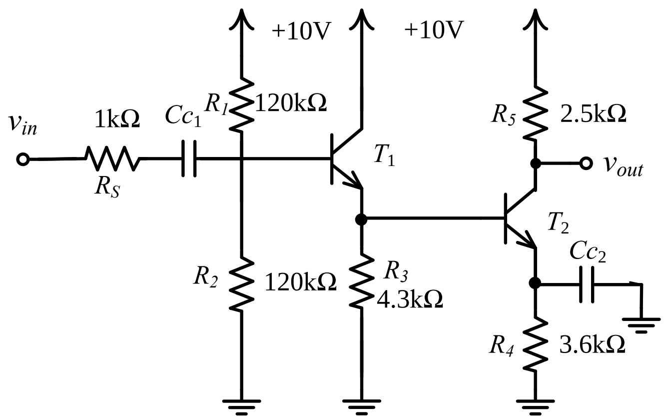
Student Name: **Instructor: Mustafa Altun**

Student ID:

EHB222E Introduction to Electronics
Homework 2

Deadline: 06/04/2015 (before the lecture)

1.



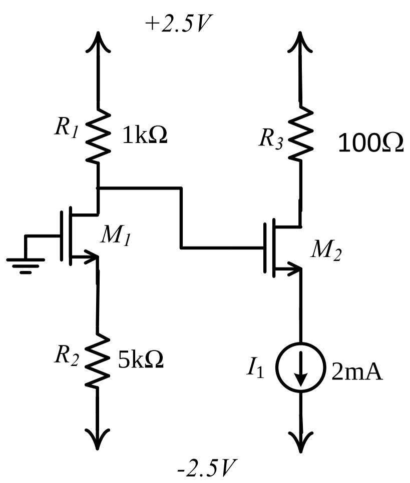
Two-stage BJT amplifier

Coupling capacitors C_{C1} and C_{C2} should take high values; you can select 1mF for both.
Transistor parameters : $\beta=300$, $V_{BE}=0.65V$, $V_T=25mV$, $V_A=100V$

a. Calculation: Carry out a DC analysis for the circuit . Find DC operating values for I_{C1}, I_{C2}, V_{CE1}, and V_{CE2}.

b. Simulation: Construct the above circuit using SPICE. Use the Fairchild 2N4124 model for the transistors. Find DC values of I_{B1}, I_{B2}, I_{C1}, I_{C2}, V_{in}, and V_{out} by performing bias point (DC operating point) analysis in SPICE. Compare the results with those calculated in part **1(a).** Do they match well? Justify your answer.

2.



Transistor parameters : $V_T=0.76V$, $k_n'=126uA/V^2$, $\lambda=0$ ($V_A=\infty$), $(W/L)_1=(W/L)_2=13u/2u$

a. Calculation: Carry out a DC analysis for the circuit . Find DC operating values for I_{D1}, I_{D2}, V_{DS1}, and V_{DS2}.

b. Simulation: Construct the above circuit using SPICE. Use the T15DN model for the transistors; connect body terminals of the transistors to their source terminals. For details of using LTspice check out the tutorial attached to the homework. Find DC values of I_{D1}, I_{D2}, V_{DS1}, and V_{DS2} by performing bias point (DC operating point) analysis in SPICE. Compare the results with those calculated in part **2(a).** Do they match well? Justify your answer.

Grading: 1(a)25 %, 1(b)25 %, 2(a)25 %, 2(b)25 %

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
+ TOX    = 1.4E-8         NSUB   = 1E17          GAMMA  = 0.5483559
+ PHI    = 0.7            VTO    = 0.7640855     DELTA  = 3.0541177
+ UO     = 662.6984452    ETA    = 3.162045E-6   THETA  = 0.1013999
+ KP     = 1.259355E-4    VMAX   = 1.442228E5    KAPPA  = 0.3
+ RSH    = 7.513418E-3    NFS    = 1E12          TPG    = 1
+ XJ     = 3E-7           LD     = 1E-13         WD     = 2.334779E-7
+ CGDO   = 2.15E-10       CGSO   = 2.15E-10      CGBO   = 1E-10
+ CJ     = 4.258447E-4    PB     = 0.9140376     MJ     = 0.435903
+ CJSW   = 3.147465E-10   MJSW   = 0.1977689     )
*
.MODEL T15DP PMOS (                       LEVEL  = 3
+ TOX    = 1.4E-8         NSUB   = 1E17          GAMMA  = 0.6243261
+ PHI    = 0.7            VTO    = -0.9444911    DELTA  = 0.1118368
+ UO     = 250            ETA    = 0             THETA  = 0.1633973
+ KP     = 3.924644E-5    VMAX   = 1E6           KAPPA  = 30.1015109
+ RSH    = 33.9672594     NFS    = 1E12          TPG    = -1
+ XJ     = 2E-7           LD     = 5E-13         WD     = 4.11531E-7
+ CGDO   = 2.34E-10       CGSO   = 2.34E-10      CGBO   = 1E-10
+ CJ     = 7.285722E-4    PB     = 0.96443       MJ     = 0.5
+ CJSW   = 2.955161E-10   MJSW   = 0.3184873     )
```

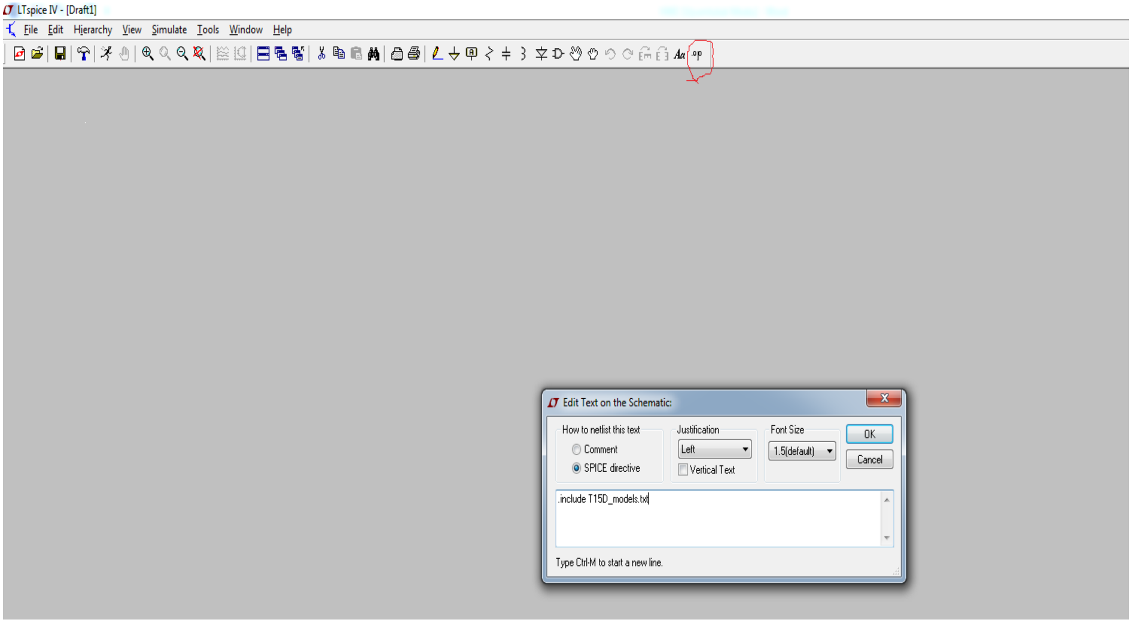
In order to use the parameter sets, shown above, in Ltspice, please follow these steps:

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

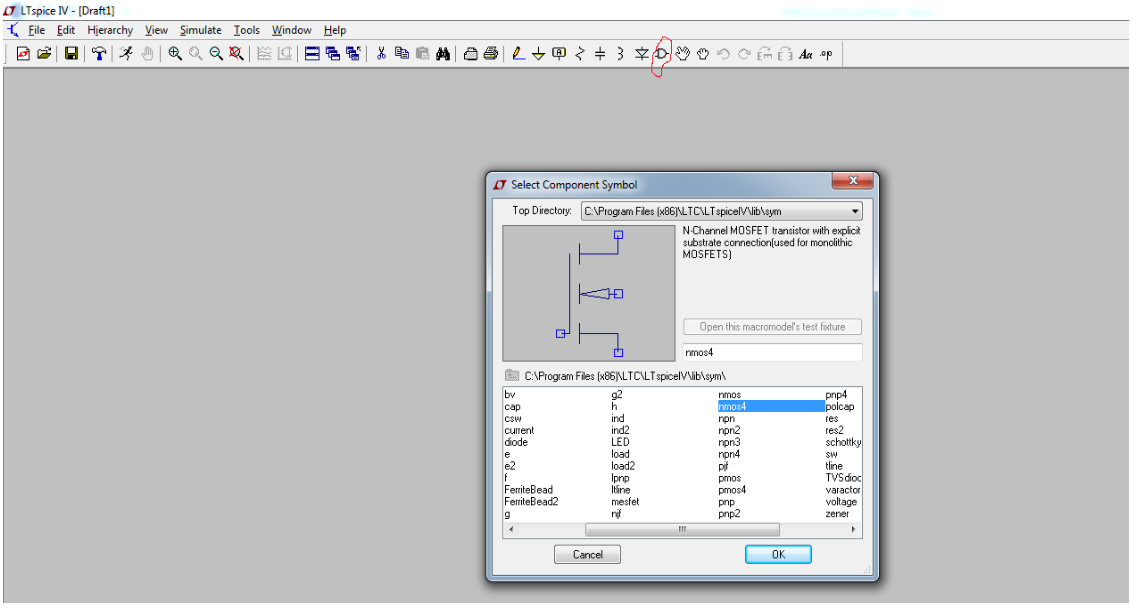
2- Copy parameters above and paste them into T15D_models.txt file.

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

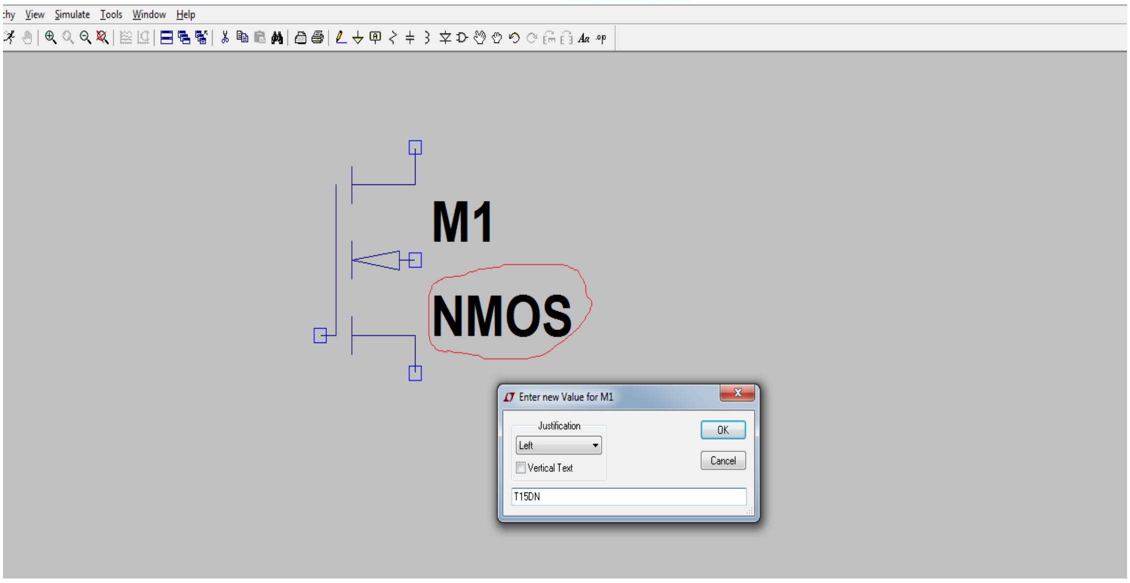
4- After click on the .op tab as shown in figure below (circled red), write '.include T15D_models.txt' into the opened window. After pressing OK, you will see a rectangle information bar. Paste it somewhere in the schematic.



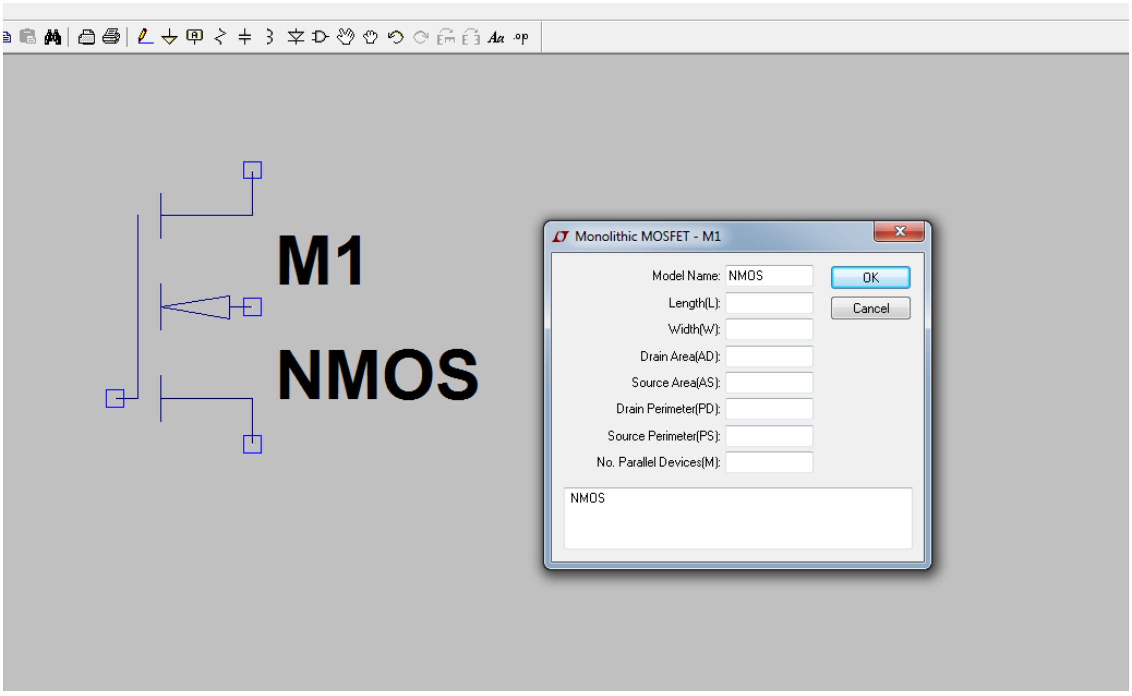
5- To add MOS transistors, click on the component tab shown below (circled red), then select NMOS4 and PMOS4.



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.