Engineering 1620 – Spring 2020

SPICE Assignment – Design a Two-Stage MOSFET Opamp

Requirements: Design a simple operational amplifier for use within an integrated circuit, estimate its low-frequency performance, and then compare your hand calculations to the response calculated for the circuit using SPICE. Such amplifiers for on-chip filters and other mixed-signal applications often use a two-stage topology similar to the example I gave in class. The first stage is a differential pair with a current mirror load and is biased by a suitable current source. The second stage is a common source amplifier with a current source load. When wider bandwidth or higher gain is needed, cascoding the current sources or even the differential pair is used.

This scheme works well when the load on the amplifier is limited to a high resistance and low total capacitance from on-chip connections. It does not work very well for off-chip connections since the capacitances associated with such connections are usually tens of picofarads and severely limit the frequency response. Worse, off-chip loads vary with the user application and a larger-than-intended capacitive load can cause an operational amplifier to become unstable and too low a resistance will kill the amplifier gain. On-chip loads are better defined. Recognizing this reality, this project is strictly designed for on-chip applications and has a well-defined range of impedance for the load.

The design goal is an amplifier that operates with and is simulated with an output load of 30K in parallel with 20pf. The resistor load is tied to 2.5 volts DC to simulate operation in the middle of the output voltage range. V_{DD} is 5.0 volts. Figure 2 shows a schematic of this dummy load. The amplifier also needs to operate properly with the capacitive part of the load reduced to 2pf. With either of these loads, the principal properties of the amplifier should meet the requirements:

- DC Gain ≥ 80 DB (X10,000)
- Phase margin at 0 DB (unity) gain ≥ 60 degrees
- The gain-bandwidth product should be as high as possible with the proper phase margin
- Power supply current $I_{DD} \leq 0.9$ mA
- Common mode rejection ratio at low frequency is > 80 DB
- Output voltage range no less than 2.5 ± 1.5 volts
- Input common mode voltage range 0 to 3 volts minimum
- Slew rate $\geq 4 \cdot 10^6$ V/sec
- Dominant pole position as high as practical given the phase margin requirement, meaning that you are trying to maximize the gain-bandwidth product.
- The amplifier still meets these specifications if the load capacitance is reduced to 2 pf.

SPICE Issues: You are designing an opamp to be built in a manufacturing process that can only make transistors with gate lengths, L, as short as 0.5 microns, a very old (ca. 1995) process. (One reason for so old a process is that the SPICE version you are using only supports simple models (SPICE Level 3) for monolithic MOSFETs.) Short devices have low values of output resistance and you will need to choose larger values for L.

To use the transistor models for this process, you will need to download a set of parameters from the class website. The models are fairly rudimentary and only apply to devices with at least 0.5 micron gate lengths. They are on the site under Course Materials/.LIB File for 0.5 um MOSFET SPICE Models. Save that page as a text file with a ".LIB" extension. Add that library to your deck with the SPICE command ".LIB "<Full Path and Name to Models>".

You will probably draw your circuit in LTSpice. In doing so, use the symbols for NMOS4 and PMOS4 as these have the parameters for length, width, number of parallel devices, etc. This software is intended for customers of Analog Devices and is oriented to power MOSFETs that use different models. NMOS and PMOS symbols are set up to use the appropriate power device models.

The Design Procedure: Some of the process parameters for hand calculations are given in the table below but calculations for gm and r_o are a little more complicated than what we did in class and are discussed in the Appendix.

The process parameters for approximate hand calculations are:

Parameter	N-Channel	P-Channel
V_{TH0}	0.70 volts	0.95 volts
$K = \mu C_{OX}/2$	75 $\mu A/V^2$	26 $\mu A/V^2$
C_{OX} (fd/m^2)	$3.6 \cdot 10^{-3}$	$3.6 \cdot 10^{-3}$
Channel length modulation constant: $\lambda = \frac{1}{V_A}$ See Appendix I as this parameter depends on L and Vov		

In designing your circuit you may use:

- Any number of transistors with your choice of device length and width. (Note: while the process can make devices with L = 0.5 um, these have low voltage gain and are used for digital designs. You will want to use longer ones.) Because of manufacturing constraints, all values of L and W must be integer multiples of 0.25 um.
- Your first stage differential amplifier transistor pair are to be PMOS devices and the manufacturing process is N-well so that pair can have their body terminals tied to the source terminals.
- You may use one ideal DC current source of 50 microamperes to bias your circuit.
- You may use one discrete capacitor with value $\leq 12 pf$ for frequency compensation.
- You may use a discrete resistor ≤ 1 K in series with the compensation capacitor to improve your phase margin and bandwidth.

Implicit in these requirements is the assumption that you will give the amplifier a dominant pole by exploiting the Miller effect with a capacitor and resistor in series between the output and input of the common source second stage. The requirement for the best possible bandwidth with a

minimum phase margin of 60 degrees means you will have to experiment with the resistor and capacitor values to adjust the frequency response.

Simulations to characterize the completed design are done with the input common mode voltage = 2.5 volts. The minimum set of such simulations is:

- .OP for quiescent conditions (you may or may not have to adjust the offset voltage to get the circuit within the quiescent output voltage range of 2.5 +/- .4 volts.)
- Bode plots (.AC frequency response with gain and phase) for full load (20pf) and light load (2pf) of:
    - Small-signal frequency response of both stages to a differential signal
    - Small-signal common mode response
- A .DC sweep over the differential input (VOFFSET in the figure) to show the output voltage range of the circuit.

Optional Simulations:

- Test the input common mode range by a .DC sweep of the VCMDC source from 0 to 3.0 volts checking that the output voltage of the first stage remains within 0.1 volts of its initial value. (This is a very liberal specification. My design held the change in output DC voltage of the second stage to under 0.7 volts.)
- Connect the opamp as a unity gain buffer and drive its input with a step function of rise time = 10 ns from 1.5 volts to 2.5 volts at the input. What is the rise time of the step and the maximum of $\frac{dv_{OUT}}{dt}$?

Figure 1 shows the very simplest possible input circuit and a way of using sources as stimulus for both common mode and differential excitation. VDELTA is the AC source for open-loop frequency response simulation. VDCCM sets the DC common mode voltage and VCM is an AC source used to determine the common mode gain and rejection ratio by simulating a 1.0 volt AC common mode input. By default, the DCCM level should be 2.5 volts during simulation of the open-loop differential gain and the common mode gain. VDELTA and VCM should never both be turned on at the same time.

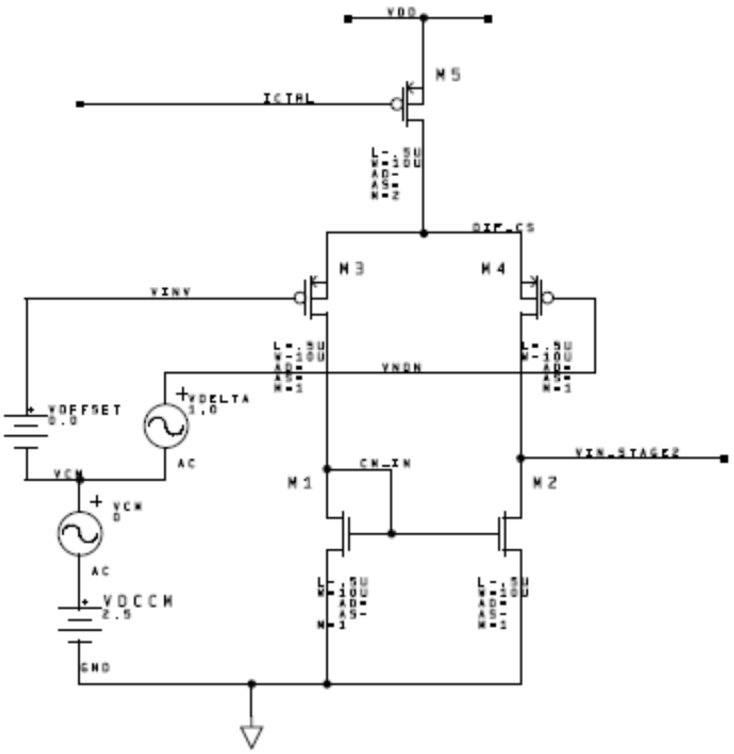


Figure 1: Simple Differential First Stage with Sources for SPICE Simulations. VCM set to 1.0 volts simulates the small signal common mode measurement. VDELTA set to 1.0 volts simulates the small signal differential open-loop response.

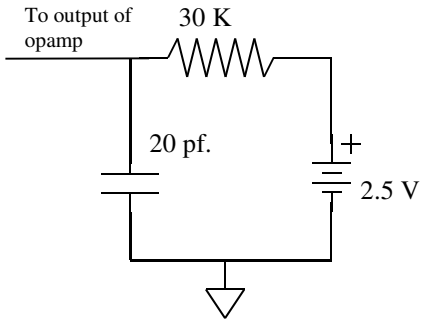


Figure 2: Dummy Load Simulates the Operating Load for Testing Opamp

Your report should include:

1. A very brief introduction describing the circuit topology and whether you used any cascoding to improve gain or bandwidth. Emphasize how much the circuit exceeds minimum requirements.
2. A clear schematic on white background annotated with the L and W values of all devices. In lieu of the schematic, you may submit the .asc file from LTSpice.
3. A table of simulated quiescent conditions for VGS, VDS, Vov, and I_D for all transistors. Compare these to your target values of I_D and Vov. You infer the simulated Vov from the simulation of VGS and the process data given above. The VDS and VGS come from LTSpice through a .op run. You probably have to delete the .AC command in order for LTSpice to display the .op data. You can save the .op data to a file to make this simple.
4. Do hand calculations of the low frequency gain of the differential first stage, the overall gain, and the CMRR. Compare these with the SPICE results. (For CMRR see Appendix II.) Do the same for slew rate. Show enough work including relevant formulas that it is clear how you did the calculation. A table of side by side values is a nice way to present that data.
5. A final table showing the properties of the finished amplifier including the measured gains (again), the slew rate, dominant pole frequency, unity gain frequency, the phase margin at unity gain, and the range of voltage over which the output follows the input without saturation or clipping. You may use your design value for the case of your actual

value of the compensation capacitor in lieu of simulating it in SPICE if you choose not to do the optional simulation.

Procedure Details:

1. You will concentrate first on sizing the transistors and biasing the circuit. Start by selecting lengths for your transistors. While all transistors that must match each other are the same length, not all transistors need to be the same. Lengths must be a multiple of 0.25 microns because of manufacturing constraints. Output resistance (r_O) depends on the channel length because its primary cause is channel length modulation. The change in length due to a change in VDS decreases as a fraction of the total length when the device is longer. The transconductance also degrades for very short devices. Unfortunately LT Spice does not report its calculated small signal device parameters so I have tabulated some data to make it easier for you to select L and to estimate g_m and r_O . The Appendix has a graph of how the maximum voltage gain for a common source amplifier varies with the channel length. The gains on that graph are ideal, that is, it assumes that the load on the common source device is a perfect current source. Your output circuit has the load resistor (30K) as well as the r_O of its current source transistor. This means your stages are likely to give gains about ½ to ¼ of the ideal values. You need gains of at least 100 in each stage and that suggests a range of L from 1.25 to 2.0 um is probably optimal.

2. Next select device overvoltages. The device "overvoltage" is defined as $V_{OV} \equiv V_{GS} - V_{TH}$. This term appears directly in the drain current equation and is also the approximate V_{DSAT} of the transistor. You choose the initial overvoltages based on two rules of thumb. There is a generally accepted range for the overvoltage from -0.05 volts to about .3 volts that represents reasonable tradeoffs between noise, gain, and frequency response. (A negative overvoltage represents subthreshold operation. Subthreshold operation of the differential pair is fairly common to improve gain but it requires very wide transistors.) Lower overvoltages raise gains but result in larger transistors. I am not convinced that LTSpice is computing the near threshold performance correctly so use V_{OV} a little above threshold for your PMOS differential pair. Low V_{OV} in the input transistors lowers noise, but it results in higher noise if used in the current mirror. So the second rule of thumb is that the current mirror overvoltage should be approximately twice that of the differential pair.

3. Make the drain currents of the current sources be integer multiples of the 50 microampere biasing source. (See Hints and Comments below for how to do this precisely in SPICE.) In the choice of quiescent drain currents and the size of the compensation capacitance, C_M, there is a complex tradeoff between slew rate (determines first stage bias), phase margin (limits values of C_M) and step response (determines second stage bias). I did not get a chance to talk about phase margin this year (2020), so I want to simplify bias selection. I have played with the choices and think the compensation capacitor will probably be 6 pf < C_M < 12 pf. Select the first stage bias to assure somewhat faster slew rate than required with the largest likely C_M. (You never design to a specification limit as you don't want process variation to put product out of spec.) The second stage current will be determined by the need to drive the 30 K load resistor (2.25 volt swing, 30 K requires 75

uA) and the sum of the 20 pf load capacitor and, C_M, the Miller capacitor at the full slew rate. Calculate the minimum stage current required assuming the minimum value for C_M and the slew rate that would give with the initial choice of first stage current. The real stage current must be somewhat bigger that this required drive current so that the common source transistor does not turn off as the amplifier slews in the positive direction. Make that current source at least 10 % higher current than the minimum. The basic formula for drain current then sets the widths of the devices.

The total number of individual devices and currents you need to choose is not very big. There are only three distinct devices – one for the PMOS current sources, one for the differential pair, and one for the NMOS mirror. To make the circuit balance with zero volts input, the current mirror and the common source device must have the same overvoltage. The easiest way to do that is to make the common source amplifier out of a number of copies of the mirror transistor. Similarly, there are only two indeterminate currents, the first stage tail current and the output stage bias.

4. If the simulated low frequency gain is sufficient, then set the resistance in series with the compensation capacitor to a trial value 1.5 times the inverse of the transconductance of the second-stage, common source amplifier device. Set a trial value of compensation capacitance and iterate the compensation in SPICE to optimize the gain-bandwidth with unity gain phase margin (60 degrees minimum).

5. Tabulate all the required simulations and consider doing the optional inverter test. You spent effort on assuring a minimum slew rate by design and this is the only test of whether you succeeded.

Hints and comments:

- You need devices with varying widths for the different currents in each part of your circuit. You make wide transistors by paralleling several copies of a small transistor so that current ratios are precise. SPICE allows you to do this with a single symbol by attaching the property “m=10”to the symbol to mean 10 transistors of that size in parallel. Use m instead of proliferating transistor symbols to create large devices.

- Be careful about suffixes on numerical values. All lengths and areas in SPICE are in meters or square meters so to specify microns you need the suffix U and for areas you usually need P after a product in square microns. You get crazy results if you forget.

- In the figure showing the use of sources for signals in simulation, the DC voltage VOFFSET can be used to sweep the input range in 50 uV steps to see the output range of the amplifier. It can also be set to a small value to offset any mismatches in the circuit that result in an output much different from 2.5 volts during small signal simulation.

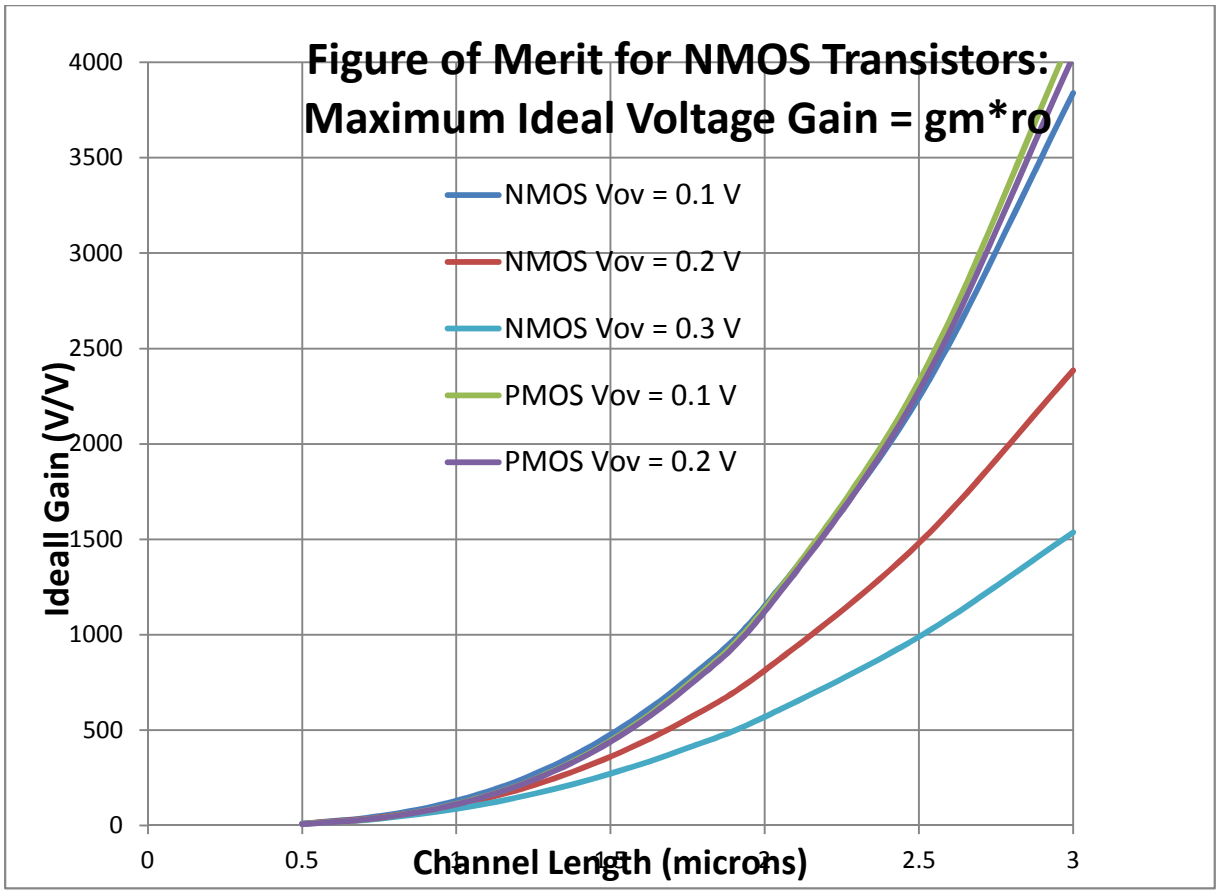
- The phase margin is simple to extract in LTSpice. Put an AC source into the inverting input of your circuit. Insert a .AC command line with frequency running 1 Hz to 100 MHz, magnitude 1 volt, and phase = 0. On the AC simulation graphs, set a cursor on the output

voltage and move it until the magnitude is 0 dB. The phase at that point is the phase margin. There is a PowerPoint file on the class website that describes why the phase margin is important.

Appendix I: Selecting Device Length and Estimating gm and r_o

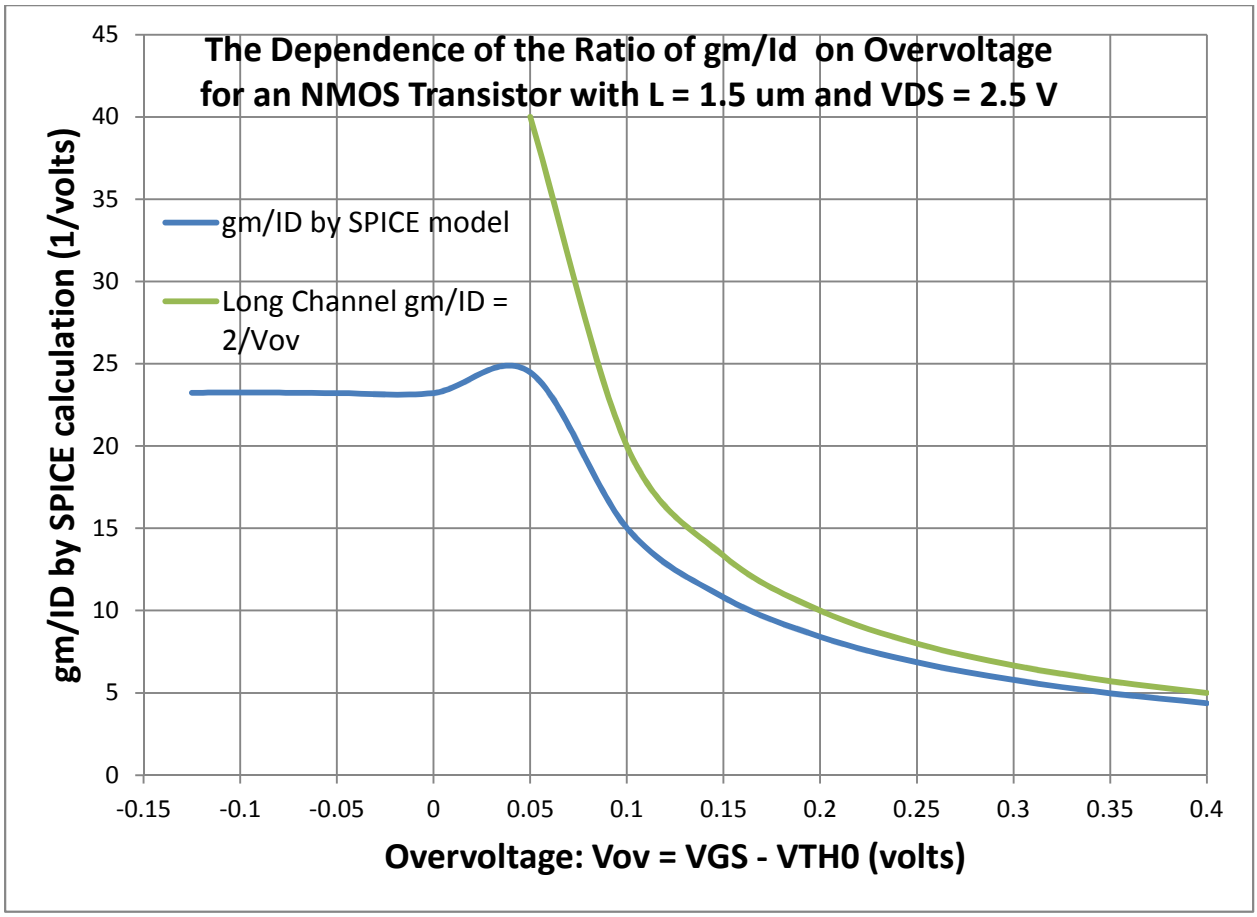
In circuits with no discrete resistors, the voltage gain is largely determined by transistor output resistances. That resistance is caused by channel length modulation and the change in length due to a change in VDS decreases as a fraction of the total length when the device gets longer. The larger L is the larger r_O is. The transconductance is directly proportional to the drain current for a given device length and overvoltage. Similarly the output resistance is inversely proportional to I_D. The product of transconductance with output resistance, $g_m r_O$, is a dimensionless figure of merit for a device. From our MOSFET I-V approximation with long-channel devices at high Vov $g_m r_O \approx \frac{2}{\lambda V_{OV}}$. Both λ and g_m have a more complex relationship to length and overvoltage in the SPICE model than we assumed in class. This Appendix makes the real relationship easier to use for hand calculations. The $g_m r_O$ product represents the voltage gain for a transistor that is biased with an ideal current source and has no other load. Thus it is the highest possible voltage gain for the transistor.

Here is a plot of that figure of merit - the ideal, common-source, voltage gain - for N and P channel devices at several different overvoltages as a function of channel length. Notice that this gain is independent of V_{OV} for P-channel devices but not for NMOS transistors. If a common source transistor is mated with a current source with comparable output resistance, then the resulting stage gain will be roughly half the ideal gain. Your amplifier needs to have gain > 80 DB (X10,000) and it will probably have a second stage gain in the range of 30 to 40 DB. (With the 30 K load and the supply current limitation any higher gain is hard to get.) That sets a minimum target gain for the first stage and the ideal gains of the figure should suggest a starting point for selecting L. Second stage lengths probably are determined by current matching requirements. Keep in mind that is does not make sense to go much longer in L than necessary as the area of the circuit grows proportionately to L^2, increasing cost and decreasing bandwidth.



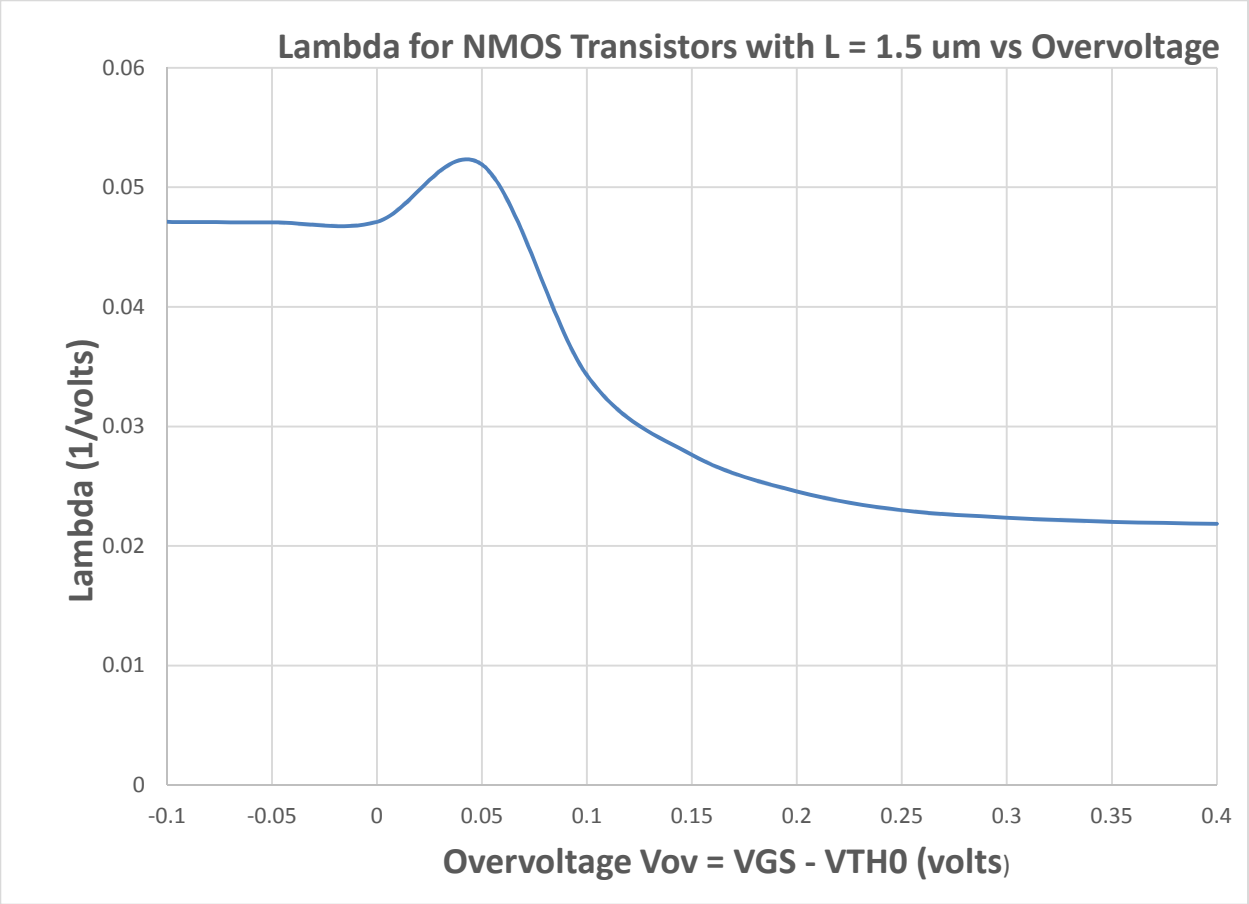
To calculate the low frequency stage gains you need a way to estimate g_m and r_O for each transistor separately. In class and in Razavi we often use the simple relation that $g_m = \frac{2I_D}{V_{OV}}$. Unfortunately this relation is not helpful when V_{OV} is low. The proportionality with I_D is still true but the coefficient becomes a more complex function of overvoltage when $V_{OV} \le 0.3$ volts. The next graph shows that relationship for an NMOS device under very specific conditions. However, the relation is essentially independent of L for $L \ge 1.0$ microns. The same value for the coefficient holds for PMOS devices as well. I do not understand the hump in the curve and suspect it is an error in the SPICE model or model parameters. Notice the deviation from the relation $g_m = \frac{2I_D}{V_{OV}}$ that is plotted on the green curve. The way the ratio levels off as the device enters subthreshold operation reflects the fact that a current exponentially dependent on a voltage has a constant ration of g_m to ID. That ratio is $\frac{q}{nkT} \simeq \frac{40}{n}$ where n is the ideality factor for the device.

Estimating the output resistance of a device requires a value for the λ parameter in the device current equation. ($r_O = \frac{1+\lambda V_{DS}}{\lambda I_D}$) For PMOS devices, λ depends primarily on the length L. The table below gives λ from SPICE simulations for several candidate values of L.



Dependence of λ on Channel Length, L, for P-channel MOSFETs	
Channel Length - L (microns)	λ
1	0.079
1.25	0.043
1.5	0.026
1.75	0.017
2	0.012
2.5	0.006
3	0.004

The output resistance of NMOS devices in this process is dependent on both L and Vov. The graph below shows the Vov dependence for a 1.5 micron long device. Estimate the value of lambda for other values of L by multiplying the value from this graph by 0.7 for L = 1.75 microns and by 0.6 for L = 2.0 microns.



Appendix II: Common Mode Gain and CMRR

A common mode input voltage induces a change in the drain currents of the input differential transistor pair that is the same in both transistors. The value of that current depends primarily on the output resistance of the current source that biases the source terminals of the differential pair. If the resistance of the current source is r_{Ocurr_source} then the drain current is approximately

$$i_{d\_cm} = \frac{v_{cm}}{2r_{Ocurr\_source}} .$$

The current mirror that converts the two drain currents into a single current output will take the difference of the two currents, substantially lowering the net common mode current into the output node of the differential amplifier. However the current mirror does not take an exact difference. For a simple mirror with no cascoding, the net current difference into the mirror output node at constant output voltage and equal differential-pair drain currents is the input current divided by

$$(1+g_m r_O)$$

where the transconductance and output resistance are those of the input mirror transistor. (*You should be able to derive this formula from the small signal model of the mirror circuit!*). The net common mode output current of the stage is $i_{cm} = \frac{v_{cm}}{2r_{Ocurr\_source}\left(1+g_m r_O\right)}$.

The net current for a differential input is $i_\Delta = g_{m\_diff} v_\Delta$ where g_{m_diff} is the transconductance of one of the transistors in the differential pair. The common mode rejection ratio is the ratio of the differential current to the common mode current for excitation by the same value of input voltage.

$$CMRR = \frac{i_\Delta}{i_{cm}} = 2g_{m\_diff} r_{Ocurr\_source}\left(1+g_m r_O\right)$$

Calculate the common mode voltage gain by dividing the differential gain by the CMRR.

Appendix III: Summary of Formulas for MOSFET Properties

All of the MOSFETs in your circuit are likely to be in saturation, the normal active region. Many of the properties of the devices at low frequency are derived from the basic ID-VGS relationship. This table summarizes these relations.

Property	**Formula**
ID vs VGS and VDS in saturation	$I_D = \frac{W}{L} K \left(V_{GS} - V_{TH}\right)^2 \left(1+\lambda V_{DS}\right)$
Transconductance	$g_m = \frac{2I_D}{V_{ov}}$ for $V_{ov} \geq .3$ V$_D$; for $V_{OV} < 0.3$ $g_m \propto I_D$ see Appendix I for the proportionality factor
Output resistance (Channel length modulation)	$r_O = \frac{\left(1+\lambda V_{DS}\right)}{\lambda I_D}$

Gain of common source stage with no source degeneration	$g_m r_{output\_node}$
Gain of differential amplifier with current mirror	$g_m \left(r_{Odiff\_transistor} \parallel r_{Omirror} \right)$ where g_m is the transconductance of one of the differential pair transistors
Slew rate of two stage opamp	$S.R. \equiv \left\| \frac{dv_{OUT}}{dt} \right\|_{MAX} = \frac{I_{BIAS}}{C_{MILLER}}$ where I_{BIAS} is the total current biasing the differential pair.
Common mode rejection ratio	$CMRR = 2 g_{m\_diff} r_{O1stStageBiasCurrentSource} \left(1 + g_m r_{Omirror\_transistor} \right)$
Dominant pole position	$f_d = \frac{1}{2\pi C_M \left(1 + A_{2ndStage} \right) r_{O1stStage}}$ See formula for Gain of differential stage for the output resistance of that stage.
Phase Margin – measuring	To determine PM, plot frequency response magnitude and phase; find the frequency at which the gain is 1X (0 dB or unity gain) and subtract the phase at that frequency from 180. Should be 60 degrees or more when you are done. This is easy to do inside LTSpice with the Bode plot and a cursor.