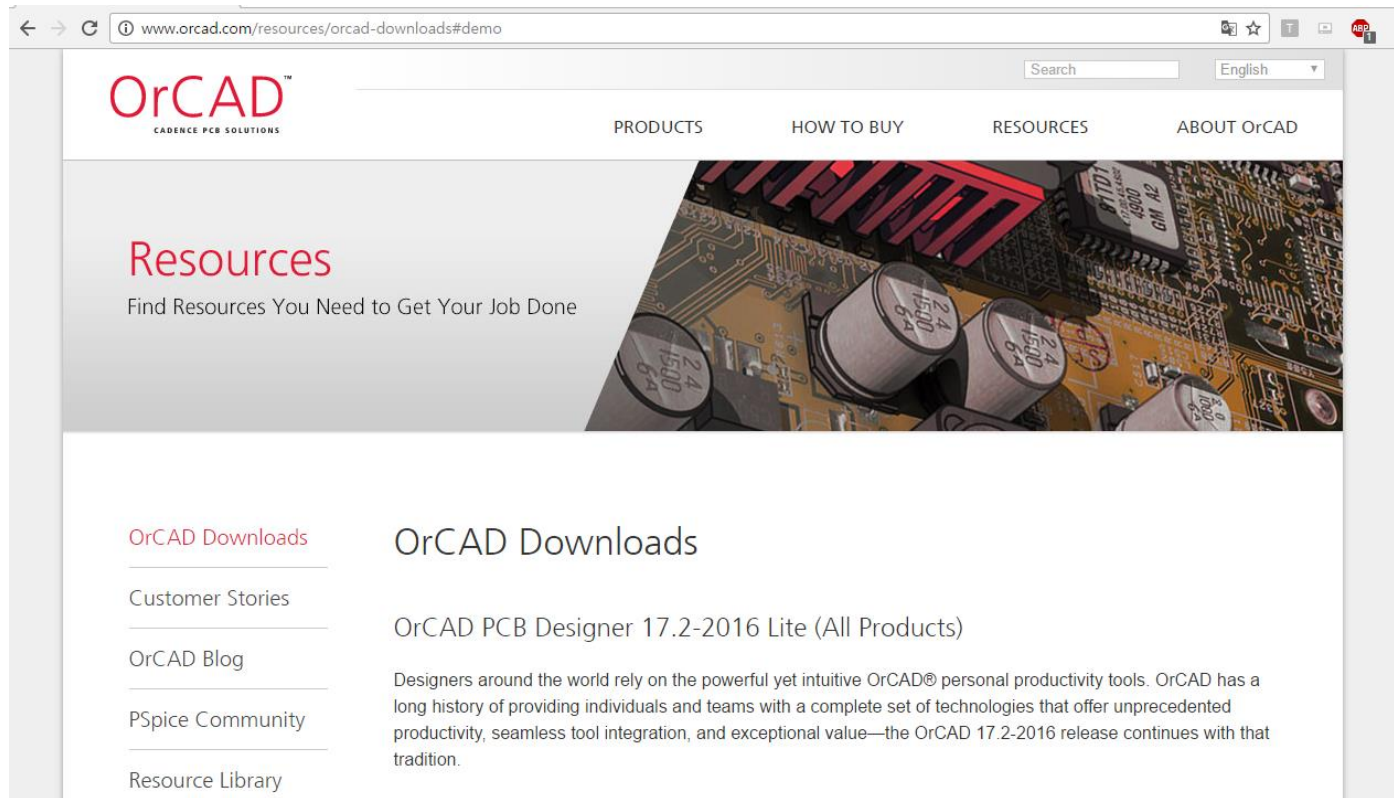


# Lect. 4: PSpice Tutorial

## ◆ OrCAD PSpice Installation

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



The screenshot shows a web browser window displaying the OrCAD website. The address bar shows the URL [www.orcad.com/resources/orcad-downloads#demo](http://www.orcad.com/resources/orcad-downloads#demo). The website header includes the OrCAD logo (OrCAD™ CADENCE PCB SOLUTIONS) and navigation links for PRODUCTS, HOW TO BUY, RESOURCES, and ABOUT OrCAD. A search bar and a language dropdown set to "English" are also visible. The main content area features a large image of a printed circuit board (PCB) with various components like capacitors and integrated circuits. Below the image, the word "Resources" is displayed in a large red font, followed by the subtitle "Find Resources You Need to Get Your Job Done". On the left side, there is a vertical menu with links: OrCAD Downloads, Customer Stories, OrCAD Blog, PSpice Community, and Resource Library. The main content area under "Resources" is titled "OrCAD Downloads" and features a link for "OrCAD PCB Designer 17.2-2016 Lite (All Products)". Below this link, a paragraph of text describes the OrCAD software's history and value.

OrCAD™  
CADENCE PCB SOLUTIONS

PRODUCTS HOW TO BUY RESOURCES ABOUT OrCAD

Resources  
Find Resources You Need to Get Your Job Done

OrCAD Downloads

Customer Stories

OrCAD Blog

PSpice Community

Resource Library

OrCAD Downloads

OrCAD PCB Designer 17.2-2016 Lite (All Products)

Designers around the world rely on the powerful yet intuitive OrCAD® personal productivity tools. OrCAD has a long history of providing individuals and teams with a complete set of technologies that offer unprecedented productivity, seamless tool integration, and exceptional value—the OrCAD 17.2-2016 release continues with that tradition.

# Lect. 4: PSpice Tutorial

## ◆ 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)

The 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](#)

# Lect. 4: PSpice Tutorial

## ◆ 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 \*

Address \*

Country \*

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

# Lect. 4: PSpice Tutorial

## ◆ 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
<b>setup</b>	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

# Lect. 4: PSpice Tutorial

## ◆ OrCAD PSPICE Installation

1. Welcome to the InstallShield Wizard for OrCAD 16.6 Lite. We advise you to turn off any virus detection programs, firewall programs, and spyware programs while running this installation. Antivirus programs, Firewall programs, and Spyware programs can cause this installation to fail.

2. License Agreement. Please read the following license agreement carefully. CADENCE DESIGN SYSTEMS, INC. SOFTWARE LICENSE AND MAINTENANCE AGREEMENT. THIS SOFTWARE LICENSE AND MAINTENANCE AGREEMENT ("AGREEMENT") IS A LEGAL DOCUMENT BETWEEN YOU AND CADENCE DESIGN SYSTEMS, INC. ("CADENCE"). PLEASE READ THIS AGREEMENT CAREFULLY BEFORE INSTALLING YOUR CADENCE SOFTWARE ("SOFTWARE"). BY USING THE SOFTWARE, YOU (EITHER AN INDIVIDUAL OR A BUSINESS ENTITY) AGREE TO BE BOUND BY THE TERMS OF THIS AGREEMENT. IF YOU DO NOT WANT TO BE BOUND BY THE TERMS OF THIS AGREEMENT, CADENCE IS UNWILLING TO LICENSE THE SOFTWARE TO YOU, IN WHICH EVENT YOU MUST PROMPTLY RETURN THE SOFTWARE AND ALL ACCOMPANYING ITEMS (INCLUDING MANUALS, BINDERS, CD-ROMS, SOFTWARE, AND ANY OTHER PRINTED MATERIALS) TO CADENCE.

3. Setup Type. Select the setup type to install. Install this application for:  Only for me (Recommended)  Anyone who uses this computer (all users)

4. Start Copying Files. Review settings before copying files. Setup has enough information to start copying the program 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: OrCAD\_Capture\_CIS\_Lite, PSpice\_Lite; Product destination path: C:\OrCAD\OrCAD\_16.6\_Lite; Working directory: C:\SPB\_Data.

5. Ready to Install the Program. The wizard is ready to begin installation. Click Install to begin the installation. If you want to review or change any of your installation settings, click Back. Click Cancel to exit the wizard.

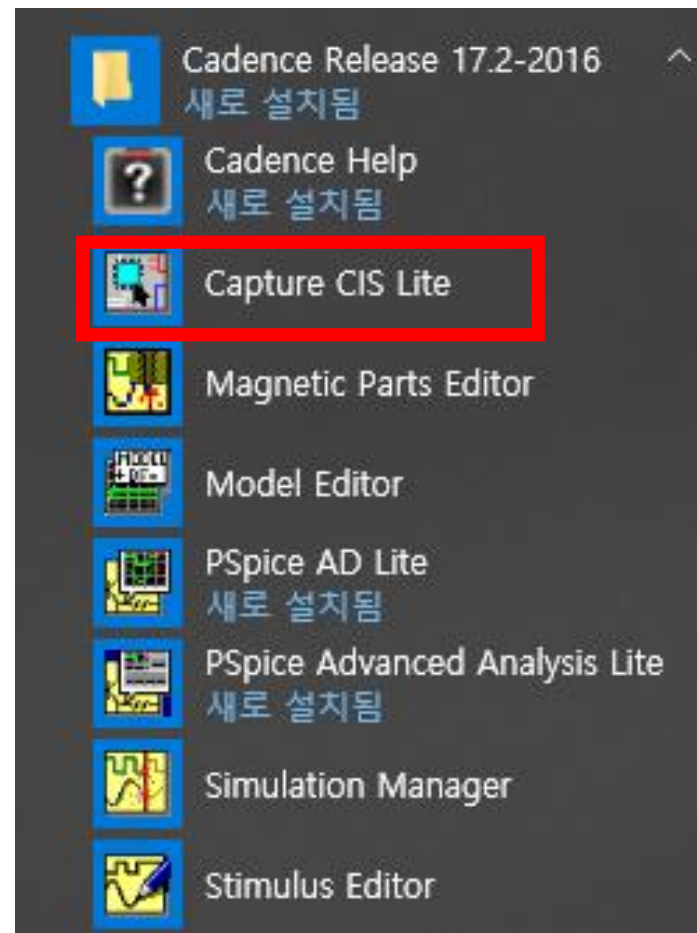
6. Setup Complete. Press F1 for help. Setup has finished installing files on your computer. Select the options you want below, then click Finish, or click Finish to complete the installation without viewing the Product Notes. View Product Notes; Open Cadence Web Page; Post installation, if you want to access other installed SPB releases, use the Cadence SPB Switch Release from the Start Menu to set the required release.

The path name should be in English!

# Lect. 4: PSpice Tutorial

## ◆ OrCAD PSPICE Installation

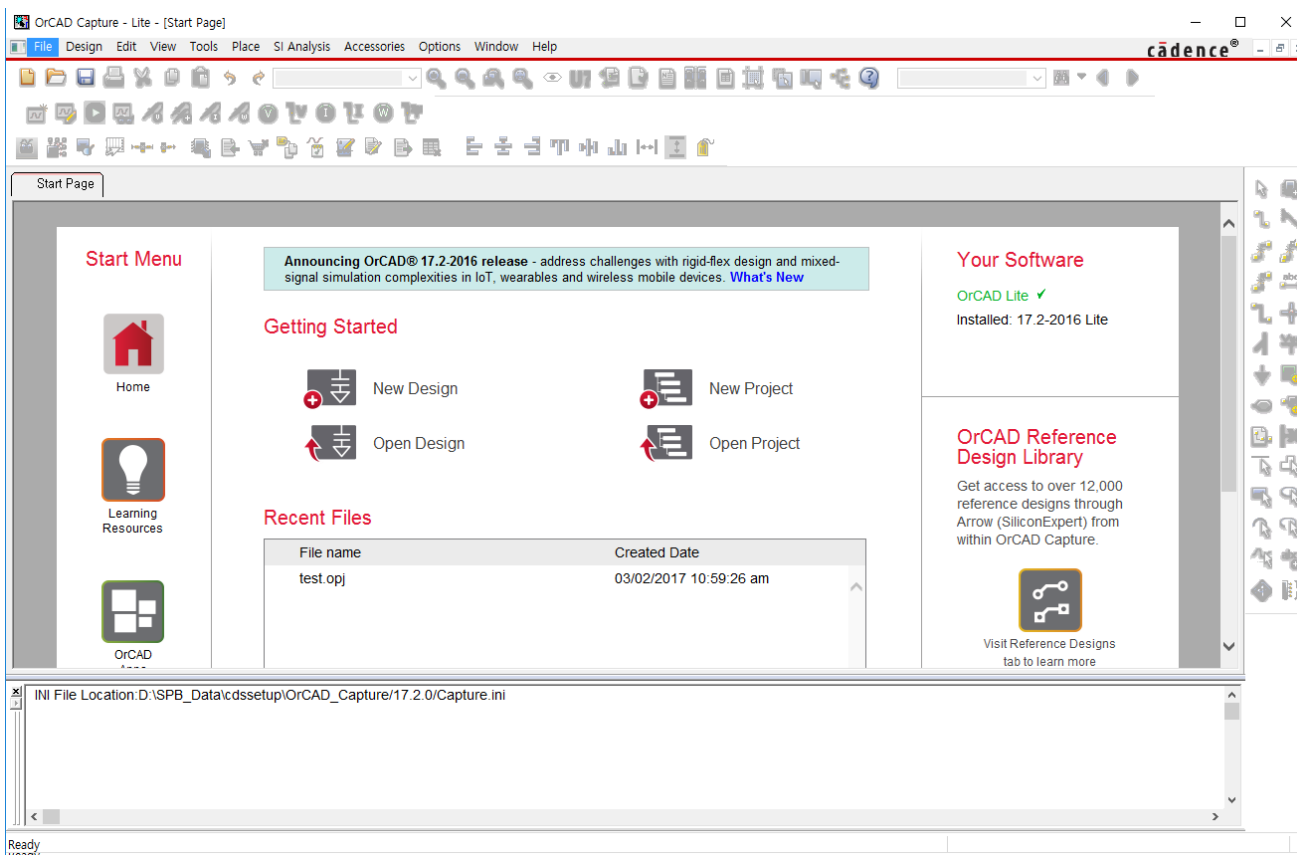
Start OrCAD Capture CIS Lite



# Lect. 4: PSpice Tutorial

## ◆ OrCAD PSpice Installation

File -> New Project to open schematics

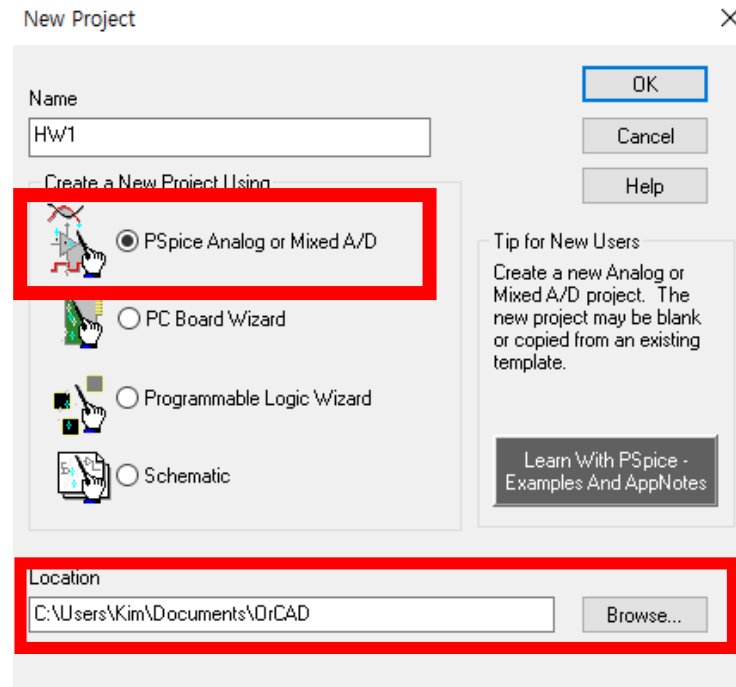


# Lect. 4: PSpice Tutorial

## ◆ OrCAD PSpice Installation

Choose PSpice Analog or Mixed A/D project

File path name should be in English!





# Lect. 4: PSpice Tutorial

## ◆ OrCAD PSpice Installation

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

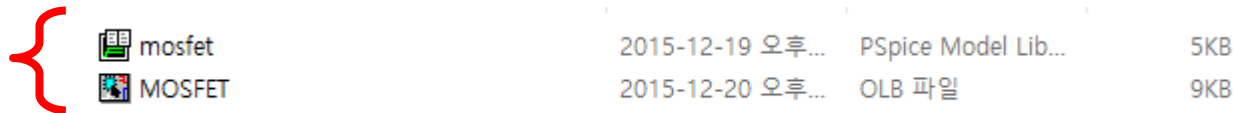
The screenshot shows the OrCAD Capture CIS - Lite interface. On the left, the 'Create PSpice Project' dialog is open, with 'Create based upon an existing project' selected and 'empty.opj' entered in the text field. The main window displays the project hierarchy for 'tutorial.dsn'. The hierarchy includes 'Design Resources', 'Schematic1', 'PAGE1', 'Design Cache', 'Library', and 'PSpice Resources'. The 'Library' folder contains several basic libraries: '#worcad#worcad\_16.6\_lite#tools#capture#library#pspice#analog.olb', '#worcad#worcad\_16.6\_lite#tools#capture#library#pspice#breakout.olb', '#worcad#worcad\_16.6\_lite#tools#capture#library#pspice#source.olb', '#worcad#worcad\_16.6\_lite#tools#capture#library#pspice#sourcstm.olb', and '#worcad#worcad\_16.6\_lite#tools#capture#library#pspice#special.olb'. A red bracket groups these libraries with the text 'Basic libraries'. A red arrow points to 'PAGE1' with the text 'Schematics included in your project (Double click PAGE1)'. The status bar at the bottom shows the file location: 'INI File Location: C:\SPB\_Data\cdssetup\OrCAD\_Capture\16.6.0\Capture.ini'.

# Lect. 4: PSpice Tutorial

## ◆ MOSFET Library Setup

Download 'MOSFET\_OrCAD.zip' file which is uploaded in YSCEC.

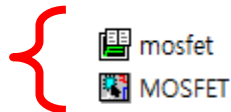
Check mosfet.lib & MOSFET.olb is in your unzipped folder.



# Lect. 4: PSpice Tutorial

## ◆ MOSFET Library Setup

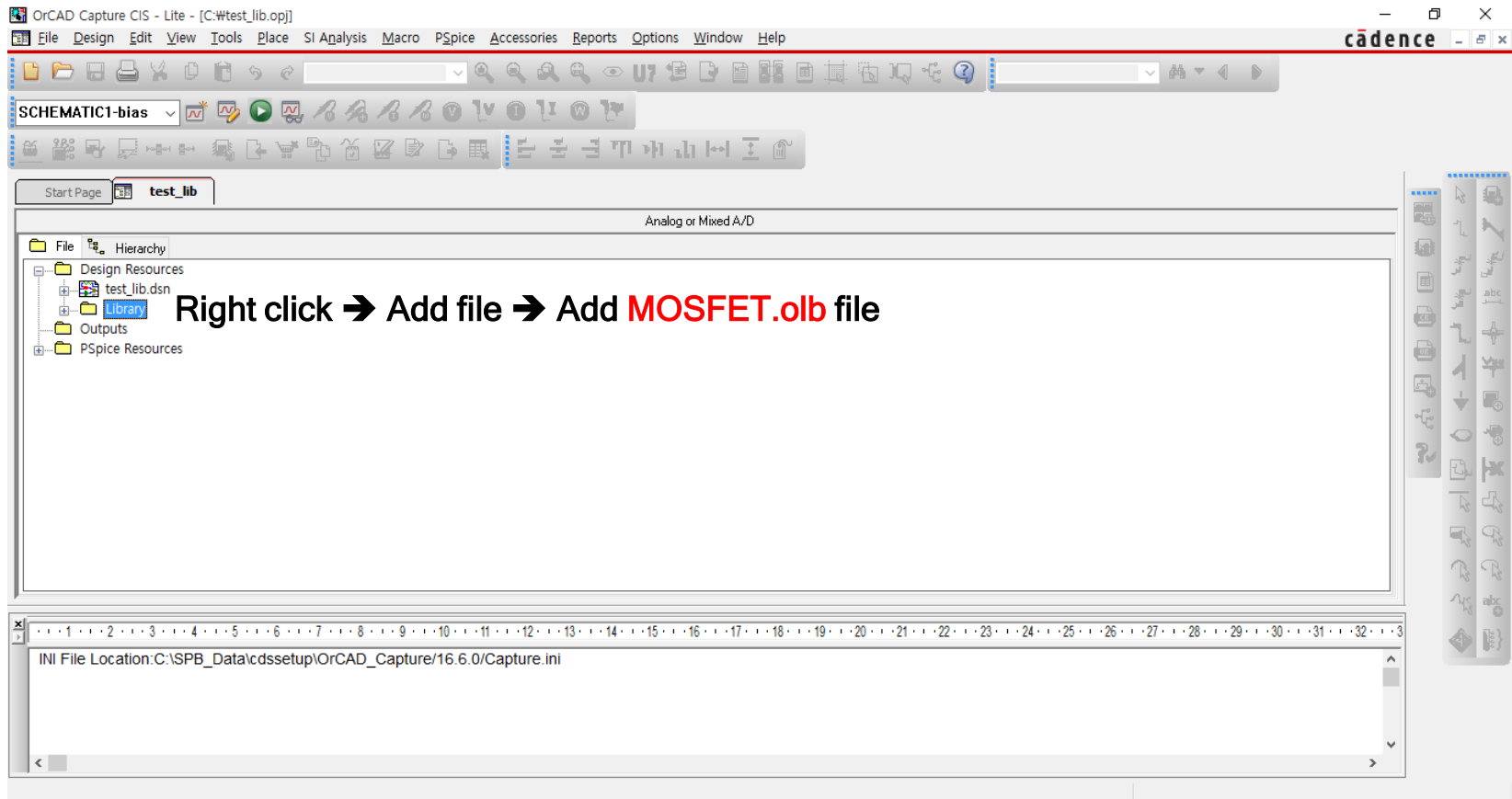
1. Download 'MOSFET\_OrCAD.zip' file uploaded in YSCEC.
2. Check mosfet.lib & MOSFET.olb is in your unzipped folder.
3. File path of this library folder must not have the Korean. (Move this folder to 'C drive' or 'My Document')



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

# Lect. 4: PSpice Tutorial

## ◆ MOSFET Library Setup



# Lect. 4: PSpice Tutorial

## ◆ MOSFET Library Setup

OrCAD Capture CIS - Lite - [C:\test\_lib.opj]

File Design Edit View Tools Place SI Analysis Macro PSpice Accessories Reports Options Window Help

SCHEMATIC1-bias

Start Page test\_lib

Analogue or Mixed A/D

File Hierarchy

- Design Resources
  - test\_lib.dsn
    - Library
- Outputs
- PSpice Resources

Right click → Add file → All file types → Add **mosfet.lib** file

→ Select file type as schematic library

Select File Type

The file '.\users\mk\downloads\mosfet\mosfet.lib' has no type associated with it.  
Please select a type from the list below:

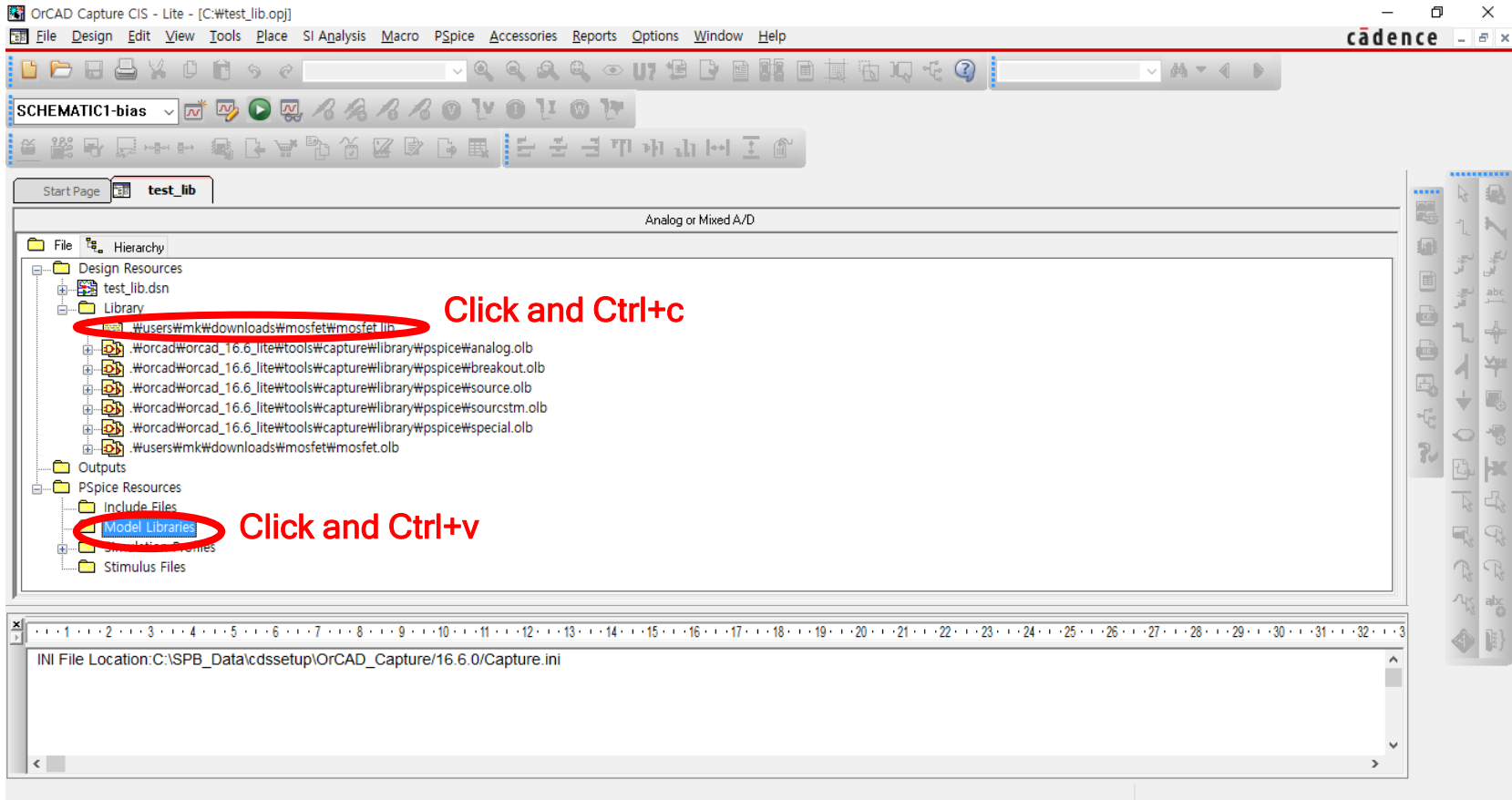
- PSpice Profile
- Report
- Schematic Design
- Schematic Library**
- Simulate Stimulus
- Standard Delay File
- Unknown
- Verilog Netlist

OK

INI File Location: C:\SPB\_Data\cdssetup\OrCAD\_Capture\16.6.0\Capture.ini

# Lect. 4: PSpice Tutorial

## ◆ MOSFET Library Setup



# Lect. 4: PSpice Tutorial

## ◆ MOSFET Library Setup

Drain

Gate

Source

M1 Bulk(Body)

ORBIT2L2N

NMOS

Source

Gate

Drain

M2 Bulk(Body)

ORBIT2L2P

PMOS

Place Part

Part

ORBIT2L2P

Part List:

ORBIT2L2N

ORBIT2L2P

Libraries:

ANALOG

BREAKOUT

Design Cache

MOSFET

SOURCE

SOURCESTM

Packaging

Parts per Pkg: 1

Part:

Type: Homogeneous

Normal Convert

+ Search for Part

INI File Location: C:\SPB\_Data\cdssetup\OrCAD\_Capture\16.6.0\Capture.ini

Double click → change W & L of MOSFETs

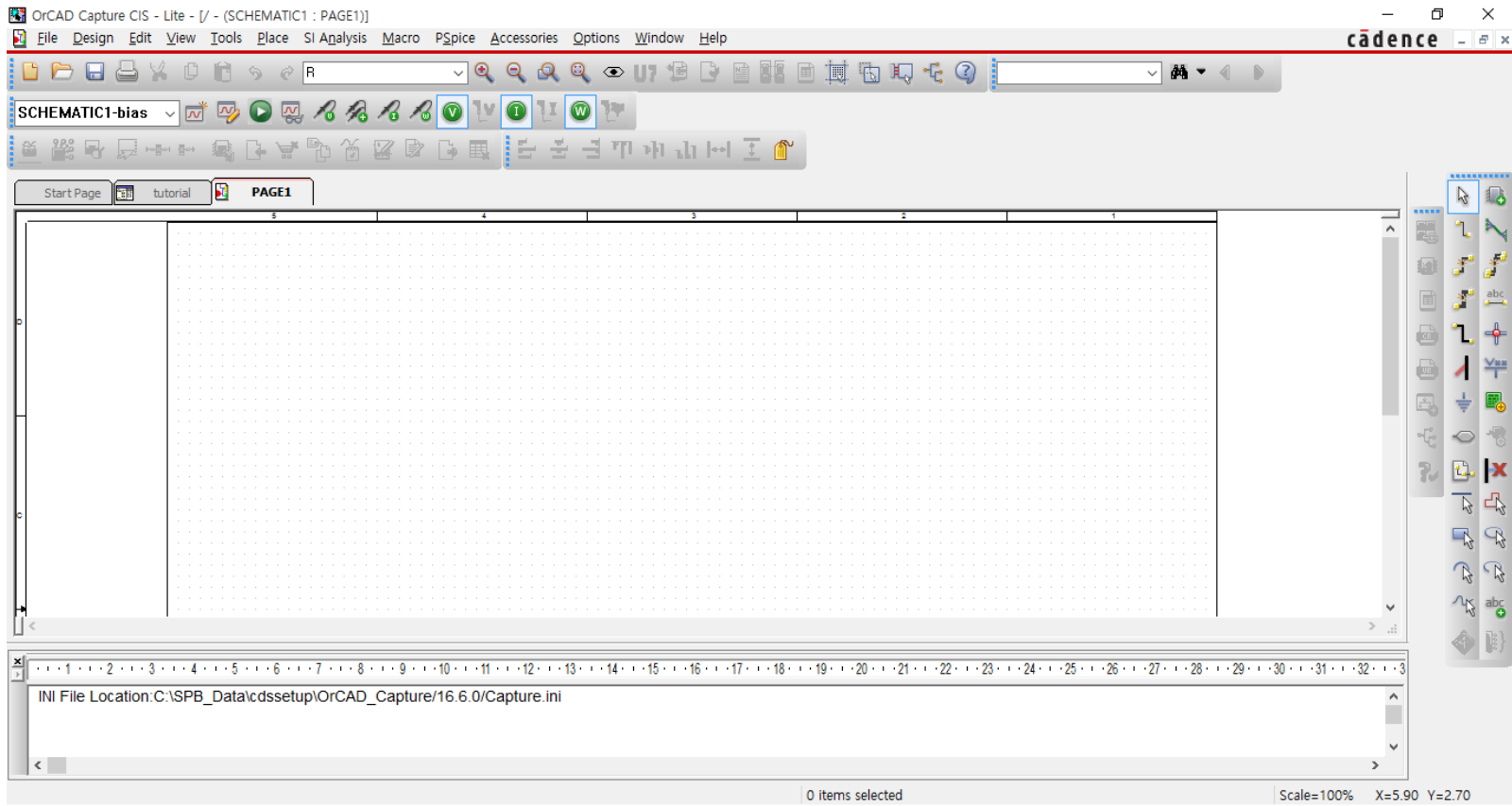
NMOS body connected to Ground

PMOS body connected to VDD

# Lect. 4: PSpice Tutorial

## ◆ PSPICE Basics

Empty schematic window is popped-up





# Lect. 4: PSpice Tutorial

## ◆ PSPICE Basics

Zoom Areas (Zoom in, out, fit)



Simulations

(setting, edit, run, results)



Markers

(current, voltage, diff. voltage, power)

Wire

Place part

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

# Lect. 4: PSpice Tutorial

## ◆ PSPICE Basics

### Key shortcuts

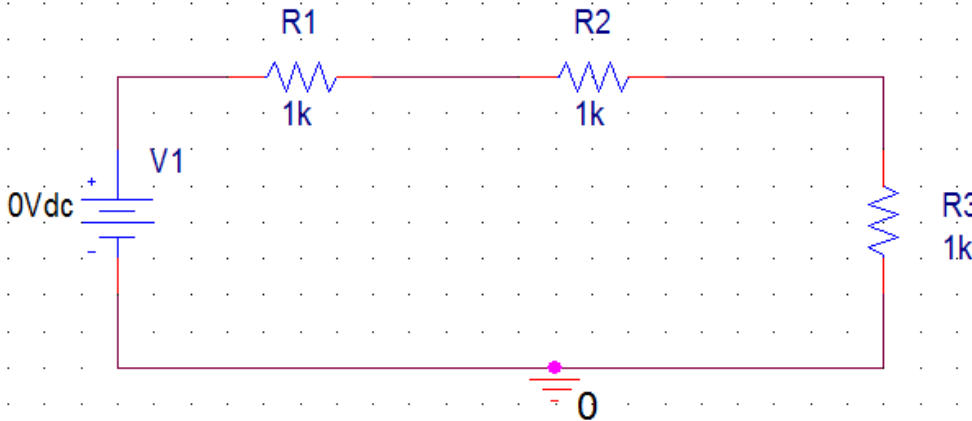
Description	Shortcut
Rotate	R
Mirror Horizontal	H
Mirror Vertical	V
Copy	Ctrl+C
Paste	Ctrl+V

# Lect. 4: PSpice Tutorial

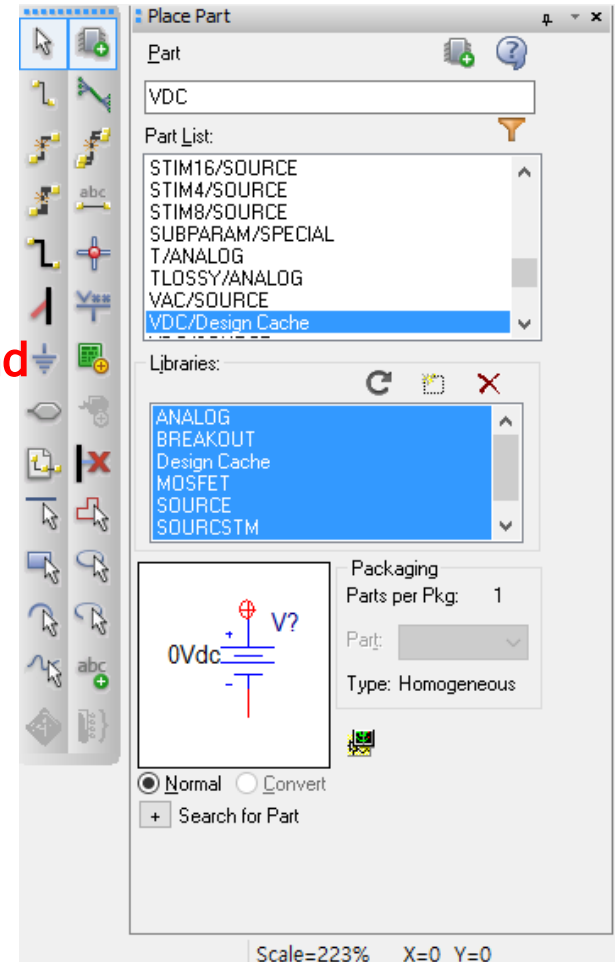
## ◆ PSPICE Basics

Draw schematic

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



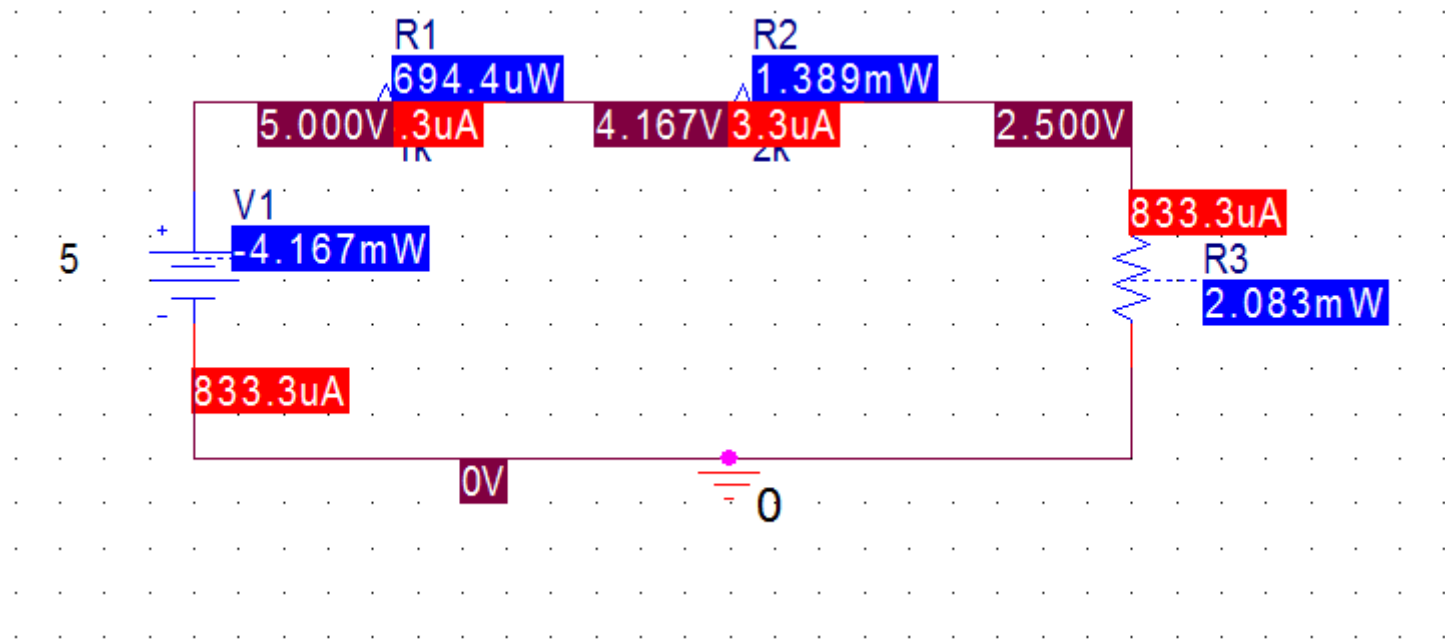
Ground



# Lect. 4: PSpice Tutorial

## ◆ Part.1 Bias point simulation

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



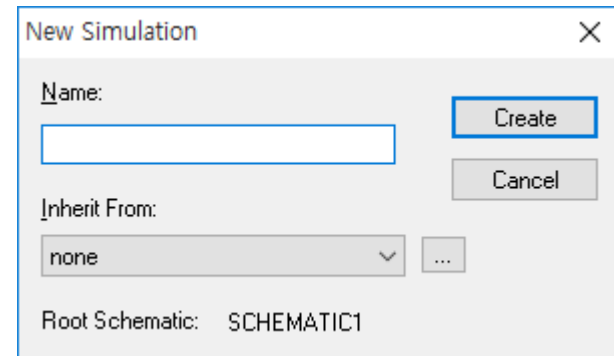
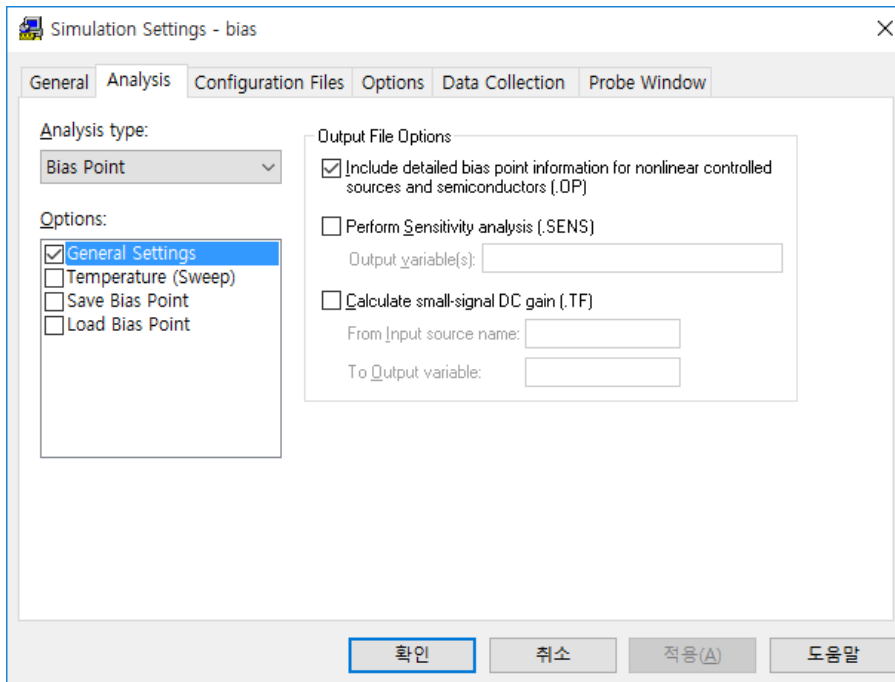
# Lect. 4: PSpice Tutorial

## ◆ Part.1 Bias point simulation



New simulation

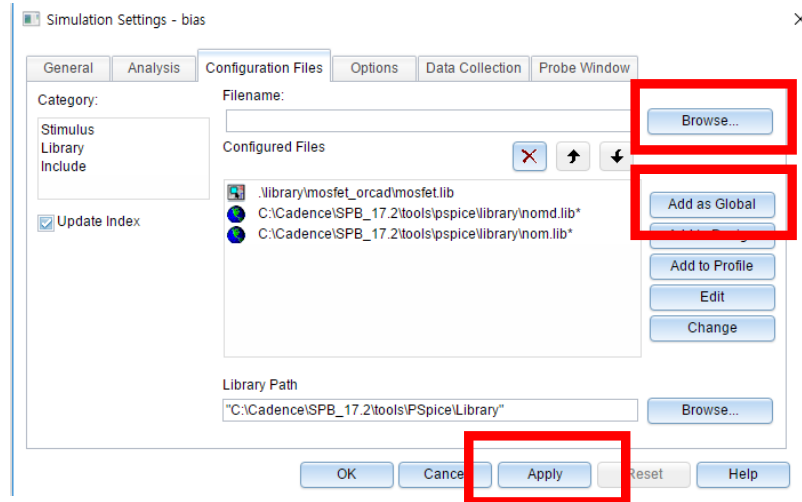
New simulation  
(Bias simulation would be already added if you have proper base project.)



# Lect. 4: PSpice Tutorial

## ◆ 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!**

# Lect. 4: PSpice Tutorial

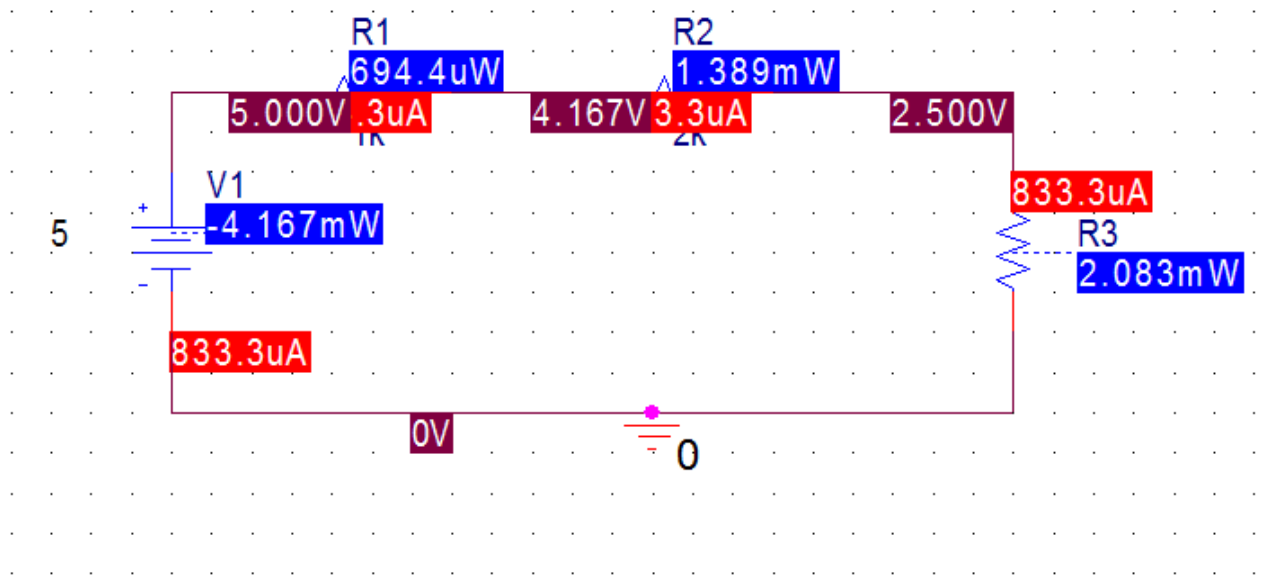
## ◆ Part.1 Bias point simulation

Run simulation



The image shows the PSpice simulation toolbar with several icons. Red arrows point to specific icons with labels: 'Run' points to the play button, 'See Voltage' points to the voltage measurement icon (V), 'See Current' points to the current measurement icon (I), and 'See Power' points to the power measurement icon (W).

Simulation Results

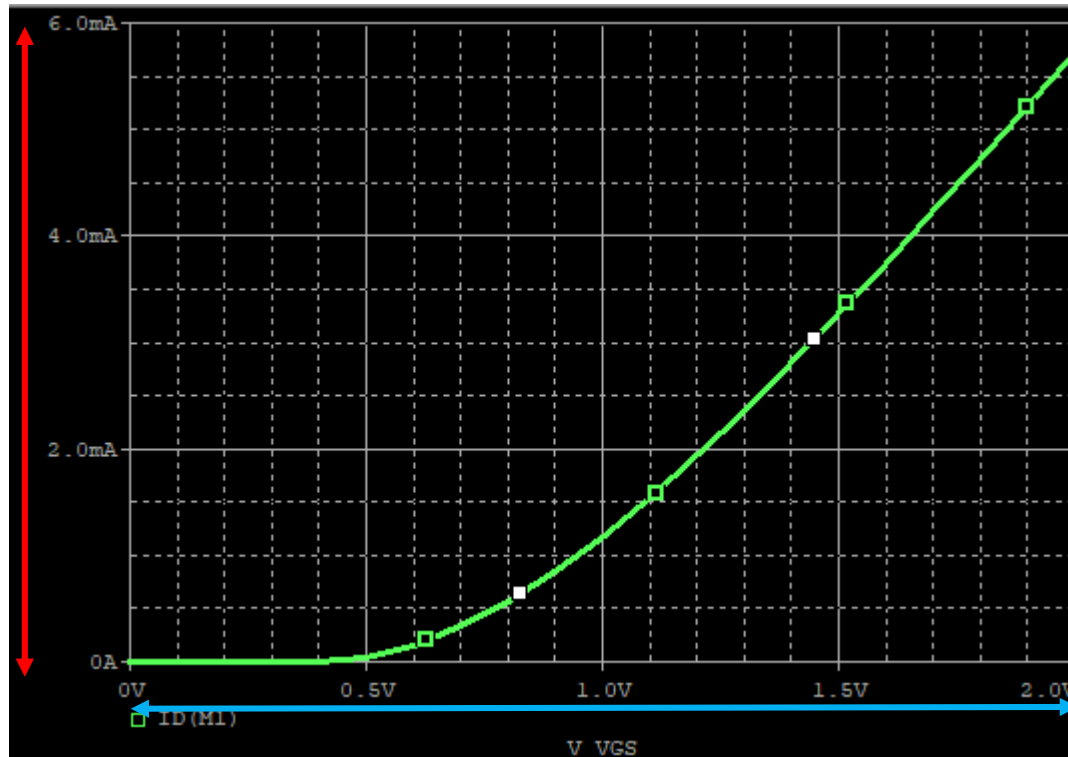


# Lect. 4: PSpice Tutorial

## ◆ 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

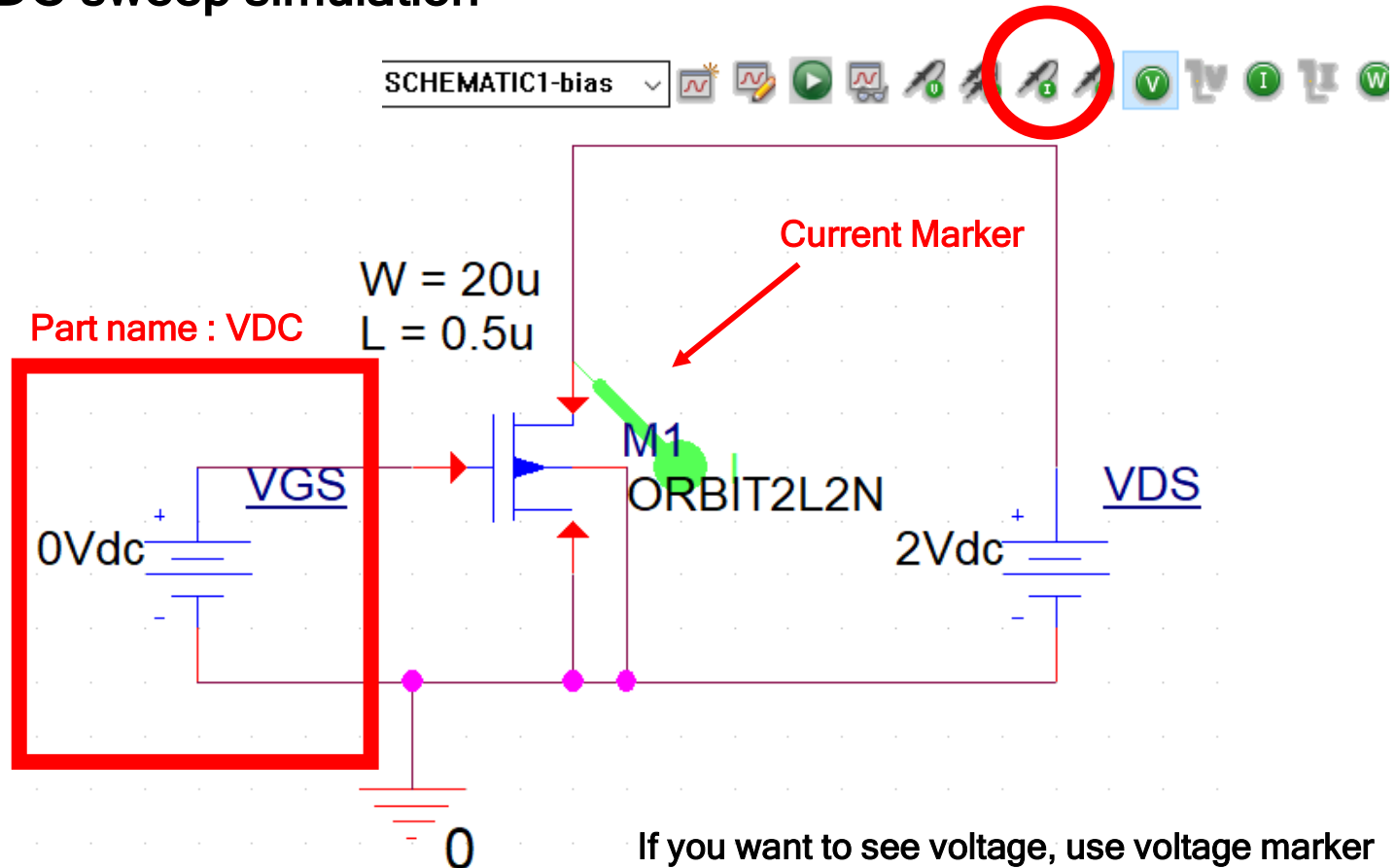


DC value  
changing



# Lect. 4: PSpice Tutorial

## ◆ Part.2 DC sweep simulation

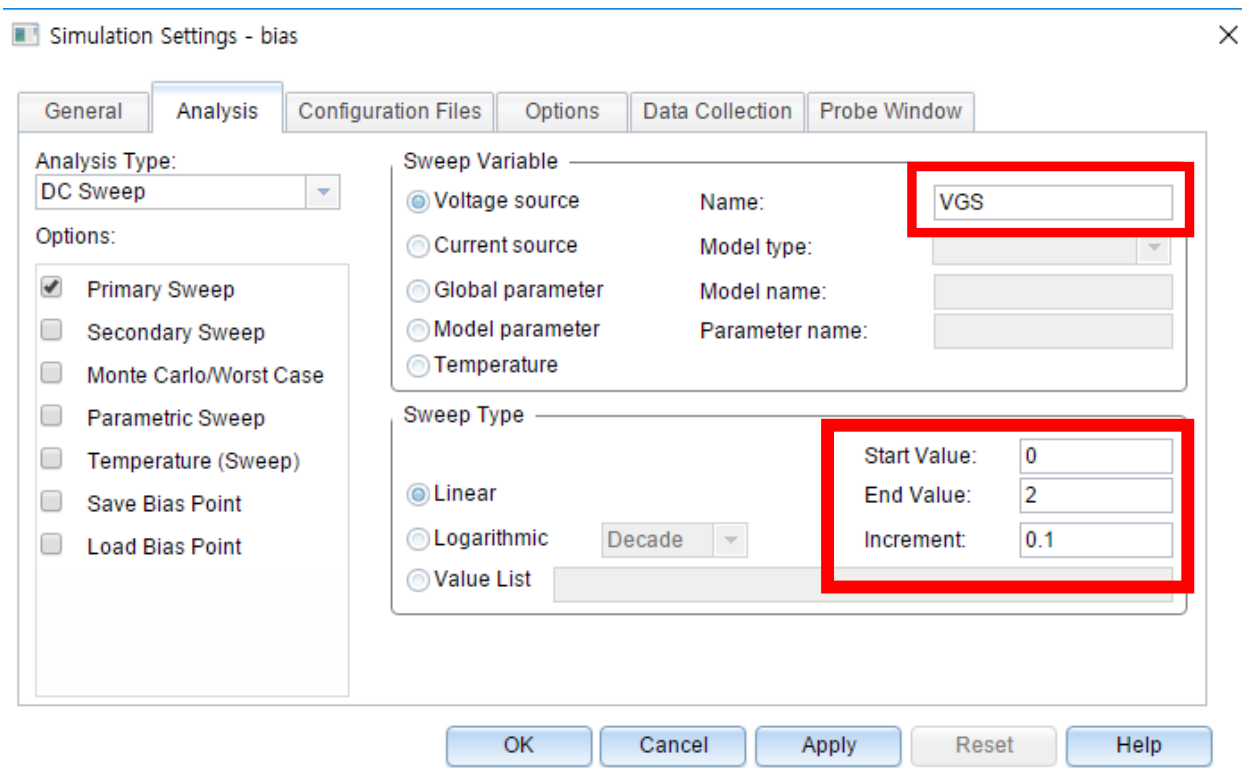


# Lect. 4: PSpice Tutorial

## ◆ Part.2 DC sweep simulation



Click Simulation Settings

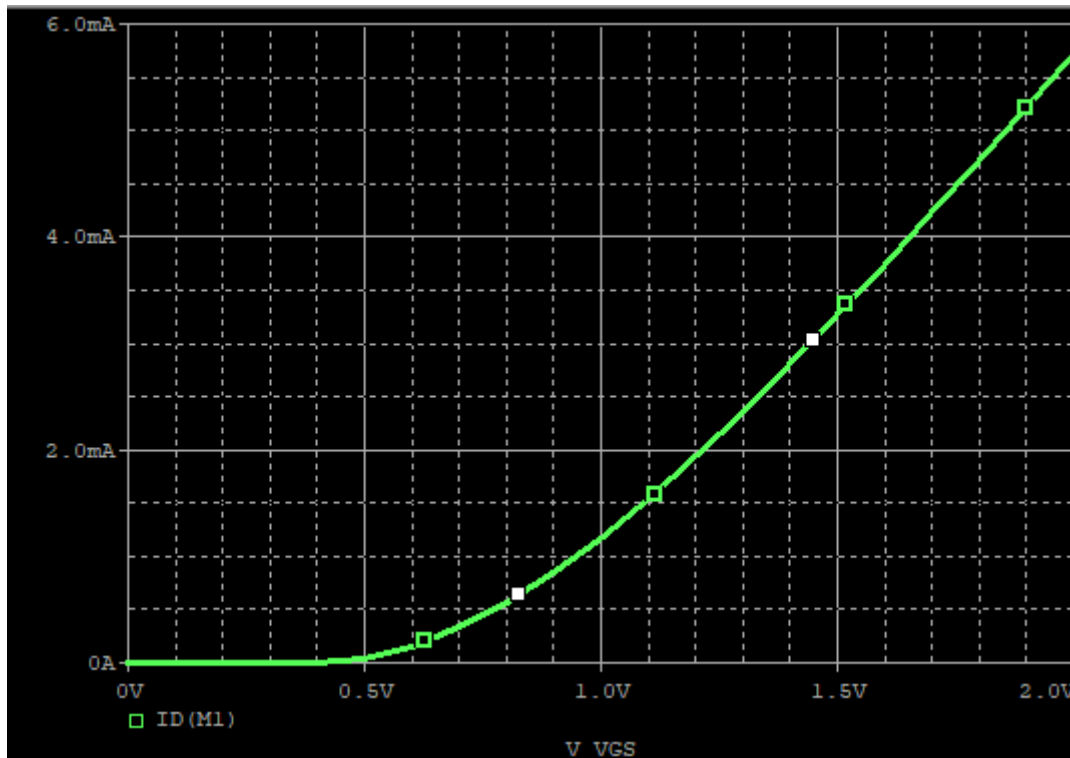


# Lect. 4: PSpice Tutorial

## ◆ Part.2 DC sweep simulation



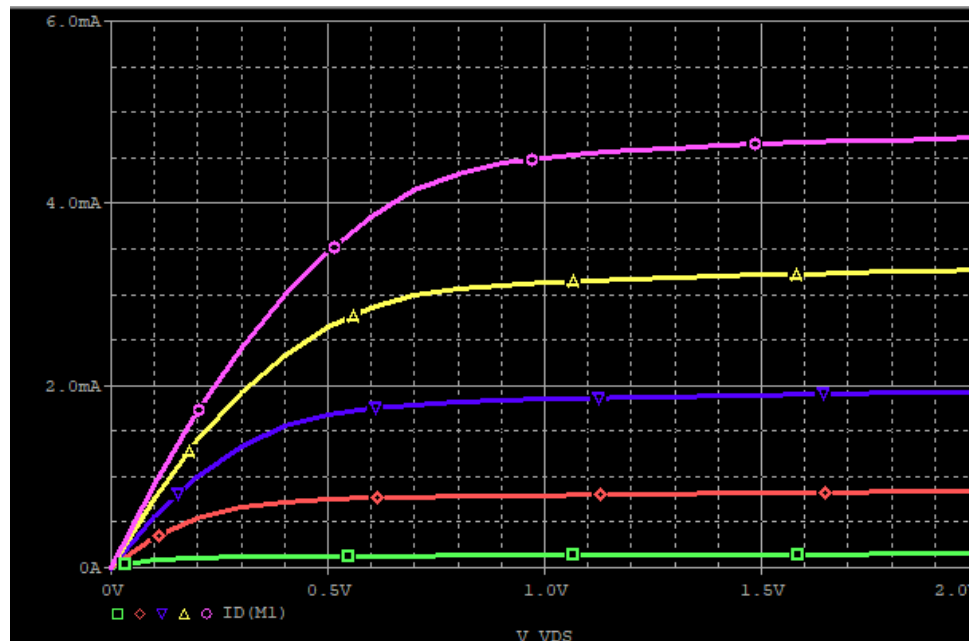
Run PSpice



# Lect. 4: PSpice Tutorial

## ◆ Part.3 Parametric Simulation

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

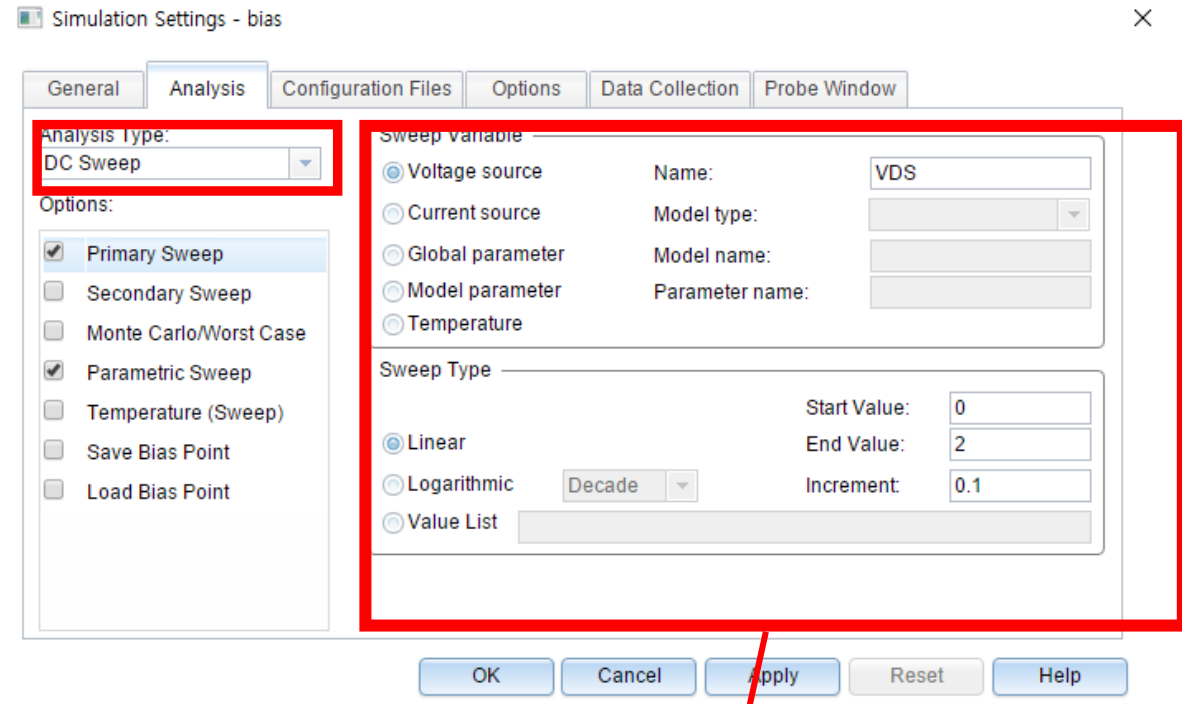


Parametric simulation can be used for not only DC sweep but also AC sweep and Transient simulation

# Lect. 4: PSpice Tutorial

## ◆ Part.3 Parametric Simulation

Choose simulation type you want to view



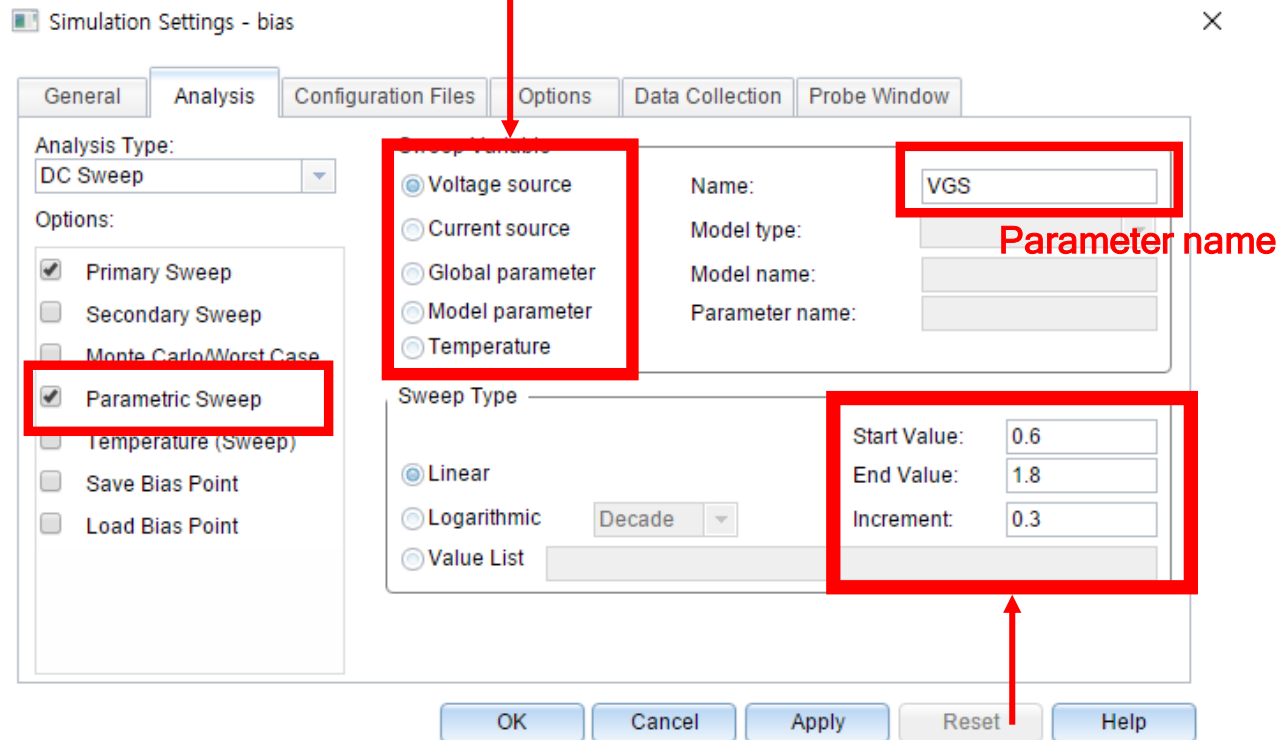
Don't forget to setup sweep condition!

# Lect. 4: PSpice Tutorial

## ◆ Part.3 Parametric Simulation

### 1. Source value change

Choose Voltage source



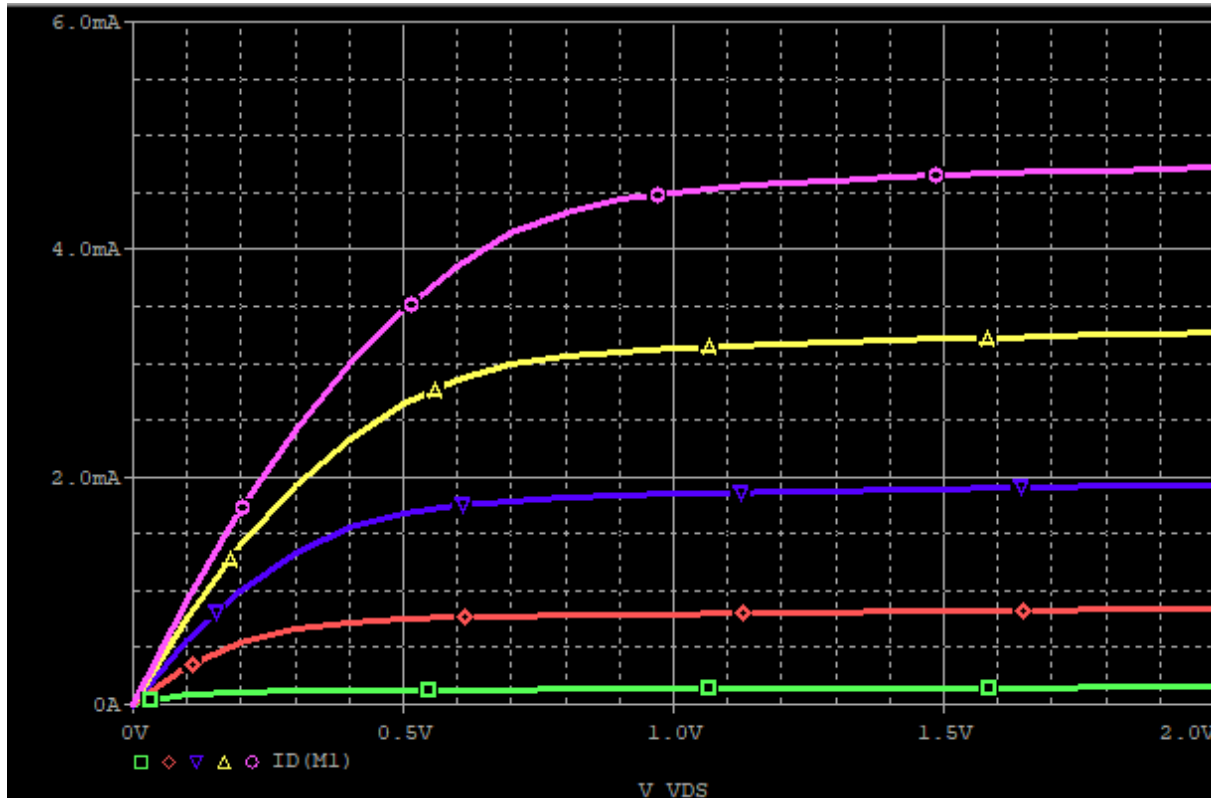
click Parametric Sweep

Parameter name

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

# Lect. 4: PSpice Tutorial

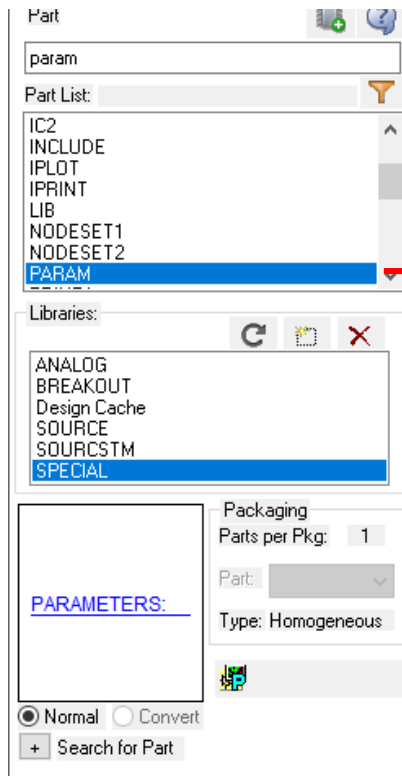
## ◆ Part.3 Parametric Simulation



# Lect. 4: PSpice Tutorial

## ◆ Part.3 Parametric Simulation

### 2. Passive element(resistor, capacitor) change



Find PARAM and place it to the schematic

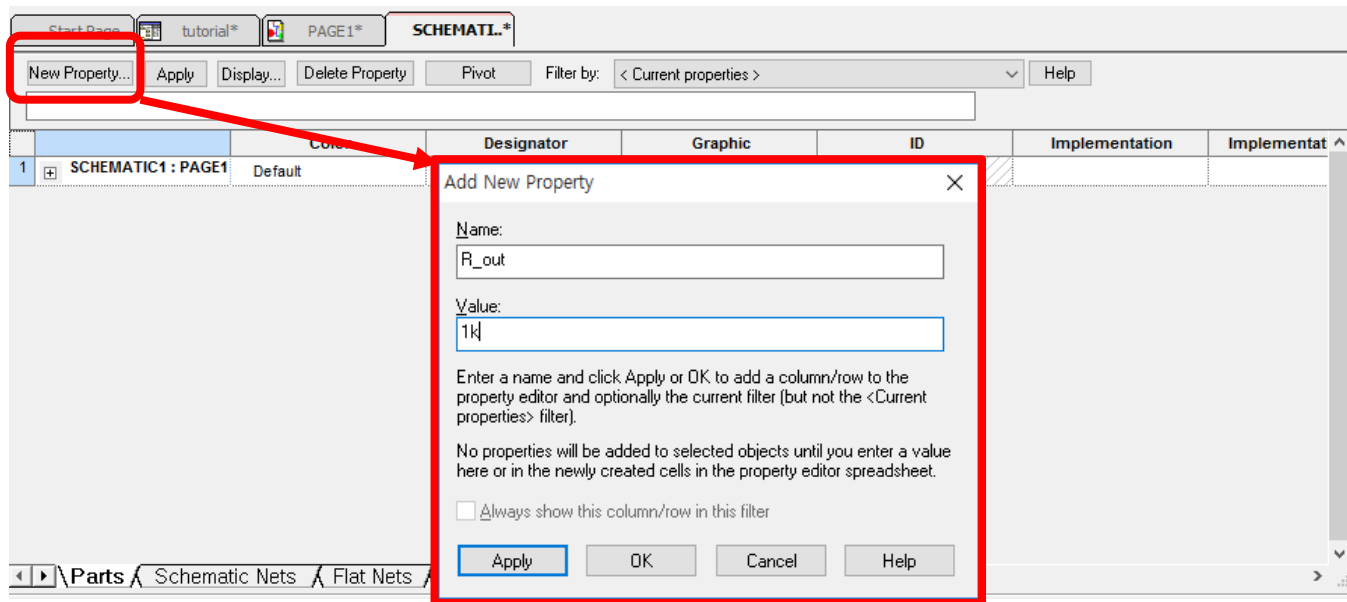


# Lect. 4: PSpice Tutorial

## ◆ Part.3 Parametric Simulation

### 2. Passive element(resistor, capacitor) change

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



# Lect. 4: PSpice Tutorial

## ◆ Part.3 Parametric Simulation

### 2. Passive element(resistor, capacitor) change

The screenshot shows the PSpice software interface. At the top, there are tabs for 'Start Page', 'tutorial\*', 'PAGE1\*', and 'SCHEMATL.\*'. Below the tabs is a toolbar with buttons for 'New Property...', 'Apply', 'Display...', 'Delete Property', 'Pivot', and 'Filter by: < Current properties >'. A table below the toolbar lists properties for 'Primitive', 'PSpiceOnly', and 'Reference'. The 'R\_out' property is highlighted in a red box, with a red arrow pointing to it from the text '① R\_out is added & change display property'. The 'R\_out' property has a value of '1k'. Below the table, there is a 'Display Properties' dialog box for 'R\_out'. The dialog box has a 'Name' field with 'R\_out', a 'Value' field with '1k', and a 'Display Format' section with radio buttons for 'Do Not Display', 'Value Only', 'Name and Value' (selected), 'Name Only', 'Both if Value Exists', and 'Value if Value Exists'. There are also fields for 'Font' (Arial 7), 'Color' (Default), 'Rotation' (0° selected), and 'Text Justification' (Default). The dialog box has 'OK', 'Cancel', and 'Help' buttons. A red arrow points from the 'Save' button in the 'SCHEMATL.\*' window to the 'Save' button in the 'Display Properties' dialog box, with the text '② Right click → Save' next to it.

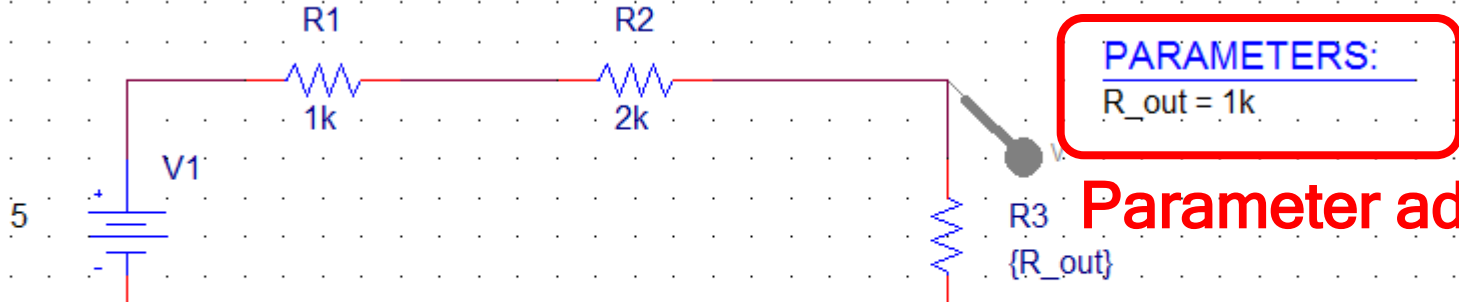
① R\_out is added & change display property

② Right click → Save

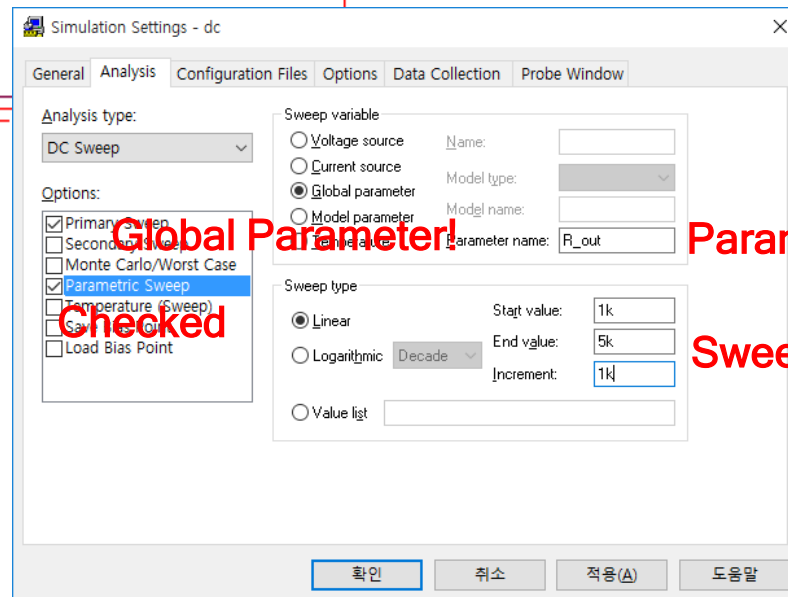
# Lect. 4: PSpice Tutorial

## ◆ Part.3 Parametric Simulation

### 2. Passive element(resistor, capacitor) change



**Parameter added!**



**Global Parameter!**

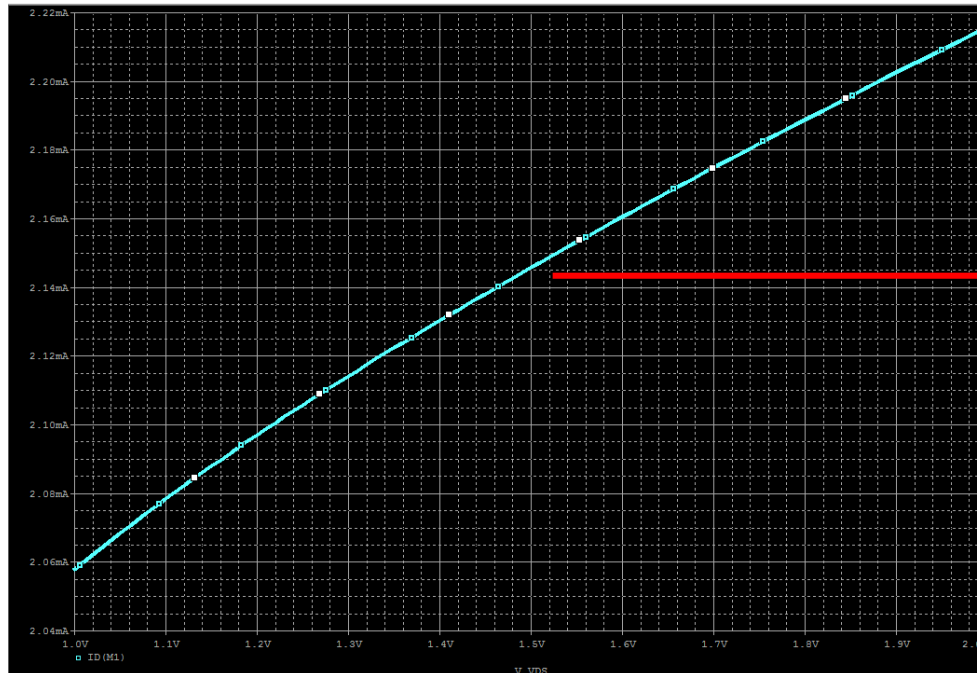
**Parameter Name**

**Checked**

**Sweep value**

# Lect. 4: PSpice Tutorial

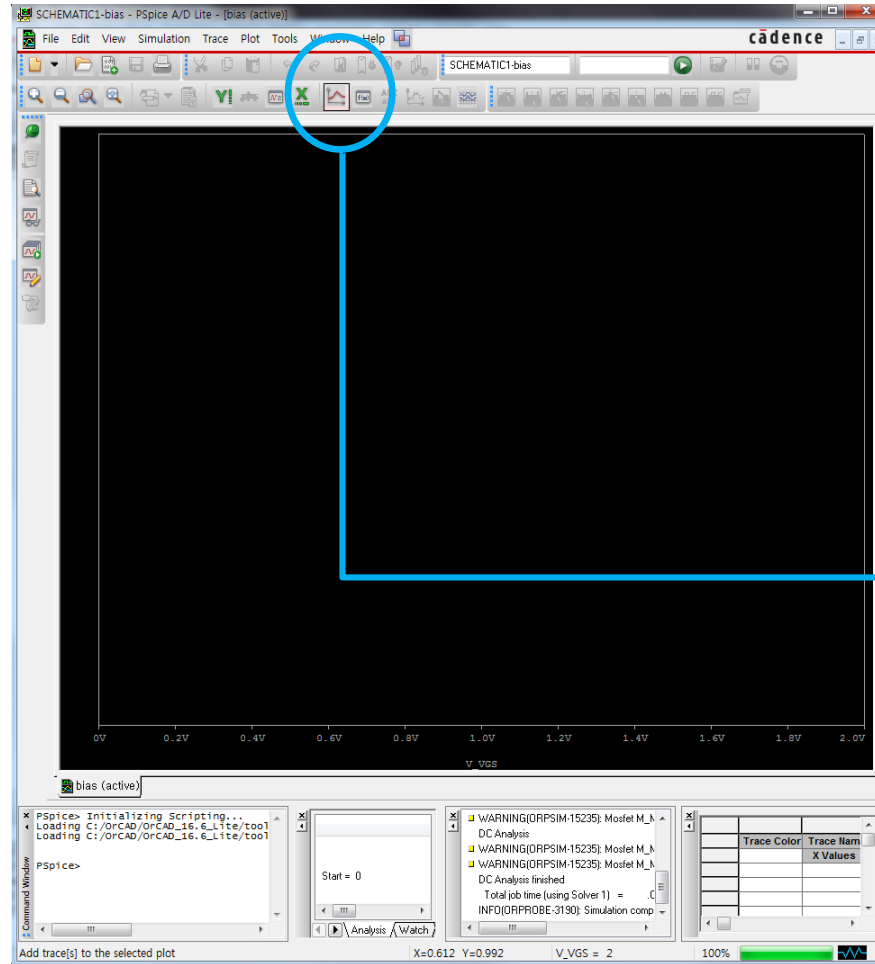
## ◆ Part.4 Function Plot



How can you plot derivative of this curve?

# Lect. 4: PSpice Tutorial

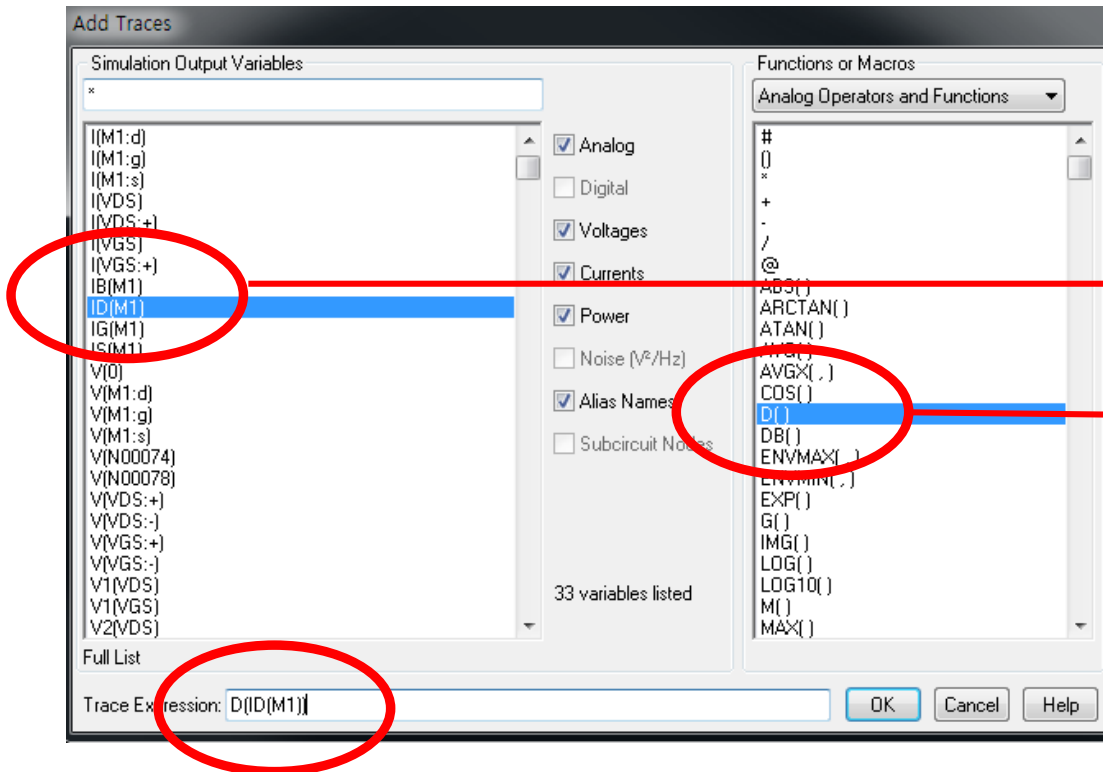
## ◆ Part.4 Function Plot



Select 'Add Trace'

# Lect. 4: PSpice Tutorial

## ◆ Part.4 Function Plot

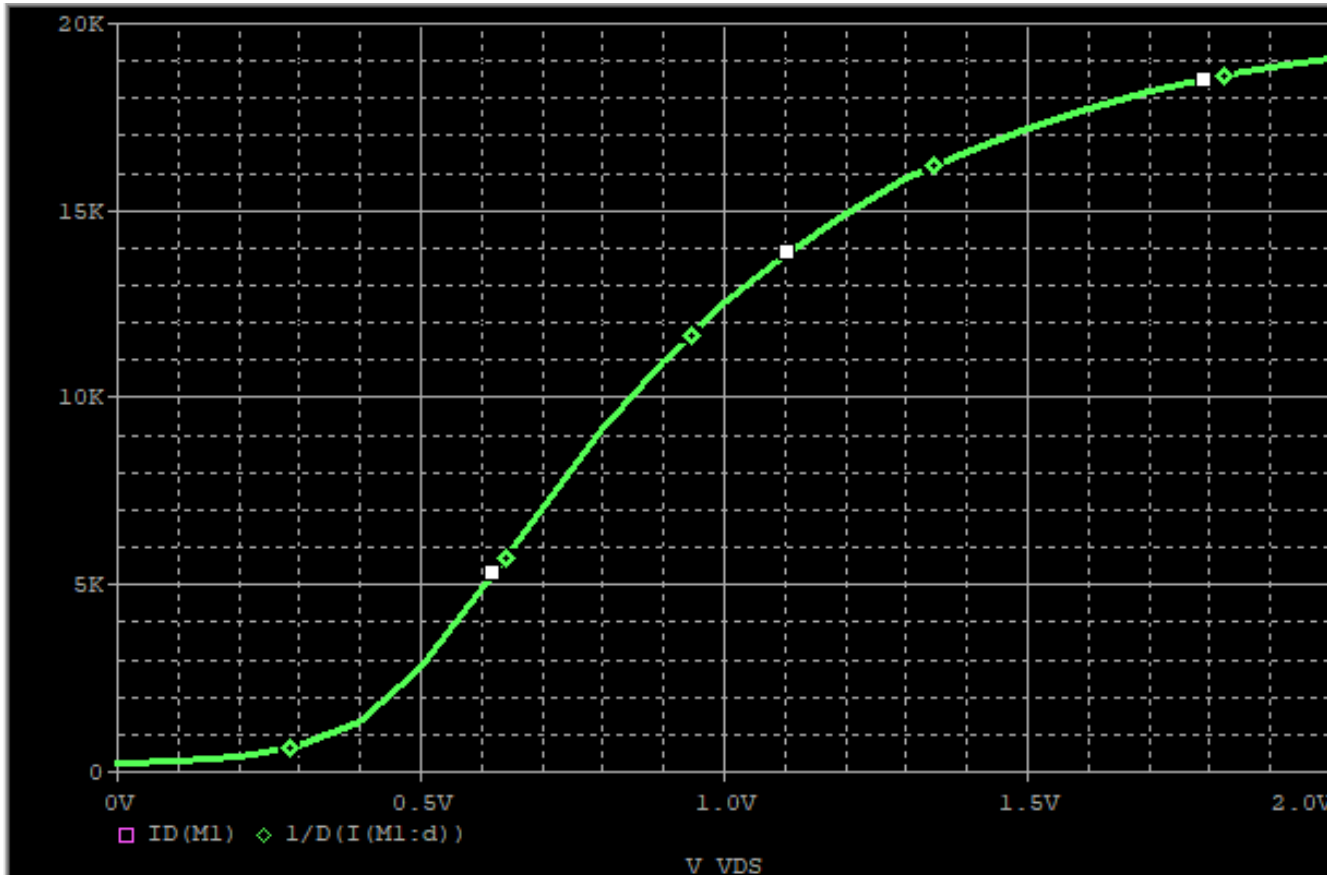


ID(M1) represents drain current of M1 MOSFET

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

# Lect. 4: PSpice Tutorial

## ◆ Part.4 Function Plot

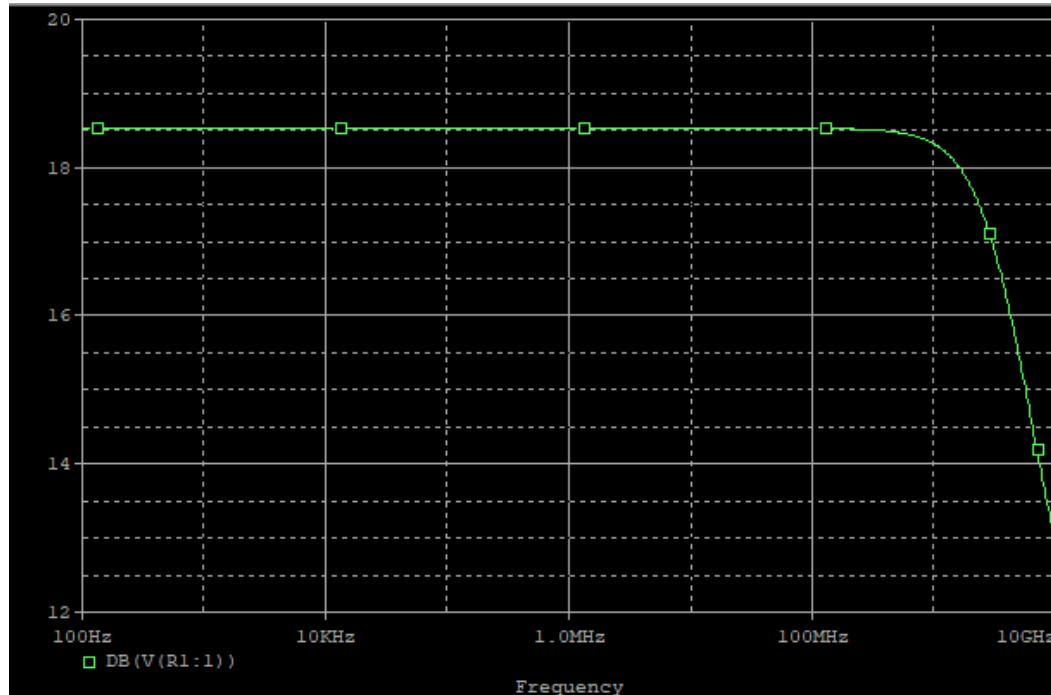


# Lect. 4: PSpice Tutorial

## ◆ Part.5 AC Simulation

AC sweep is used for seeing **frequency response** at a specific node .

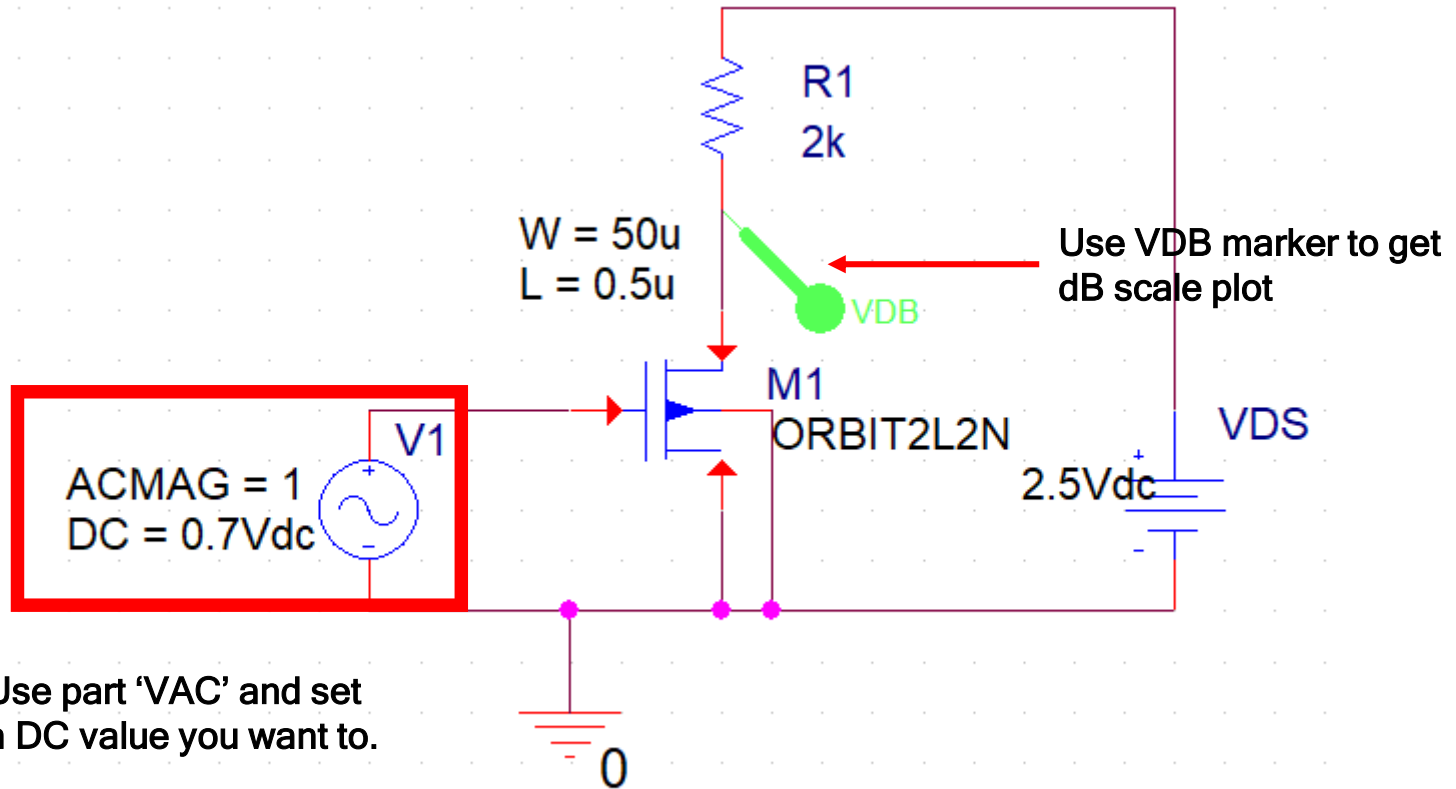
(Don't confuse it; it is not used for time-domain simulation)





# Lect. 4: PSpice Tutorial

## ◆ Part.5 AC Simulation



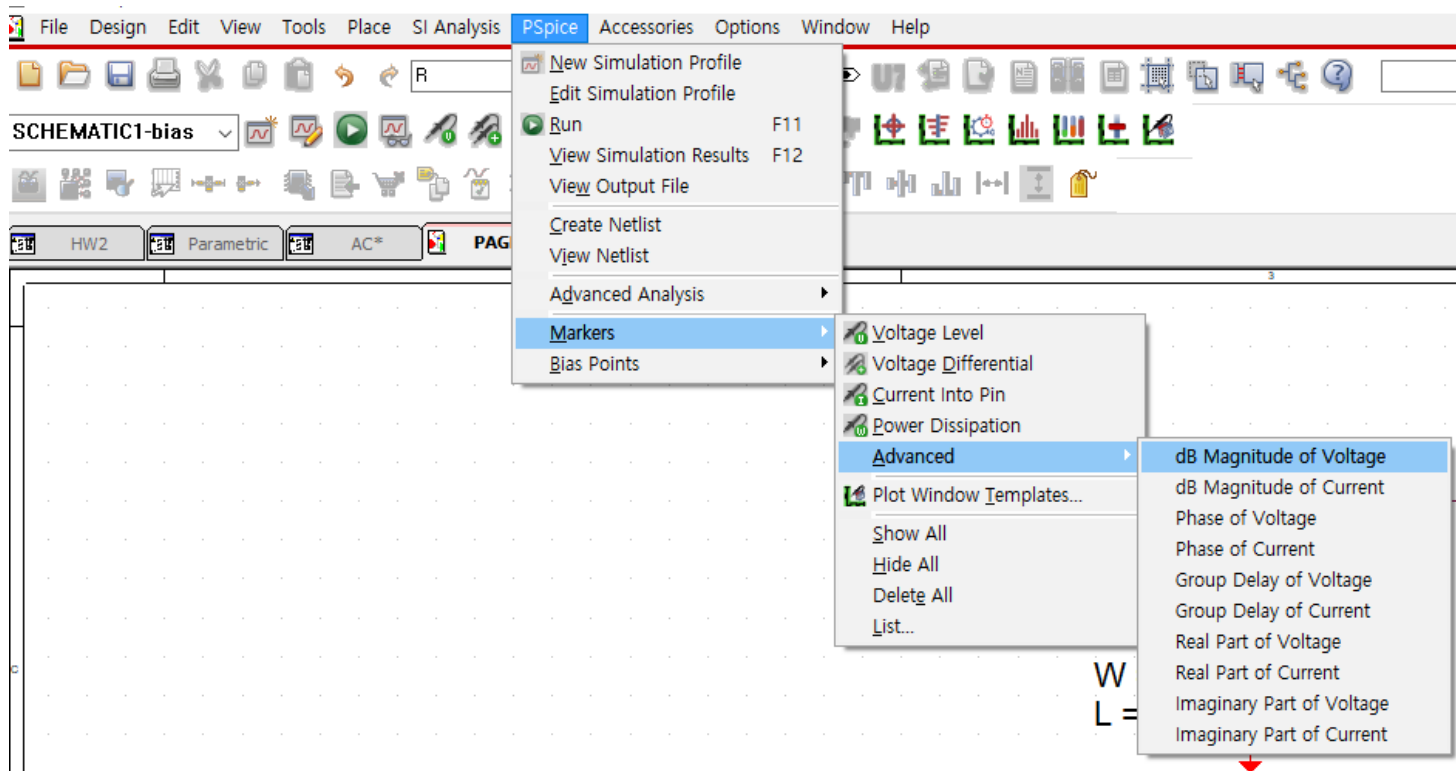
Use part 'VAC' and set a DC value you want to.

Make sure that ACMAG is set to 1.

# Lect. 4: PSpice Tutorial

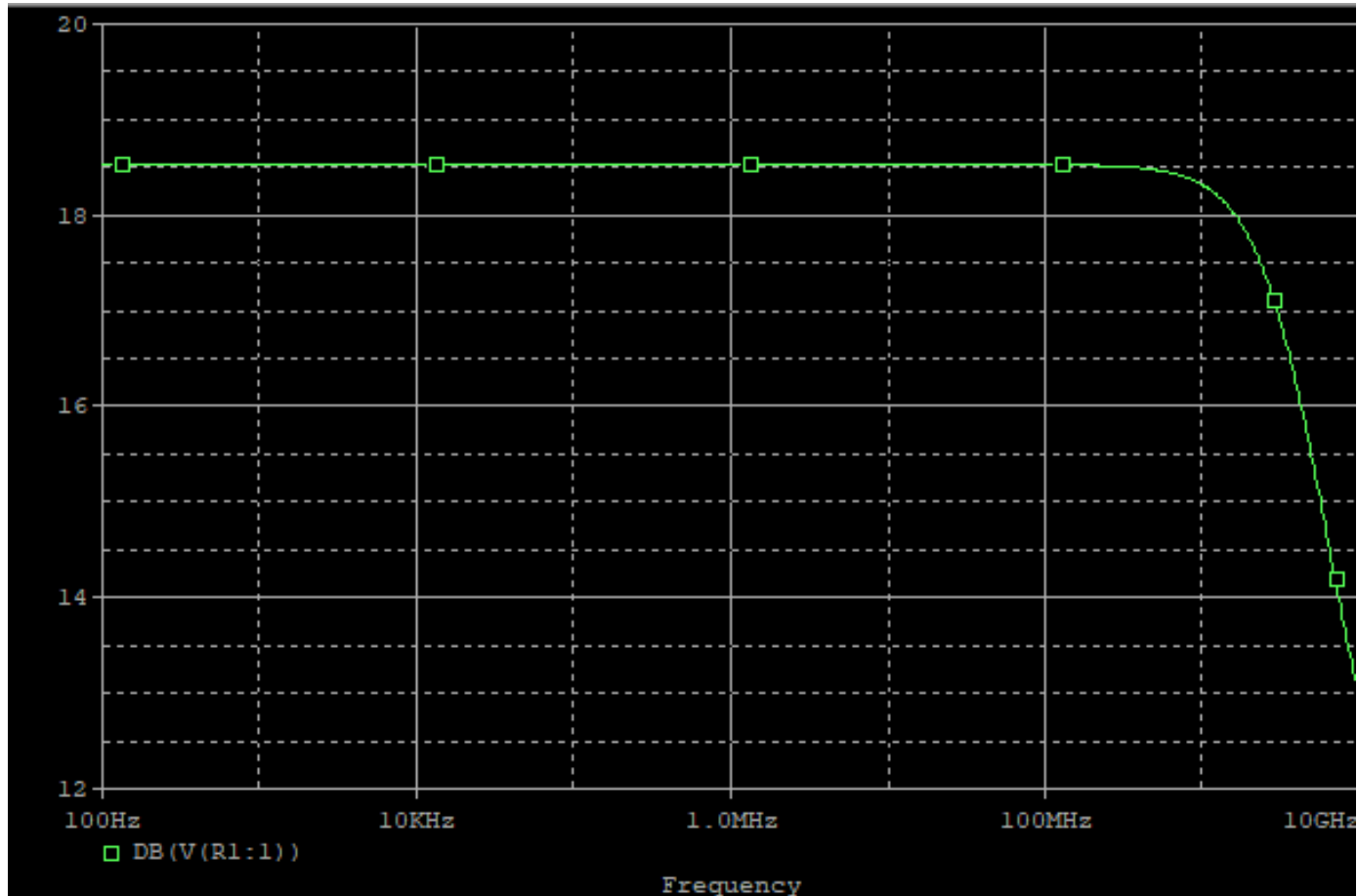
## ◆ Part.5 AC Simulation

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.)



# Lect. 4: PSpice Tutorial

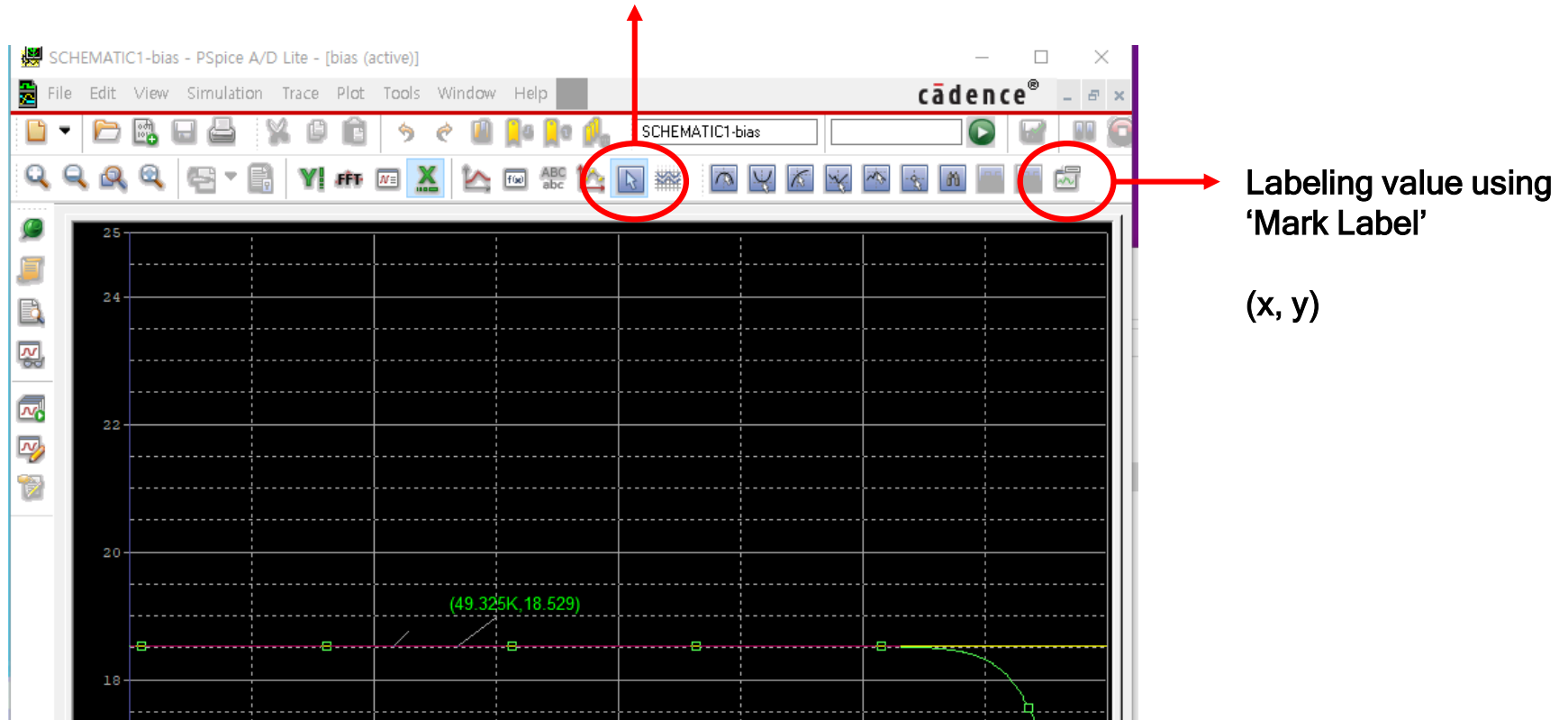
## ◆ Part.5 AC Simulation



# Lect. 4: PSpice Tutorial

## ◆ Part.5 AC Simulation

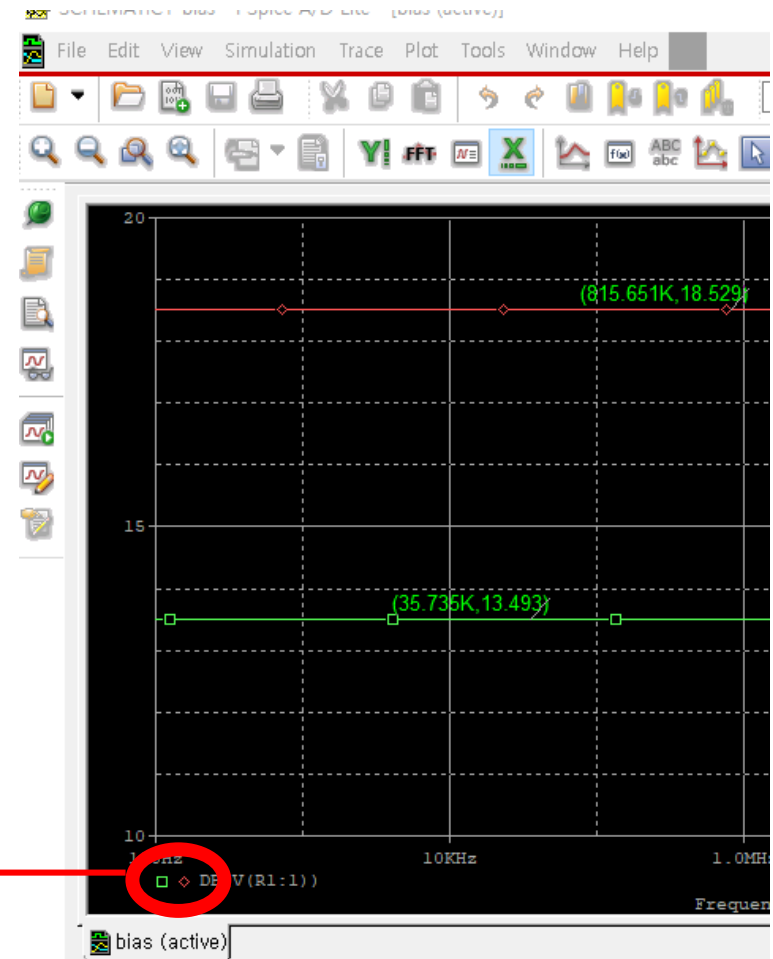
Use 'Toggle cursor' you want to see specific point (cursor can be dragged from the left)



# Lect. 4: PSpice Tutorial

## ◆ Part.5 AC Simulation

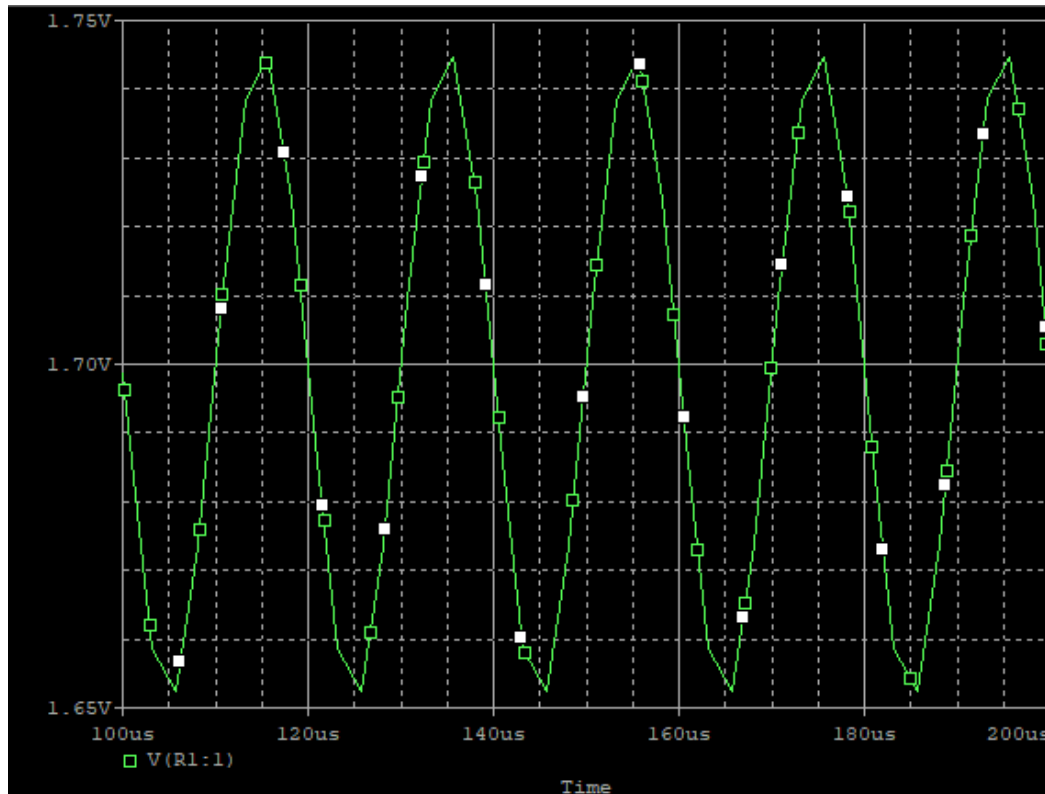
If you want to change cursor when using parametric simulation, Select the color you want to see and re-drag the cursor from the left.



# Lect. 4: PSpice Tutorial

## ◆ Part.6 Transient Simulation

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

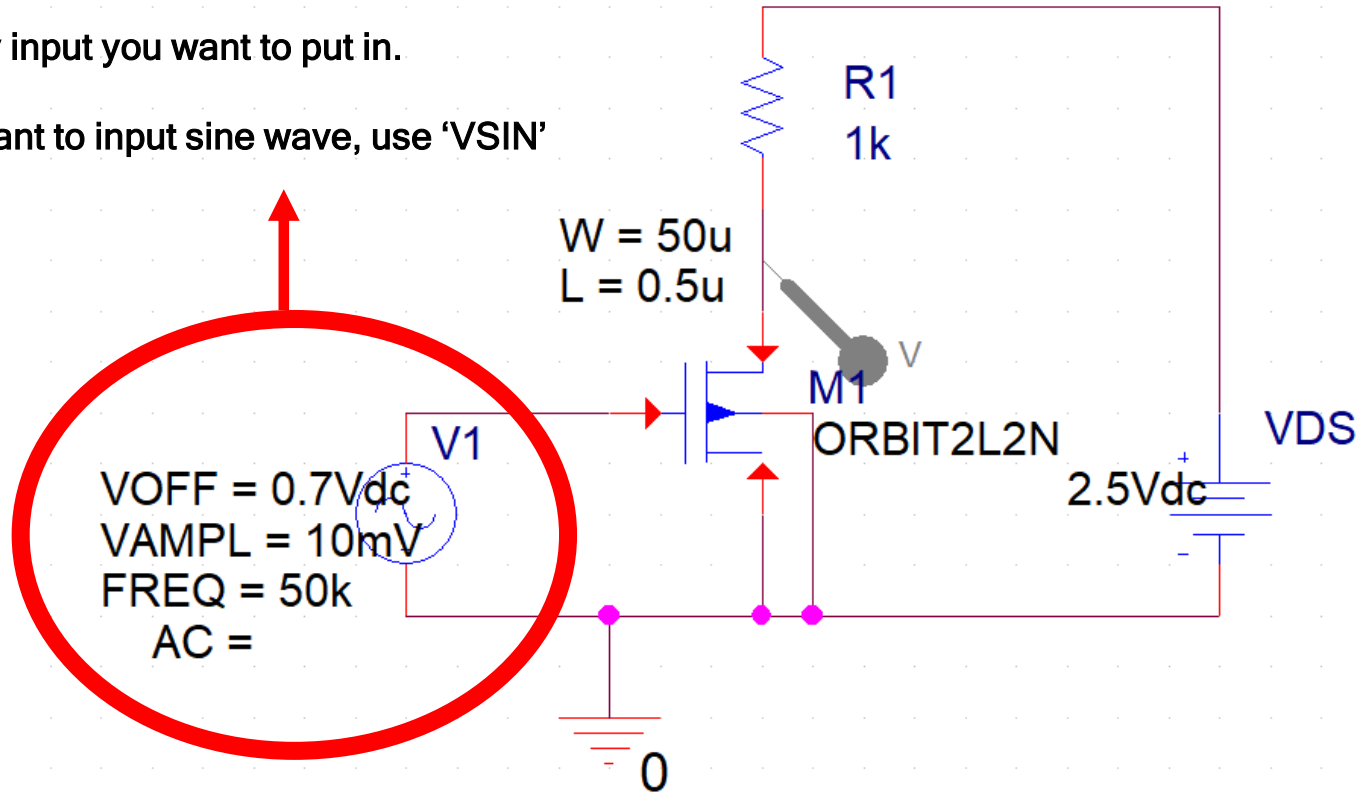


# Lect. 4: PSpice Tutorial

## ◆ Part.6 Transient Simulation

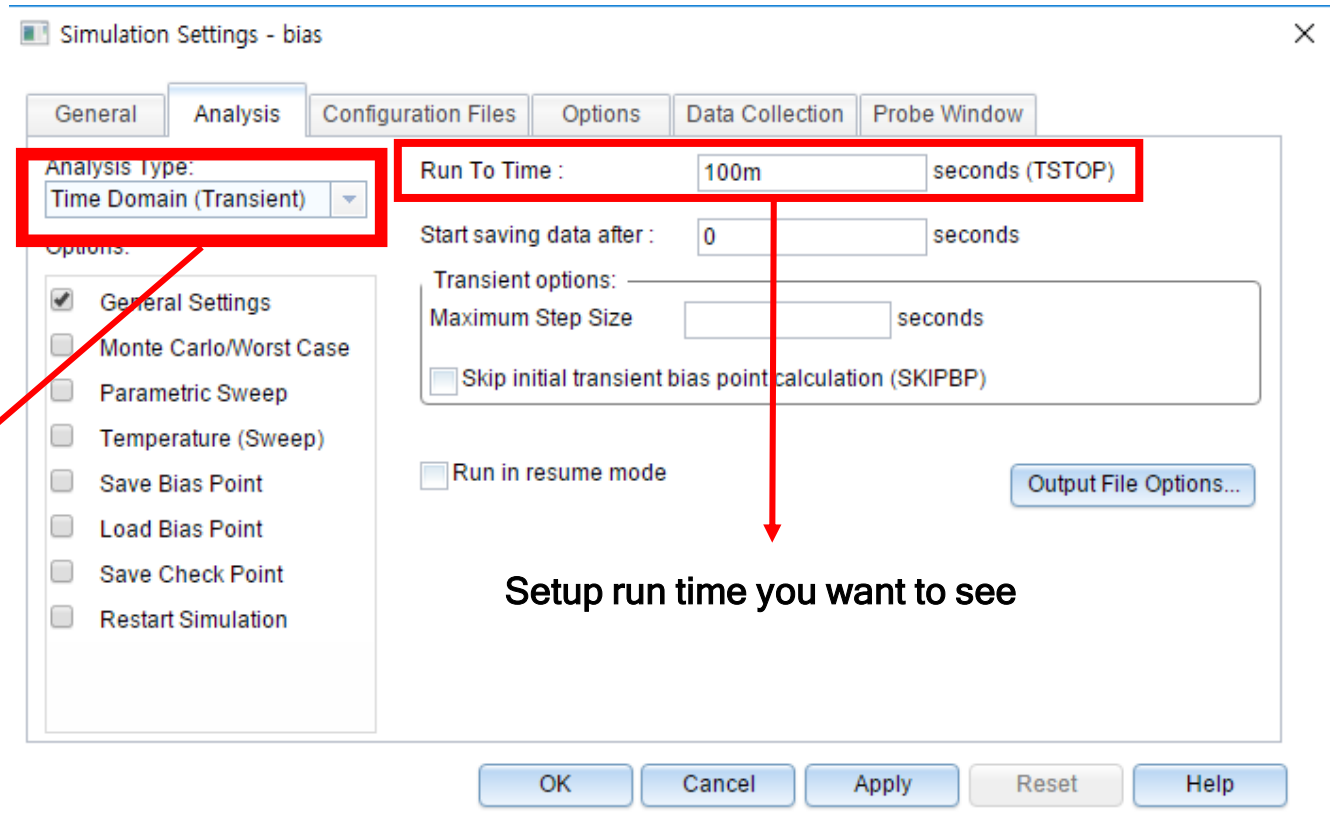
Use any input you want to put in.

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



# Lect. 4: PSpice Tutorial

## ◆ Part.6 Transient Simulation



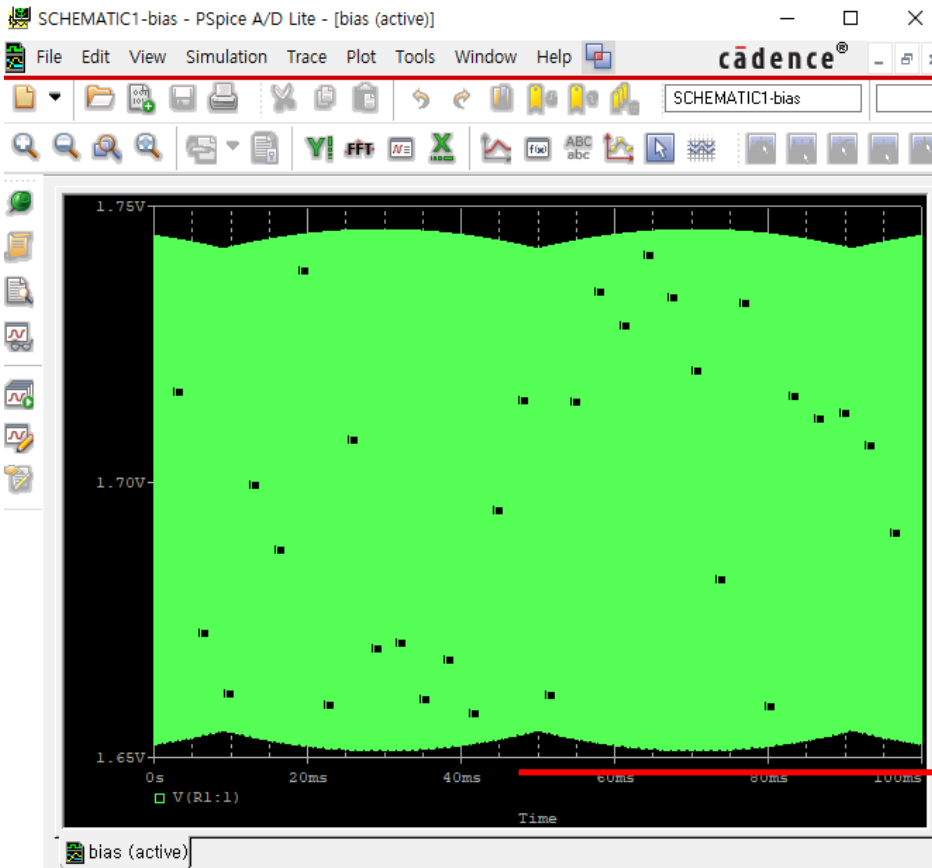
Select Time-domain

Setup run time you want to see



# Lect. 4: PSpice Tutorial

## ◆ Part.6 Transient Simulation

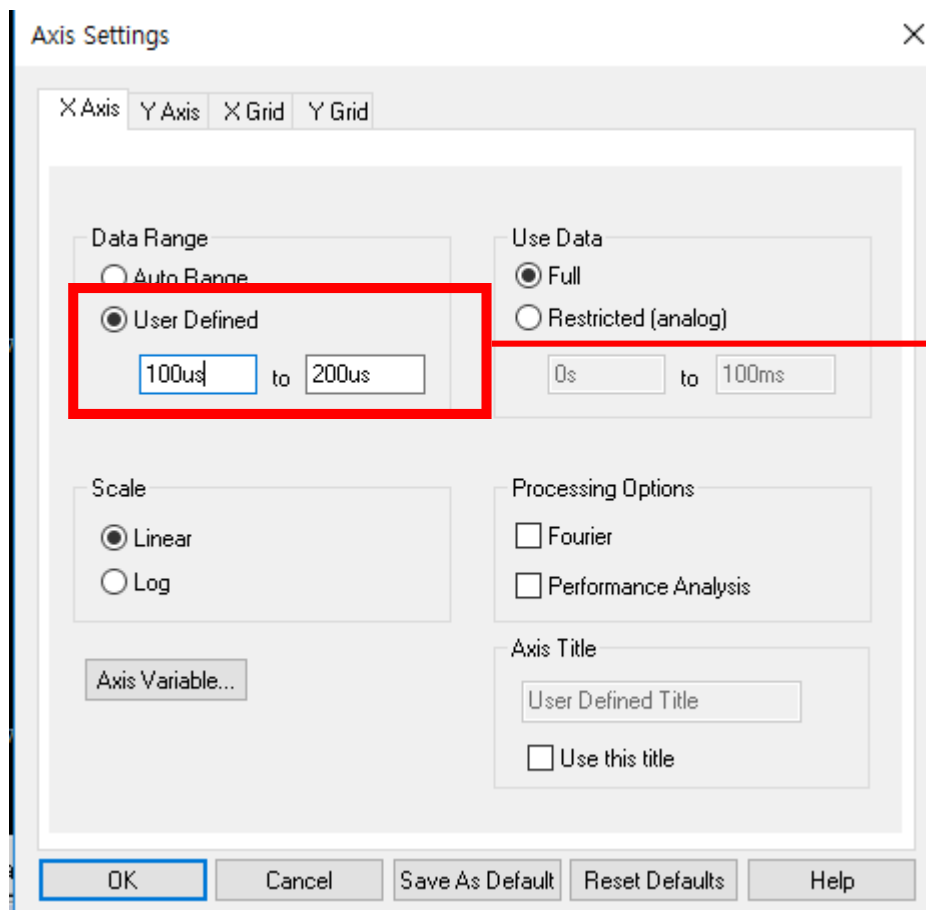


If you run too much time, signal can not be seen well.

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

# Lect. 4: PSpice Tutorial

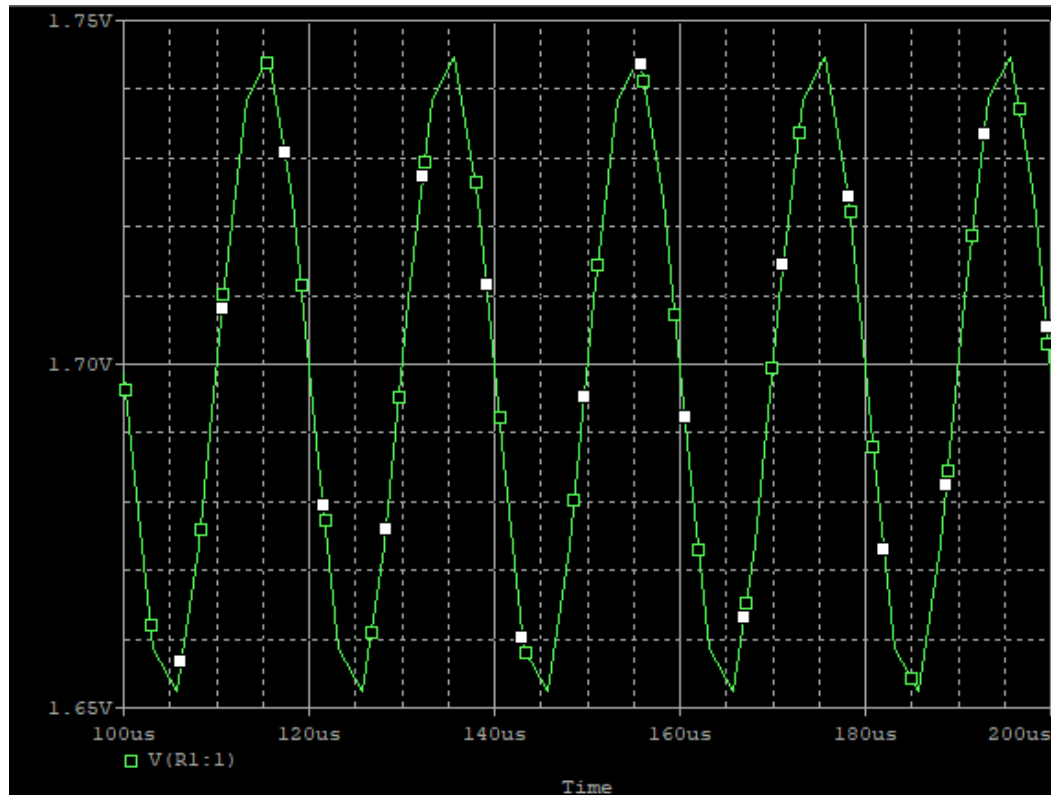
## ◆ Part.6 Transient Simulation



Click user defined and set up the time range

# Lect. 4: PSpice Tutorial

## ◆ Part.6 Transient Simulation



# Lect. 4: PSpice Tutorial

## ◆ Homework

For NMOS having  $W= 10 \mu\text{m}$  and  $L= 0.25 \mu\text{m}$ ,

- 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  $\lambda$  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.