# EE 140/240A Linear Integrated Circuits Spring 2020

### 1 Introduction

For this lab, you may consult the professor, the TAs, your friends, the textbook, the internet, and any other living or inanimate objects, with the exception of your peers' lab reports. You may obtain data in pairs, but must **submit your own written report**. Be concise. Hand calculations should be to 1 or at most 2 digits of precision. **Don't use a calculator—I won't let you use one on the exam and it's good to get in practice.** 

#### 2 Lab: Folded Cascode Design

The goal is to design and simulate a PMOS input folded cascode for your programmable gain amplifier (PGA). The figures below show the topology circuitry of a PMOS input folded cascode (Is this topology reasonable in our technology? If not, how would you modify it? *Hint: Do you have the headroom for a stacked current mirror in the cascode?*). You also need to design the bias circuit to bias your amplifier. The constant- $g_m$  circuit from Lab 4 is good candidate to start with. In any case, you might want to start with longer channel lengths (e.g.  $L = 1\mu m$ ) to keep the devices in quadratic mode where they are easier to analyze. There are many ways to bias this circuit, this is only one of many possibilities.



Figure 1: PMOS Input Folded Cascode

Specification	Requirement
$V_{DD}$	1.2V
Settling Accuracy $(f = \frac{1}{8})$	< 0.4%
Settling Time $(f = \frac{1}{8})^{T}$	< 5µs
$C_L$	400fF
Input Common Mode Range	Includes ground
Temperature	$-40^{\circ}C < T < +85^{\circ}C$

At a high level, you'll be:

- (1) Using hand calculation to design your amplifier based on the spec you're given.
- (2) Verifying that your bias points are correct with a DC simulation. If simulated values are very far off from what you calculated by hand, STOP. Try to find the source of the discrepancy—if your DC bias is wrong, your amplifier will never behave the way you expect.
- (3) Checking the AC parameters of your amplifier. Again, if simulated values are vastly different from your hand calculations, you should look for the source of the discrepancy.
- (4) Verifying the stability of your amplifier in feedback.

### 2.1 Hand Calculation

- (1) Calculate the requirements for open loop gain and unity gain bandwidth.
- (2) Choose device lengths ( $L = 1 \mu m$  is a good place to start) and overdrive voltages.
- (3) Calculate the transconductance required for the bandwidth and the current given your choice of overdrive voltage.
- (4) Calculate widths for all transistors. Start with the current mirror for the input diff pair and work your way toward the output. The table below should give you a reasonable place to start with hand calculations using the quadratic model.

	$ V_{th} $	$\lambda @L = 1 \mu m$	$\mu C_{ox}$
ne	0.65V	0.14 1/V	$140 \mu A/V^2$
nel	0.35V	0.16 1/V	$150\mu A/V^2$
pe	0.65V	0.09 1/V	$30\mu A/V^2$
pel	0.30V	0.15 1/V	$60\mu A/V^2$

### 2.2 DC Simulation

- (1) Embrace good hierarchy practices. Like your inverter for Lab 2, you'll want to have your amplifier in a cell of its own. We've provided a starter cell in library where you can populate the schematic. Feel free to modify the symbol as you like.
  - a. Make sure you can see the ee140\_gsi library in Cadence. If it isn't there, in your Library Manager click Edit → Edit Library Path. From there, Edit → Add Library → add library ee140\_gsi in /home/ff/ee140/sp20/cadence/ee140\_gsi. Afterward, press OK or File → Save to save the result in your cds.lib

<ul> <li>Library Path</li> </ul>	Editor:ngkai/ee140_TA/cadence/cds.lib [NameS		)  °	A	dd Library	×
<u>File Edit View D</u>	esign Manager <u>H</u> elp	cādence	Library			
Libraries			Name	ee140_gsi		
Ubrary 19 VLG_PRIMITVES 20 D_CELLS_HD 21 D_CELLS_HDLL 22 D_CELLS_HDMV 23 GATES_HD 24 GATES_HDSV 25 avTech 26 cds DefTechLb 27 xfab_connectLib 28 Cadence Libs 29 Digital.bs	Path /.xiti/setup/x_all/diglibs/VLG_PRIMITI/VES/cade nce_ICG1/VLG_PRIMITIVES /.xiti/setup/xt018/diglibs/D_CELLS_HD/cadence_ICG1/D_CELLS_HD /.xiti/setup/xt018/diglibs/D_CELLS_HDL/cadence_ICG1/D_CELLS_HDL /.xiti/setup/xt018/diglibs/GATES_HD/wadence_ICG1/D_CELLS_HDMV /.xiti/setup/xt018/diglibs/GATES_HDS/cadence_ICG1/GATES_HD /.xiti/setup/xt018/diglibs/GATES_HDS/vadence_ICG1/GATES_HDS/ SASSURAHOME/tools/assura/etc/avtech/avTech S CDSHOME/tools/all/etc/cdsDefTechLb xkti/setup/xt_all/diglibs/kab_connectLibs/ /.xiti/setup/x_all/cadence/xenv/lib/lbs/Cadence_ICG1/xtab_connectLibs /.xiti/setup/x_all/cadence/xenv/lib/lbs/CadenceLibs		Use	Mapped Name tory	ee140_ggi	
30 AnalogLibs 31 XFABLibs 32 iab4 33 iab2 34 iab5 35 To add a new Ibrary de to the library in the Pat When done, select the	/ xkit/setup/x_all/cadence/xenv/lib/lbs/AnalogLibs / xkit/setup/x_all/cadence/xenv/lib/lbs/XFABLibs lab4 lab2 lab5 finition, type the name in the Library column and the path h column. Press RETURN to continue adding libraries. menu 'Fie -> Save' to save your edits.		/home Design Use Use	ff/ee140/fa19/cadence Manager NONE No DM	OK Apply Can	I Help

Figure 2: Library Manager

- b. Copy pmos\_cascode with right click + Copy in the Library Manager.
- c. In the pop-up menu make sure to provide your library name under "To". By default it would be ee140\_gsi, which you need to replace with the name of your own library.

<ul> <li>Copy Cell</li> </ul>					
From					
Library	ee140_gsi				
Cell	zz_pmos_cascode				
То	Use your design libarary				
Library	lab5				
Cell	zz_pmos_cascode				
Options					
Cop	/ Hierarchical				
	✓ Skip Libraries TECH_XT018_HD US_8ths VLG_PRIMITIVES XFAB Libs ahdlLib analogLib avTech basic cdsDefTechLib functional lab2 lab4 lab5 sample sbaLib xfab_connectLib				
	Extra Views				
🗹 Cop	All Views				
	Views To Copy schematic spectre_ac				
🔄 Upd	of Entire Library	•			
Database Integrity     Experimental Action of the second sec					
Add To Category Cells *					
ОК	Apply Cancel He	elp			

Figure 3: Copy Cell

d. If you get a window showing the Copy Problems, click on Overwrite All and then click on OK. This would finish copying the cell to your library.

			Copy Pro	oblems		×
Destination Libra	ry:		la	ab5		
One or more of the following problems have occurred: <ol> <li>The source file does not exist.</li> <li>A destination file will be overwritten.</li> <li>A destination file is opened for edit.</li> <li>A destination file is opened for edit.</li> <li>A destination file is opened for edit.</li> </ol> <li>A destination file conflicts with another.</li> <li>A source library file is not valid in a different library.</li> <li>A source conserver sollwing moster file has zero size.</li>						
From Library	From Cell	From View	To Cell	To View	Error	Action
ee140_gsi	pmos_cascode_wra	data.dm	pmos_cascode_wr	data.dm	Would Overwrite	Overwrite
ee140_gsi	pmos_cascode_wra	symbol	pmos_cascode_wr	symbol	Would Overwrite	Overwrite
ee140_gsi	1	data.dm	1	data.dm	Would Overwrite	Overwrite
OK Fix Errors Overwrite All Cancel Help						

Figure 4: Copy Problems

- (2) Update the schematic of the pmos\_cascode you copied with your own design. Check and save.
- (3) Copy zz\_pmos\_cascode\_openLoop into your own library. We'll use this to characterize your amplifier in an open loop configuration.
- (4) The amp in your copied test bench currently points to the cell in ee140\_gsi. Replace the amplifier in zz\_pmos\_cascode\_openLoop with the cell from your own library.
- (5) We've provided a starter measurement setup for you in zz\_pmos\_cascode/openLoop/spectre\_state1. Make sure the "dc" checkbox is ticked in the ADE window.
- (6) Run the simulation.
- (7) To have Cadence show you the DC node voltages on your schematic: Results → Annotate → DC Node Voltages
- (8) To have Cadence show you the DC operating points in your schematic: Results  $\rightarrow$  Annotate  $\rightarrow$  DC Operating Points
- (9) Ensure that your bias network's DC node voltages check out first, then check the main body of your amplifier next.
- (10) If there is a large discrepancy between your DC simulation and hand calculation, **STOP**! Go through systematically to try and find the source of the discrepancy. If your DC bias is wrong, nothing else will work.

#### 2.3 AC Simulation

- (1) In the same testbench as your DC simulation (zz\_pmos\_cascode\_openLoop), make sure the "ac" checkbox is ticked.
- (2) Run the simulation.
- (3) You should now see plots of your AC magnitude and phase, and we've also included bandwidth and unity gain frequency calculations (these are specialized functions in Cadence).
- (4) Once again, if there's a large discrepancy in your AC simulation and hand calculation, try and find where the discrepancy is coming from. Chances are it's a result of biasing problems or a miscalculation of  $g_m$ ,  $r_o$ , etc.

## 2.4 Stability Simulation

Stability analysis is a powerful tool that has more capabilities than standard AC analysis because it doesn't mask loading effects of parasitic elements like AC analysis can when calculating phase margin. It calculates the frequency response of the loop gain

Loop Gain = 
$$T = A_0 f$$

- (1) Copy zz\_pmos\_cascode\_closedLoop from the ee140\_gsi library to your own.
- (2) Make sure the amp in your copied testbench points to the cell in *your* library and not ee140\_gsi. Change it if necessary.

- (3) The variable "BETA" is the same thing as f (some textbooks use  $\beta$  instead of f).
- (4) Make sure the "stb" checkbox is ticked.
- (5) Run the simulation.
- (6) Click Results → Direct Plot → Main Form. In the window that pops up, choose "stb" if it isn't already selected under "Analysis".
- (7) To find phase margin, select "Phase Margin". This is how the TAs added the output stb\_PM to your test bench.
- (8) Once you have an estimate of phase margin, put your device in unity gain feedback and see if the response to a step input looks like you expect.

#### 3 Deliverables

- (1) Show how you calculated gain and bandwidth requirements.
- (2) Schematic of opamp and bias generation with sizes and currents annotated. **Do not use Cadence** schematics for submission. You should either
  - Draw neatly by hand
  - Use some nice schematic drawing software (circuitikz in LATEX, Adobe Illustrator, Digi-Key Scheme-It, etc.)
- (3) Bode plots of amplifier frequency response (mag and phase) under different temperatures with DC gain and bandwidth annotated. **Do not use default Cadence plots for submission**. You should either
  - Export and plot with Python, MATLAB, Excel, etc. with readable axis labels
  - Modify your Cadence plot settings:
    - Graph  $\rightarrow$  Properties  $\rightarrow$  Set background to white
    - Graph  $\rightarrow$  Properties  $\rightarrow$  Graph Options  $\rightarrow$  Font  $\rightarrow$  Fixed [Sony]  $\rightarrow$  Size 18
    - Right click on traces  $\rightarrow$  Width  $\rightarrow$  Extra Thick
- (4) The following table:

Temperature	DC Gain	Unity Gain Bandwidth	Phase Margin @ $f = 1$
$-40^{\circ}\mathrm{C}$			
27°C			
85°C			