Analog IC Design in Nanoscale CMOS

Tutorial – Using MOS models & Simulating in LTspice

Lokesh Rajendran / Hooman Reyhani

August 2021

Introduction

- The following are the steps for installing LTspice and using the MOS models provided in the course webpage. Please download the models for the process technology of your choice from here.
  - ➢ This tutorial will be using 0.18µm technology.
  - ➢ The methods specified here remain the same for all planar MOS technology – for FinFET's additional steps are required.
- LTspice is a free SPICE simulator available from Analog Devices. The software can be downloaded from here.
  - ➢ Please choose the package suitable for your operating system.
  - ➢ This tutorial uses Windows10 package.

Entering Schematic Diagrams in LTspice

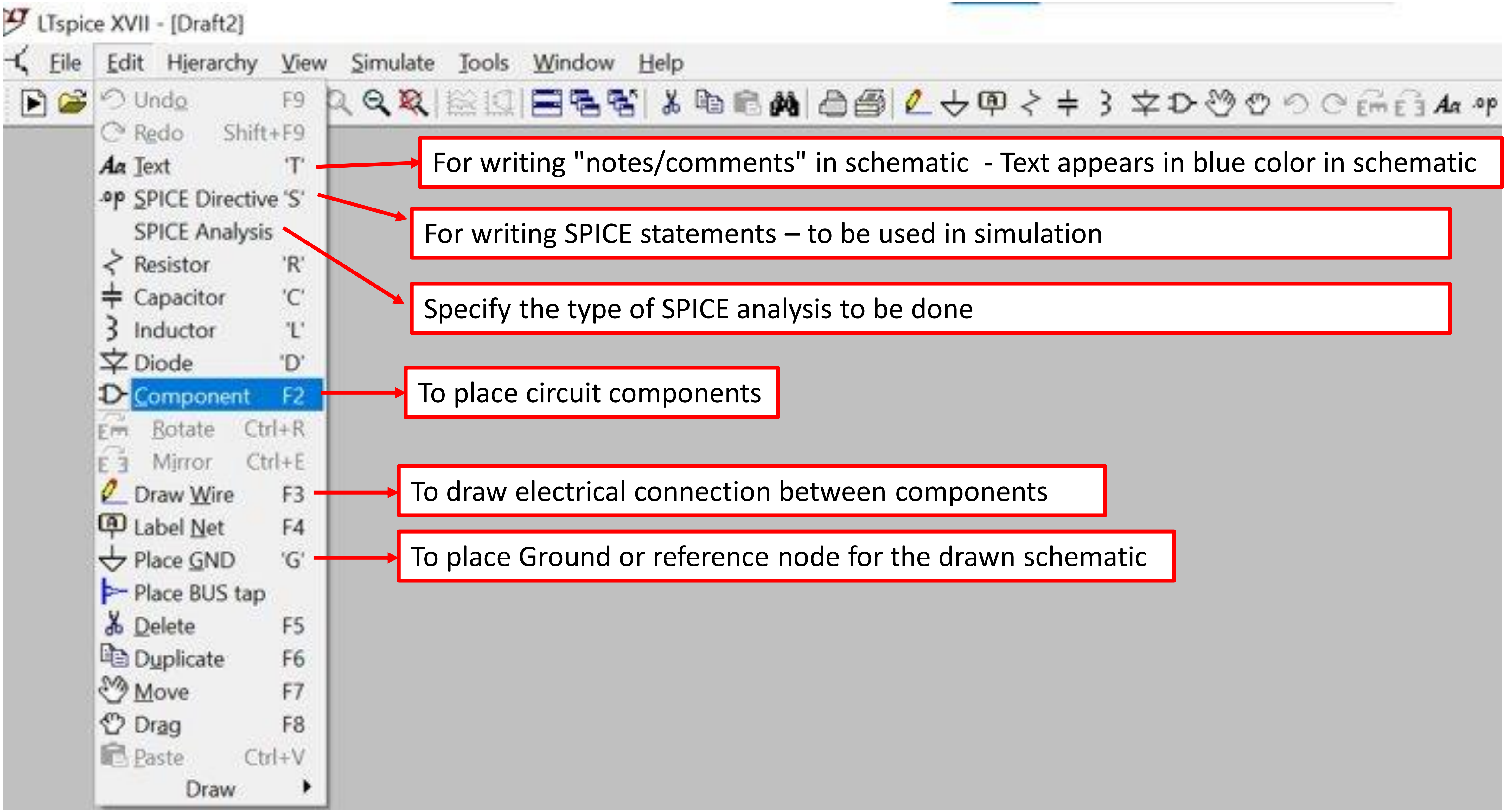
- Choose “File>New schematic” in LTspice for a blank schematic page.
- Choose “Edit > Component”. Select “nmos4” for choosing a 4 terminal NMOS. Similarly "pmos4" for a 4 terminal PMOS.
- Place the symbol in the schematic.
  - ➢ *See 'APPENDIX' slide on LTspice snapshots for various options.*
- “Right click” on the nmos symbol and provide the model's name.
- “nmos” for NMOS & “pmos” for PMOS for 0.18µm technology file.
- Specify "Width" & "Length" for the MOSFET.
  - ➢ To make W & L values visible in schematic – "CTRL + Right click" on MOS symbol. Check the values button.

Running a simple 'DC sweep' simulation

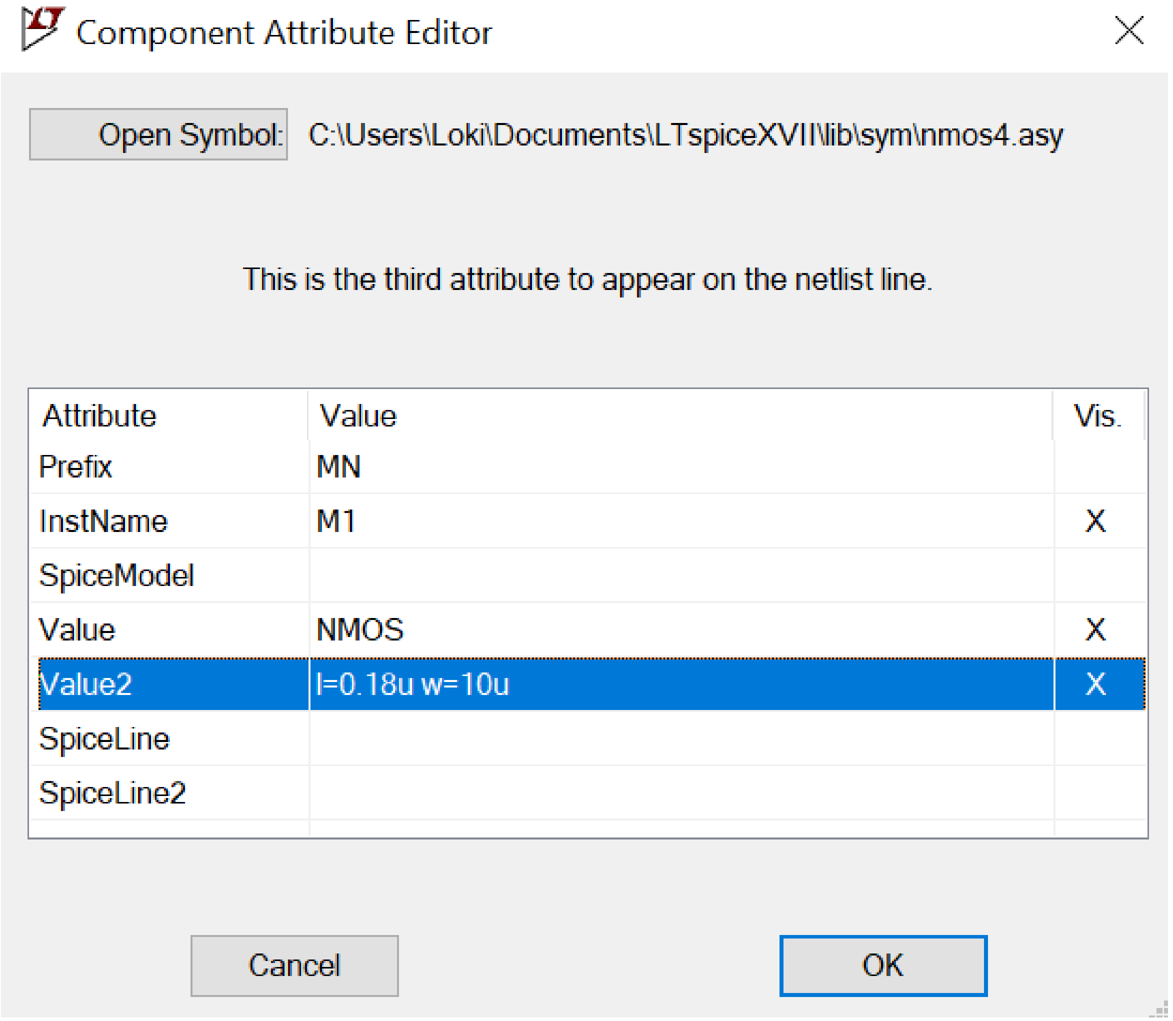
- Complete schematic diagram and wire all components.
- Choose "Edit > SPICE Directive". Use the ".include" SPICE directive and provide the path to the downloaded ".inc" model file.
  - Example:

    .include C:\...\p18\p18_cmos_models_tt.inc
- Now the schematic is ready for simulation.
- To specify the type of SPICE analysis select "Edit > SPICE Analysis".
- After saving the schematic, the simulation can be run by simply choosing the "Simulate > Run" option.

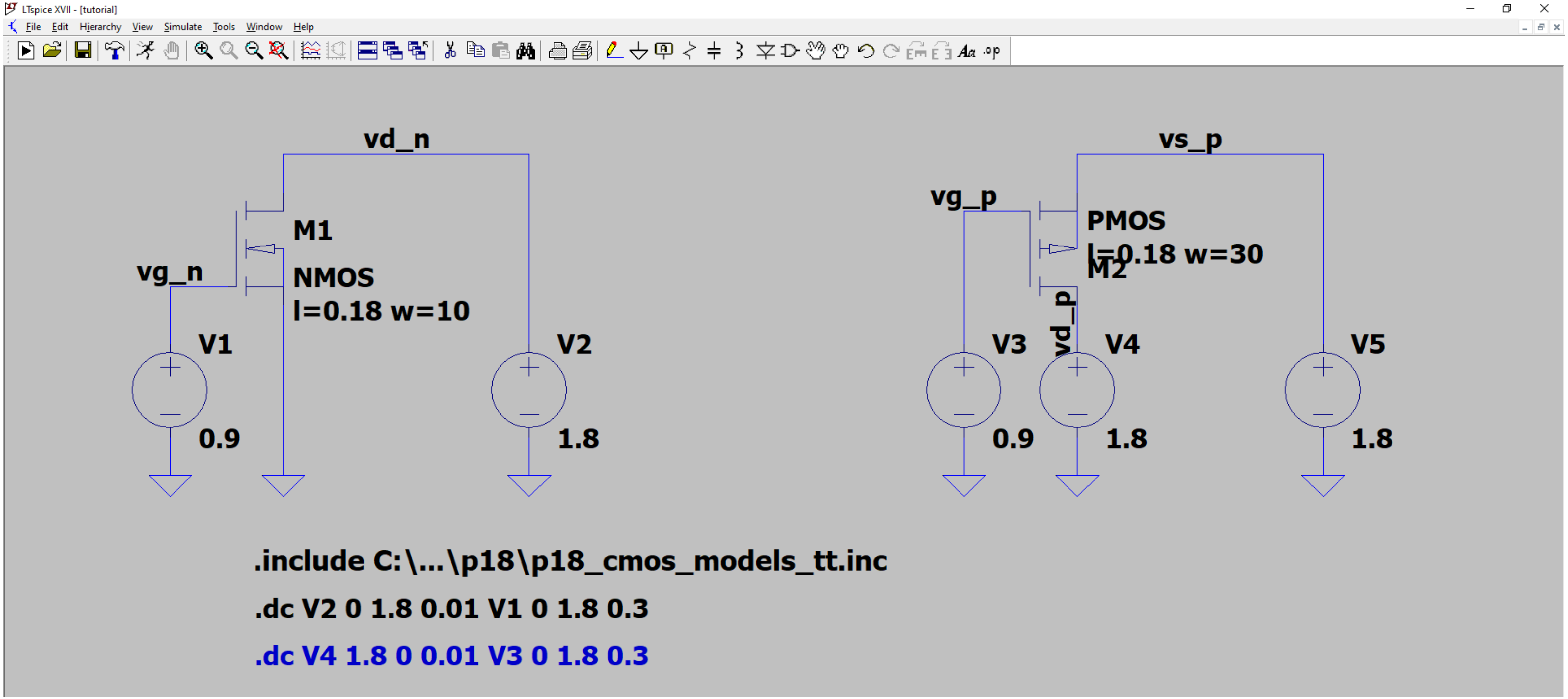
APPENDIX


LTspice XVII - [Draft2]
File
Edit
Hierarchy
View
Simulate
Tools
Window
Help
Undo F9
Redo Shift+F9
Text 'T'
SPICE Directive 'S'
SPICE Analysis
Resistor 'R'
Capacitor 'C'
Inductor 'L'
Diode 'D'
Component F2
Rotate Ctrl+R
Mirror Ctrl+E
Draw Wire F3
Label Net F4
Place GND 'G'
Place BUS tap
Delete F5
Duplicate F6
Move F7
Drag F8
Paste Ctrl+V
Draw
For writing "notes/comments" in schematic - Text appears in blue color in schematic
For writing SPICE statements – to be used in simulation
Specify the type of SPICE analysis to be done
To place circuit components
To draw electrical connection between components
To place Ground or reference node for the drawn schematic

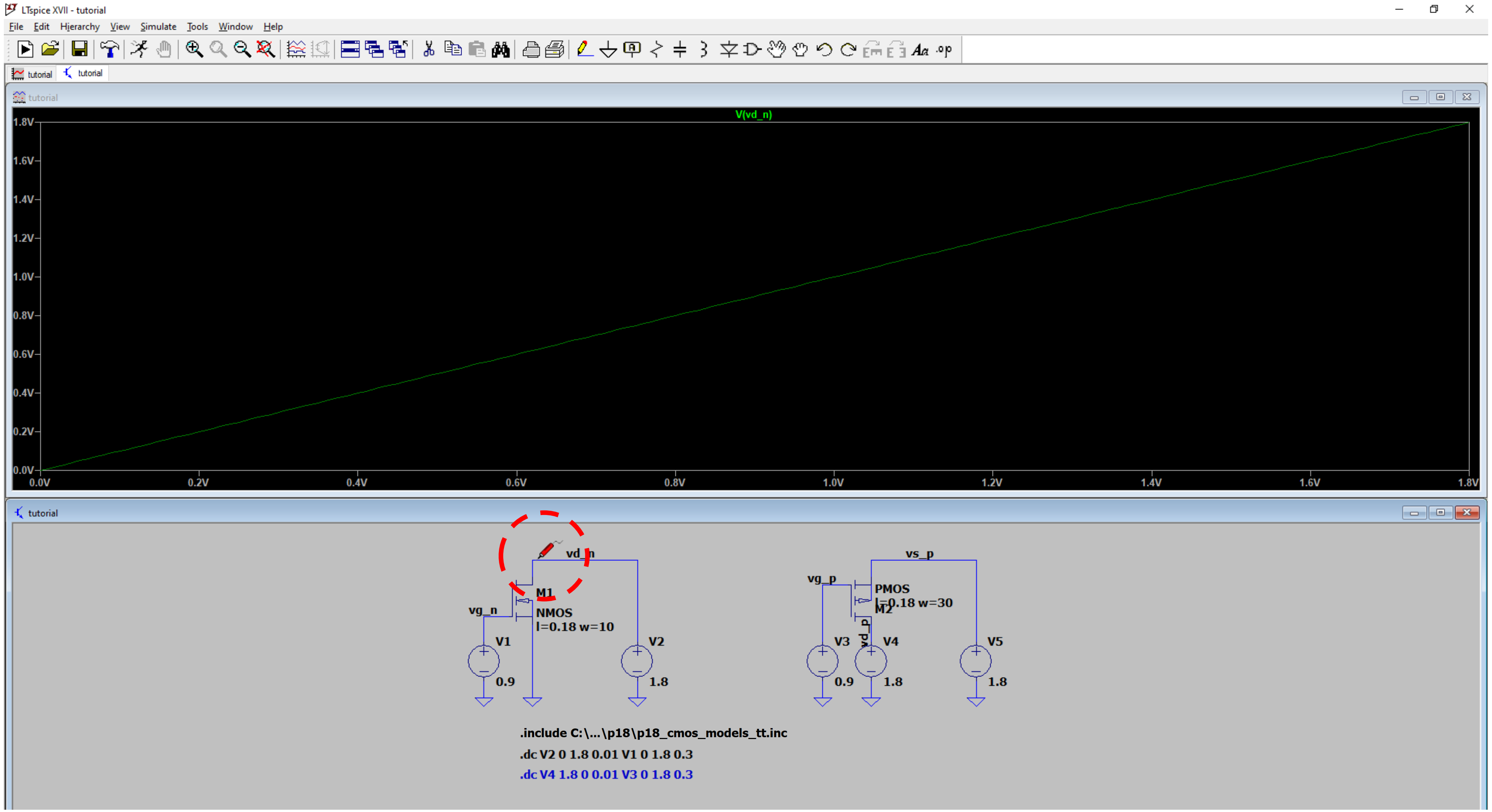
For making the W & L values visible in schematic – "CTRL + Right click" on the MOS symbol. Then check the values button.



Example schematic for plotting NMOS & PMOS – ID versus VDS characterisitcs



Plotting of signals after running simulation – VOLTAGE PROBE



Plotting of signals after running simulation – CURRENT PROBE