Analog Integrated Circuit

Lab: II

Task 1-Current mirror circuit

a Start the Orcad Capture CIS program and create a new project named "Lab02". Draw the circuit shown in Fig. 2 with Capture. The circuit shows a simple current mirror consisting of a diode connected transistor M2 which is biased by a DC current source Iref. M1 and M2 have identical gate-source voltages. RL is the load resistor. Use the PSpice MOS model "MbreakN" from the BREAKOUT library for the MOS transistor and modify the name of transistor from "MbreakN" to "nmos_tsmc". Follow the instructions from previous lab to add the the "nmos_tsmc" model to the library. Parameter "M" is the multiplicity of the transistor and shows how many transistors are connected in parallel. In this circuit the value of the load resistor has been defined as a "variable" which can be swept (changed) using DC Sweep simulations.

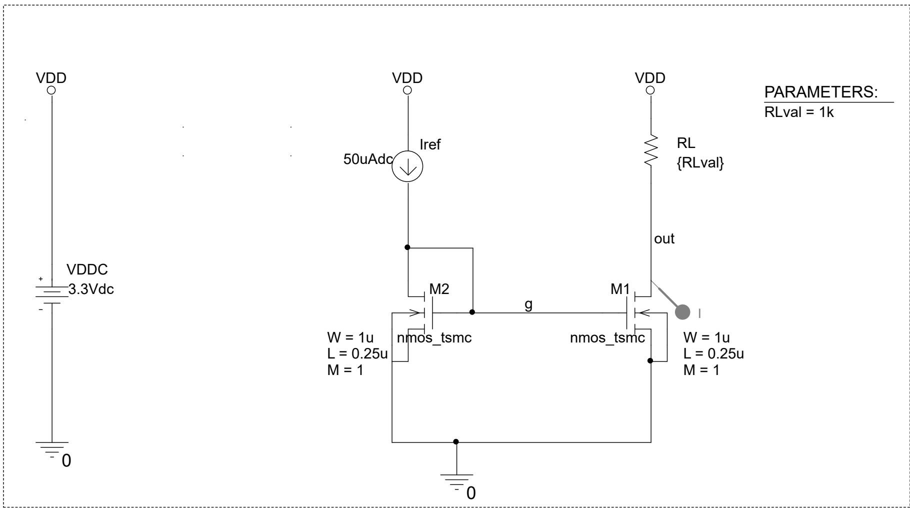

Figure 1: Simple current mirror circuit.

Please follow the instructions below to setup a parameterized sweep simulation in PSpice.

Parametric Sweep simulation

In order to do a sweep of the values of resistor RL, you need to change its value from a fixed number to a variable. In this circuit we have chosen the name RLval. Please note that the text RLval is enclosed in curly brackets .[1]

[1]Reference: engineering.purdue.edu

Now go to the Place Part menu, under the SPECIAL library, select a “part” called PARAM. If this library is not on your list, you will have to add it using the Add Library... button. It is called special.olb in the pspice folder. Now place this selected “part”(PARAM) near your “variable” load resistor. Double left-clicking on it, brings up the Property Editor. Left-click on the New Column button give the variable a name, in this case we are using the variable name RLval, and a value (one that is reasonable for your design). This value will be used as the default condition if you choose to do another type of simulation on this design. Use the value 2. Apply it and then highlight this new column. Left-click Display... and then activate the button labeled Name and Value. When you close out the [Property Editor] your schematic should look like the schematic shown in Fig. 2.

b Determine theoretically the maximum load resistance $R_{L,max}$ for which the current the load (I_L) is approximately equal to the reference current ($I_L \approx I_{Ref}$). Assume for simplicity that the channel length modulation can be neglected and use the transistor parameters (e.g. V_{th}) that you found in your previous lab.

c Perform a parametric sweep analysis to sweep the value of the load resistance in a range of 1 kOhm to a value larger than the $R_{L,max}$ and plot I_L versus R_L. Can you verify your theoretical prediction in the previous part?

d Modify the plot to draw I_L as a function of V_{out}. Using this plot, find out the voltage range of V_{out} where the transistor M1 is working in saturation. Compare this result with the result in part b.

Hint: In the probe window go to Plot→Axis Settings→ Select X-Axis tab→ Axis Variable... →Modify Trace Expression to V(out).

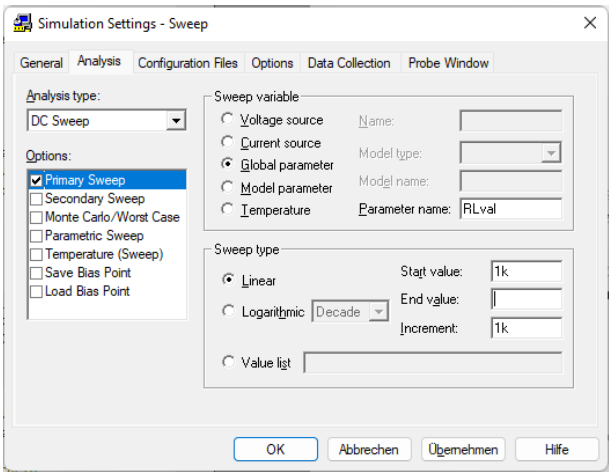

Figure 2: DC-Sweep analysis setup to sweep the value of the load resisttor RLval. Choose appropriate value for ”End value” as described in part c.

e Modify the circuit as shown in Fig. 3. Vt is an AC voltage source. Please note that apart from the AC amplitude, which is only relevant while an AC analysis is performed, the DC value of this voltage source should be specified (The default value is 0V). Set the AC amplitude equal to 1.0V and the DC value equal to 1.5V.

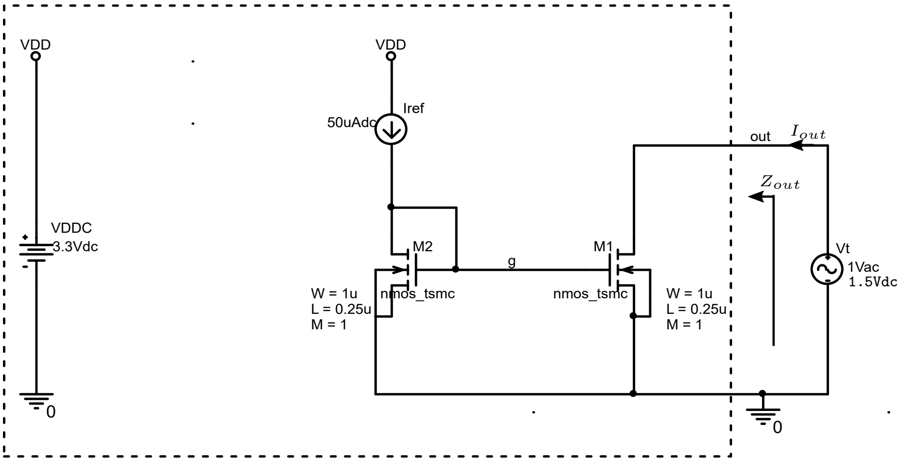

Figure 3: Simulation setup for systematic analysis of the current source. Looking into the node out and ground, the whole circuit consisting of M1, M2, and Iref and VDD can be in small signal replaced by an impudence $Z_{out}(j\omega)$ which at lower frequencies can be approximated by the DC small signal resistor $Z_{out}(0) = R_{out}$). Why?

f Use a DC Sweep analysis to sweep the the DC value of Vt ($V_{out,DC}$) from 0 to 3V and plot $I_{out,DC}$ versus $V_{out,DC}$. Save this plot as a data file "sweep-simple-currentmirror.dat" for later comparison in Task2.
Hint: In probe window go to File → Save as →Choose a suitable name e.g. ".dat" is a PSpice data file which can be used to reload simulation data and append it to an existing simulation results afterwards.

Use the derivative function in PSpice to plot the $\frac{dI_{out,DC}}{dV_{out,DC}}$. The small signal DC resistance for any arbitrary DC operating in the range $0V \leq V_{out,DC} \leq 3V$ point is given as:

$$R_{out} = \left(\frac{dI_{out,DC}}{dV_{out,DC}}\right)^{-1}$$

Determine the minimum DC voltage $V_{out,DC}$ for which the circuit works a a "current source" (i.e.g. the current remains almost constant, or equivalently the slope of the curve is "small". Argument why in this range M1 should work in saturation. Determine R_{out} for $V_{out,DC} = 1.5V$ and $V_{out,DC} = 0.1.5V$. Explain why the value of R_{out} for these two output voltage values is considerably different.

g Now perform an AC analysis to plot the magnitude and phase of the impedance for the small signal circuit looking into node out and ground as a function of frequency $Z_{out}(j\omega)$. Determine the small signal DC impedance R_{out} approximately.

Hint: The impedance is defined as $Z_{out}(j\omega) = \frac{V_{out,AC}}{I_{out,AC}}$. Run the AC-Analysis simulation and plot two traces with the following expressions ABS(-1/I(Vt)) and P(-1/I(Vt)).
Justify why this expressions can be used to find out the magnitude and phase of $Z_{out}(j\omega)$! Explain why do we need a minus sign (-) in the expressions. To determine R_{out}, one can approximately determine the magnitude of the impudence at a very "low" frequency (e.g at 1Hz): $R_{out} \approx Z_{out}(j2\pi \times 1Hz)$.

h Now set the DC value of Vt to 0.1.5V ($V_{out,DC} = 0.1.5V$) and determine R_{out}.

i compare the values you determined for R_{out} in part g (for $V_{out,DC} = 1.0V$) and h ($V_{out,DC} = 0.1.5V$) , with the results of part f.

j R_{out} can also be found using a DC-Bias point analysis (see also for more details Project Part I, page 5, parts l and m). Setup a DC bias Analysis as shown in Fig. 4 and run the simulation. Please note that in this setup we treat the circuit as an amplifier with an input voltage Vt and output voltage defined at node out (V(out)).

After running the simulation, the PSpice A/D Demo window opens. In the upper toolbar click in "View" → "Output File" to view output simulation file.
Scroll to the bottom of the output file to find the results of the small-signal analysis below the line `**** SMALL-SIGNAL CHARACTERISTICS`. The value of "INPUT RESISTANCE AT V_Vt =" corresponds to R_{out}.[2]

k Draw the small signal equivalent circuit of the whole circuit and calculate the small signal resistance of the circuit. Use the for small signal parameters of the transistors obtained in the DC-Bias simulation.

[2] In this simulation, we are only interested in the input impedance of this "amplifier" with corresponds to R_{out}. Justify the other calculated parameters gain and output impedance are consistent.

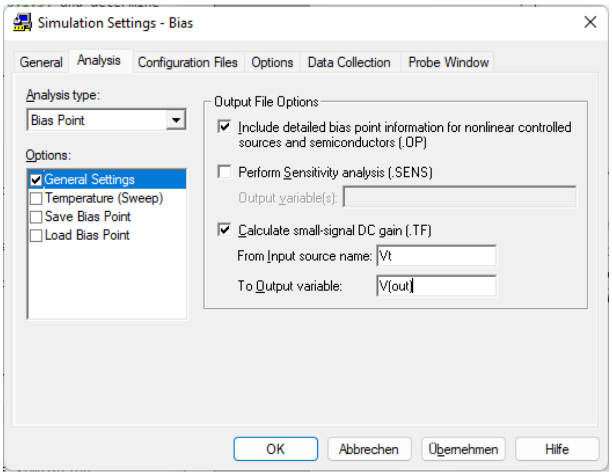

Figure 4: DC-Bias point simulation setup to determine the DC-operating point, small signal parameter, and determine R_{out}.

Task 2-Cascode current mirror

a In order to improve R_{out} one can use a "cascode" current source topology as shown in Fig. 5. Create a new schematic in the current PSpice project and draw the schematic as shown in Fig. 5.

b Similar to Task 1-f, perform a DC Sweep analysis to sweep the the DC value of Vt ($V_{out,DC}$) from 0 to 3V and plot $I_{out,DC}$ versus $V_{out,DC}$. Determine the minimum DC voltage $V_{out,DC,min}$ for which the circuit works a a "current source" (i.g. the current remains almost constant, or equivalently the slope of the curve is "small". Argument why in this range M1 and M3 should work in saturation.

c Append the $I_{out,DC}$ versus $V_{out,DC}$ for the simple current mirror to the simulation results of cascode current mirror. Discuss the major differences between these two curves.

Hint: To append a plot from another simulation results go to File → Append Waveform(.DAT)...→Browse and then select the data file you want to append to the graph (in this case "sweep-simple-currentmirror.dat").

d Perform a DC-Bias simulation, as described in Taks 1-f to determine R_{out} at $V_{out,DC} = 1.5V$ and compare the results with the results of the simple current mirror.

e Draw the small signal equivalent circuit of the whole circuit and calculate the small signal resistance of the circuit. Use the for small signal parameters of the transistors obtained in the DC-Bias simulation.

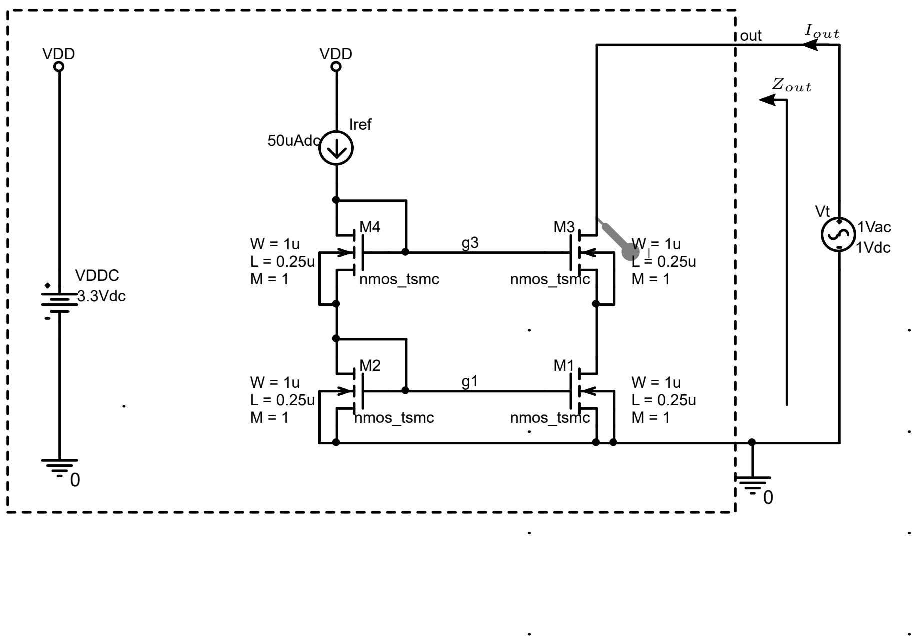

Figure 5: Simulation setup for systematic analysis of the *cascode* current mirror circuit.