HSPICE Homework #3 of「類比積體電路導論」

作業繳交截止日期: **Oct. 17, 2024,18:00**
(上傳E3 數位平台繳交)

Fig.1 are the differential amplifier circuits with (a) passive load, (b) active load, (c) current-source load. Perform the HSPICE simulations using device parameters of 0.18um CMOS technology. Please use n_18_mm and p_18_mm. VDD = 3V, $(W/L)_{1,2}$ = 45/0.3(um/um), $(W/L)_{3,4}$ = 10/0.3(um/um), $(W/L)_5$ = 75/1(um/um), R_D = 5kΩ, Vbs=0.6V, Vb=2V, input common mode voltage V_{icm}=1V, temperature at 25°C and in TT corner. From the simulation results,

(1) Find the common mode gain Acm = Vx/V_{in+}, the differential mode gain Adm = $(Vx-Vy)/V_{in+}-V_{in-}$, and CMRR= Adm/ Acm in Fig.1(a), (b). You should use Hspice to simulate the result.

	Acm (V/V)	Adm (V/V)	CMRR (V/V)
(a)			
(b)			

(2) Please derive an appropriate sinewave amplitude (100 kHz) based on the maximum output swing and the gain obtained from the SPICE simulation. Explain the method to determine the maximum output swing. Then, use this sinewave amplitude as the input signal to plot the transient (30 μs) output signal. Check the operating point of each MOSFET to ensure it is in the saturation region.

(3) Compare the differences (DC gain, Rout, output swing) between the circuit in Fig.1(b) and Fig.1(c).

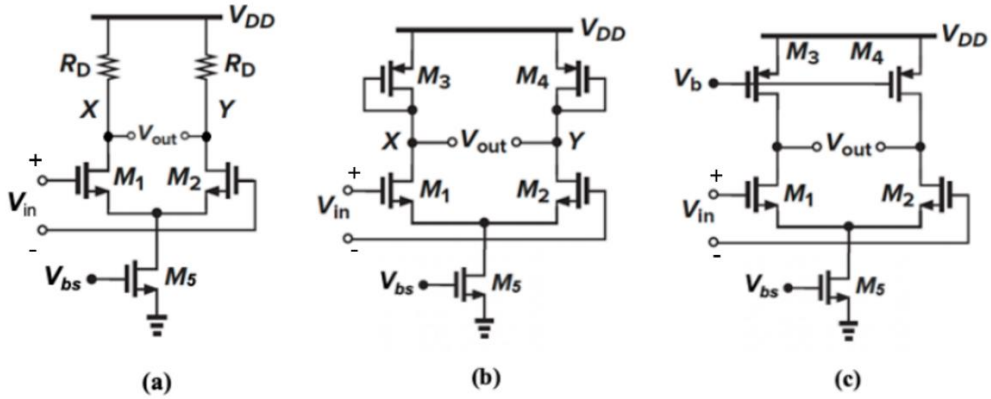

Figure 1

HSPICE_HW3 作業需要有以下幾點：

1. HSPICE Code (截圖，**不同電路請區分開來**)
2. Simulation Result (波形繳交背景請用白色，並且波形標示和呈現要清楚)
3. 每個小題的單獨說明

   問題(1) 模擬結果 Fig.1(a),(b)、表格

   問題(2) 波型(包含輸入與輸出)、說明 for Fig.1(a),(b),(c)

   問題(3) 說明
4. 以.pdf 的格式上傳
5. 檔案名稱用「Hspice_HW3_自己的學號」(例如：Hspice_HW3_0811541)，於作業繳交截止日期(發布後隔週禮拜四晚上 6 點)前，上傳到指定的 E3 數位平台繳交，遲交一周內打八折，一周後不計分。