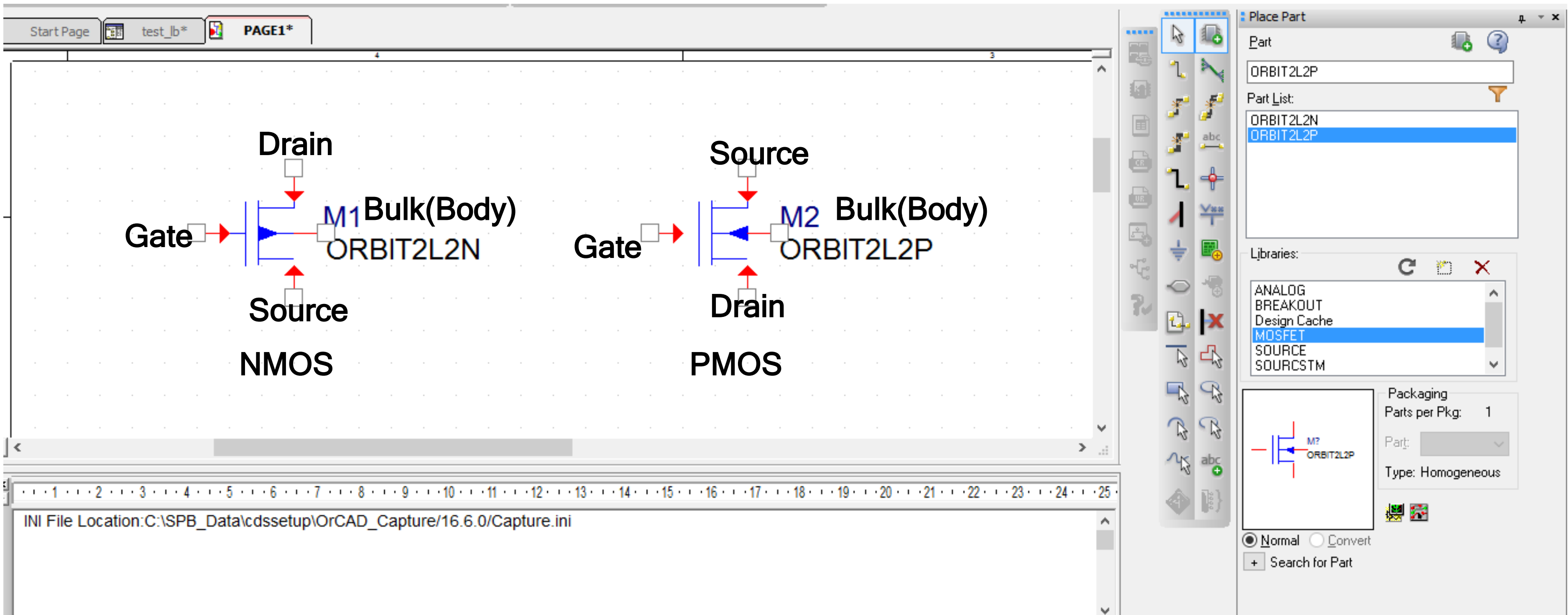


◆ MOSFET Library Setup



W.-Y. Choi

◆ MOSFET Library Setup



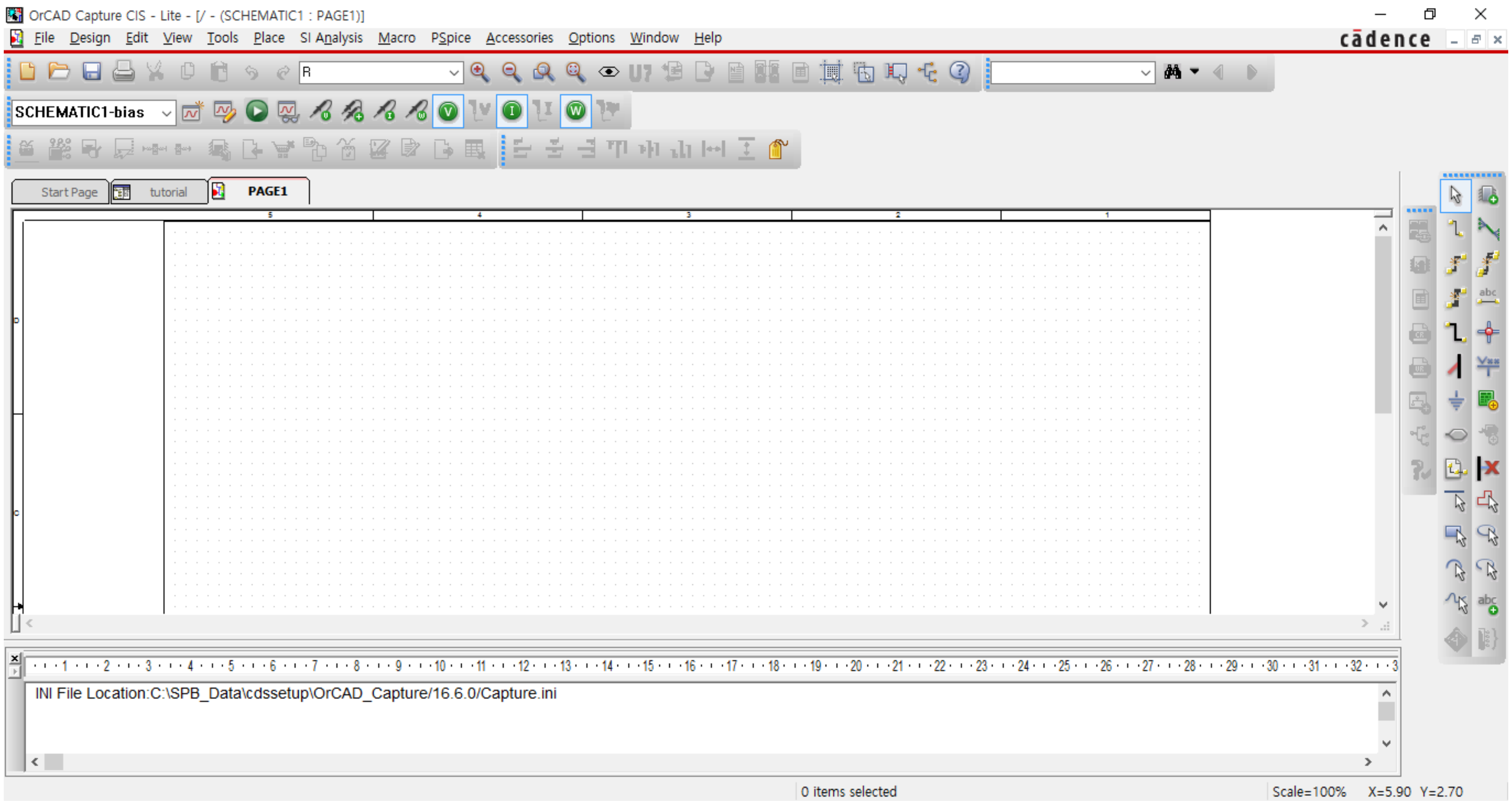
Double click ➔ change W & L of MOSFETs

NMOS body connected to Ground

PMOS body connected to VDD

◆ PSPICE Basics

Empty schematic window is popped-up



W.-Y. Choi

◆ PSPICE Basics

Zoom Areas(Zoom in, out, fit)

Simulations
(setting, edit, run, results)

Markers
(current, voltage, diff. voltage, power))

Place part

Wire

Ground

- Wire: draw wires(W)
- Place part: place instances(P)
  - Resistor: R, Capacitor: C, Inductor: L
  - DC voltage source: VDC
  - Sinusoidal voltage source: Vsin
  - AC voltage source: VAC
- Voltage/Current/Power Marker: Show output as marked port

◆ PSPICE Basics

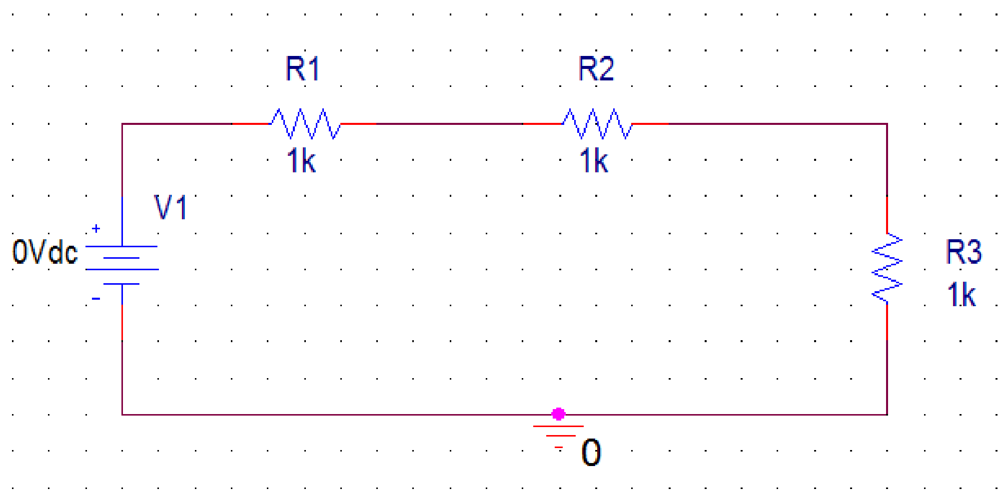
Key shortcuts

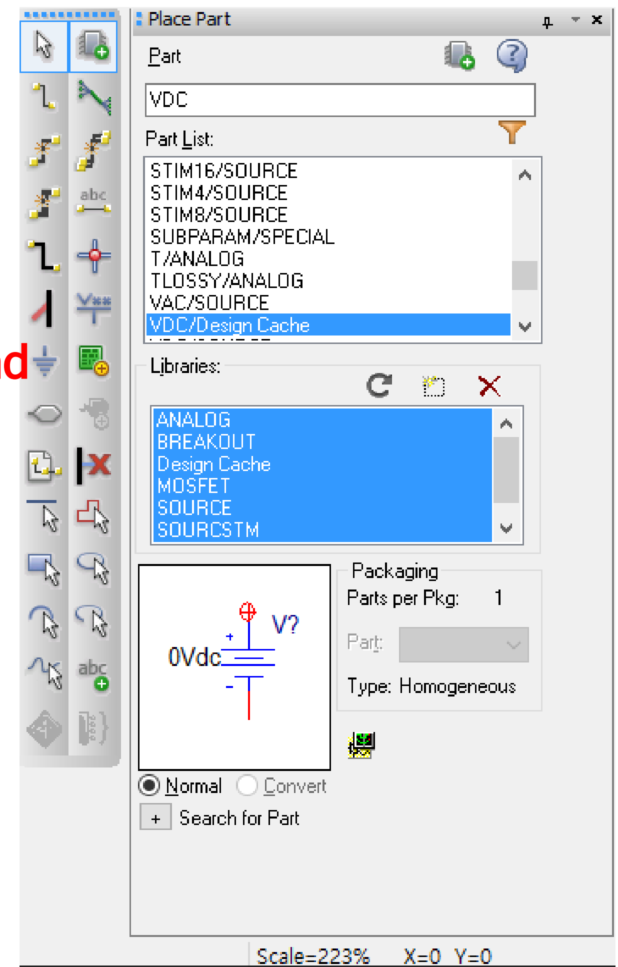
Description	Shortcut
Rotate	R
Mirror Horizontal	H
Mirror Vertical	V
Copy	Ctrl+C
Paste	Ctrl+V

◆ PSPICE Basics

Draw schematic

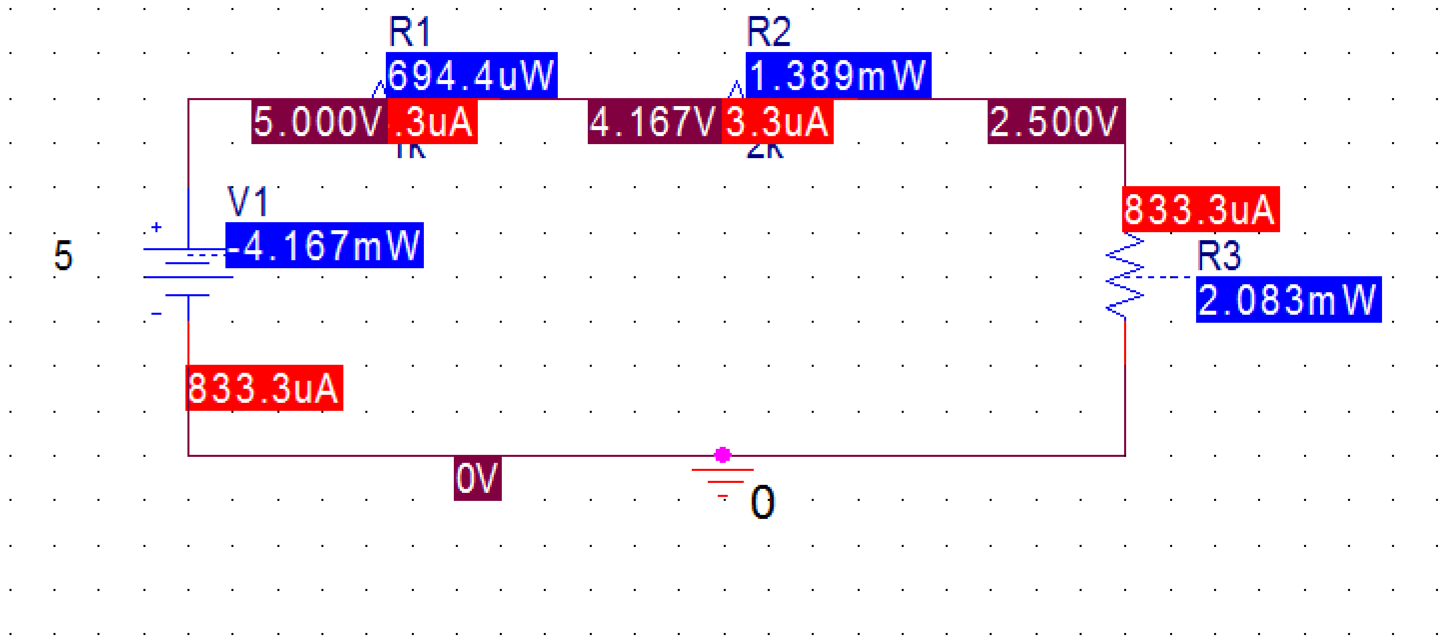
- Use place part to place instances
- Ground
- Double click parts: change values
- Use wire to connect(W)
- Save schematic





◆ Part.1 Bias point simulation

Bias point simulation is used for checking each node's Average Voltage, Current, Power at specific DC point

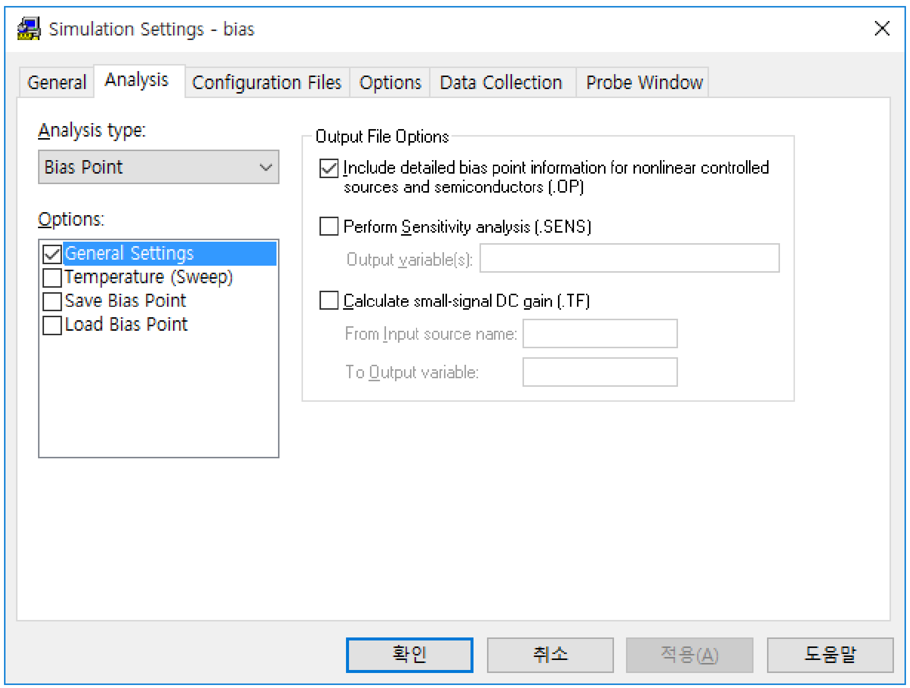


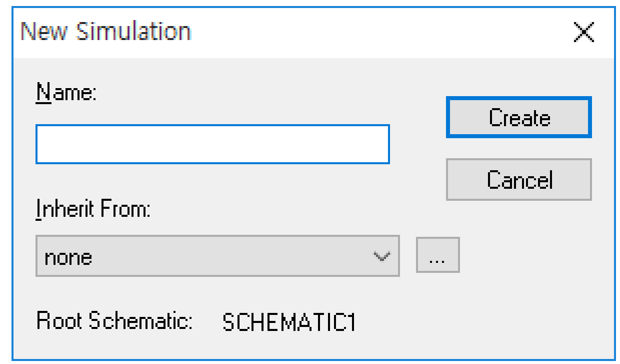
W.-Y. Choi

◆ Part.1 Bias point simulation

New simulation

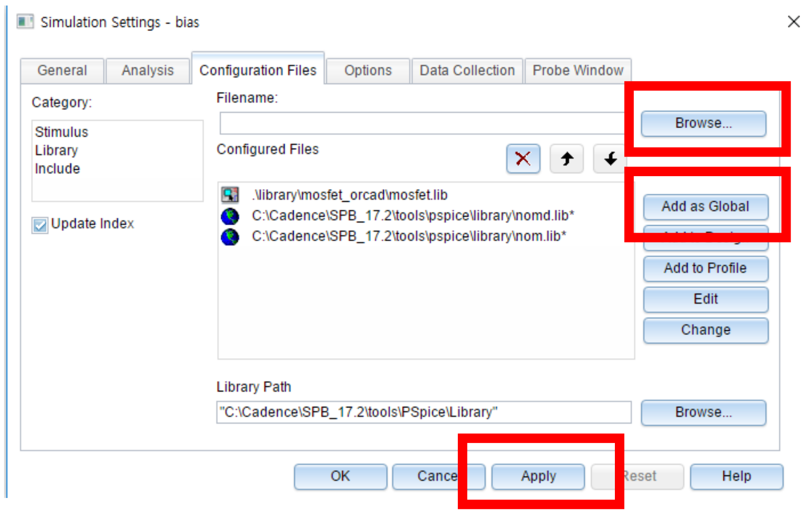
New simulation
(Bias simulation would be already added if you have proper base project.)





◆ Part.1 Bias point simulation

Library Setting



Browse file

C:/Cadence/SPB_17.2/tools/PSpice/Library/nom.lib

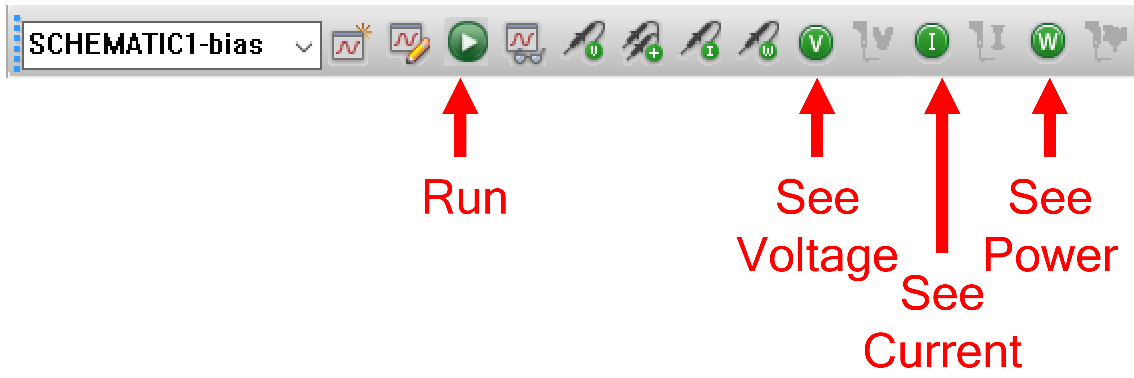
C:/Cadence/SPB_17.2/tools/PSpice/Library/nomd.lib

Click Add as Global and Apply

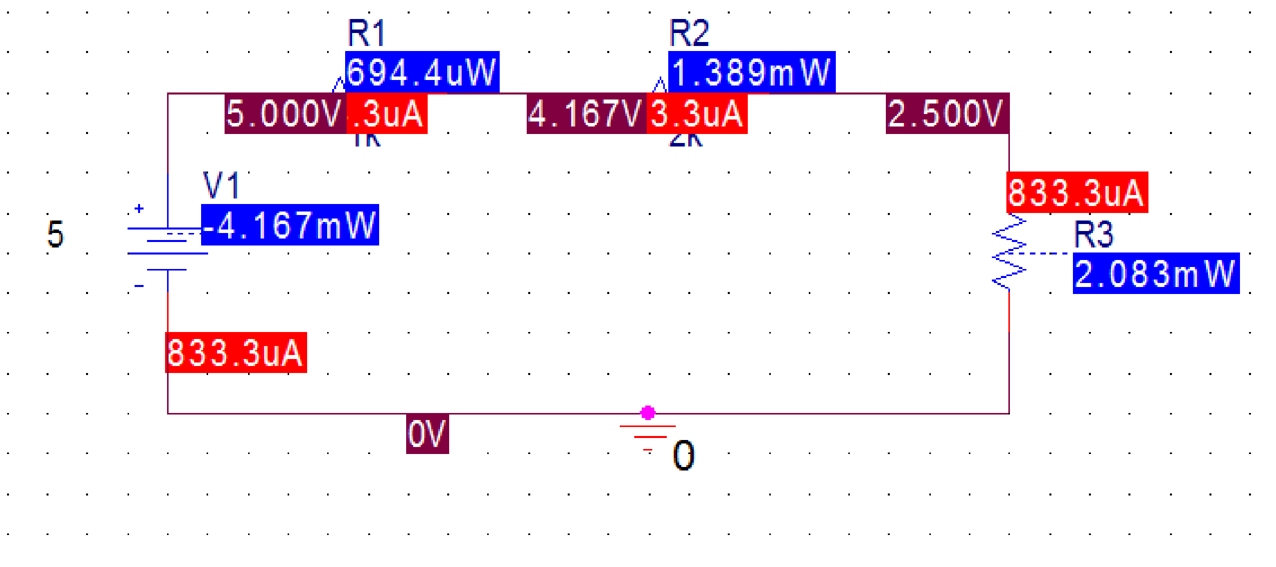
Library Setting needs for all simulation!

◆ Part.1 Bias point simulation

Run simulation



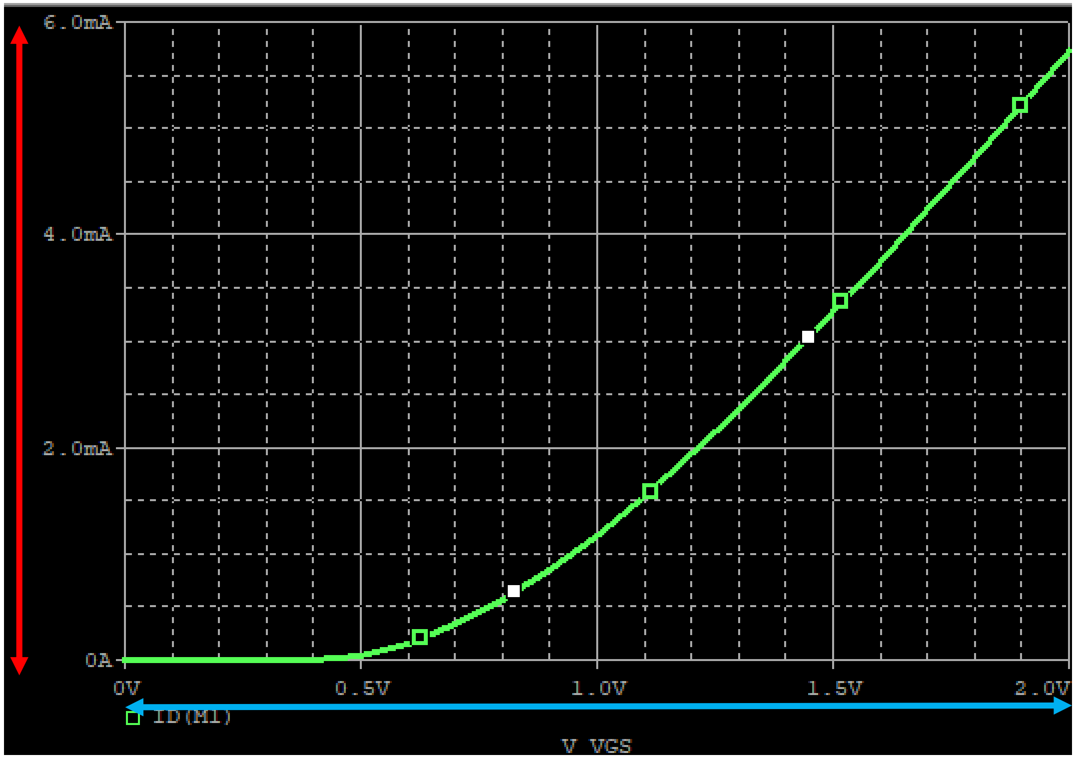
Simulation Results



◆ Part.2 DC sweep simulation

DC sweep is used for seeing specific point's voltage or current variation when changing DC value

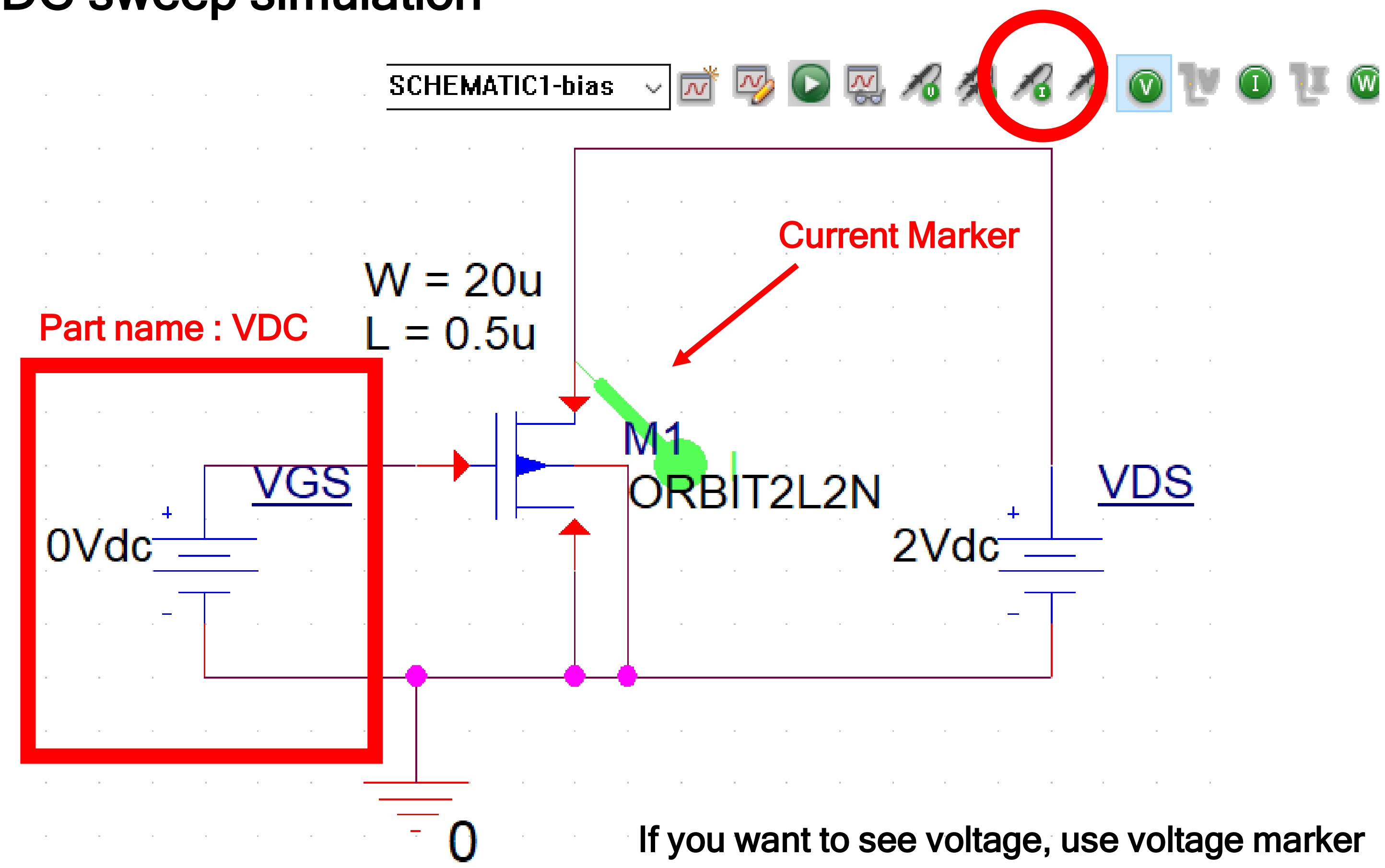
Specific point's Variation



DC value changing

W.-Y. Choi

◆ Part.2 DC sweep simulation

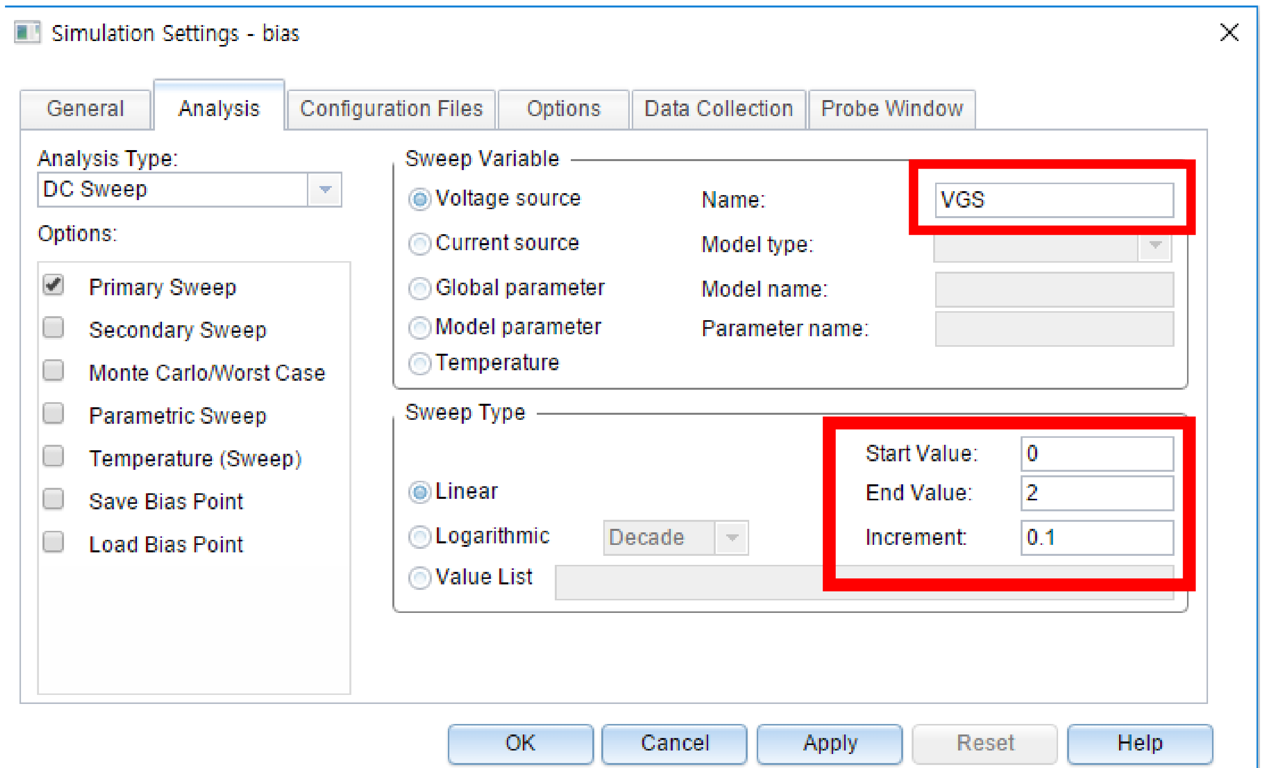


If you want to see voltage, use voltage marker

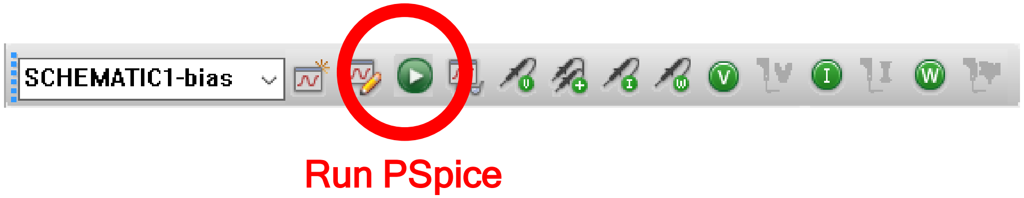
Lect. 4: PSpice Tutorial

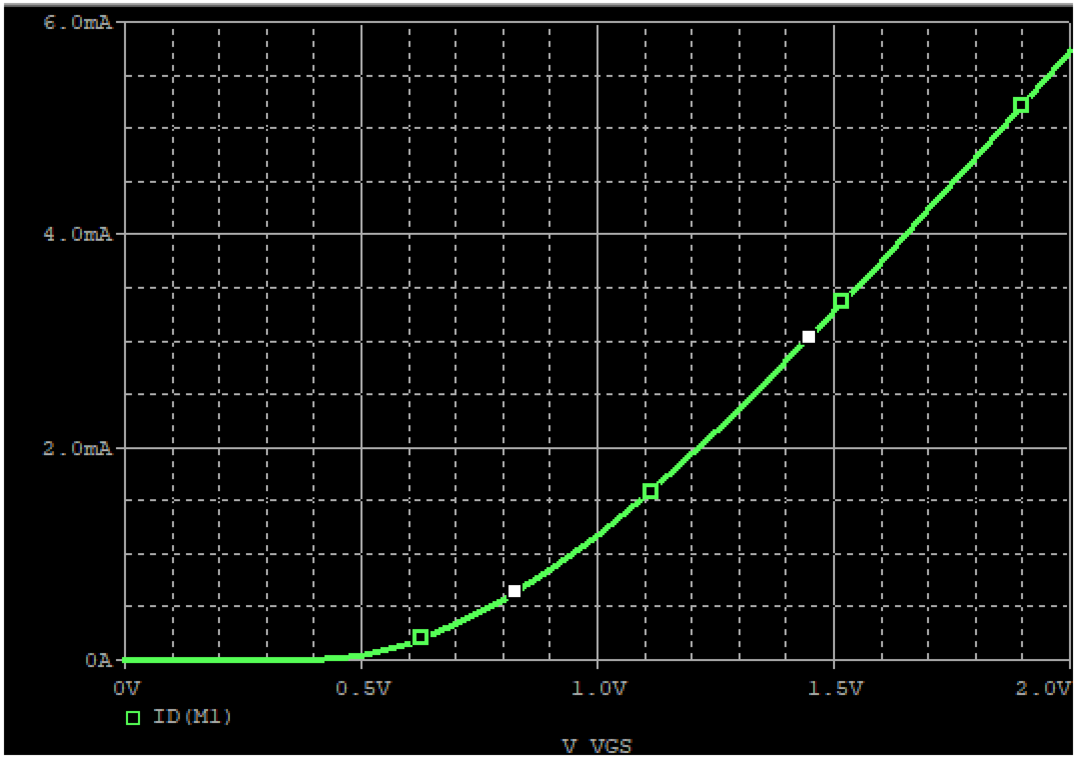
◆ Part.2 DC sweep simulation

Click Simulation Settings



◆ Part.2 DC sweep simulation

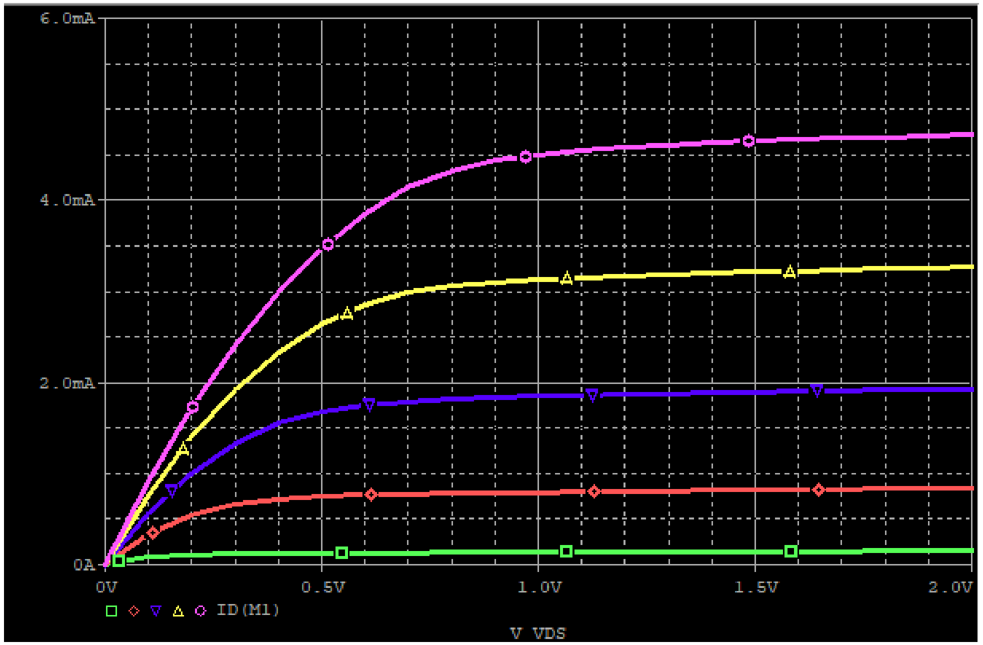




W.-Y. Choi

◆ Part.3 Parametric Simulation

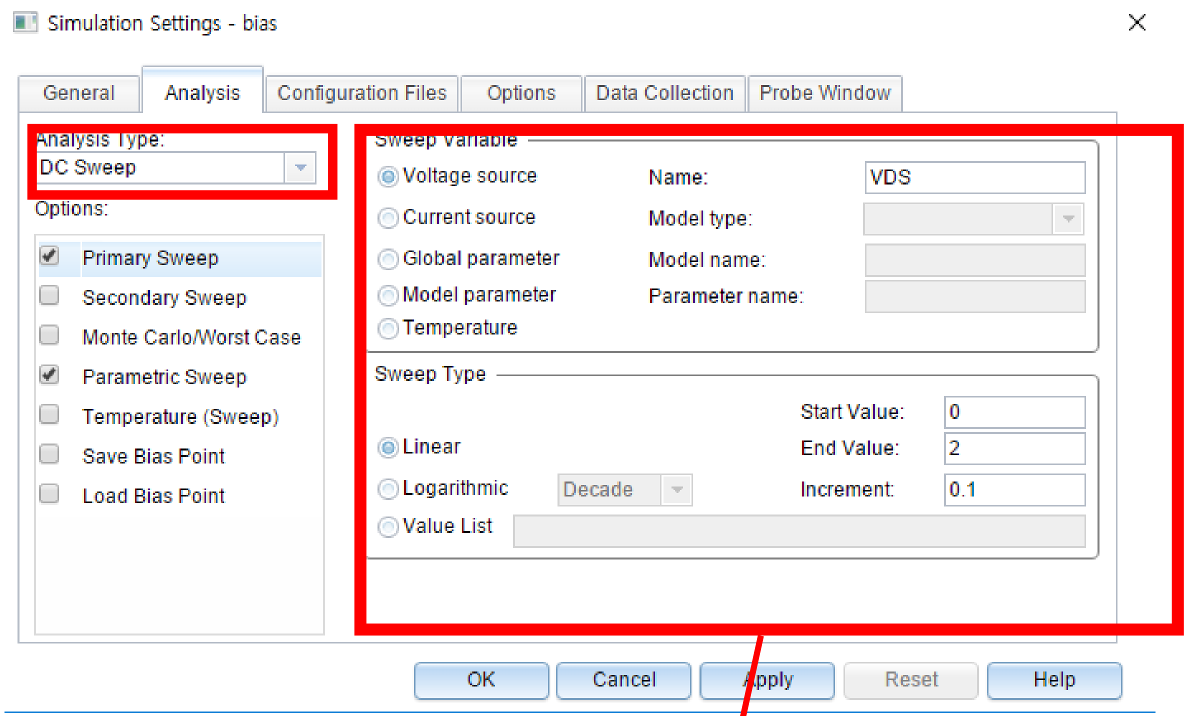
Parametric simulation is an option with which you can view simulation results for varying values of a specific parameter.



Parametric simulation can be used for not only DC sweep but also AC sweep and Transient simulation

◆ Part.3 Parametric Simulation

Choose simulation type you want to view



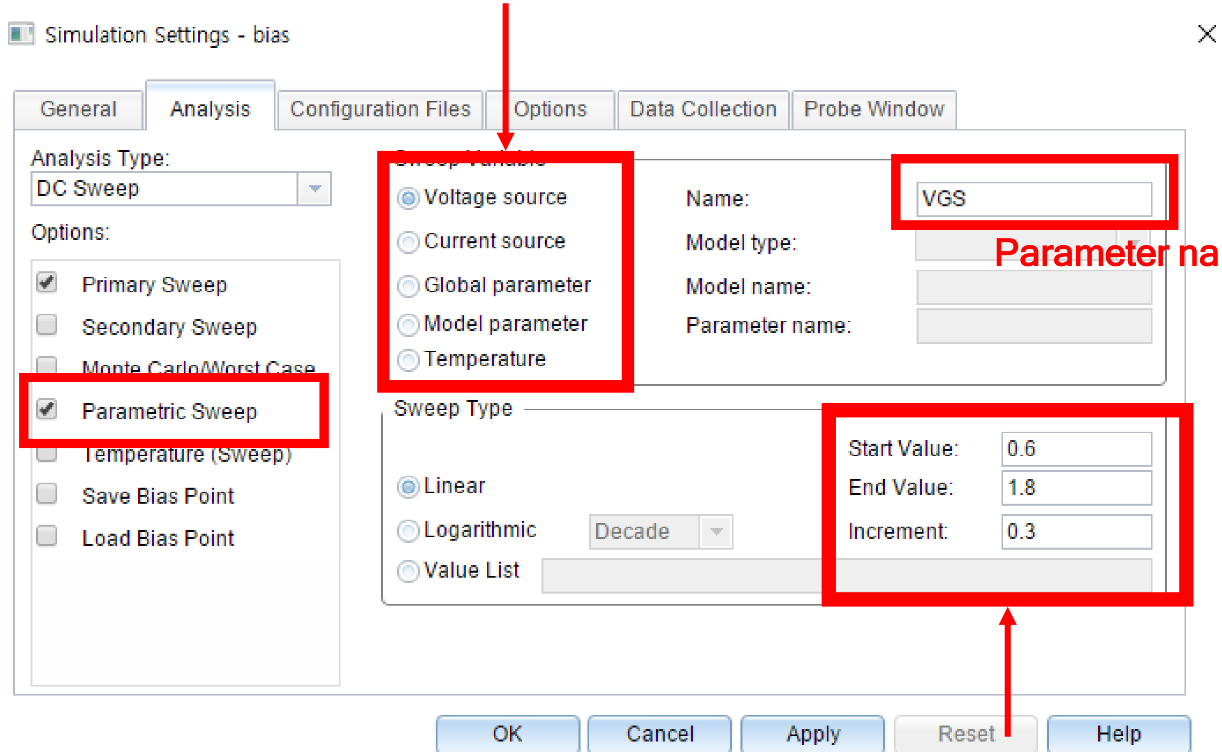
Don't forget to setup sweep condition!

◆ Part.3 Parametric Simulation

1. Source value change

Choose Voltage source

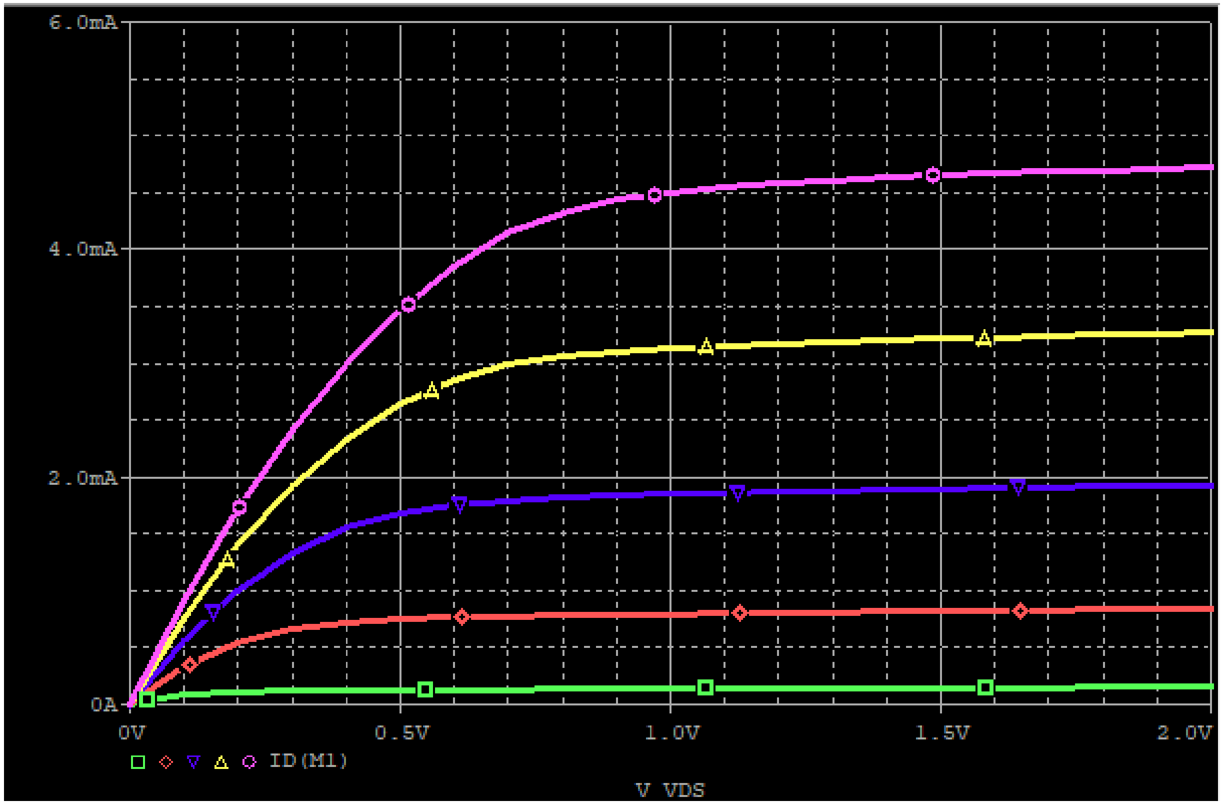
click Parametric Sweep



Parameter name

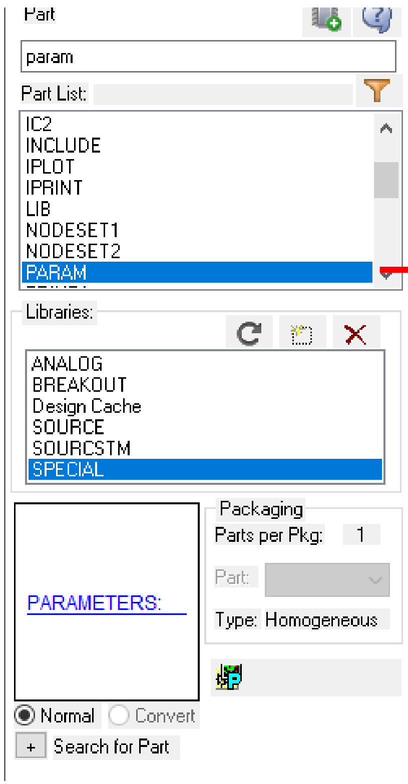
Changing value
(At this condition, VGS may be set to 0.6, 0.9, 1.2, 1.5, 1.8)

◆ Part.3 Parametric Simulation



◆ Part.3 Parametric Simulation

2. Passive element(resistor, capacitor) change

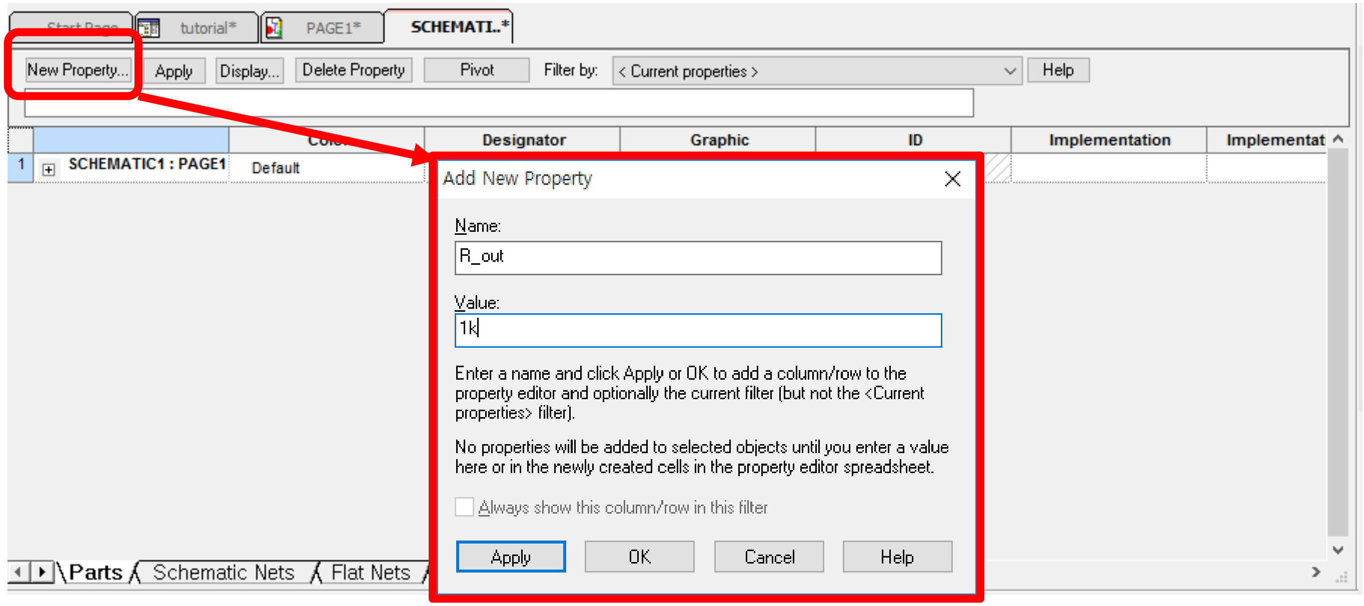


Find PARAM and place it to the schematic

◆ Part.3 Parametric Simulation

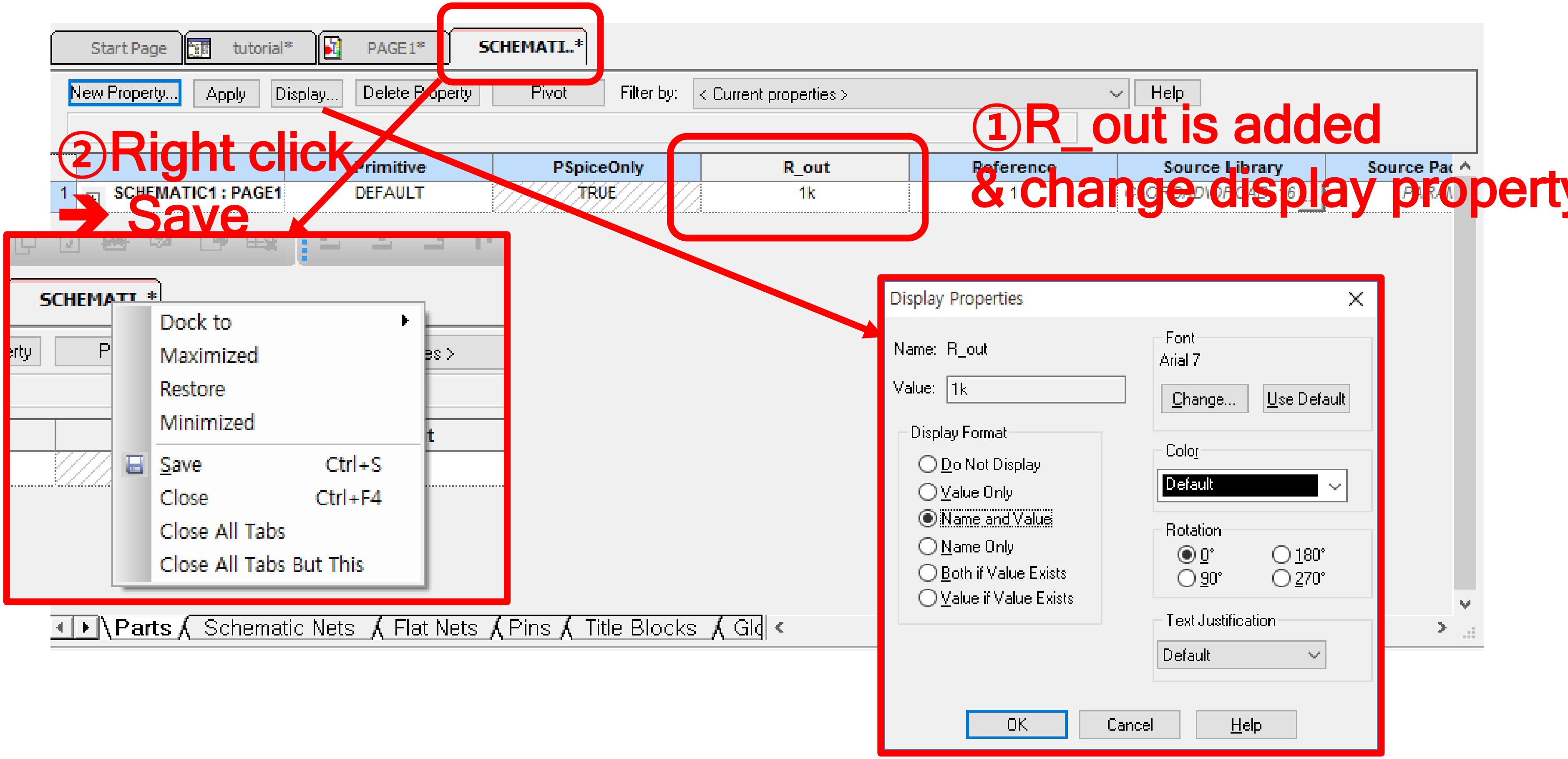
2. Passive element(resistor, capacitor) change

Double click PARAM element at schematic and select 'New Property..'



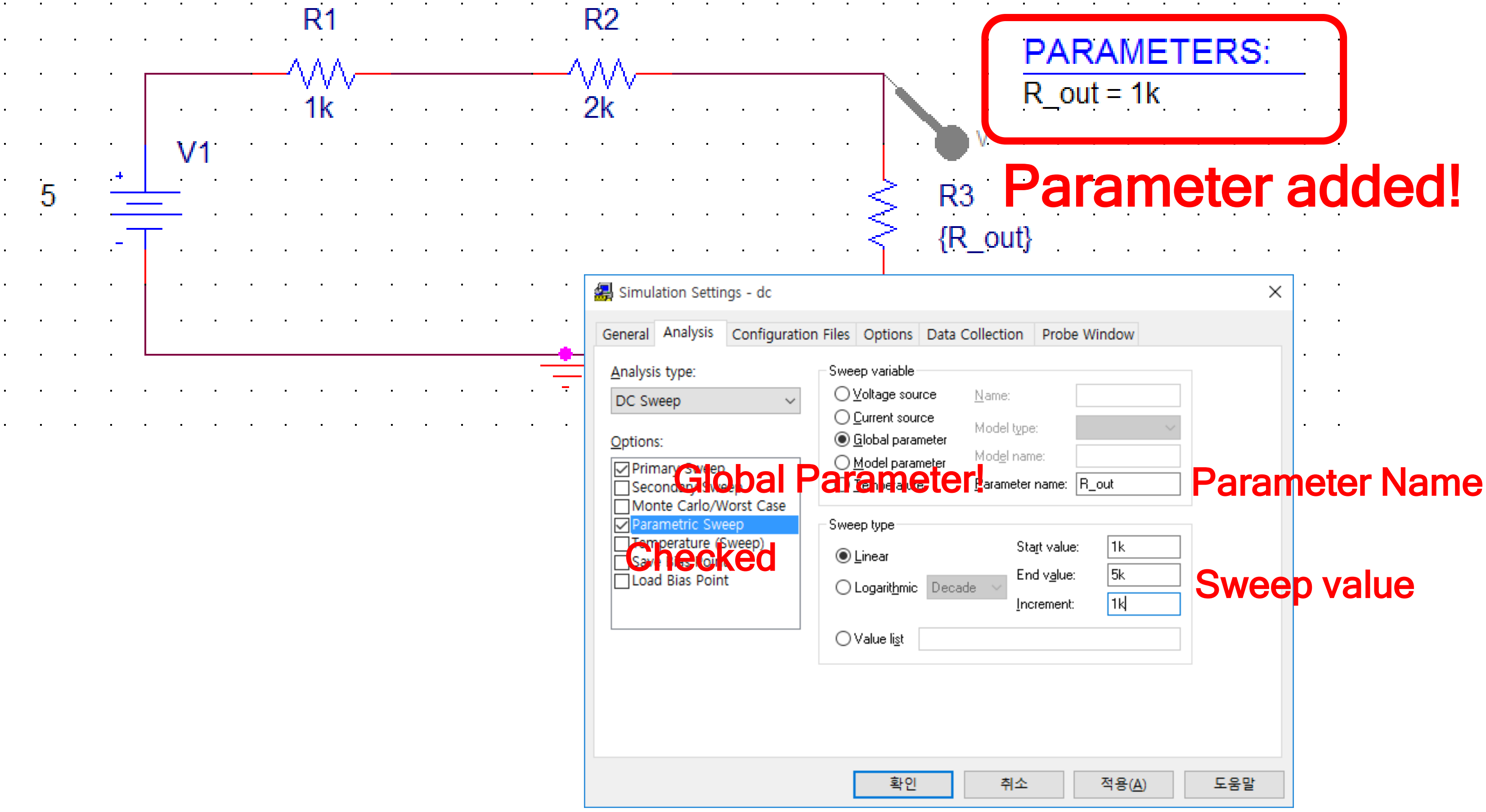
◆ Part.3 Parametric Simulation

2. Passive element(resistor, capacitor) change



◆ Part.3 Parametric Simulation

2. Passive element(resistor, capacitor) change



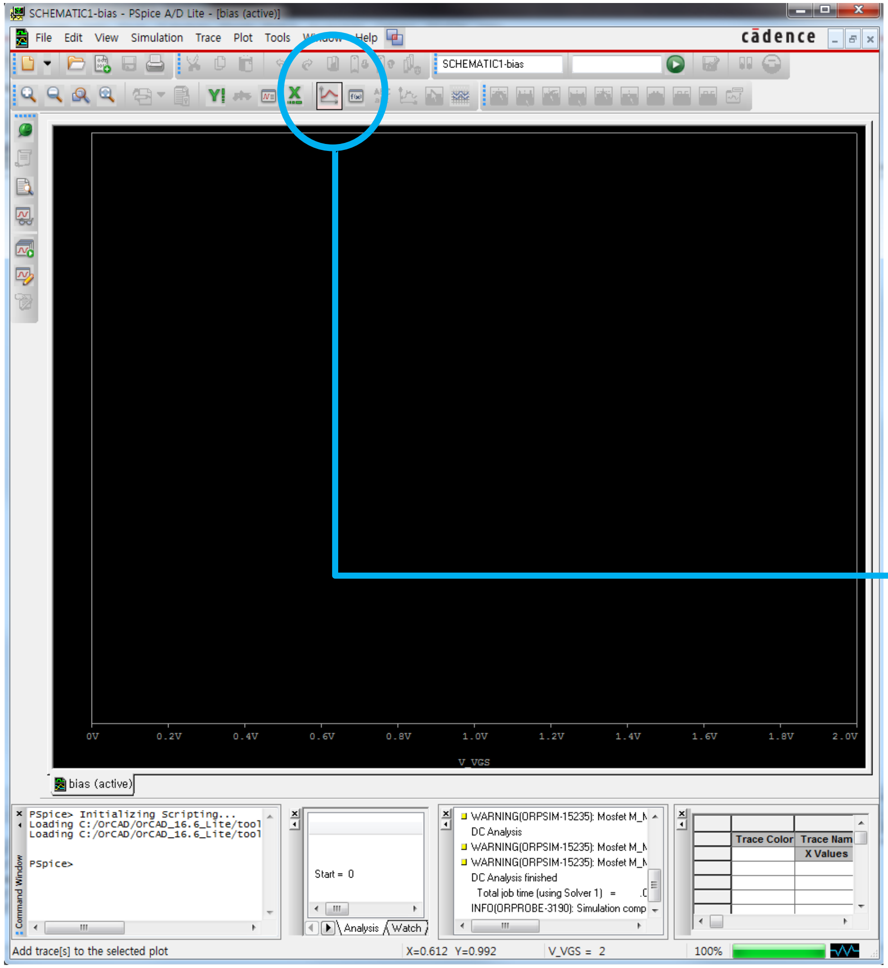
◆ Part.4 Function Plot

How can you plot derivative of this curve?

W.-Y. Choi

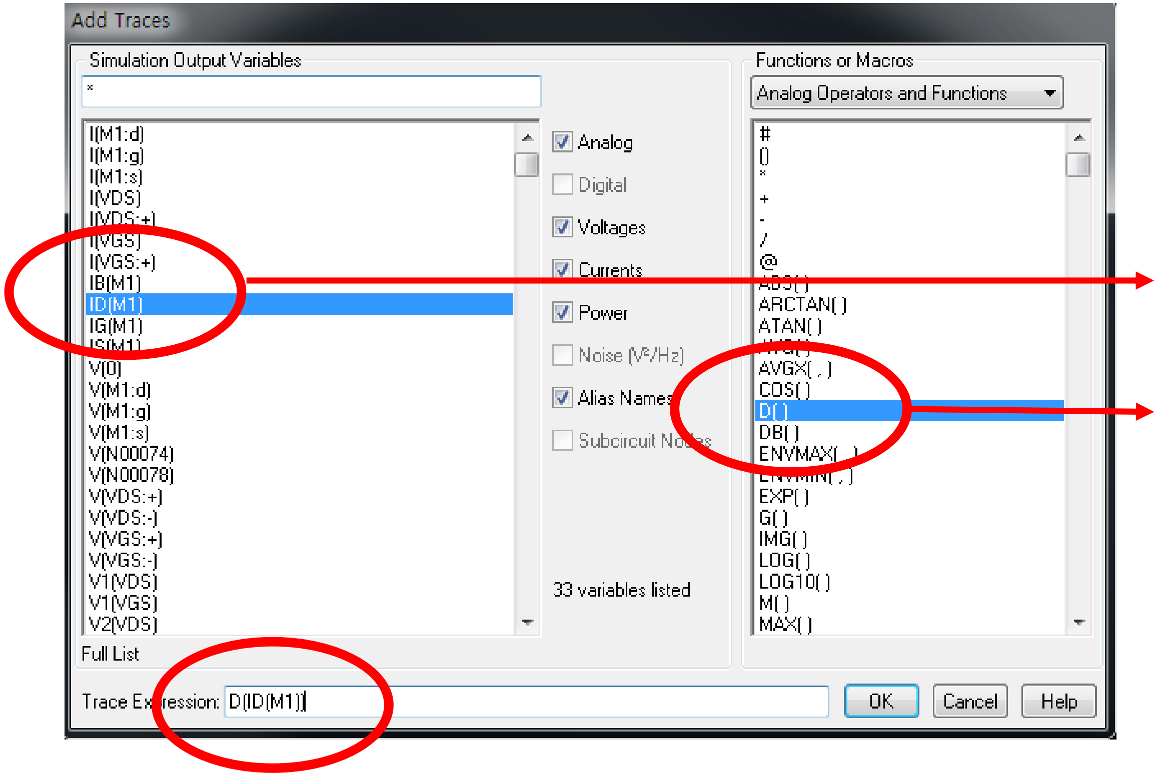
Lect. 4: PSpice Tutorial

◆ Part.4 Function Plot



Select 'Add Trace'

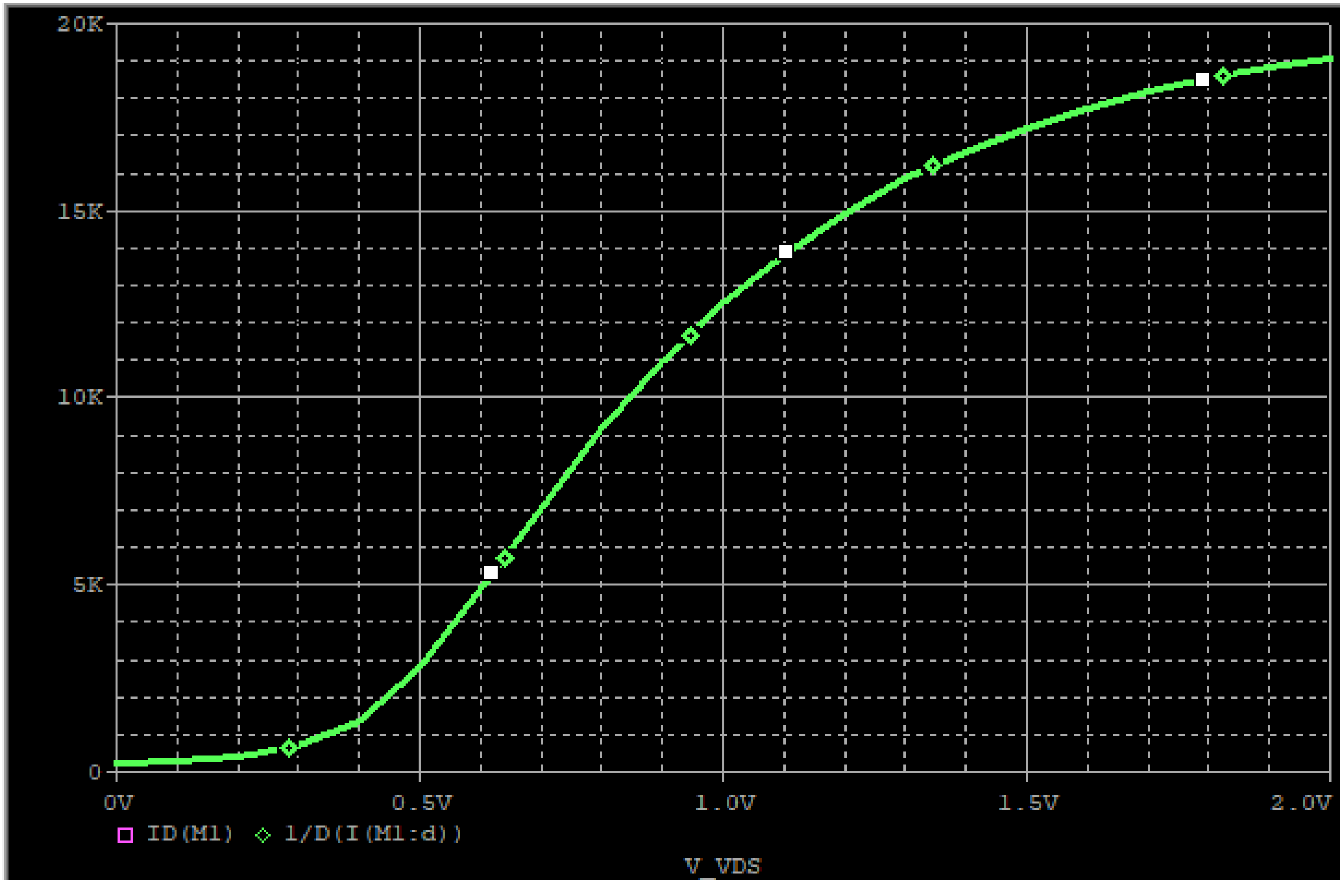
◆ Part.4 Function Plot



ID(M1) represents drain current of M1 MOSFET

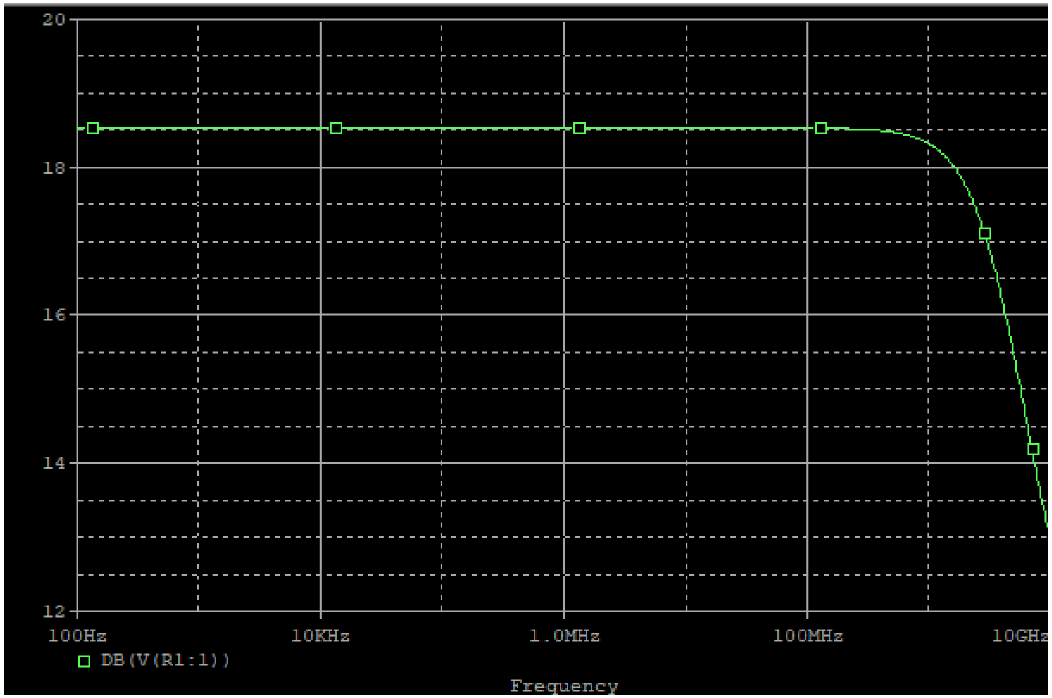
D(y) function calculates derivative of y in terms of x-axis' variable

◆ Part.4 Function Plot

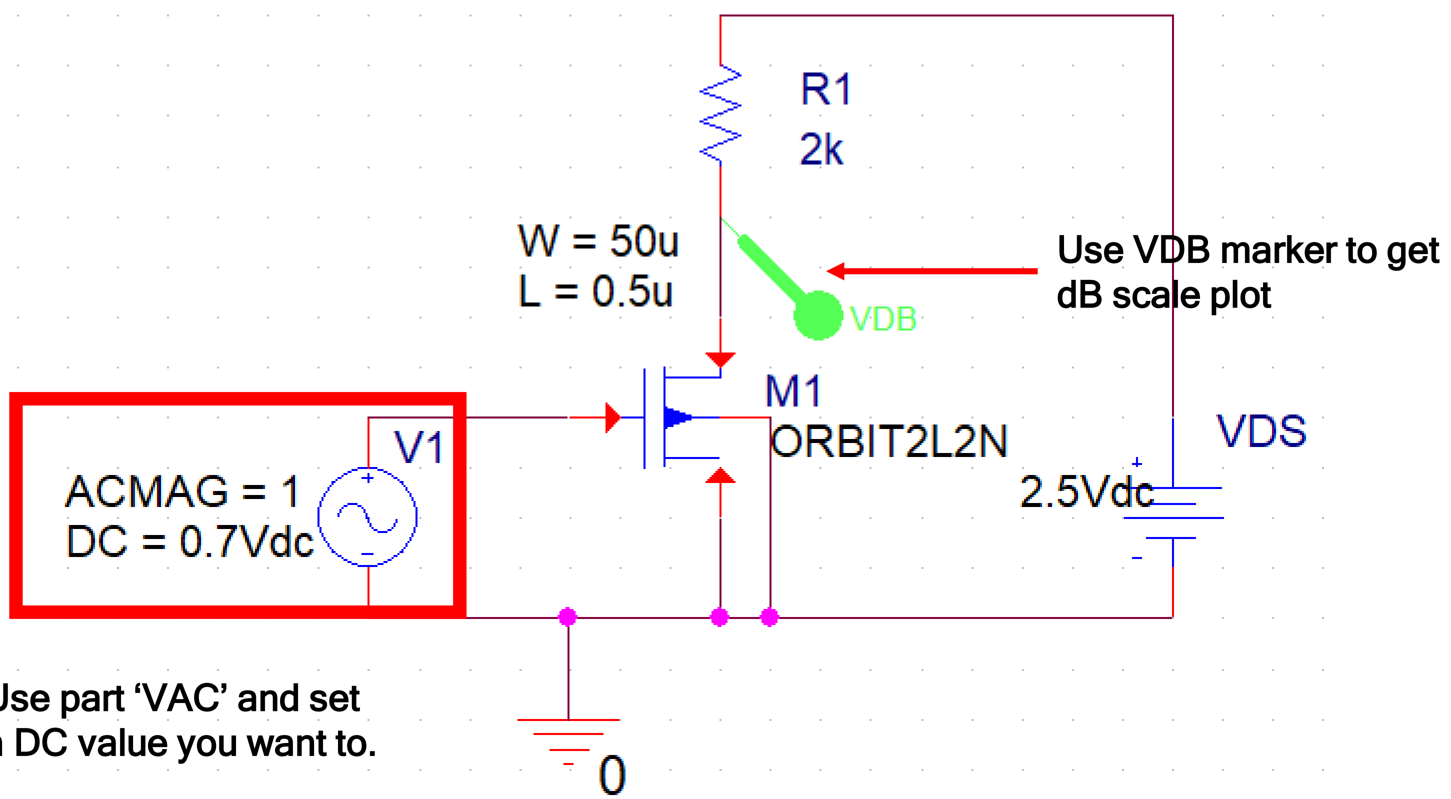


◆ Part.5 AC Simulation

AC sweep is used for seeing frequency response at a specific node .
(Don't confuse it; it is not used for time-domain simulation)



◆ Part.5 AC Simulation

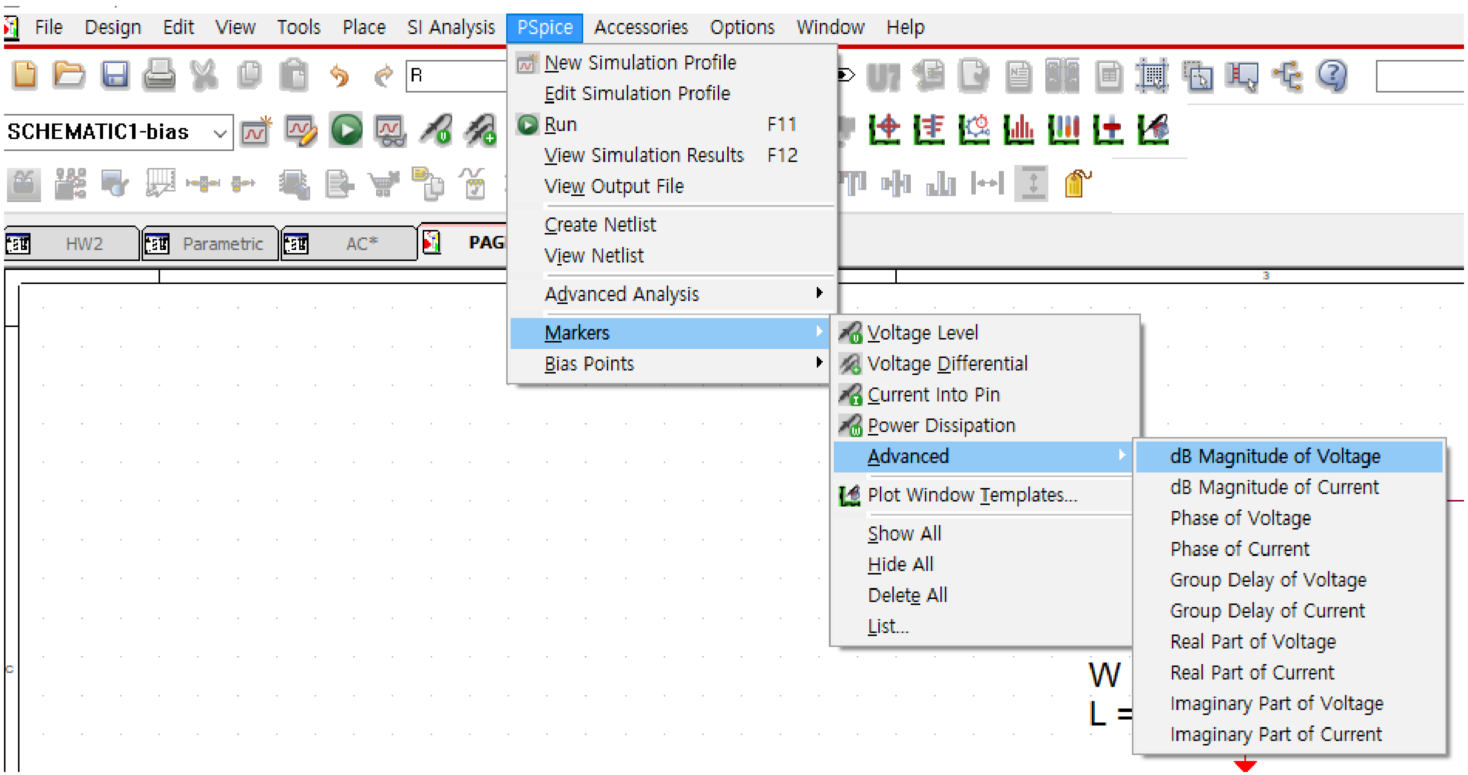


Use part ‘VAC’ and set
a DC value you want to.

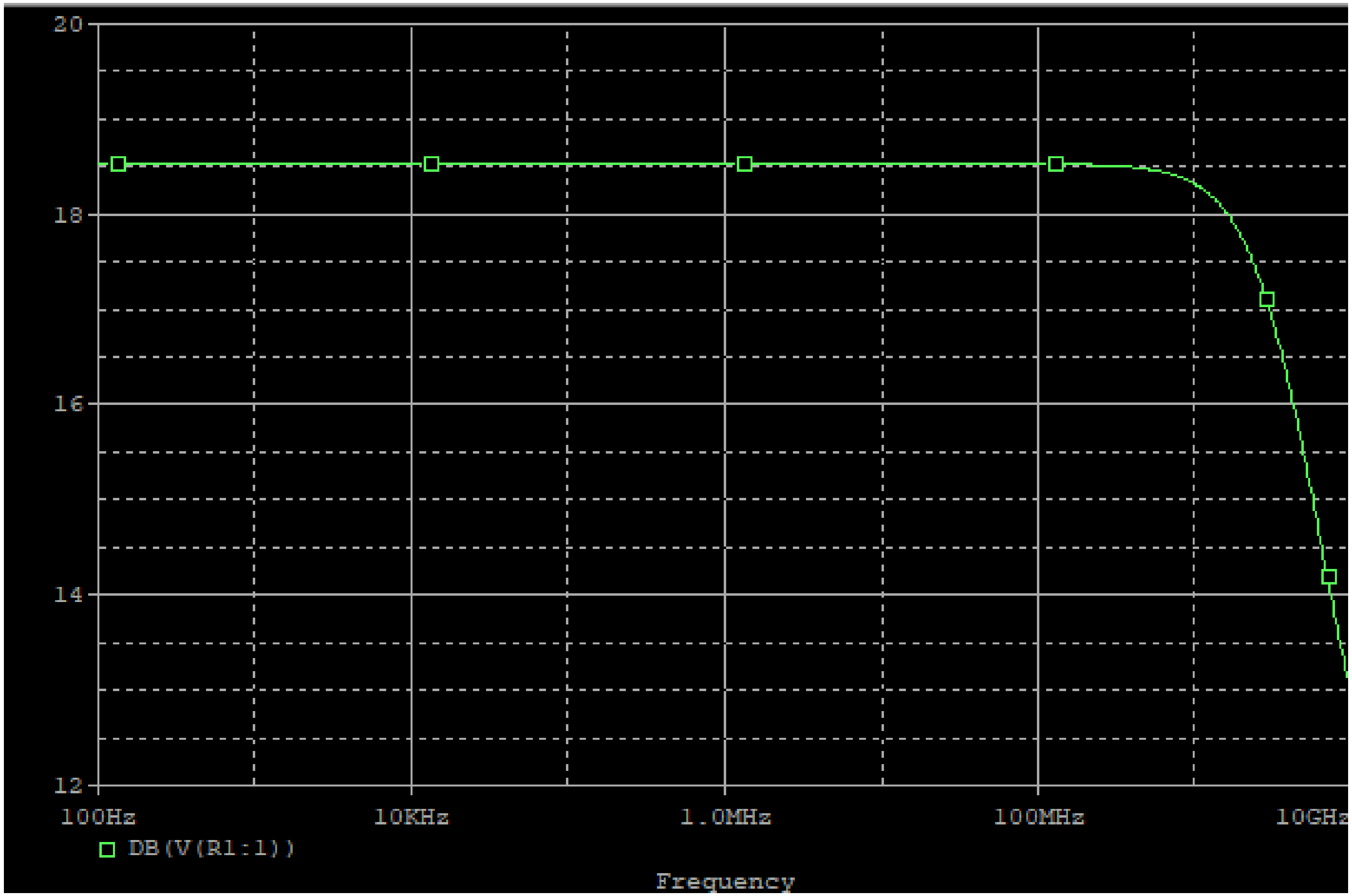
Make sure that ACMAG is set to 1.

◆ Part.5 AC Simulation

If you want to place VDB marker, just simulate it first (AC sweep).
and then, you can see dB marker is activated (If you don't simulate it before, it might be deactivated.)

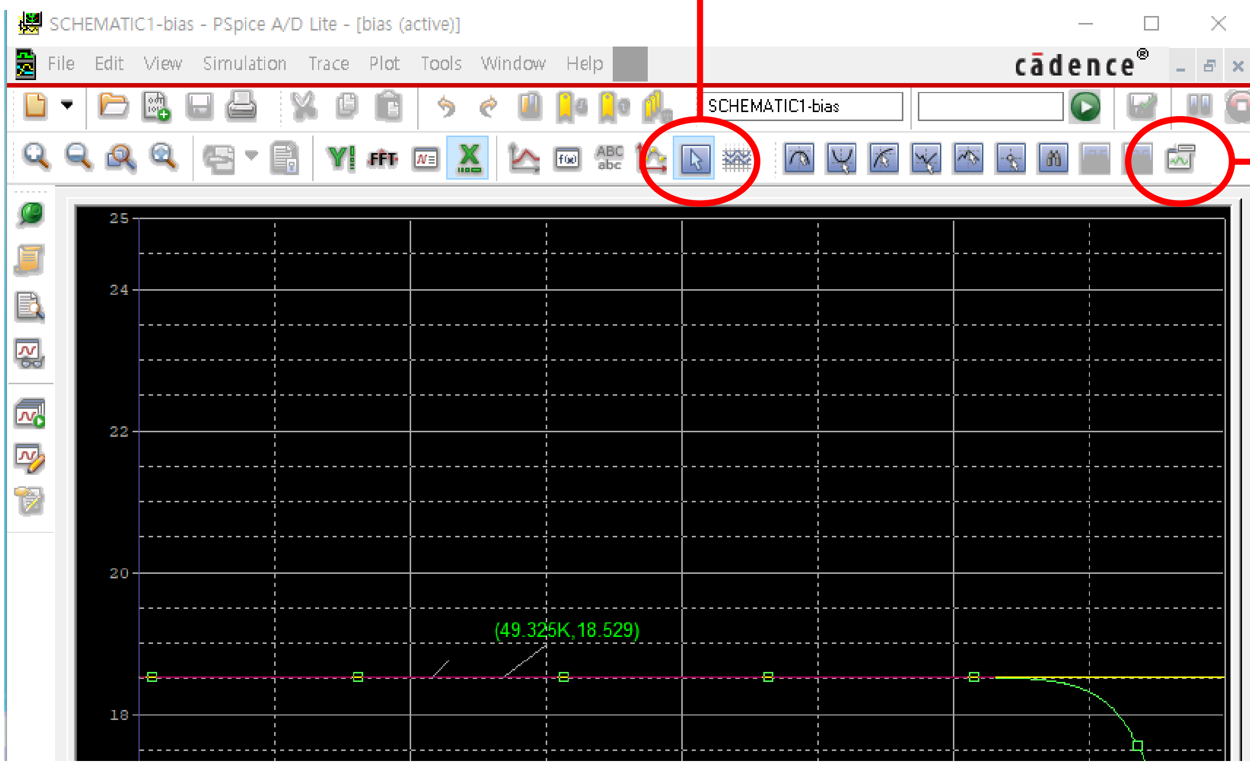


◆ Part.5 AC Simulation



◆ Part.5 AC Simulation

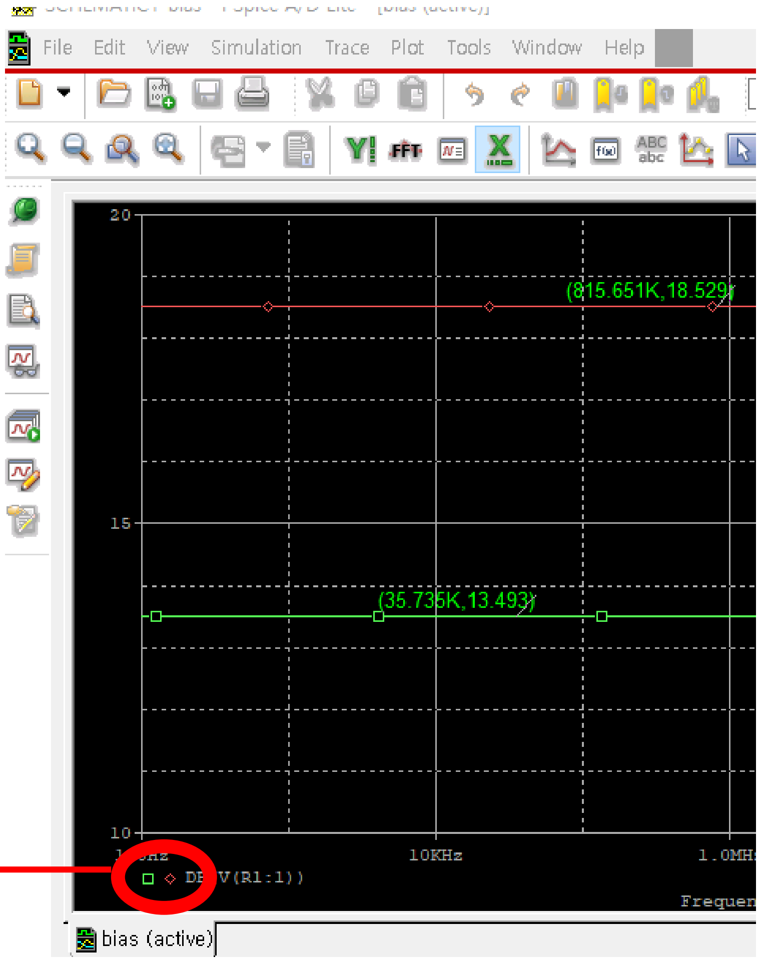
Use 'Toggle cursor' you want to see specific point (cursor can be dragged from the left)



Labeling value using 'Mark Label'

(x, y)

◆ Part.5 AC Simulation

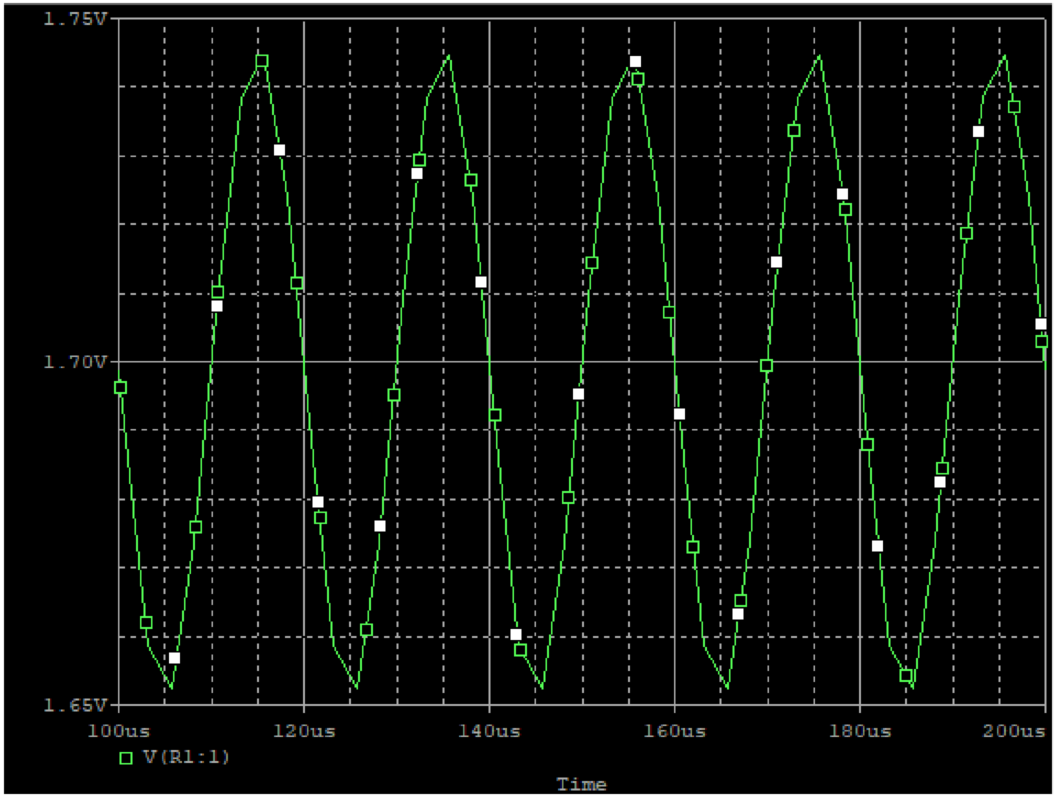


If you want to change cursor
when using parametric simulation,
Select the color you want to see and re-drag
the cursor from the left.

W.-Y. Choi

◆ Part.6 Transient Simulation

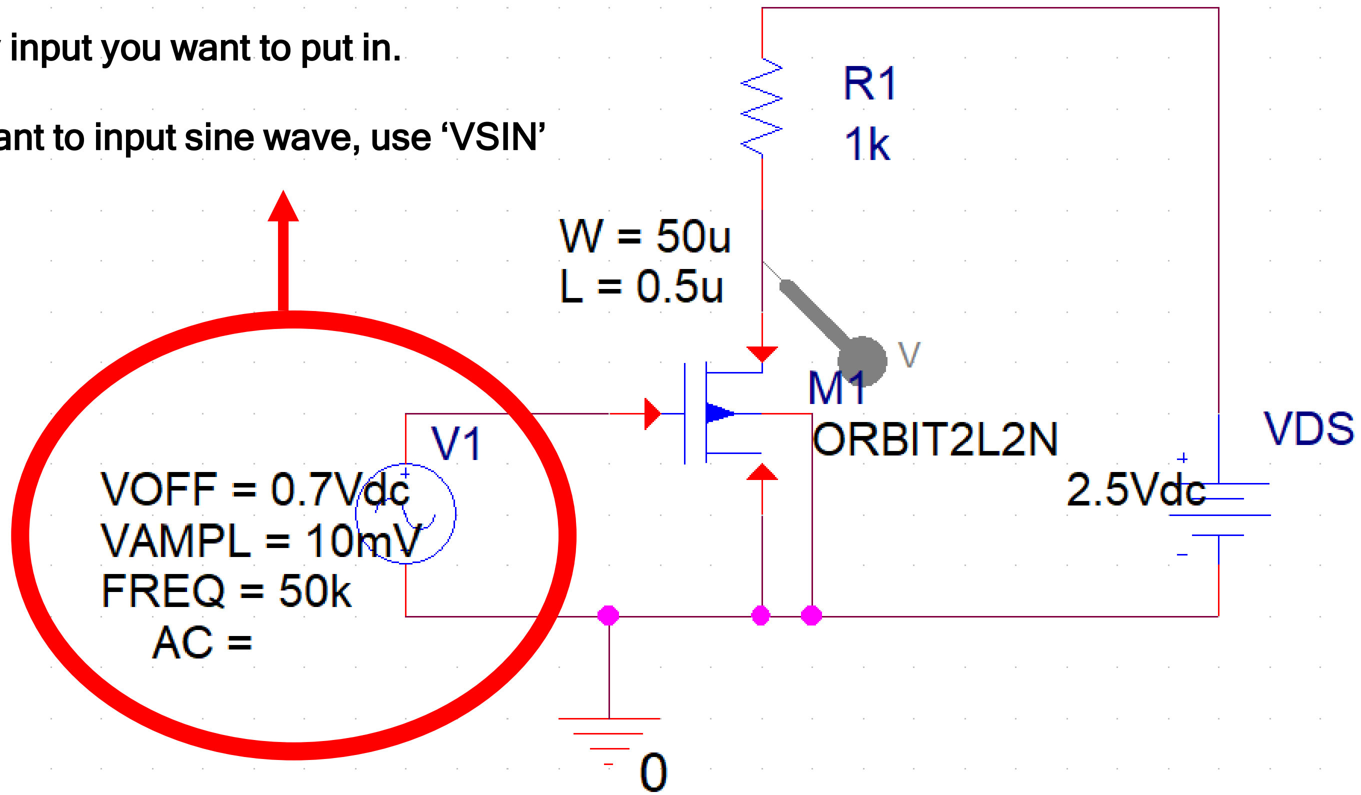
Transient simulation is used for time-domain responses at a specific node .



◆ Part.6 Transient Simulation

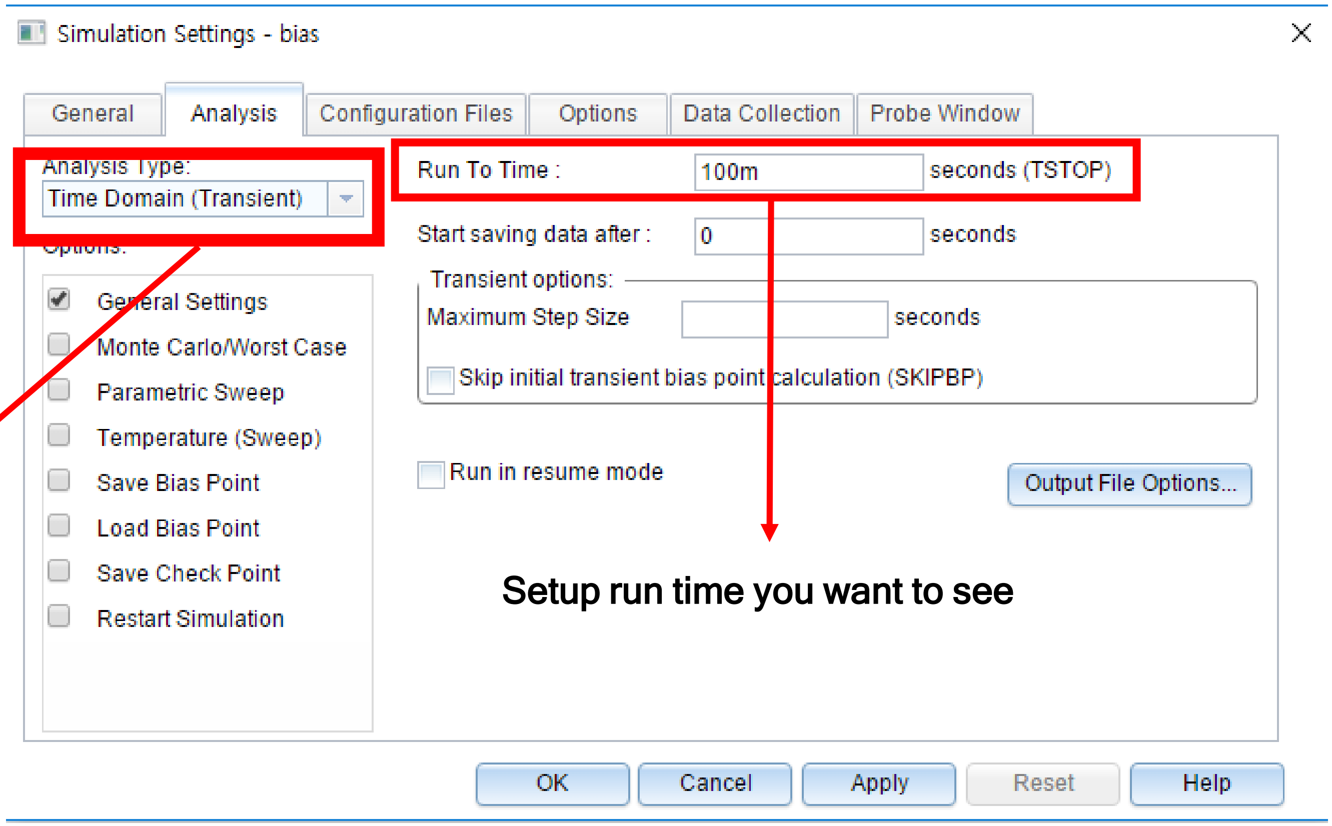
Use any input you want to put in.

If you want to input sine wave, use 'VSIN'



W.-Y. Choi

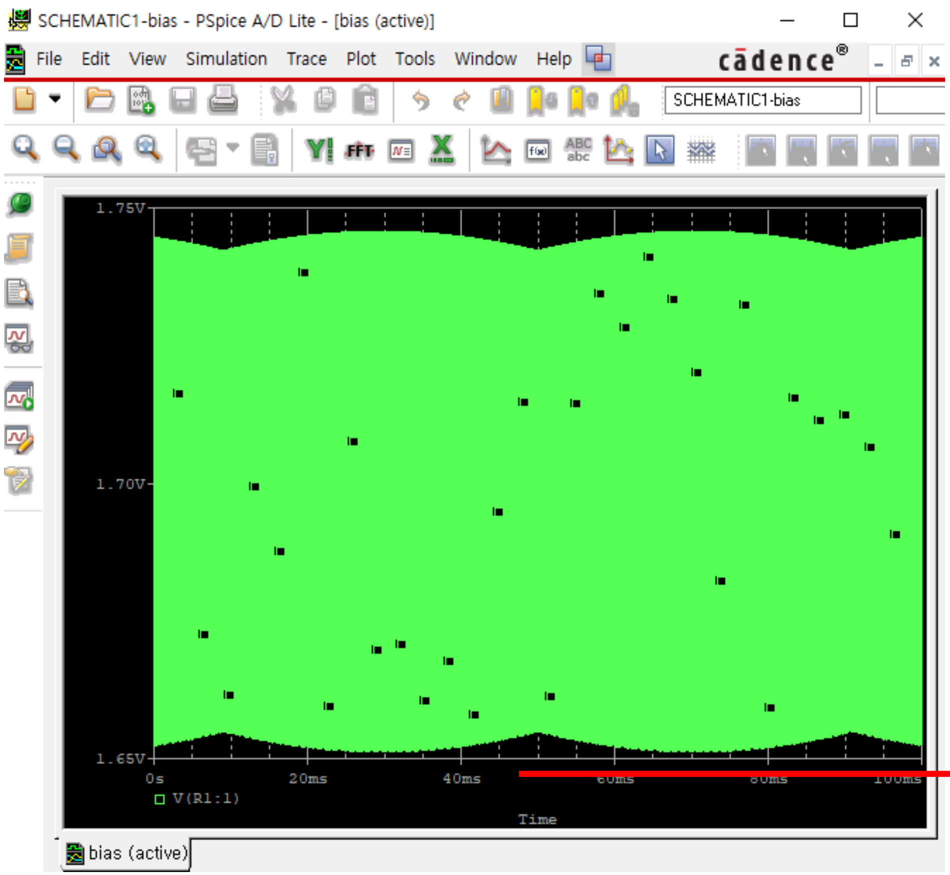
◆ Part.6 Transient Simulation



Select Time-domain

Setup run time you want to see

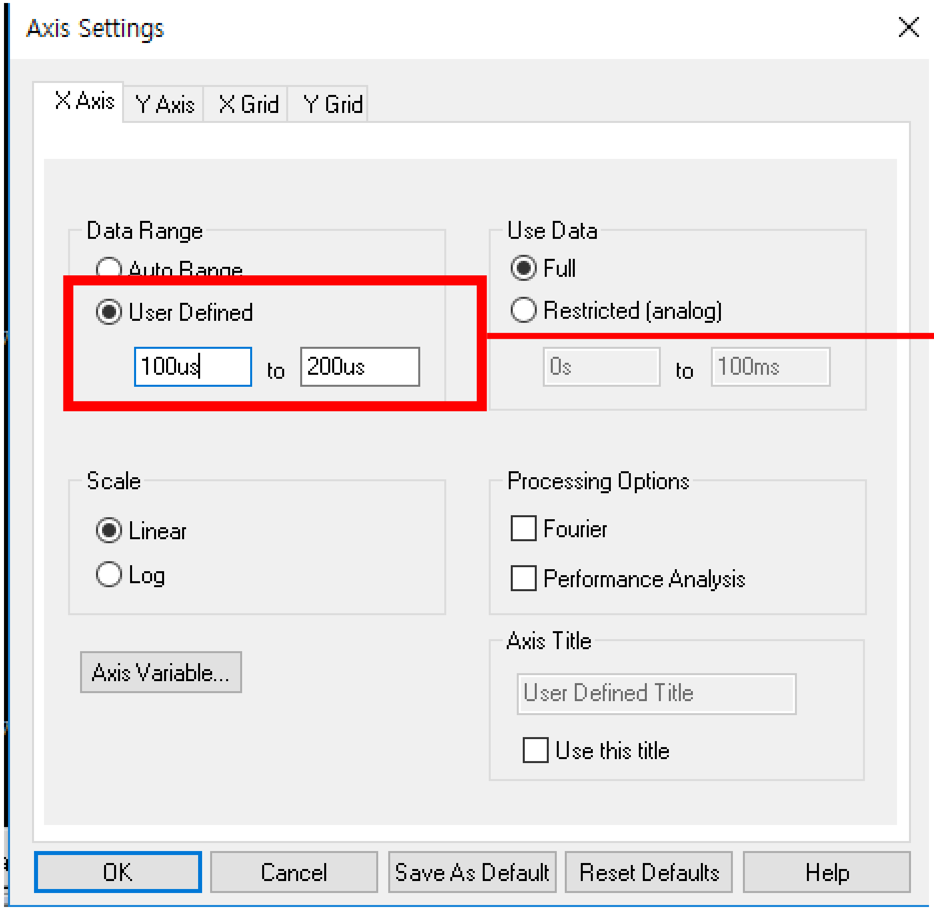
◆ Part.6 Transient Simulation



If you run too much time, signal can not be seen well.

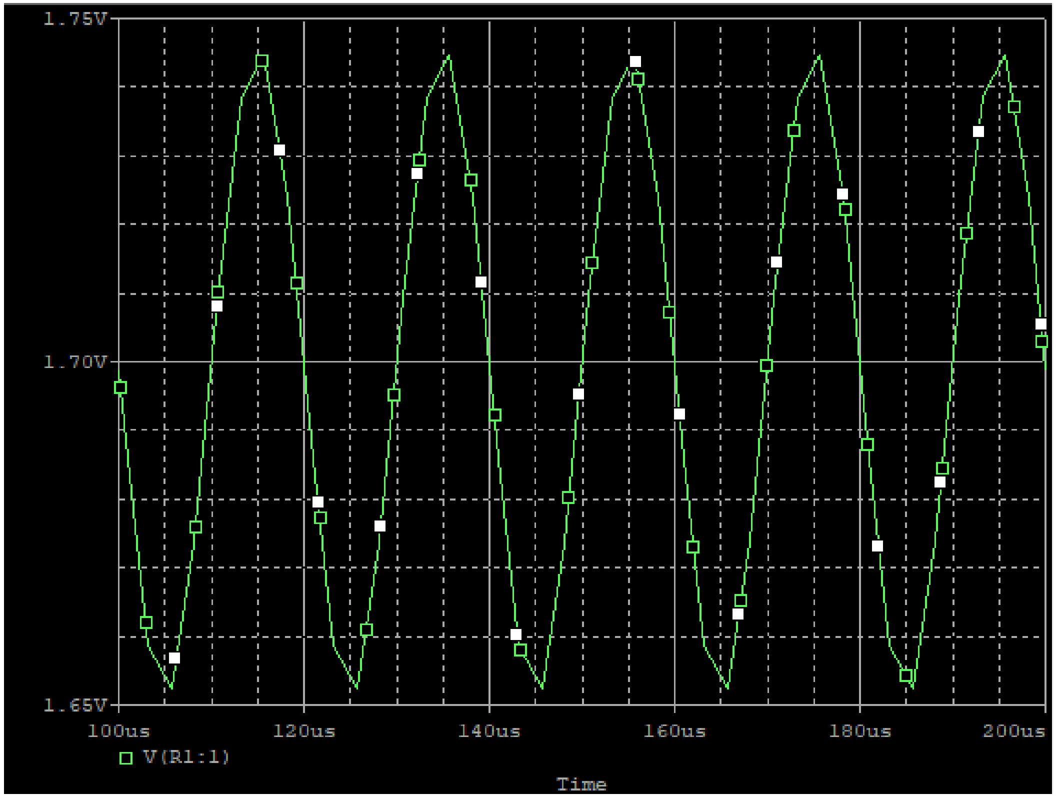
Double-click x-axis to set up time-range

◆ Part.6 Transient Simulation



Click user defined and set up the time range

◆ Part.6 Transient Simulation



Lect. 4: PSpice Tutorial

◆ Homework

For NMOS having W= 10 μm and L= 0.25 μm,

1) Determine V_{TH} by plotting $I_D - V_{GS}$ curve (Sweep V_{GS} from 0 V to 0.6 V at V_{DS} = 1.8 V)

2) Plot $I_D - V_{DS}$ curve with various V_{GS} values from 0.7 V to 1.6 V with increment of 0.3 V. For V_{DS} , sweep 0 V to 1.2 V.

3) Determine λ for the transistor at V_{GS} = 1.4 V. For this, plot $I_D - V_{DS}$ curve for V_{DS} ranging from 1.0 V to 2.0 V and determine the slope.

4) Determine the numerical value of $\mu_n C_{OX}$ for the transistor using the results obtained in 3)

5) Plot g_m for V_{DS} = 1.8 V for V_{GS} ranging from 0 V to 2.0 V. Compare your result with the equation given in the lecture note when V_{GS} is 1.4 V.