**Student Name:** **Instructor: Mustafa Altun**

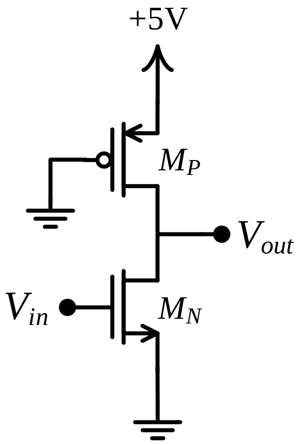
**Student ID:**

# EHB322E Digital Electronic Circuits
# Homework 1

Deadline: 06/03/2017 (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:* $k_p' = \mu_p c_{ox}$=48uA/V$^2$, $k_n' = \mu_n c_{ox}$=156uA/V$^2$, $V_{TN}$=0.7V, $V_{TP}$=-0.95V, $W_P$=3.2u, $L_P$=0.6u, $L_N$=0.6u.




Pseudo NMOS Inverter

- **a)** Find the minimum value of $\mathbf{W_N}$ to satisfy that $V_{out}$ = 0.2V when $V_{in}$= 5V applied.
- **b)** Find the switching threshold value of $\mathbf{V_M}$.
- **c)** Find the **static power** consumption of the inverter for $V_{in}$=0V and $V_{in}$=5V.
- **d)** Suppose that a load capacitor of 10pF is connected to the output. Find the value of the propagation delay $\mathbf{t_{PLH}}$.

**2) SIMULATION:** Construct the above circuit using SPICE. Connect body terminals of transistor to their source terminals. Select $W_P$=3.2u, $L_P$=0.6u, $L_N$=0.6u. 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 $\mathbf{W_N}$ to satisfy that $V_{OL}$ $V_{out}$ = 0.2V when $V_{in}$= 5V applied.
- **b)** Sketch voltage transfer curve of the inverter; find noise margin values of $\mathbf{NM_L}$ and $\mathbf{NM_H}$; find the switching threshold value of $\mathbf{V_M}$.
- **c)** Find the **static power** consumption of the inverter for $V_{in}$=0V and $V_{in}$=5V.
- **d)** Suppose that a load capacitor of 10pF is connected to the output. Find the value of the propagation delay $\mathbf{t_{PLH}}$. ,
- **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
```

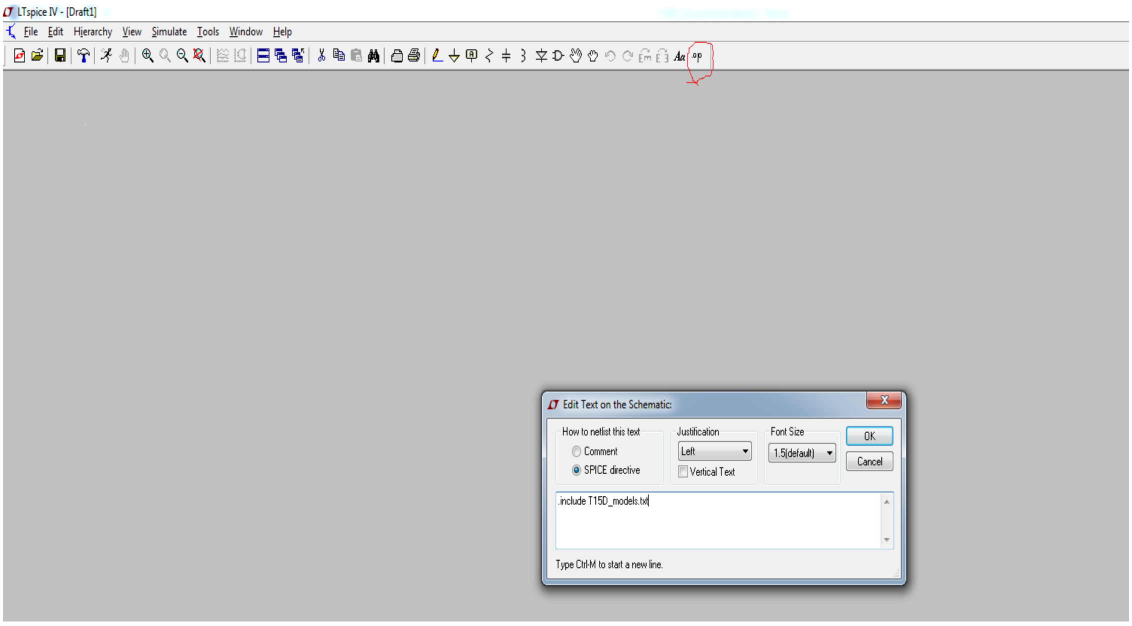
In order to use the parameter sets, shown above, in Ltspice, please follow these steps:

1- Create a .txt file named T15D_models.txt.

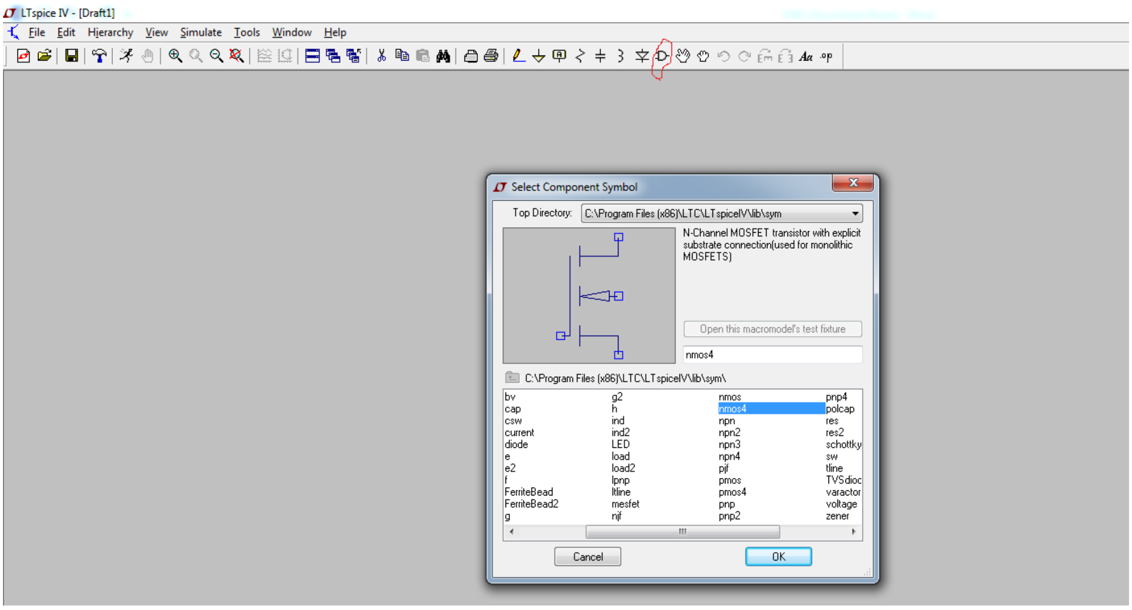
2- Copy parameters above and paste them into T15D_models.txt file.

3- Then place T15D_models.txt file into the LTspice folder (or the folder where your project saved).

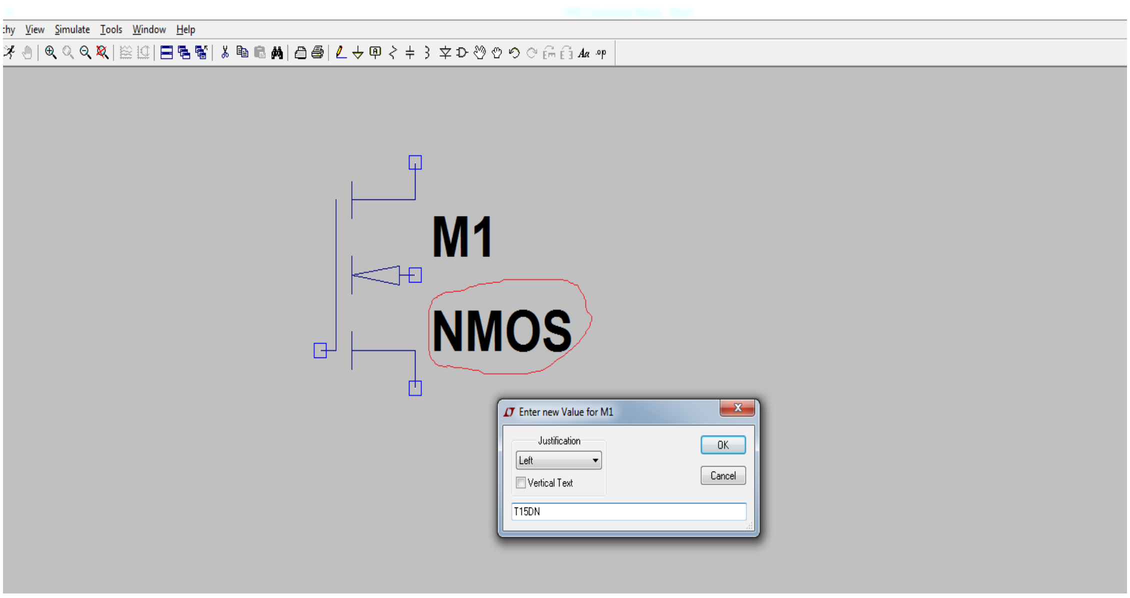
4- After click on the .op tab as shown in figure below (circled red), write '.include T15D_models.txt' into the opened window. After pressing OK, you will see a rectangle information bar. Paste it somewhere in the schematic.



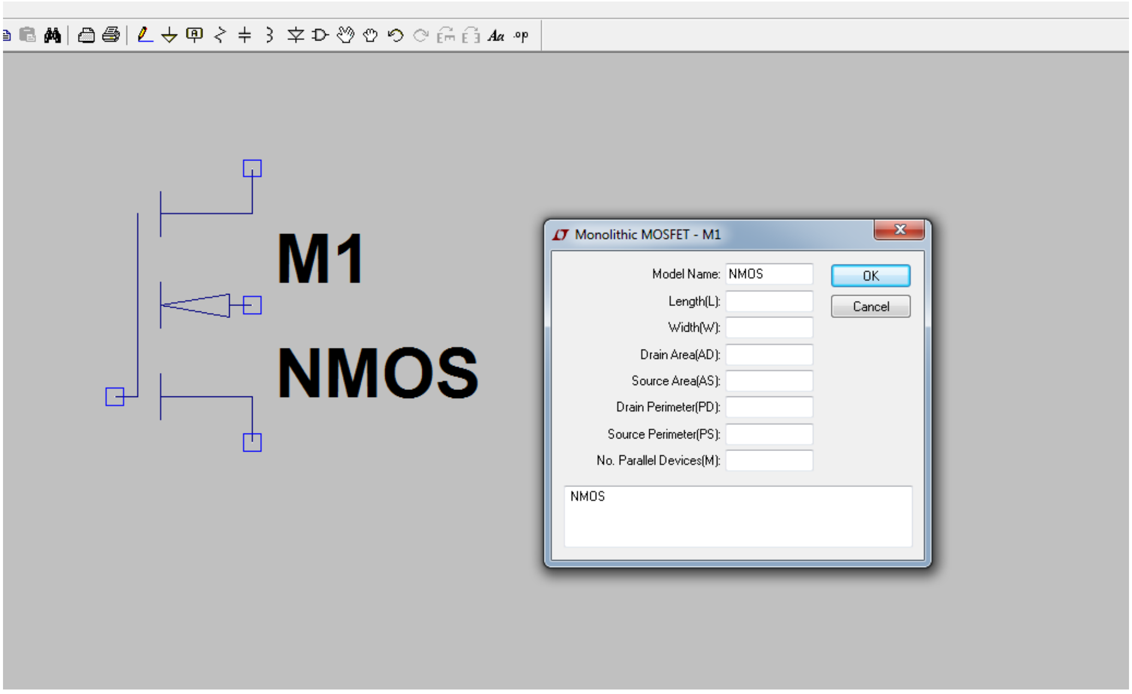

5- To add MOS transistors, click on the component tab shown below (circled red), then select NMOS4 and PMOS4.




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.