

Homework 6

Due: Thursday, 10/12/00

Problem 1: Textbook Problem 3.2. Derive A_v in terms of I_{bias} , W , and L . Use, from Example 3.2 pg. 139, $r_{on}=r_{ds-n}=8000L/I_{bias}$ and $r_{op}=r_{ds-p}=12000L/I_{bias}$. Comment on how bias current and transistor size affect overall gain.

Problem 2: Textbook Problem 3.6: Begin with the small signal model shown in Figure 3.7 and use the “test source” method to find R_{out} .

For the problems below, use the following SPICE MOS Model parameters and $V_{DD}=5V$ for all calculations and SPICE simulations. For all calculations, you may ignore channel length modulation effect (assume $\lambda=0$) and assume the LAMBDA in the SPICE model is sufficient for all these problems.

SPICE MOS Model Statements for $L=2\mu m$ transistors.

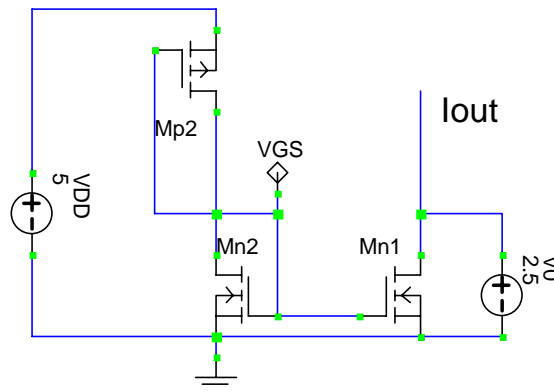
```
.model cmosn nmos level = 1 vto = 0.77 kp = 7.7e-5 gamma = 0.71 phi = 0.73
```

```
lambda = 0.0625 tox = 3e-8 ld = 2e-7 u0 = 670 nsub = 2e16
```

```
.model cmosp pmos level = 1 vto = -1.1 kp = 2.1e-5 gamma = 0.355 phi = 0.66
```

```
lambda = 0.053 tox = 3e-8 ld = 5e-8 u0 = 180 nsub = 5e15
```

Problem 3: Simple Current Mirror



a) The circuit shown here is a simple current mirror with a “diode connected” pMOS load acting as the reference current source. If $W=4\mu m$ and $L=2\mu m$ for all transistors, calculate $V_{GS1}=V_{GS2}$. Notice that $|V_{GS}|$ of the pMOS device will be $V_{DD}-V_{GS2}$. Ignore the output supply, VO , for parts a) and b).

b) What will the value of I_{out} be if $V_{DS1} = V_{DS2}$?

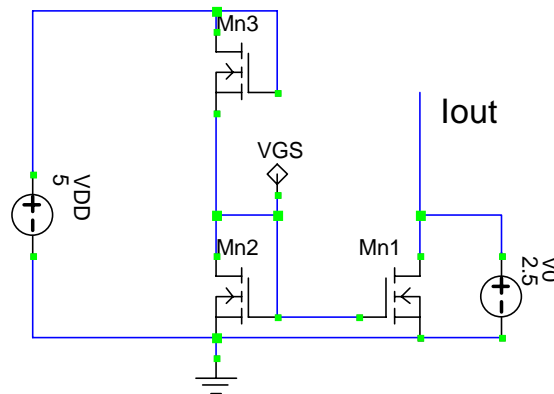
c) Verify your calculations with a SPICE simulation. Results should include standard SPICE report (schematic, netlist, and appropriate output file), with the value of $V_{GS1}=V_{GS2}$, $I(Mn2)$, and I_{out} clearly specified. (see SPICE notes below).

NOTE: To do this problem in SPICE you must have a load device connected to $Mn1$. Rather than connecting a resistor or transistor, simply connect a voltage supply to the drain of $Mn1$ and set the voltage to 2.5V.

d) Adjust the output voltage so that $V_{DS1}=V_{DS2}$, simulate again, and compare results. Comment on the change in the current I_{out} .

Problem 4:

Repeat Problem 1 after replacing the pMOS current reference load with a diode connected nMOS device, as shown below. Comment on the differences.



Problem 5:

a) Based on the relationships discovered in Problem 2, determine the W/L ratio needed for the current reference load (Mn3) to set the output current to $100\mu\text{A}$. Assume that $V_{DS1}=V_{DS2}$ for your calculations, and only change $(W/L)_3$.

*Note, you will need to reduce $V_{GS1}=V_{GS2}$ (relative to Problem 2) which you will do by making more of the voltage drop across Mn3. To do this, you will need to make Mn3 more “resistive” which means you will have to increase L of Mn3. In other words, if L_3 is larger than L_2 , then V_{GS3} must be larger than V_{GS2} since $I_{D2}=I_{D3}$.

*Use only integer values for W and L . For example, if you calculate you need $L=10.24\mu\text{m}$, then round the value to $10\mu\text{m}$. Circuit designers are limited to the resolution with which we can set W and L .

b) Verify your design in SPICE. Be sure to set the output voltage so that $V_{DS1}=V_{DS2}$. If your result is not close, check your calculations. Otherwise, you do not need to make further adjustments in SPICE to make $I_{out}=100\mu\text{A}$. Just report the value you obtained.

Notes on Using SPICE for this assignment:

1) SPICE Analysis Methods:

From the *Simulation / Set Up Simulations* menu in B2Spice, you can select from a variety of analysis to be performed on your circuit. Each are useful for different cases or to obtain different results. Here is some explanation. Also see the posted Using SPICE notes.

.OP: Operating Point Analysis. Will show a list of node voltages and power supply currents. Very useful for checking the bias conditions of your circuit. You must have this turned on to use the Show Op Point Values setting from the View menu.

.SHOW: Show Device Parameters. Will produce a list of the bias currents and voltages as well as small signal parameter values for each device in your circuit. You can get information for only select devices by typing the name of the device in the box after

clicking on the .SHOW option. This analysis is very useful for ensuring all of your devices are operating in the active/saturation region.

.TRAN: Transient Analysis. This will sweep the circuit with time as a variable. It is generally used in conjunction with a time-varying voltage or current source to measure the response of the circuit to this input.

.DC: DC Analysis. This will sweep the DC level of a chosen voltage or current source. This is generally done to measure the I-V characteristics of a device or circuit or to analyze the effect of altering bias source.

.AC: AC Analysis. This analysis will apply a small signal perturbation to all sources set to AC (by selecting the Use AC box in the properties window) and will sweep the frequency of the AC signal. This is the best method to perform small signal analysis to measure, for example, small signal gain.

2) Measuring Currents:

Currents are not a normal output from B2Spice. To measure them there are two options.

(A) –the recommended method- Use the .SHOW analysis (in Simulation Setup) to show the currents, voltages, etc. for each transistor. The current through the transistor will be shown as 'id', in units of amps.

(B) Place a 1Ω resistor in series with the transistor you want to measure and plot the voltage across this resistor. Since $V=IR$, the voltage will equal the current with $R=1\Omega$.

3) Troubleshooting Common Problems:

(A) **pMOS devices** have their source and bulk connected to the *highest* potential, rather than the lowest as with nMOS devices. You will need to “flip” (right-click device and select ‘flip vertical’) each pMOS device so that the source=bulk terminal is pointing up toward the VDD power supply.

(B) **Node Conflicts:** B2Spice is notorious for messing up node number; two nodes that are connected will have different node number and act as though they are not connected. You should always check for this problem if you are having any trouble. Go to *View / Show Node Numbers*

and the numbers will be displayed. You will also get an error message if there is a conflict. To eliminate conflicts, here are a few suggestions.

- 1) disconnect and re-wire the problem node(s).
- 2) place a marker on the node to force it to a specific value (use Set Node Index under Marker properties –right-click the Marker).
- 3) place a very small ($<1\Omega$) resistor between problem nodes to essentially short them.