

# University of Michigan EECS 311: Electronic Circuits Fall 2010 Cadence Tutorial



## S&K Demo: Non-Inverting Amplifier

- Build a non-inverting amplifier in Cadence with a gain of 2 V/V
- Used the circuit in Fig. 1 in the S&K paper



$$A_{v} = 1 + \frac{R_{3}}{R_{2}}$$
$$R_{0} = 2k\Omega$$
$$R_{1} = 8k\Omega$$
$$C_{0} = C_{1} = 20nF$$
$$R_{2} = R_{3} = 10k\Omega$$



## Setting-Up the Cadence Environment

You must do this setup before running Cadence for the first time, but only once.

- Login to a Linux machine in one of the CAEN labs. To connect remotely from a Windows computer or personal computer, login remotely to login.engin.umich.edu.
- 2. Go to your home directory, then create a link from your home directory to your class AFS space. Replace <uniqname> with your UMICH uniqname >> cd ~

>> ln -s /afs/umich.edu/class/eecs311/f10/students/<uniqname> eecs311

- 3. Use your link to get to your class space on the AFS file server anytime >> cd ~/eecs311
- 4. Copy all of the Cadence setup files to your AFS space
  - >> cp /afs/umich.edu/class/eecs311/f10/cadence/setup\_working\_dir/\* ~/eecs311
  - >> cp /afs/umich.edu/class/eecs311/f10/cadence/setup\_working\_dir/.\* ~/eecs311



# Setting-Up the Cadence Environment

**Aside:** For those who have used Cadence in a class before, check your home directory for the following two files:

- ~/.cdsinit
- ~/.cdsenv

If they exist, you may need to rename them to ~/.cdsinit.tmp and ~/.cdsenv.tmp and remove any Cadence-related modifications you made to your ~/.cshrc file.

The default .cshrc file can be found at /usr/caen/skel/std.cshrc. It will need to be renamed to .cshrc.

**Note:** If you created these files for a class you are currently taking, you will have to undo these operations before launching Cadence for this other class.



# Launching Cadence & Creating a Library

- Login to a Linux machine in one of the CAEN labs. To connect remotely from a Windows computer or personal computer, login remotely to login.engin.umich.edu.
- 2. Launch Cadence from your AFS directory. The Command Interface Window (CIW) will pop up.

```
>> cd ~/eecs311
```

```
>> icfb &
```

3. Launch the Library Manager by selecting Tools > Library Manager...

💥 icfb - Log: /afs/umich.edu/user/v				
File Tools Options			Help	1
Loadi Conversion Tool Box				14
Loadi Loadi <b>Library Manager</b>				
Loadi Library Path Editor Loadi VPCM				
I Verilog Integration				
mouse VHDL Tool Box	M :	R:		
Synopsys Integration				



# Launching Cadence & Creating a Library

The *EECS311Lib* and *EECS311Examples* libraries should appear in the Library Manager.

4. Select File > New > Library... from the Library Manager to create a new library

New 🖻	Library			
Ojien ~	Cell View	Cell	View	
Open (Read-Only) 🔿	Category		I	
Load Defaults	11			
Save Defaults				
Open Shell Window 🏻 🌱				
Exit ^	•			
tunctional rfExamples				
rfLib				
	]]			
Messages				
			s/wentzlof/libManager.log	11



# Launching Cadence & Creating a Library

5. Enter *tutorial* for the Library Name and click OK

Name	tutorial
Directory	
•••	
/eecs31	1/f08/students/wentzlof
	1/f08/students/wentzlof <mark>Manager</mark>
	Manager

6. Select "Don't need a techfile" and click OK



7. The library *tutorial* should now appear in the Library Manager.



#### Creating a New Schematic

 Click on the *tutorial* library in the Library Manager to select it, then go to File > New > Cell View...

💹 Library Manager: Dire	ctory/eed	cs311/f08/students/wentzlof	- 🗆 🛛
File Edit View Des	ign Manager		<u>H</u> elp
New >	Library		
The second secon		TION	
and the second s	Category	I	
Load Defaults Save Defaults Open Shell Window ^p Exit ^x functional rfExamples rfLib tutorial			
		s.lib file at '/afs/umich.edu/class/eecs311/f08/stud at /afs/umich.edu/class/eecs311/f08/students/wentzlo	



#### Creating a New Schematic

 Type *amplifier* for the Cell Name, and choose Composer-Schematic for the Tool. The View Name will default to schematic, then click OK.

🗙 Create New File 💷 🗵							
ок	Cancel	Defaults	Help				
Library N	lame 🗌	tutoria	_				
Cell Nam	e a	amplifier					
View Nar	ne 🛛	schematic					
Tool		Composer-Schematic =					
Library path file							
10/students/jkbrown/cadence/cds.lib							

10. The new schematic will open. To add a part to the schematic, go to Add > Instance... or by hit the 'i' key.

Cmd	:				Sel	: 0									
Tools	Des	ign	W	fino	low	Edit	Add	Check	Sheet	Ор	tion	s	Mi	<b>jra</b> t	te
		VC.					Insta	ance							
							Wire	(narrov	v)						
ala							Wire	(wide)							
*							10000	Name.		35					
							Pin.			8					
$\oplus^2$							52.00	1		32					
~							Bloc			35					
G.2							Net	Express	sion	34					
3							Sold	er Dot							
							Note	р. — — — — — — — — — — — — — — — — — — —		2					
$\sim$							Pate	hcord		194					
Daw							N 8	24 - 24. 2	91 - Ni						
1															



11. To find a part, click Browse from the Add Instance dialog box

📉 Add I	nstance		
Hide	Cancel	Defaults	Help
Library	I		Browse
Cell	Ĭ		
View	symbol		
Names	Ĭ.		
Array		Rows 1	Columns 1
Rotat	e	Sideways	Upside Down

12. The Library Browser will pop up. Select EECS311Lib > lab1\_opamp > symbol, then click Close.

- Library	Category	Cell	View
EECS311Lib		lab1_opamp	jsymbol
EECS311Lib US_Sths ahdlLib basic cdsDefTechLib functional rfExamples rfLib tutorial		lab1_opamp	symbol.



- 13. Place the lab1\_opamp on the schematic
- 14. Exit the current command by pressing the 'Esc' key





15. Add the remaining instances using Add > Instance...

- Resistors (analogLib > res)
- Capacitors (analogLib > cap)
- Pulse generator (analogLib > vpulse)
- Ground connections (analogLib > gnd)
- 16. Connect instances as shown in the figure on the next slide.
- 17. Go to Add > Wire Name... to name the *Vin* and *Vout* nets
- **Aside:** To copy an instance, go to Edit > Copy or by hit the 'c' key.
- **Aside:** To rotate an instance, go to Edit > Rotate or by hit the 'r' key.
- **Aside:** To edit the properties of an instance, select the component and either go to Edit > Properties or press the 'q' key.
- **Aside:** To move an instance, click and drag the component, press the 'm' key, or press shift+'m' to break wire connections then move it.
- **Aside:** To auto-zoom the schematic, go to Window > Fit or press the 'f' key.



#### Below is the final circuit schematic.





## **Editing Properties**

18. Edit the properties of the pulse input to have the following properties:

AC magnitude = 1 V DC voltage = 1 V Voltage 1 = 0 V Voltage 2 = Ampl V Delay time = 100u s Rise time = 1n s Fall time = 1n s Pulse width = 2m s Period = 4m s

Aside: 'Voltage 2' is used for transient simulations. By giving it a variable name, it can be swept/changed later.Note: Units are added automatically.

ок	Cancel Apply	Defaults Previous Next	He				
Apply To only current instance Show system user CDF							
	Browse	Reset Instance Labels Display					
	Property	Value	Display				
	Library Name	analogLib]	off 😑				
	Cell Name	vpulse	off 🗆				
	View Name	symbol	off =				
	Instance Name	<b>A</b> Q <sup>®</sup>	off 😑				
	User Property	Add Delete Modify Master Value Local Value	Display				
	lvsignore	TRUE	off =				
	CDF Parameter	Value	Display				
AC ma	gnitude	1 V <u>ě</u>	off =				
AC pha	ise	Ĭ.	off =				
DC vol	tage	1 V <u>í</u>	off =				
Voltag	e 1	0 <b>v</b>	off 😑				
Voltag	e 2	Ampl V	off 😑				
Delay	time	100u š	off 😑				
Rise ti	me	1n š	off =				
Fall tin	ne	1n š	off =				
Pulse v	width	2m si	off 😑				
Period		4m si	off 💷				



## Setting Analog Environment

- 19. Click the 'Check and Save' button in the upper left corner of the schematic window to save the schematic design
- 20. Go to Tools > Analog Environment to setup simulations

🗙 Virtuoso® Analog Design Er	ivironment (1)	X
Status: Ready	T=27 C Simulator: spectr	e 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	Ļ
Library tutorial	# Type Arguments Enable	⊐ AC ■ TRAN
Cell amplifier View schematic	1 dc t yes	
Design Variables	Outputs	l∎″
# Name Value	<pre># Name/Signal/Expr Value Plot Save March</pre>	34
		8
		8
>	Plotting mode: Replace =	$\sim$



# Setting Design Variables

- 21. Load variable names from the schematic by selecting Variables > Copy From Cellview
- 22. Select Variables > Edit ... to edit the variable values or double-click on them in the environment window.
- 23. Give the *Ampl* variable a value of 1 (Vpk)

🔀 Editi	🗙 Editing Design Variables Virtuoso® Analog Design Enviro 💶 💷 💻 🏎							
ок	Cancel	Apply	Apply & Run Simulatio	n			Help	
Selected Variable					able of De	esign Varial	bles	
Name		Ampl		#	Name	Value		
Value (	Value (Expr)				Ampl			
Add	Delete	Change						
Cellviev	w Variab	les Coj						



#### **Setting Analyses**

- 24. Go to Analyses > Choose... or click the "Choose analysis" button on the right-hand side of the Analog Environment Window
- 25. Select dc as the Analysis type, then select "Save DC Operating Point" and ensure that 'Enabled' is selected at the bottom

🔲 Che	oosing	Analyse	s Virtuo	oso® Analo	g Des 🗕				
ок	Cancel	Defaults	Apply			Help			
Analy	sis <	> tran	🔶 dc	🔷 ac	$\diamondsuit$ noise				
	<	) xf	$\diamond$ sens	$\diamondsuit$ dcmatch	🔷 stb				
	<	) pz	$\diamondsuit$ sp	🔷 envip	$\Diamond$ pss				
	<	> pac	$\Diamond$ pnoise	⇔pxf	$\diamondsuit$ psp				
	<	>qpss	🔷 qpac	🔷 qpnoise	🔷 qpxf				
	<	db2b							
Swee	ep Variat	ble							
	Fempera								
	Design V								
	Component Parameter								
<b>•</b>	vlodel Pa	rameter							
Enabl	led 🔳				Options	<b>.</b>			



#### **Setting Analyses**

26. Select ac as the Analysis type. Set the Sweep Variable to 'Frequency' and sweep over a range from 1 Hz to 10k Hz. Set the Sweep Type to 'Logarithmic' with 100 Points Per Decade. Ensure that 'Enabled' is selected at the bottom.

ок	Cancel	Defaults	Apply		F		
Analy	rsis <	> tran		🔶 ac	noise		
	~	xf	$\diamond$ sens	🔷 dcmatch	🔷 stb		
	<	> pz	$\diamondsuit$ sp	🔷 envip	⇔pss		
	<	pac	🔷 pstb	🔷 pnoise	$\diamondsuit$ pxf		
	<	psp	🔷 qpss	🔷 qpac	$\diamondsuit$ qpnoise		
	<	qpxf	♦ dbsb				
			AC Analy	sis			
Swee	ep Variab	le					
- 🔶 F	requency	у					
ंा	) Design Va	ariable					
৾৾৾ঀ	Temperature						
✓ Component Parameter							
$\sim$	Componei		eter				
~	Compone Model Pa	nt Parame	eter				
~		nt Parame	eter				
۰¢		nt Parame rameter	eter				
ې لې کې ا	dodel Pa	nt Parame rameter			104		
⇒ N Swee ◆ S	vlodel Pa ep Range	nt Parame rameter <sup>IP</sup> St	eter art <u>1</u>	Stop	1014		
⇒ N Swee ◆ S ◇ C	Model Par ep Range Start-Sto Center-Sp	nt Parame rameter <sup>IP</sup> St		Stop	1014		
<ul> <li>♦ Swee</li> <li>♦ S</li> <li>♦ S</li> <li>♦ Swee</li> </ul>	Model Par ep Range Start-Sto Center-Sp ep Type	nt Parame rameter IP St pan	art <u>1</u>	Stop ts Per Decade	-		
<ul> <li>♦ Swee</li> <li>♦ S</li> <li>♦ S</li> <li>♦ Swee</li> </ul>	Model Par ep Range Start-Sto Center-Sp	nt Parame rameter IP St pan	art ⊥ั ♦ Poin		10k]		
<ul> <li>♦ Swee</li> <li>♦ S</li> <li>♦ S</li> <li>♦ Swee</li> </ul>	Model Par ep Range Start-Sto Center-Sp ep Type	nt Parame rameter IP St pan	art ⊥ั ♦ Poin	ts Per Decade	-		
Swee Swee Swee Loga	Model Pau ep Range Start-Sto Center-Sp ep Type arithmic	nt Parame rameter IP St pan	art 1	ts Per Decade	-		
Swee Swee Swee Loga Add 5	Model Pau ep Range Start-Sto Center-Sp ep Type arithmic	nt Parame rameter pp St pan	art 1	ts Per Decade	-		
Swee Swee Swee Loga Add 5	Model Pau ep Range Start-Sto Center-Sj ep Type arithmic - Specific I	nt Parame rameter pp St pan	art 1	ts Per Decade	-		



#### **Setting Analyses**

27. Select tran as the Analysis type. Set the stop time to be 2m s and ensure that 'Enabled' is selected at the bottom

🔀 Choo	sing Ana	lyses Vir	tuoso® Analo	og Design Env		x	
ОК	Cancel	Defaults	Apply			Help	
Analy	sis ┥	• tran	⇔dc	$\diamondsuit$ ac	🔷 noise		
	<	≻xf	$\diamond$ sens	🔷 🔷 dcmatch	🔷 stb		
	<	>pz	🔷 sp	🔷 envip	🔷 pss 👘		
	<	>pac	🔷 pstb	🔷 pnoise	🔷 pxf 👘		
	<	>psp	🔷 qpss	🔷 qpac	🔷 qpnoise		
	<	qpxf	🔷 qpsp				
	Transient Analysis						
Stop	Stop Time 2m						
Accur	Accuracy Defaults (empreset)						
	_ conservative _ moderate _ liberal						
	_ Transient Noise						
Enabl	ed 🔳				Options.		



## **Running Simulations**

28. Select Simulation > Netlist and Run or hit the "Netlist and Run" icon on the righthand side of the Environment window to run all 3 simulations

Sta	atus: Re	eady					T=27	C Simulati	or: spectr	ю 3	
Sess	ion Se	tup Analyses	Varia	ables Outµ	outs Sir	nulation	Results	Tools		Help	
	De	esign				Analy	'ses			۲Ţ.	
Librar	<b>y</b> tuto	orial	#	Туре	Arg	uments.			Enable	⊐ AC F TRAN	
Cell	ampl	ifier	1	de	t				yes	⊐ DC	
View		ematic	2 3	ac tran	1 0	1M 2m	100	Loga	yes yes	T T X Y Z	
	Design	Variables				Outp	uts			Œ,	
# 1	Name	Value	#	Name/Si	mal/Exp	pr 1	Value :	Plot Save	March		Netlist and Run
1 7	Ampl	1									
										₩ 4	
										8	
						Plottii	ng mode:	Replac	:e =	1AZ	
>										Ł₩	



#### **View Results**

29. To see the DC voltages and currents within the circuit, select Results > Annotate > DC Node Voltages & Results > Annotate > DC Operating Points



The opamp has roughly equal DC voltage at its terminals with no DC current going into either input. The opamp also has a gain of roughly Av = 2 V/V as expected.



## **Viewing Results**

- 30. To view the small signal response, select Results > Direct Plot > AC Magnitude, then select the *Vout* wire in the schematic window and press 'Esc'. (left figure)
- 31. Select Results > Direct Plot > AC dB20, then select the *Vout* wire in the schematic window and press 'Esc'. (right figure)



Once again, the opamp has a gain of roughly Av = 2 V/V as expected. Because it is non-ideal, the opamp has a finite frequency response.



#### **Viewing Results**

- 32. To see the transient response, select Results > Direct Plot > Transient Signal and select both *Vin* and *Vout* in the schematic window and press 'Esc'. (left figure)
- 33. Go to Zoom > X-Zoom or press 'x' to zoom-in the plot. (right figure)



Once again, the opamp has a gain of roughly Av = 2 V/V after the step response. By zooming-in, we see that the delay is  $135\mu s$ , which matches the result from the class demo.



## S&K Demo: Non-Inverting Amplifier v2

- Build a non-inverting amplifier in Cadence with a gain of 2 V/V
- Used the circuit in Fig. 1 in the S&K paper
  - Used different resistors



$$A_{v} = 1 + \frac{R_{3}}{R_{2}}$$
$$R_{0} = 9.5k\Omega$$
$$R_{1} = 500\Omega$$
$$C_{0} = C_{1} = 20nF$$
$$R_{2} = R_{3} = 10k\Omega$$



## Viewing Results v2

34. To see the transient response, select Results > Direct Plot > Transient Signal and select both *Vin* and *Vout* in the schematic window and press 'Esc'.



Once again, the opamp has a gain of roughly Av = 2 V/V after the step response, but now we have oscillations like seen in the class demo.



#### Saving Analog Environment Setup

35. In the Analog Environment window, go to Session > Save State .... Choose 'Cellview' for the Save State Option, then type a filename for the State.

🔻 Saving State Vir	tuoso® Analog Design E	nvironment (1) 🛛 🗌 🗙
OK Cancel Apply		Help
Save State Option	🔷 Directory 🔶 Cellv	iew
Directory Options		
State Save Directory	~/.ertiet_stated	Browse
Save As	statel	
Existing States		
Cellview Options		
Library	tutorial 🔤	
Cell	amplifier]	wse
State	spectre_statel 💆	
Description		
Nonej		
<u></u>		
What to Save		
<b>.</b>	. Vedebler	Select All Clear All
■ Analyses ■ Model Setup	Variables Simulation Files	<ul> <li>Outputs</li> <li>Environment Options</li> </ul>
Simulator Options	_	Waveform Setup
🔳 Graphical Stimuli	Conditions Setup	Results Display Setup
Device Checking S	Setup 🔳 Cosimulation Options	Distributed Processing



#### Saving Results

- There are at least 3 ways to save your plots:
- Good 1. Hit the 'Print Screen' key. In Linux, a window should pop-up that allows you to save the entire screen.
- Better 2. In the plot window, go to File > Save as Image..., hit the 'Browse...' button, define the file name and path, click 'Ok', and click 'Ok' again. This will save the plot as an image.
- Best 3. In the plot window, select each trace, then go to Tools > Table..., select 'Value', and click 'Ok'. You should see a speadsheet window open with data. To save the data points, go to File > Save as CSV..., define the file name and path, and click 'Save'. This will save the data, so that you can generate a clean plot in Matlab.



# Using the Linux Terminal

• For those new to Linux, there are many commands to learn within the terminal, which can make your life easier. Below are just a few.

```
% change directory
>> cd
>> <any command> --help
                                   % brings up help info on <any command>
>> <any command> ~/...
                                   % ~ starts a path from your home directory
>> <any command> ./...
                                   % . starts a path from your current directory
>> <any command> &
                                   % & launches a process in the background
                                   % list all files & folders in a directory
>> ls -a
>> ps -u <uniqname>
                                   % lists yours processes & process IDs
                                   % kills a process (like Cadence if it freezes)
>> kill <process ID>
>> ln -s
                                   % create a symbolic link
                                   % Copy
>> cp
                                   % make directory
>> mkdir
                                   % remove files
>> rm
                                   % remove directory
>> rmdir
                                   % move or rename files or directory
>> mv
>> gedit <filename> &
                                   % opens a very Windows-like text editor
>> ps2pdf
                                   % converts a postscript file to a pdf
```

- You can search the web and find more commands pretty easily
  - <u>ss64.com/bash/</u> has a pretty good list