

1. Use composer and hspice to simulate the common source and source follower as shown at Fig. 1 with  $V_{dd}=1.8V$ . Every one of MOS's  $c_{gtot}$ ,  $c_{stot}$ ,  $c_{dtot}$ , and  $c_{btot}$  cannot have more than 80fF under your design. (70%)
  - (a) Design a common source with gain  $A_1 > 120$  and plot the output noise's spectrum as shown at Fig. 1. (a). Identify the corner frequency and pole of thermal noise. (10%)
  - (b) Design a source follower with gain  $A_2 > 0.8$  and plot the output noise's spectrum as shown at Fig. 1. (b). Identify the corner frequency and pole of thermal noise. (10%)
  - (c) Compare (a) with (b). Which one has higher output flicker noise? Which one has higher thermal noise's pole? Why? (10%)
  - (d) Find the total output noise power of CS and SF by integrating the spectrum. Which one is bigger? Why? (10%)
  - (e) Find the total input referred noise power of CS and SF based on the results of (a), (b), and (d). Which one is bigger? Why? (10%)

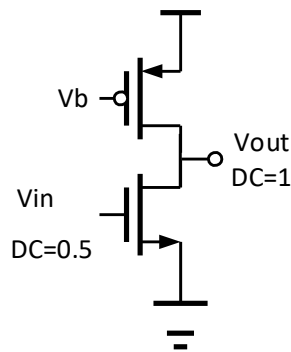


Fig. 1. (a)

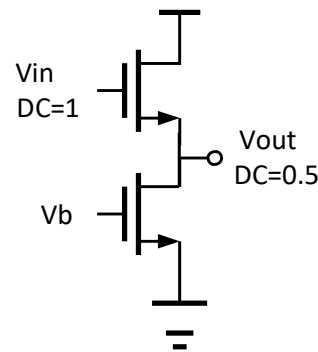


Fig. 1. (b)

- (f) Connect CS and SF as shown at Fig. 1. (c) and find the total input referred noise power. (You are allowed to "slightly" adjust the bias condition to make sure the overall gain equal to  $A_1 * A_2$ .) (5%)
- (g) Connect SF and CS as shown at Fig. 1. (d) and find the total input referred noise power. (You are allowed to "slightly" adjust the bias condition to make sure the overall gain equal to  $A_1 * A_2$ .) (5%)
- (h) Comment on what makes the difference between (f) and (g). (10%)



Fig. 1. (c)

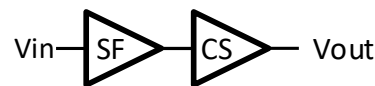


Fig. 1. (d)

**Before you start to simulate the .ac noise**

1. Find the title "Flicker and Thermal noise Model Parameters" in "cic018.l".
2. Use the "replace all" to change value of the parameter into  $KF=1E-29$ .
3. Hint: use ".noise V(vout) vin 10000"

2. Design a differential to single-ended amplifier as shown in Fig. 2 with  $V_{DD}=1.8V$ ,  $C_L=3p$  and use current mirror to generate  $V_b$ . (30%)
- (a) Design a differential to single-ended amplifier to get voltage gain  $A_v > 40dB$ . (10%)
- (b) Use the .op's parameters to calculate the pole of output node, mirrored node, and pseudo ground node. (10%) Use the .pz to check your calculation and simulate the bode plot. (10%)

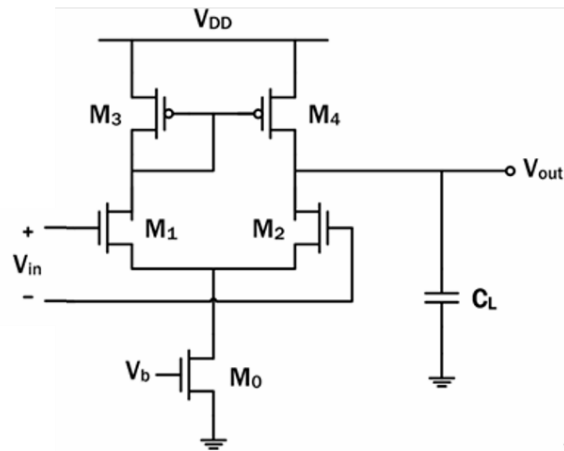


Fig. 2.

**The following should be included in your report (a) schematic (b) HSPICE netlist & simulation file (c) waveform with cursor values (d) comments.**