ENGI 5131 --- Tutorial 1

Using the Schematic Editor and Analog Environment Lakehead University

> Greg Toombs, Carlos Christoffersen Winter 2009, 2013



1. Starting Cadence

To start Cadence, open a terminal and type

mkdir design cd design /CMC/bin/goCadence

The following dialog box should appear:

💿 cm	c_kits_vie	W	\odot \land \times
<u>F</u> ile			<u>H</u> elp
CMC			search
You must be a designer/prototy	pe level sub	scriber of CMC Microsyst	ems to use.
To purchase a subscription	on go to ww	w.cmc.ca or call 613-530-	4666
💠 startCds_ga911.2.3			\square
♦ startCds_ibm130nm.1.8.0.	0		
\diamond startCds_ibm130nm.1.8.0.	1		
startCds_ibm130nm.1.8.0.	2		
startCds_ibm130nm.1.8.0.	4		
startCds_ibm130nm.1.8.0.	4_oa		
startCds_st65nm.1.0			
startCds_st90nm.3.0			
startCds_tezzaron.2010q2			
startCds_tezzaron.2011q2			
startCds_tezzaron.2011q2 startCds_tezzaron.2011q2	-		
 startCds_tezzaron.2011q2 startCds_tezzaron.2011q2 			
startCds_tsmc65nm_gp.1.	-		
startCds tsmc65nm lp.1.5			
 startCds_tsmc90nm_pr1. 			
 startCds tsmc180nm.5.2 			
♦ startCds tsmc350nm.4.3			Å
P.	run		Quit

After selecting "run", you should now see the ICFB log and a "What's New" window appear. Close "What's New".

[Tip] If anything goes terribly wrong in Cadence during any part of design or simulation, the ICFB log will be a good place to check. Closing the ICFB log quits Cadence.

2. Creating the Library and Cell

At the top of the ICFB log window, select Tools \rightarrow Library Manager. At the top of the Library Manager window, select File \rightarrow New \rightarrow Library. In the Name field, enter "tutorial1", and select OK.

A new window will appear called "Technology File for New Library". We want to implement the new

project in the .18µm (or "point eighteen") CMOS technology, so select "Attach to an existing techfile", and select OK. Another new window will appear called "Attach Design Library to Technology File". Beside "Technology Library" there is a dropdown box whose default selection is "CMCpcells". Change that to "cmosp18" and select OK.

You should see the Library Manager again. Under the Library heading, you should be able to scroll down and see "tutorial1", your new library. Select it. At the top of the Library Manager window, select File \rightarrow New \rightarrow Cell View. The library name should show "tutorial1". For the Cell Name field, type "csamp". The view name should show "schematic" and the tool should show "Composer-Schematic", the visual circuit editor for the Cadence tool suite. The Virtuoso Schematic Editor should appear, and the Library Manager should show "csamp" under the Cell heading. You can close the Library Manager.

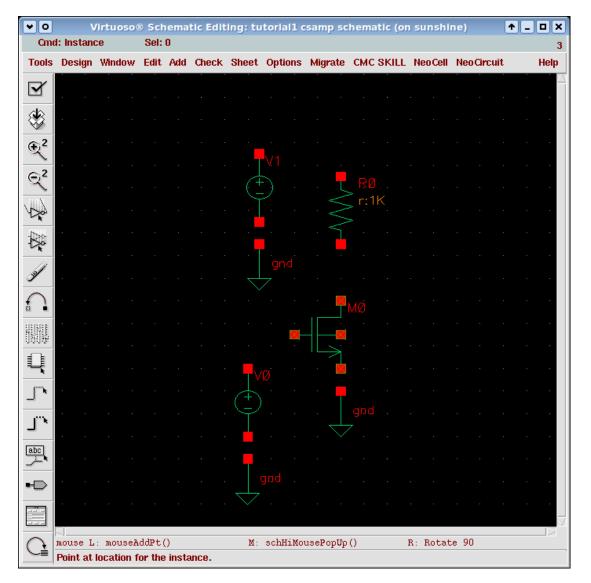


Figure 1: The circuit, with all component instances placed but no connections.

3. Entering the Circuit

To create the common-source amplifier, the transistors, voltage supply, resistor and grounds must first be added. On the toolbar to the left of the Virtuoso Schematic Editor, select the button with the icon of an eight-pin chip with the tool tip "Instance", or simply press the "i" key. If you want to get an idea of the different components available you can select Browse, but if you already know exactly what you

need you can type in the Library, Cell and View fields.

In the new Add Instance window, enter "analogLib" for the Library, "nmos4" for the Cell, and "symbol" for the View. The other fields do not need to be modified. Select "Hide" to get back to the schematic. Your cursor icon is now the symbol of a transistor. Every time you left-click, it will place an NMOS transistor. It will keep doing this until you press the escape key. We only want one transistor, so click once in the middle of the schematic and then press escape.

Using the same steps, add two analogLib/vdc/symbol, one analogLib/res/symbol, and three analogLib/gnd/symbol so that your circuit looks like Figure 1.

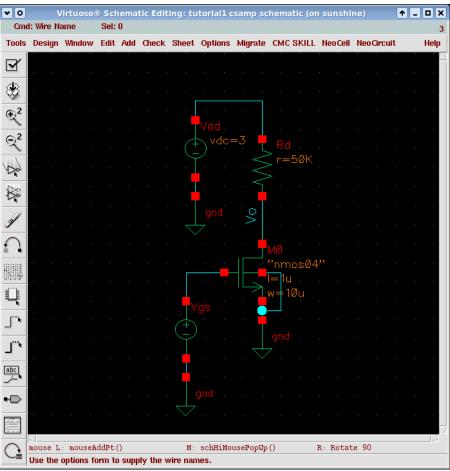


Figure 2: The finished circuit, with properties, nets and net names.

To create the connections (a.k.a. nets a.k.a. wires) between components, move the mouse cursor over a red terminal. Tap the W key, then left click where you want the wire to end. Once you are done, click on the check mark on the left toolbar with the tool tip "check and save". If you have missed any nets, there will be flashing yellow terminals.

To set the properties of a component, drag a box over it to select it. Then select the ninth button on the left toolbar that looks like an audio mixer (Property), or press Q. Set the following properties:

Supply voltage

- Instance Name: Vdd
- DC voltage: 3

Gate-source voltage

• Instance Name: Vgs

Drain resistor

- Instance Name: Rd •
- Resistance: 50k •

Transistor

- Model name: nmos04
- Width: 10u
- Length: 1u

Naming nets is very important to keep track of signals in complicated circuits. To name the output net, click on the "abc" button on the left toolbar (Wire Name). In the Names field, type "Vo", then press enter. Left-click on the net Figure 3: The simulation setup. above the drain of the transistor.

O Choosing Analyses> Virtuoso® Analog Desig ◆ □ OK Cancel Defaults Apply He Analysis tran ♦ dc ↓ ac ↓ noise ↓ xf sens ↓ dcmatch ↓ sth ↓ pz ↓ sps ↓ envip ↓ pss ↓ pz ↓ psh ↓ pnoise ↓ pxf ↓ psp ↓ qpss ↓ qpsp ↓ pss ↓ oppiose ↓ qpxf ↓ qpsp □ DC Analysis Save DC Operating Point □ Hysteresis Sweep □ Component Name /Vgg □ Design Variable Select Component □ □ Component Parameter Parameter Name dd Model Parameter Stop ፪ Sweep Range ↓ Step Size □ ↓ Step Size □ □ ↓ Center-Span Start ④ 100 Sweep Type ↓ Step Size □ □ ↓ Center-Span Step Size □ □ Add Specific Points □ □ □	v 0	Choosi	ing Analys	ses V	irtuoso®	Analog	Desic 🕇	
xf sens dcmatch stb pz sp envlp pss pac pst proise proise pxf psp qpss qpac qpnoise qpxf qpsp DC Analysis Save DC Operating Point Hysteresis Sweep Sweep Variable Design Variable Component Name /Vgs. Select Component Parameter Name dd. Sweep Range Start g. stop g. Start g. stop g. Sweep Type Linear		1	((I				Hel
xf sens dcmatch stb pz sp envlp pss pac pst proise proise pxf psp qpss qpac qpnoise qpxf qpsp DC Analysis Save DC Operating Point Hysteresis Sweep Sweep Variable Design Variable Component Name /Vgs. Select Component Parameter Name dd. Sweep Range Start g. stop g. Start g. stop g. Sweep Type Linear	Analy	/sis	🔿 tran	♦ dc	\sim	ac	⇔noise	
 ◇ pac ◇ pstb ◇ pnoise ◇ pxf ◇ psp ◇ qpss ◇ qpac ◇ qpsp ◇ qpnoise ◇ qpxf ◇ qpsp DC Analysis Save DC Operating Point Hysteresis Sweep Sweep Variable Component Name /Vgš Select Component Parameter Name dd Sweep Range ♦ Start-Stop Start (stop stop Sweep Type Step Size 100(Number of Steps 			×.		~		Y.	
 ◇ pac ◇ pstb ◇ pnoise ◇ pxf ◇ psp ◇ qpss ◇ qpac ◇ qpsp ◇ qpnoise ◇ qpxf ◇ qpsp DC Analysis Save DC Operating Point Hysteresis Sweep Sweep Variable Component Name /Vgš Select Component Parameter Name dd Sweep Range ◆ Start-Stop Start (stop stop Sweep Type Step Size 1000(Number of Steps 			◇ pz	⇒ sp	\sim	envlp	♦ pss	
 ↓ qpnoise ↓ qpxf ↓ qpsp DC Analysis Save DC Operating Point Hysteresis Sweep Sweep Variable Temperature Design Variable Component Name //Vgš Select Component Parameter Name dd Model Parameter Sweep Range ♦ Start-Stop Start @ Stop § Center-Span Sweep Type Step Size 1000 Mumber of Steps				⇔pst	b 🔆	pnoise	⇔pxf	
DC Analysis Save DC Operating Point Hysteresis Sweep Sweep Variable Temperature Design Variable Component Name Obsign Variable Component Name Component Name Parameter Name Model Parameter Model Parameter Sweep Range Start-Stop Center-Span Sweep Type Linear Number of Steps			🔷 psp	⇔qps	s 💠	qpac		
Save DC Operating Point Hysteresis Sweep Sweep Variable Temperature Design Variable Component Name Vgš Select Component Parameter Name dd Sweep Range Sweep Range Sweep Range Sweep Type Linear Number of Steps			🔷 qpnoise	\diamondsuit qpx	f 🔷	qpsp		
Hysteresis Sweep □ Sweep Variable Component Name /Vgš □ Design Variable Select Component □ Component Parameter Parameter Name dd □ Model Parameter Parameter Name dd Sweep Range Start-Stop Start ✓ Center-Span Start ① Stop ③ Sweep Type Step Size 100① Add Specific Points □				DC Ana	lysis			
Sweep Variable Component Name /Vgš Design Variable Select Component Component Parameter Parameter Name dd Model Parameter Parameter Name dd Sweep Range Start-Stop Start Stop Center-Span Start Stop 3 Sweep Type Step Size 1000 1000 Add Specific Points	Save	DC Op	erating Poir	nt 🗌				
□ Temperature Component Name /Vgš □ Design Variable Select Component □ Component Parameter Parameter Name dd ■ Model Parameter Parameter Name dd Sweep Range Start-Stop Start 3 ✓ Center-Span Start Ø 3 Sweep Type > Step Size 1000 1000 Add Specific Points	Hyst	eresis 3	Sweep		I			
		Compon	ent Parame	eter Pa				
Start g Stop 3 Sweep Type Step Size 10001 Linear Add Specific Points 10001	Swee	ep Rang	e					
✓ Step Size 10001 Linear → Number of Steps Add Specific Points			ં ડા	art 🧕		Stop	3	
Linear Number of Steps	Swee	ер Туре	1	<u>^</u>				
Add Specific Points		inear	-			2tono	1000	
				₩ Mu	mper of a	areha		
Enabled Doptions	Add 3	Specific	Points 🗌					
	Enab	led 🔳					Options	

Once all of the above steps are done, press "check and save" again. The ICFB log should show "Schematic check completed with no errors". The finished circuit is shown in Figure 2. During the next simulation steps, keep the schematic editor window open.

4. Setting up the Simulation

At the top of the schematic editor window, select Tools \rightarrow Analog Environment. The Virtuoso Analog Design Environment window (the Cadence circuit simulator) will appear. Select Setup \rightarrow Model Libraries. In the Model Library Setup window, in the Model Library File field, enter

/CMC/kits/models/textbook_mos.mod

This model controls the behaviour of transistors in simulation. Once the file name is typed in, select Add, then OK.

To set up the independent variable in the simulation (time, or a certain voltage, or a certain device parameter for instance) you need to go to the Choosing Analyses window. This can be reached with Analysis \rightarrow Choose, or the AC/TRAN/DC button on the toolbar. Choose DC analysis. Under the Sweep Variable options, select Component Parameter.

Click on Select Component. Switch to the schematic editor window. Move the cursor over Vgs. When a dotted yellow border appears, left-click. Select "dc" and then OK. Back in the Choosing Analyses window, "/Vgs" should appear for the Component Name, and "dc" should appear for the Parameter Name. (If you know exactly what you're doing, you could just type these in.)

Still in the Choosing Analyses window, in the Sweep Range options, make sure "Start-Stop" is selected, and then enter 0 for Start and 3 for Stop.

The simulator will be considering the circuit's behaviour for values of Vgs between 0V-3V. The Sweep Type determines how many values there are and how they are spaced. For many purposes, "automatic" is fine, but we want higher resolution (at the cost of slower simulation). Select a linear sweep type with 1000 steps.

The Choosing Analyses window should look like Figure 3. Select OK.

5. Running the Simulation

We need to specify which circuit variables should be monitored. At the top of the Virtuoso Analog Design Environment window, select Outputs \rightarrow To Be Plotted \rightarrow Select On Schematic. Switch to the schematic editor, and click on the Vo net. It will change colours and become a dashed line instead of a solid one. Back in the analog window under Outputs, you should see Vo.

To simulate, press the green light icon (Netlist and Run) or select Simulation \rightarrow Netlist and Run. You should get an output similar to that of Figure 4.

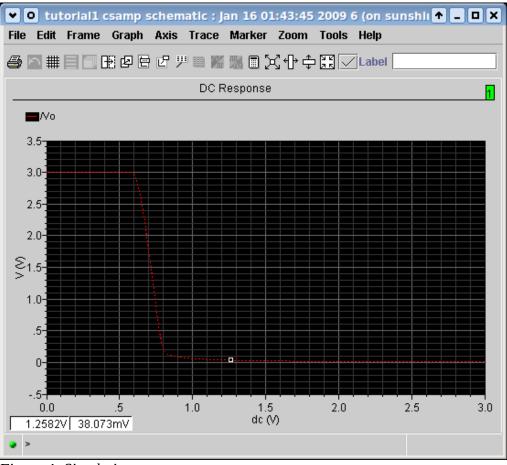


Figure 4: Simulation output.

By moving around the cursor, you can see x and y values on the lower left. Using two points in the transition region of the output, manually calculate the maximum small-signal gain:

$$A_{V} = \frac{V_{O2} - V_{O1}}{V_{GS2} - V_{GS1}}$$

It should be somewhere around -17.8.

6. (Optional) Automatic Gain Determination

It would be good to verify the accuracy of your calculation, and to get some practise with WaveScan functions. WaveScan is the Cadence tool to take care of plotting and mathematical analysis of simulation results.

Return to the analog window. In the Choosing Analyses window, change the range from 0V-3V to 0.5V-1V. (This increases the resolution of the simulation over the transition region of the output.) Click on the toolbar button with the check marks (Setup Outputs). In the Name field, enter Max Av. In the Expression field, enter

```
ymin(deriv(VS("/Vo")))
```

This takes the minimum value of the instantaneous derivative of the output voltage with respect to the input voltage. Select Add, then OK. Press "Netlist and Run" again. In the Outputs section of the analog window, beside Max Av, -17.83 appears.

There are many very useful WaveScan functions available to help analyses; a full guide PDF can be found on the course website.

7. Save and Close

At the top of the analog window, select Session \rightarrow Save State. Enter "tutorial1" in the Save As field and select OK. Close all Cadence windows. When the Save Display Information prompt appears, hit Cancel.

Debugging Note

It is important to close Cadence when you are done with it. If there has been a crash and the software aborted abnormally there are two important things you must do.

- 1. Look for and delete any files called "core" in your circuit directories. These are memory dumps that are created after crashes and take up a **huge** amount of space on the disk. Eventually they will exhaust your space quota and nothing will work.
- 2. Look for and delete any files with the extension ".cdslck". These are lock files that Cadence creates when the program is running. If the program dies abnormally, this lock file will not be deleted, and you will not be able to save any changes to your design the next time the program is run unless you delete it yourself.