

EHB322E Digital Electronic Circuits

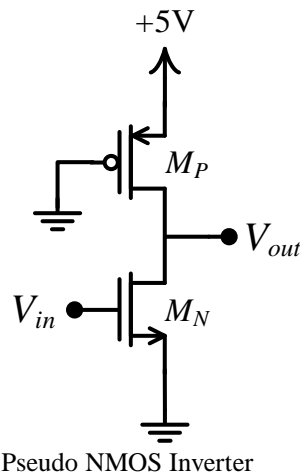
Homework 1

Deadline: 05/03/2018 (before 13:30)

Consider a pseudo NMOS inverter shown below.

- 1) **CALCULATION:** Use the following parameters for your calculations. Neglect Early effect (V_A is infinite).

Transistor parameters: $k_p' = \mu_p C_{ox} = 48 \mu\text{A}/\text{V}^2$, $k_n' = \mu_n C_{ox} = 156 \mu\text{A}/\text{V}^2$, $V_{TN} = 0.7\text{V}$, $V_{TP} = -0.95\text{V}$, $W_P = 3.2\mu$, $L_P = 0.6\mu$, $L_N = 0.6\mu$.



- a) Find the minimum value of W_N to satisfy that $V_{out} = 0.2\text{V}$ when $V_{in} = 5\text{V}$ applied.
 - b) Find the switching threshold value of V_M .
 - c) Find the **static power** consumption of the inverter for $V_{in} = 0\text{V}$ and $V_{in} = 5\text{V}$.
 - d) Suppose that a load capacitor of 10pF is connected to the output. Find the value of the propagation delay t_{PLH} .
- 2) **SIMULATION:** Construct the above circuit using SPICE. Connect body terminals of transistor to their source terminals. Select $W_P = 3.2\mu$, $L_P = 0.6\mu$, $L_N = 0.6\mu$. Use T15DN and T15DP spice models for NMOS and PMOS transistors, respectively. For details of using LTspice check out the tutorial attached to the homework.
- a) Find the minimum value of W_N to satisfy that $V_{OL} V_{out} = 0.2\text{V}$ when $V_{in} = 5\text{V}$ applied.
 - b) Sketch voltage transfer curve of the inverter; find noise margin values of NM_L and NM_H ; find the switching threshold value of V_M .
 - c) Find the **static power** consumption of the inverter for $V_{in} = 0\text{V}$ and $V_{in} = 5\text{V}$.
 - d) Suppose that a load capacitor of 10pF is connected to the output. Find the value of the propagation delay t_{PLH} .
 - e) Compare the simulation results derived from 2(a), 2(b), 2(c), and 2(d) with those calculated in the first part. Justify your answer.

Grading: 1(a)15%, 1(b)15%, 1(c)10%, 1(d)10%

2(a)10%, 2(b)15%, 2(c)5%, 2(d)10%, 2(e)10%

Note: Do not forget to attach SPICE output file prints to your homework!

Mini LTspice Tutorial

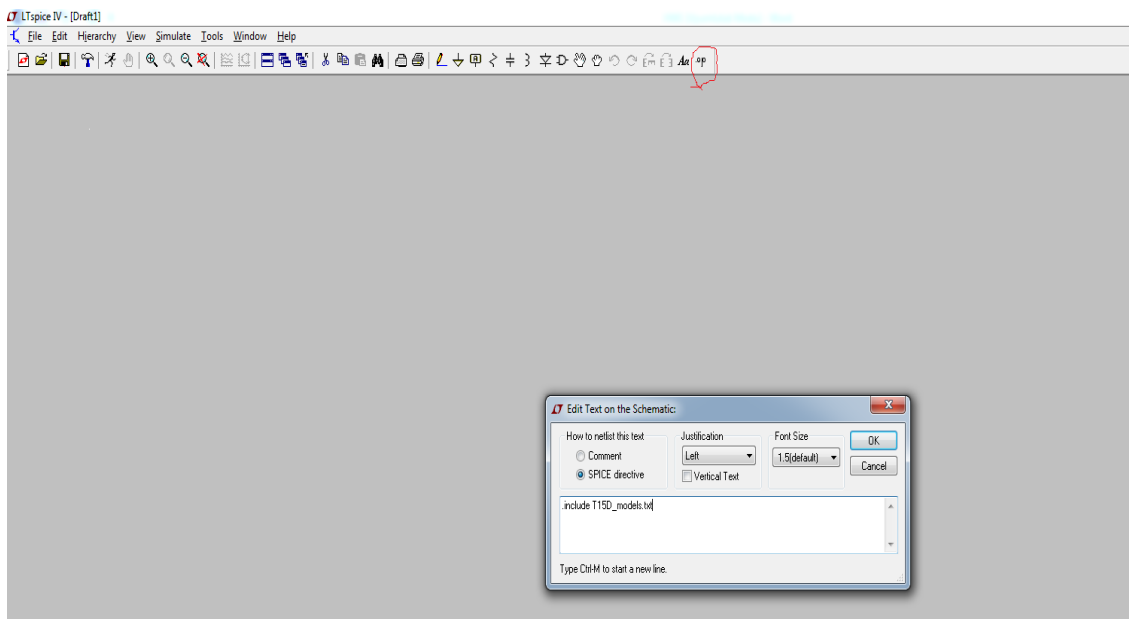
Model parameters for NMOS and PMOS transistors are given below.

```
.MODEL T15DN NMOS LEVEL=3 PHI=0.7 TOX=9.5E-09 XJ=0.2U TPG=1
+ VTO=0.7 DELTA=8.8E-01 LD=5E-08 KP=1.56E-04
+ UO=420 THETA=2.3E-01 RSH=2.0E+00 GAMMA=0.62
+ NSUB=1.40E+17 NFS=7.20E+11 VMAX=1.8E+05 ETA=2.125E-02
+ KAPPA=1E-01 CGDO=3.0E-10 CGSO=3.0E-10
+ CGBO=4.5E-10 CJ=5.50E-04 MJ=0.6 CJSW=3E-10
+ MJSW=0.35 PB=1.1
```

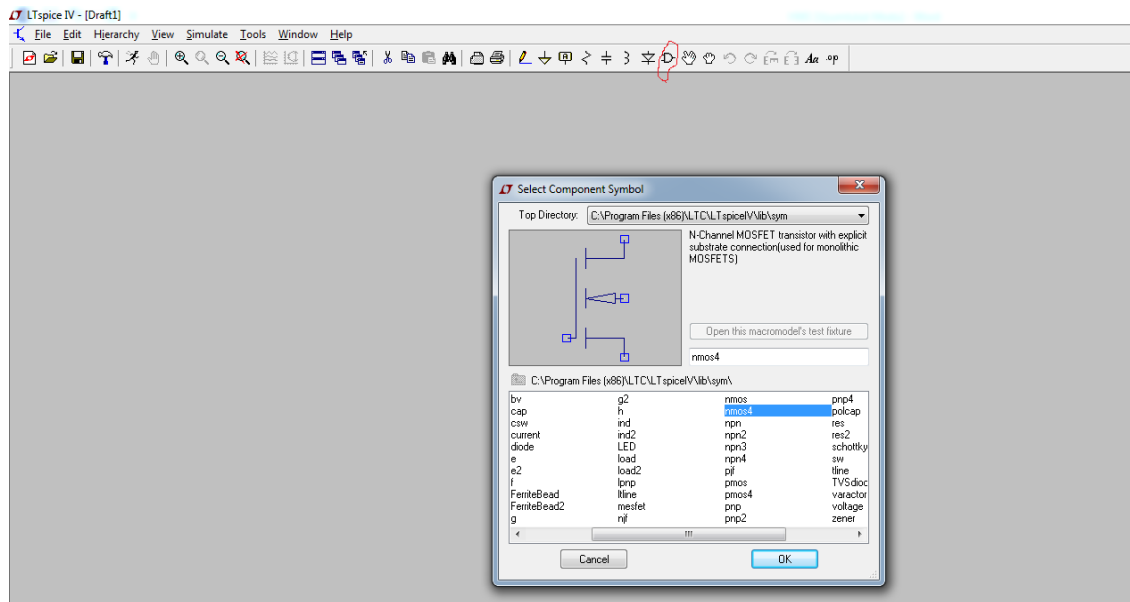
```
.MODEL T15DP PMOS LEVEL=3 PHI=0.7 TOX=9.5E-09 XJ=0.2U TPG=-1
+ VTO=-0.95 DELTA=2.5E-01 LD=7E-08 KP=4.8E-05
+ UO=130 THETA=2.0E-01 RSH=2.5E+00 GAMMA=0.52
+ NSUB=1.0E+17 NFS=6.50E+11 VMAX=3.0E+05 ETA=2.5E-02
+ KAPPA=8.0E+00 CGDO=3.5E-10 CGSO=3.5E-10
+ CGBO=4.5E-10 CJ=9.50E-04 MJ=0.5 CJSW=2E-10
+ MJSW=0.25 PB=1
```

In order to use the parameter sets, shown above, in Ltspice, please follow these steps:

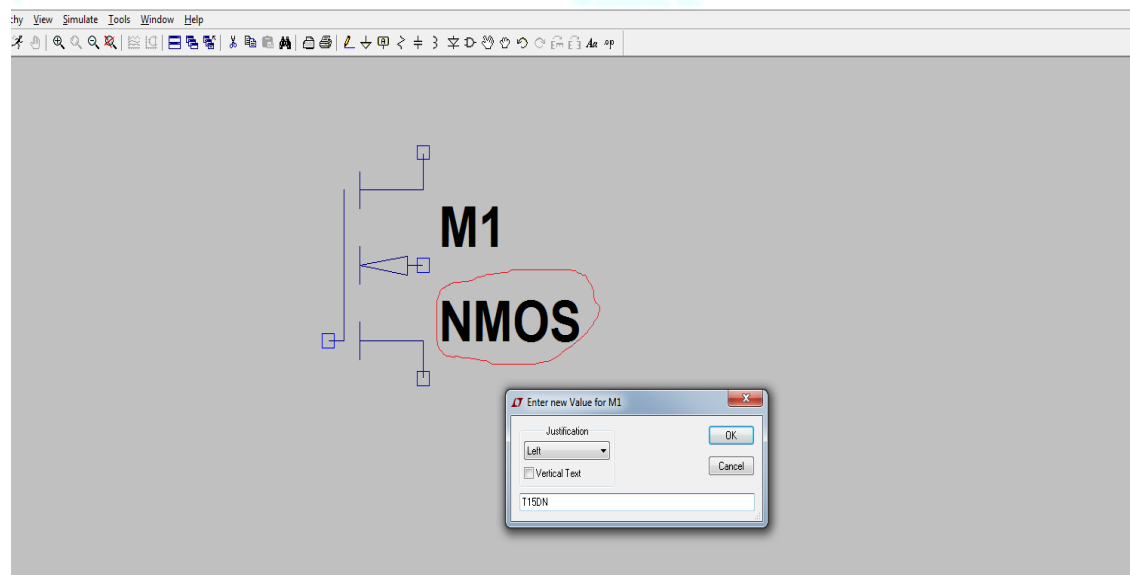
- 1- Create a .txt file named T15D_models.txt.
- 2- Copy parameters above and paste them into T15D_models.txt file.
- 3- Then place T15D_models.txt file into the LTspice folder (or the folder where your project saved).
- 4- After click on the .op tab as shown in figure below (circled red), write 'include T15D_models.txt' into the opened window. After pressing OK, you will see a rectangle information bar. Paste it somewhere in the schematic.



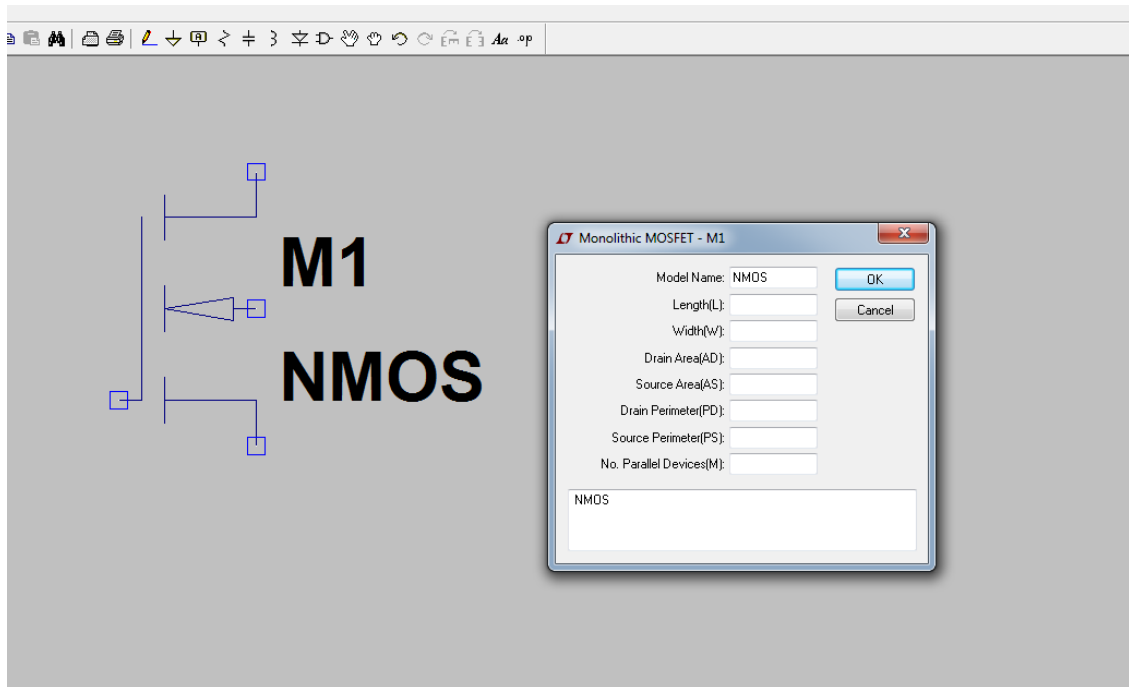
- 5- To add MOS transistors, click on the component tab shown below (circled red), then select NMOS4 and PMOS4.



- 6- After selecting the transistor, right click on the transistor; name it as T15DN for NMOS and T15DP for PMOS



- 7- To enter W and L parameter values, right click on the transistor and write W and L values.



Note: Your simulation results can be slightly different from hand calculations because of the probable mismatches between calculation and simulation parameters.