VE311 Electronic Circuit Homework 5

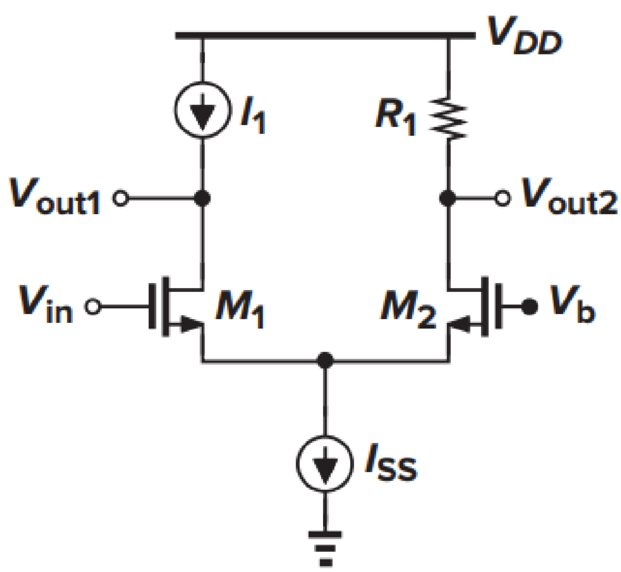
Due: **December 6th**

Note:

1) *Please use A4 size paper or page.*
2) *Please clearly state out your final result for each question.*
3) *Please attach the screenshot of Pspice simulation result if necessary.*

Question 1. Differential Amplifier Circuits 1

[20pts] For a differential amplifier circuits shown below, assume that I_1 and I_{SS} are ideal and λ, $\gamma > 0$, determine the value of V_{out1}/V_{in} and V_{out2}/V_{in}.



We apply small signal analysis to this question, for M_2 since we replace both I_1 and I_SS with open-circuit, V_{out2} would always be 0. Then $V_{out2}/V_{in} = 0$.

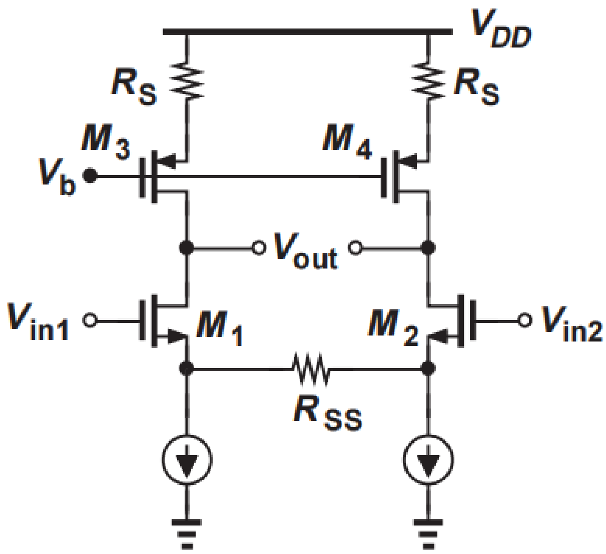
As for M_1, the following equation could be established using small signal analysis:

$$\frac{V_{out1}}{r_{o1}} + g_{m1}V_{in1} = 0$$

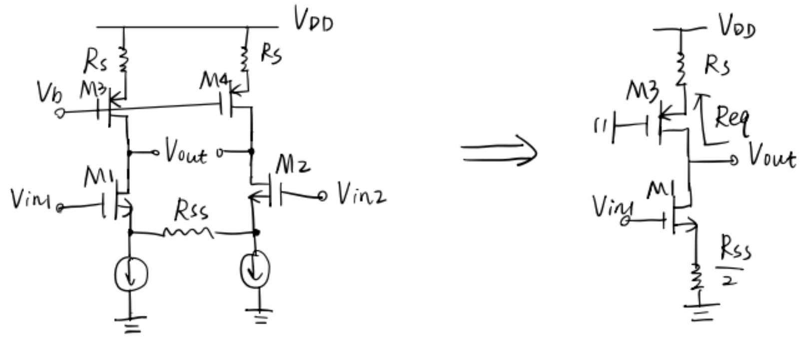
$$\frac{V_{out1}}{V_{in}} = -g_{m1}r_{o1}$$

Question 2. Differential Amplifier Circuits 2

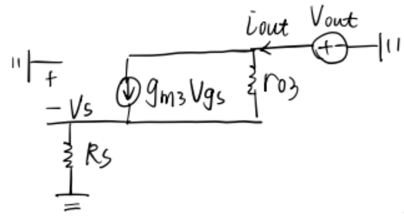
[40pts] Calculate the differential voltage gain of the circuits shown below. Assume perfect symmetry and $\lambda > 0$.You may need to compute the gain as $A_v = -G_m R_{out}$. Hint: use half circuit and R_{SS} can be divided into $\frac{R_{SS}}{2}$ in your half circuit.



First, apply half circuit method:



For Req:

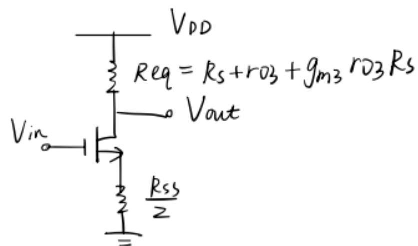


by KCL: $i_{out} = \frac{V_s}{R_s} = g_{m3}V_{gs} + \frac{V_{out} - V_s}{r_{o3}}$

by KVL: $V_{gs} = -V_s$

$$\Rightarrow R_{eq} = \frac{V_{out}}{i_{out}} = R_s + r_{o3} + g_{m3} r_{o3} R_s$$

$\therefore$ the circuit can be simpify as a CS stage with source degeneration:



$$\therefore R_{out} = (R_s + r_{o3} + g_{m3} r_{o3} R_s) \,||\, (\frac{R_{ss}}{2} + r_{o1} + g_{m1} r_{o1} \frac{R_{ss}}{2})$$

$$G_m = \frac{2g_{m1}}{R_{ss}} (\frac{1}{g_{m1}} \,||\, r_{o1} \,||\, \frac{R_{ss}}{2})$$

$$\therefore A_v = -G_m \cdot R_{out} = \frac{-2g_{m1}}{R_{ss}} (\frac{1}{g_{m1}} \,||\, r_{o1} \,||\, \frac{R_{ss}}{2}) \cdot (R_s + r_{o3} + g_{m3} r_{o3} R_s) \,||\, (\frac{R_{ss}}{2} + r_{o1} + g_{m1} r_{o1} \frac{R_{ss}}{2})$$

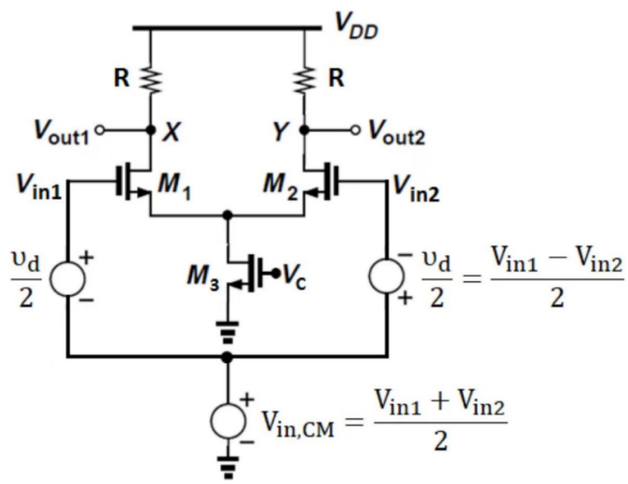
Question 3. Differential Amplifier with PSPICE Simulation

[40pts] For the circuit shown below, use NMOS SPICE model except for $\lambda = 0$, $\eta = 0$

(a) For $V_{DD} = 5V$, $V_{in,CM} = 2.2V$, $V_C = 1.2V$, $R = 5k\Omega$, $(\frac{W}{L})_1 = (\frac{W}{L})_2 = \frac{50\mu m}{2\mu m}$, $(\frac{W}{L})_3 = \frac{100\mu m}{2\mu m}$, what are the values for A_{DM} and A_{CM} ?

(b) Plot $(V_{out1} - V_{out2})$ as a function of $(V_{in1} - V_{in2})$ from -4 V to 4 V (i.e. $\frac{v_d}{2}$ from -2 V to 2 V). Confirm whether A_{DM} in (a) is consistent with the simulation result here. (Hint: To generate both $\frac{v_d}{2}$ and $-\frac{v_d}{2}$, you may need to use a Voltage Controlled Voltage Source(VCVS). You can find it in Pspice component)

(c) Plot $V_{out,CM}$ as a function of $V_{in,CM}$ from 0 V to 5 V, while $v_d = 0$. Confirm whether A_{CM} in (a) is consistent with the simulation result here.



NMOS Model

LEVEL = 1	VTO = 0.7	GAMMA = 0.45	PHI = 0.9
NSUB = 9e+14	LD = 0.08e−6	UO = 350	LAMBDA = 0.1
TOX = 9e−9	PB = 0.9	CJ = 0.56e−3	CJSW = 0.35e−11
MJ = 0.45	MJSW = 0.2	CGDO = 0.4e−9	JS = 1.0e−8

(a). Since M_3 could be treat as an ideal current source in this case,

$$I_{SS} = \frac{1}{2}\mu_n C_{ox}(\frac{W}{L_{eff}})_3(V_c - V_{th})^2 = 9.12 \times 10^{-4}A$$

Then,

$$A_{DM} = -g_{m1,2}R_D = -9.12, A_{CM} = 0$$

Simulation for (b) and (c) are shown below.