Cadence Tutorial

Colin Weltin-Wu

Step 1

Before anything you need to modify your .bash_profile file in you root directory. Open the file ~/.bash_profile in your favorite editor, and it should look something like this:

User specific environment and startup programs

PATH=\\$PATH:\\$HOME/bin

export PATH unset USERNAME

.bash_profile

You must add the line PATH=/usr/cad/mmsim/tools/bin:\$PATH before the PATH statement, so it looks like this:

```
# .bash_profile
```

```
# User specific environment and startup programs
PATH=/usr/cad/mmsim/tools/bin:$PATH
PATH=$PATH:$HOME/bin
```

export PATH unset USERNAME

Save and exit. Check that the line is now there by typing more .bash_profile and the prompt.

Now create a Cadence directory for your files. You can name it "4312" for example, and create it with mkdir 4312 at the prompt. Then change to that directory with cd 4312. Now

you have to set up some environment parameters. Run the following commands, replacing **<username>** with your login username:

```
cp ~tod/4312/display.drf ~<username>/4312/.
cp ~tod/4312/.cdsinit ~<username>/4312/.
cp ~tod/4312/cds.lib ~<username>/4312/.
cp ~tod/4312/.cdsenv ~<username>/.
```

Now log out and log back in to make sure all the changes take effect. Once you're logged back in, cd to the Cadence directory and run cadence with icfb &. The icfb window is the main Cadence window and also serves as the console where error messages appear.

	icfb - Log: /home/user2/fall	07/cw2319/CD5.log.1		٥×
File Tools Options I	BM_PDK		Help	1
registering "Symbol" *WARNING* "Layout": registering "Layout" > "IBM_PDK Menu added"	unregistering user menu function	mgcLoadCalibreMenu		T T
Ι				
mouse L:	M :	R :		
>				

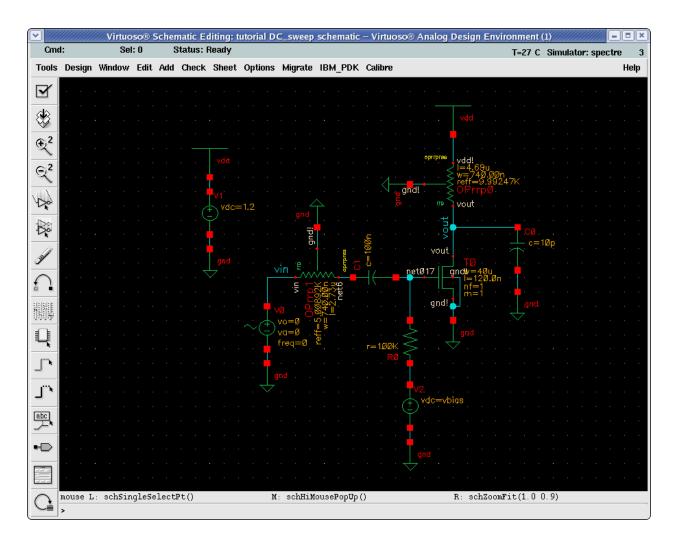
Step 2

Once you have the icfb window, click Tools -> Library Manager to bring up the Library Manager. This is where all your designs are stored in a hierarchical fashion. To create a new project, go to File -> New -> Library.. and name it anything you want, for example tutorial. When it asks for a techfile, select "Don't Need a Techfile".

Step 3

¥	Crea	te New Fil	e 🗴
ок	Cancel	Defaults	Help
Library N	ame t	utorial	i
Cell Name	9 D	C_sweepį́	
View Nan	ne ^{so}	chematic	
Tool	Co	mposer-S	chematic $=$
Library pa	ath file		
me/user2	2/fall07,	/cw2319/4	312/cds.libį́

Now that we have a new library, let's put something in it. The first thing we want to simulate is a DC sweep, so go again to File -> New -> Cell View.. and create a new Schematic view with cell name "DC_sweep", and make sure it's in the tutorial library. Once you click OK, a new virtuoso schematic editing window should come up.



Now we want to build a simple circuit to simulate, shown above. Cadence uses all three mouse buttons along many keyboard shortcuts. There is a contextual list of what each mouse button does at the bottom of the window. Let's put a MOSFET down first. Press i short for "instantiate" to bring up the instantiation dialog.

☑ /////	Add Instance	////////
Hide	Cancel Defaults	Help
Library	Ι	Browse
Cell	Y	
View	Ĭ.	
Names	y	
Array	Rows 1 Columns	1
	Rotate Sideways Upside	Down

If you remember the parameters of the part you want to use you can type it directly into the fields, otherwise browse for an "nfet" symbol in the bicmos8hp library.

Library	Cell	View	
bicmos8hp	infet	Isymbol	
E4312 US_8ths analogLib basic bicmos8hp cdSDefTechLib esd8hp ieee ripper simprim std synopsys tsmc025 tutorial unisim	havar havar_inh ind ind_inh kqres_inh langecoupler logo_IBM logo_IBM logo_PN lvsres meanderline2 min mim_inh nTiedown ncap ncap_inh nfet nfet_inh nfet nfet_inh nfet nfet_inh nfet nfet_inh nscot nsres nsres_inh nwCont	auCdl auLvs hspiceD hspiceS spectre symbol	

Note that the Instance window enlarges to show all the parameters associated with the item you have chosen to instantiate. Change the width of a single finger to 40u M and leave everything else the same. Cadence accepts standard engineering suffixes of units to simplify data entry.

Now when you bring the mouse back over the Virtuoso window, a MOSFET is being dragged. Left click to place it down.

Step 5

Now instantiate a resistor **oprrpres** from the same library. Don't change anything, just instantiate it above the MOSFET. We will be building a CS amplifier. If you want to rotate a part while dragging, right-click. The third terminal of the resistor is the substrate, which adds a parasitic capacitance from the layout.

Now we show another way to specify the item parameters: by left-clicking to select an object, then pressing q to bring up the Object Properties dialog.

By unchecking Specify res by Geometry? you can set the resistance to 10k. Then click OK.

OK Cancel Apply De	faults Previous Next	
Apply To only curre Show system	ent instance m ■ user ■ CDF	
Browse	Reset Instance Labels Display	
Property	Value	Display
Library Name	bicmos8hp	off 🗕
Cell Name	oprrpreš	value =
View Name	symbol	off 😑
Instance Name	OPrrpŰ	off =
	Add Delete Modify	
CDF Parameter	Value	Display
Specify res by geometry?		off =
Backplate	sub =	off 🗆
Resistance (total)	9.99247K Ohms	off =
Resistance	9.99247K Ohms	off =
Width >= 0.74u	740.00n M	off =
Length >= 1.10u	4.69u M	off 🗕
Number of Series Bars	1 <u>į́</u>	off =
Parallel Bars	1	off =
Multiplicity	1	off 🗕
Sub Resistance	50 Ohmsį	off =
Device temperature rise al	oov 0	off =
Use self heating?		off 😑
	oprrpreš	off =

Once a cell is on the schematic, instead of instantiating a new one each time, you can simply copy existing cells by using the c key, then clicking the item you'd like to copy. Cadence has "sticky" keys, which means your command will work indefinitely unless you cancel out of it, or switch to a different command. Two useful keys are the ESC key, which gets out of whatever you're doing, and CTL-d, which deselects everything. (regions you may have inadvertently drag-selected)

Instantiate a gate resistor (the input source resistance) by copying and rotating the resistor, and set its value to $5k\Omega$. Don't be alarmed if the values change to something close but strange, Cadence rounds to the closest value possible within the constraints of layout, i.e. a resistor length of $9.2323\mu m$ is impossible so rounding may be required.

Step 6

Items such as ideal passive elements, voltage and current sources and the like are all in the analogLib library. Instantiate a DC power source with a vdc cell set to a DC Voltage of 1.2V. Instantiate an input voltage source vsin and set its parameters as shown, leaving all the others untouched for now:

			Edit Obje	ect Propertie	s	
ок	Cancel	Apply	Defaults Pre	vious Next		He
Apply Show	То		urrent – 🗌 ins stem 🔳 user	stance		
		Browse	Reset In:	stance Labels	Display	
	Proper	ty		Value		Displa
	Library	/ Name	analogLib)		off
	Cell Na	ame	vsinį			off
	View 1	lame	symbolį			off
	Instan	ce Name	VŬ			off
			Add	Delete	Modify	
	User F	roperty	Master		Local Value	Displa
	lvsign	ore	TRUE	Ĭ		off
	CDF Pa	arameter		Value		Displa
AC ma	gnitude		1 V			off
AC pha	ase		Q			off
DC vo	Itage		0 V <u>í</u>			off
Offse	t voltage		0 V			off
Amplit	ude		0 V			off
Freque	ency		0 Hz			off

For the DC blocking cap and the load cap, use the ideal element cap in the analogLib, and set them to 100n and 10p, respectively. For the bias resistor, use an ideal resistor res also in analogLib, with value 100k.

Now instantiate the Vdd and Gnd symbols, which are called vdd and gnd, respectively. If you want to view what you've done, f will fit the window to the full schematic, z will let you zoom into a selected region, and CTL-z will zoom out of a selected region.

Step 7

Now we want to wire it up. Press w to use the wire tool, and it's pretty easy. Cadence will try to snap to points that are attachable, or you can double-click to terminate a wire mid-route. Wire up the circuit, don't forget to wire up the substrate terminal of the MOSFET. When Cadence netlists the schematic to produce something to simulate, it will assign net names that aren't always readable. To specify wire names, press the 1 key (lowercase "L") to bring up the label dialog. List all the names you want to enter separated by a space, i.e.

			Add Wire Na	ame		///////
Hide	Cancel	Defaults				Help
Wire N	ame I	let Expression	ı			
Names		vin vout				
Font Hei	ght	0.0625	Bus Ex	pansion	ϕ off \diamondsuit on	
Font Sty	/le	stick =	Placem	ent	\blacklozenge single \diamondsuit	multiple
Justifica	tion	lowerCenter	Purpos	e	🔶 label 😞 al	lias
Entry St	tyle	fixed offset $_$	Ванияе	Display	🔶 horizoutal	🔷 vertical
				Sho	w Offset Defa	ults
Rota	te					

Now click the wires you'd like to label. Your schematic should now be done and look like the picture in the beginning.

Now click the checkbox right under **Tools** to check and save the schematic. If there are any errors or warnings, Cadence will tell you. An Error will for sure cause the simulation to not run, whereas a Warning is less severe and depending on the particular case is allowable. This schematic should return nothing, because it should have zero of each.



	oso® Analog Design Environment (1)	
Status: Ready	T=25 C Simulator: hspice	D 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	Ł
Library tutorial Cell DC sweep	# Type Arguments Enable	⇒ AC F TRAN ⇒ DC
View schematic		I I T X Y 2
Design Variables	Outputs	I ∎
# Name ∀alue	# Name/Signal/Expr Value Plot Save March	
		8
>	Plotting mode: Replace =	\sim

Now we want to run a DC simulation. Go to Tools -> Analog Environment to bring up the simulation window. The first thing we need to do is specify the simulation engine and the directory for the simulation. Go to Setup -> Simulator/Directory/Host... and choose the simulator spectre and the project directory /tmp. Once this finishes, we need to specify the model files for our devices. Do this by going to Setup -> Model Libraries... and then entering the following path for Model Library File:

/usr/tech/bicmos8hp/IBM_PDK/bicmos8hp/V1.1.0.0HP/Spectre/models/allModels.scs

Then BE SURE to click "Add" to make the line appear above. Then click OK to exit.

Next we import the variables from the schematic. The easiest way to do this is have Cadence grab them from the schematic by going to Variables -> Copy From Cellview which should cause vbias to appear in the lower left window. Cadence needs all variables initialized to some value (even if it is a sweep variable and will change) so double-click vbias, enter 0 for its value, click Change then OK to exit.

Step 9

Next go to Analysis -> Choose to setup our DC sweep. Configure it as shown in the picture, except you should sweep from 0 to 1.2V instead (I made a mistake):

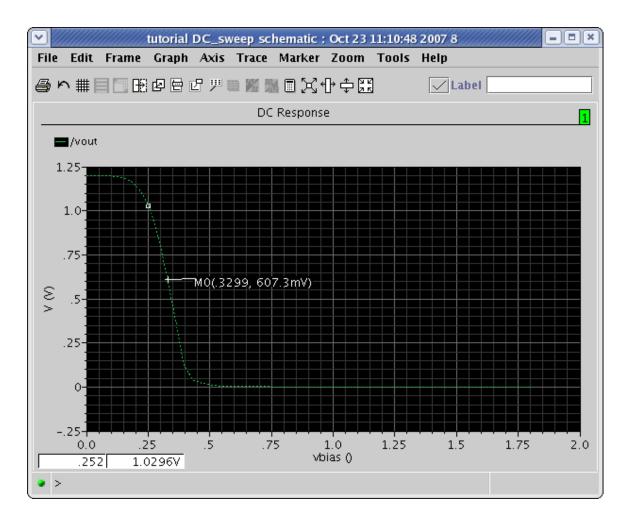
Choosing	g Analyses	Virtuoso®	Analog Desi	gn Environ	n 💌
OK Canc	el Defaults	Apply			Help
Analysis	🔷 tran	🔶 dc	🔷 ac	🔷 noise	
	⇔xf	\diamondsuit sens	\diamond dcmatch	🔷 stb	
	\diamondsuit pz	🔷 sp	🔷 envip	⇔pss	
	\diamondsuit pac	◇pstb	\diamondsuit pnoise	⇔pxf	
	\diamond psp	🔷 qpss	\diamondsuit qpac	\diamondsuit qpnoise	
	¢ qpxf	🔷 qpsp			
		DC Analysis			
Save DC O	perating Poi	nt 🗌			
Hysteresis	Sweep				
· ·		Variable eter	e Name ^{vbi:} Select Design		
Sweep Ran	ge				
🔶 Start-S	Stop	n di		1.0	_
🔷 Center-		tart 🖞	Stop	1.8	
Sweep Typ	0				
Automatic	-				
Add Specifi	c Points 🔄				
Enabled 🔳				Options.	

Now setup which variables you'd like to see plotted. There are again many ways to do this, but in the beginning it's easiest to just choose them on the schematic. Go to

Outputs -> To Be Plotted -> Select On Schematic then click vout in the schematic window. It should highlight. Press ESC otherwise you will continue to select nets, which may not be desirable. The simulator window should look like this:

Virt	1050® Analog Design Environment (1)	
Status: Ready	T=27 C Simulator: spectr	ne 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	Ł
Library tutorial	# Type Arguments Enable	⊐ AC F TRAN
Cell DC_sweep View schematic	1 dc 0 1.8 Auto Star yes	
Design Variables	Outputs	[‡ ′
# Name Value	# Name/Signal/Expr Value Plot Save March	
1 vbias O	1 vout yes allv no	3
>	Plotting mode: Replace =	\sim

Now just click the third button from the bottom on the right, "Netlist and Run". A notification will pop up but just choose "Don't show again" and click OK, and spectre will run. You should get a plot window as so:



I placed a marker where we should set vbias, around 0.33V to get a DC output voltage of 0.6V which is the midpoint of Vdd. Once we've set the bias condition, we can do further analyses.

Now we can analyze the AC performance of the amplifier. Go back to the simulation window, and first update vbias to 0.33V, which will bias the transistor in saturation. Now go back to Analysis -> Choose... and this time select ac analysis, and set it up like this:

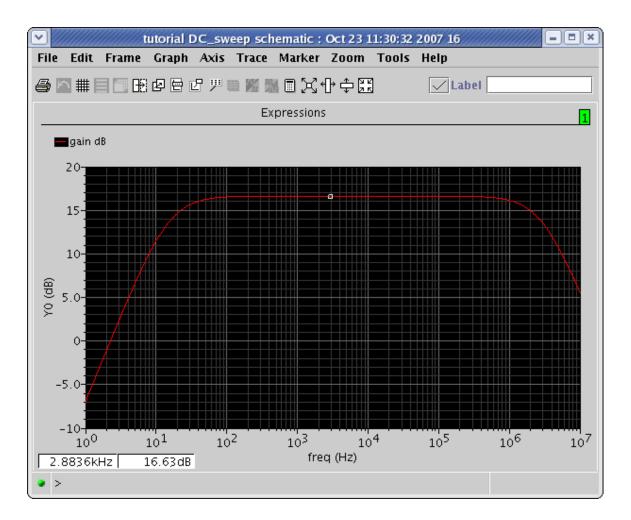
OK Cancel Defaults Apply Help Analysis tran dc ac noise × xf sens dcmatch stb > pz sp envlp pss > pac pstb pnoise pxf > psp qpsf qps qpac Oesign Variable Temperature Ac Analysis Sweep Variable Temperature Ocmponent Parameter Model Parameter Start-Stop Center-Span Start-Stop Center-Span Start-Stop Center-Span Start 10 Sweep Type Logarithmic None 50	Cho	oosing	Analyses ·	- Virtuoso®	Analog Desi	gn Environ	n 🗵
xf sens domatch stb pz sp envlp pss pac pstb pnoise pxf psp qpss qpac qpnoise qpxf qpsp Capac qpxf qpsp Capac qpxf qpsp Sweep Variable Frequency Design Variable Temperature Component Parameter Model Parameter Start 10 stop 1000 Start-Stop Center-Span Start 10 stop 1000 Sweep Type Logarithmic Picture Specialized Analyses None	ок	Cance	el Defaults	Apply			Help
pz sp psc pstb psp qpss qpsf qpsp qpsf qpsp Frequency Design Variable Frequenty Component Parameter Model Parameter Model Parameter Sweep Range Start 10 stop Center-Span Start 10 stop Start 10 stop 50 50 60 60 60 60 70	Analy	sis	🔷 tran	⇔dc	🔶 ac	🔷 noise	
pac pstb pnoise pxf psp qpss qpac qpnoise qpxf qpsp qpac qpnoise AC Analysis Sweep Variable Frequency Design Variable Temperature Component Parameter Model Parameter Model Parameter Sweep Range \$tart-Stop Center-Span Start Sweep Type Points Per Decade Logarithmic Sumber of Steps Specialized Analyses None			⇔xf	\diamond sens	\diamond dcmatch	⇔stb	
> psp qpss qpac qpnoise > qpxf qpsp AC Analysis Sweep Variable > Frequency > Design Variable > Temperature > Component Parameter > Model Parameter > Model Parameter > Sweep Range • Start-Stop > Center-Span Sweep Type Logarithmic > Number of Steps 50 Add Specific Points			🔷 pz	🔷 sp	\diamondsuit envip	⇔pss	
AC Analysis AC Analysis Sweep Variable Frequency Design Variable Temperature Component Parameter Model Parameter Sweep Range Sweep Range Sweep Range Points Per Decade Logarithmic Fo Add Specific Points Specialized Analyses None			\diamondsuit pac	\diamondsuit pstb	🔷 pnoise	🔷 pxf	
AC Analysis Sweep Variable Frequency Design Variable Temperature Component Parameter Model Parameter Model Parameter Sweep Range Start-Stop Center-Span Sweep Type Logarithmic Number of Steps Specialized Analyses None			🔷 psp	\diamondsuit qpss	🔷 qpac	\Diamond qpnoise	
Sweep Variable Frequency Design Variable Temperature Component Parameter Model Parameter Model Parameter Sweep Range Start-Stop Center-Span Sweep Type Logarithmic Number of Steps 50 Add Specific Points Specialized Analyses None			¢ qpxf	🔷 qpsp			
 Frequency Design Variable Temperature Component Parameter Model Parameter Model Parameter Start-Stop Center-Span Start Start Stop 100g 100g 50 				AC Analysis	Ş		
Design Variable Temperature Component Parameter Model Parameter Model Parameter Sweep Range Start-Stop Center-Span Sweep Type Points Per Decade Number of Steps Add Specific Points Specialized Analyses None	Swee	p Varia	able				
Temperature Component Parameter Model Parameter Sweep Range Start-Stop Center-Span Sweep Type Logarithmic Points Per Decade Summer of Steps Add Specific Points None None None None None None None None	🔶 F	requen	су				
Component Parameter Model Parameter Sweep Range Start-Stop Center-Span Sweep Type Logarithmic Points Per Decade Number of Steps Add Specific Points Specialized Analyses None	⇒ D	esign '	Variable				
Model Parameter Sweep Range Start-Stop Center-Span Sweep Type Logarithmic Number of Steps Add Specific Points Specialized Analyses None	ंर	emper	ature				
Sweep Range Start-Stop Center-Span Sweep Type Logarithmic Points Per Decade Number of Steps Add Specific Points Specialized Analyses None	_	ompon	ent Parame	ter			
Start-Stop Center-Span Start 10 1000½ Sweep Type Logarithmic Points Per Decade Number of Steps 50 Add Specific Points Specialized Analyses 50	⇒⊾	lodel P	arameter				
Start-Stop Center-Span Start 10 1000½ Sweep Type Logarithmic Points Per Decade Number of Steps 50 Add Specific Points Specialized Analyses 50	Swee	p Rano	16				
Start 10 Stop 100M Sweep Type Points Per Decade 50 Logarithmic Number of Steps 50 Add Specific Points Specialized Analyses None Image: Content of Steps	A S	tart-Si	ton				
Sweep Type Points Per Decade Number of Steps 50 Add Specific Points Specialized Analyses None 	· ·		' St	art 10	Stop	1001	
Add Specific Points Specialized Analyses None None	~ ~		open				
Logarithmic	Swee	р Туре	9	🔺 Points	Por Decade		
Add Specific Points	Loga	rithmic	:	•		50	
Specialized Analyses None				Vitunio	er ut atepa		
None	Add S	pecific	: Points 🔄				
	Speci	alized <i>i</i>	Analyses				
Enabled Options			None				
Enabled Doptions							
	Enabl	ed 🔳				Options.	

Double-click the DC analysis and in the lower-left corner de-select **enabled** so that only our AC analysis will run.

Now let's look at the gain in dB, so we double-click the **vout** under **Outputs** of the simulator window, and change it to this:

OK Cancel Apply	Hel
Selected Output	Table Of Outputs
lame (opt.) gain dr	# Name/Signal/Expr Value Plot Save March
dB20(abs(VF("/vout")/VF("/vin")))	1 gain dB yes
Calculator Open Get Expression Close	
/ill Be IPlotted/Evaluated	

The syntax of the calculator takes some getting used to. Netlisting and running this should produce the gain plot like this:



The midband gain is 16.6dB, which isn't great but amplifies nonetheless.

We can also do a transient analysis. From the gain plot, a frequency of 10kHz is in-band for the amplifier. So we can go back to our vsin source (in Virtuoso) and change it like this:

			Edit O	bject Propertie	s		
ок	Cancel	Apply	Defaults Pr	evious Next		He	
Apply	То	only c	urrent = _ in	stance =			
Show		_ sy	stem 🔳 user	r 🔳 CDF			
		Browse	Reset In	istance Labels	Display		
	Property			Value			
Library Name Cell Name		analogLi	bị		off =		
		vsirį			off =		
View Name Instance Name			symbolį	symbol			
			a AQ	AŬ			
			Add	Delete	Modify		
	User Property		Master		Local Value	Display	
lvsignore			TRUE	1		off 💷	
	CDF Pa	aramete	r	Value		Display	
AC magnitude			1 V			off =	
AC phase			Q			off =	
DC voltage			0 V			off =	
Offset voltage			đν			off =	
Amplitude			5m V			off =	
Amplit	wae						
Amplit Freque			10K H	Iz		off 😑	

The first three parameters, AC magnitude, AC phase, and DC voltage are for AC analysis, whereas the latter three are corresponding parameters but for transient analyses. For the transient analysis we need to ensure we're still operating in small-signal (for the purpose of this example, but you can simulate anything you'd like) so I chose a peak amplitude of 5mV. Check and save the schematic to be sure the simulator updates, otherwise when you try to netlist and run you will get an unsuccessful warning in the icfb window.

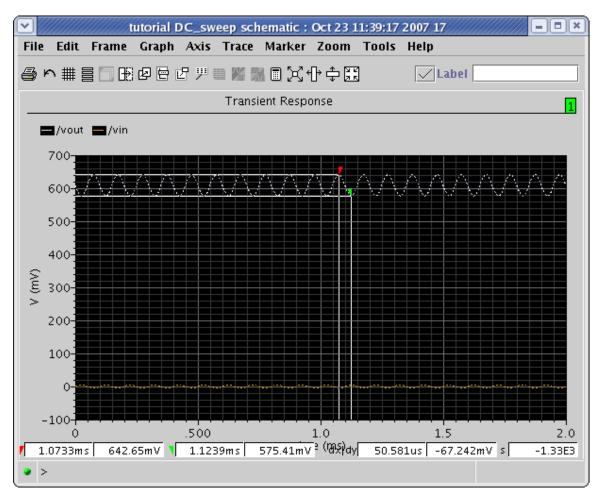
In the simulation setup, we now choose a transient analysis:

ок	() () () () () () () () () ()	Defaults	((B Analog Desi	girlEntrien	n He	
Analy	sis ∢	• tran		⇒ac	\diamondsuit noise		
	<	×f	\diamondsuit sens	\bigcirc dcmatch	⇔stb		
	<	pz	🔷 sp	🔷 envip	\Diamond pss		
	<	pac	\diamondsuit pstb	🔷 pnoise	\diamondsuit pxf		
	<	psp	\diamondsuit qpss	🔷 qpac	\diamondsuit qpnoise		
	<	qpxf	🔷 qpsp				
Transient Analysis							
		aults (em ative 🗌 n	preset) noderate _	liberal			
□ Tr	ansient	Noise					
	ed 🔳				Options.		

We should also add back the input and output waves to the Outputs window, and deactivate the unneeded Analyses and Outputs. The simulator should now look like this:

Virtu	oso® Analog Design Environment (1)	
Status: Ready	T=27 C Simulator: spectr	ne 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	÷Ę
Library tutorial	# Type Arguments Enable	⊐ AC ■ TRAN
Cell DC_sweep View schematic	1 dc 0 1.8 Auto Star no 2 ac 1 10M 50 Loga no 3 tran 0 2m yes	
Design Variables	Outputs	ŒĽ
# Name Value	# Name/Signal/Expr Value Plot Save March	
1 v bias 330m	1 gain dB no 2 vin yes allv no 3 vout yes allv no	- ₩ ₩
> Results in /tmp/DC_sweep	Plotting mode: Replace =	

The simulation output is here.



I have marked off the peak-peak output, which is 67mV. The input peak-peak voltage is 10mV (5mV peak), so the gain magnitude is 6.7, which corresponds to the 16.6dB seen in the AC analysis.