ENGI 5131 --- Monte Carlo Analysis

Lakehead University

Wai Wong, Carlos Christoffersen Winter 2013, updated May 2017

cādence™

Objective:

To gain familiarity with basic setup of the Cadence Spectre Analog Design Environment and the Monte Carlo analysis available there in.

1. Introduction

Monte Carlo analysis is commonly used to predict the effect of random variations of CMOS process parameters in the performance of a new design. For example, the tolerance rating of some capacitors and resistors can range as much as $\pm 15\%$. Often it is impractical to include the tolerance of every component in hand calculations; fortunately, the Monte Carlo analysis incorporated in a circuit simulator can effectively predict the performance of a circuit in various working conditions. If the performance of a circuit does not meet expectations, consider to replace some components or redesign the circuit.

2. Schematic

Create a simple current mirror schematic as shown below:



Replace the transistor standard models with the statistical models with mismatch information. In the schematic editor select Edit->Search->Replace:

		Scher	natic Rep	place		↑	□ ×
Apply	Cancel	Defaults	Replace	Skip	ReplaceAll		Help
Search I	For	model	-		nch		
Replace	With	model			nch_mis		
Search 3	Scope	cellview	-				
Zoom To Preview	Object Changes	_ (No Edits)		S	ave Change	es 🗌	
		<< 	lide Obje	ct Filte	er		
Sch	ematic Ol	ojects			i	ali 📃 none	
	wire	🗌 pin		insta	nce 📃 no	ote shape	
	wire name	e 📃 pin na	ame 📃	mark	er 📃 no	ote text	
	flight	_ label					
Instance Objects 🔄 all 🔄 none							
📕 name 🔄 pin 🔄 pin name 🔄 label							

Initially, select "other" in the "Search For" list ant type "model". Show the object filter and select the fields shown above. Then press "Replace" once for every instance. Close the Replace window and press "F6" to redraw the schematic. The new mode name appear next to each transistor. If your design has additional transistor models, perform a similar model replacement for each model (for example, use pch_mis for thin-oxide PMOS).

3. Monte Carlo Set up

1. Start the ADE and set up a DC sweep as shown below. Make sure to check the "save DC operating point" box.

Cho	oosing A	nalyses -	- Virtu	0508	Analog Des	sign Env 🛧	□ ×
ок	Cancel	Defaults	Apply				Help
Analy	sis <	> tran	🔶 dc		⇔ac	\diamondsuit noise	
	<	xf	\diamondsuit se	ns	\diamond dcmatc	h 🔷 stb	
	<	> pz	🔷 sp		🔷 envip	⇔pss	
	<	pac	⇔ps	tb	\diamondsuit pnoise	♦ pxf	
	<	psp	\diamond qp	SS	🔷 qpac	\diamond qpnoise	e
	<	qpxf	♦ P	sp	<> hb	\bigcirc hbac	
	<	hbnoise					
			DC An	alysis			
Save	DC Ope	rating Poir	nt 📕				
Hyste	eresis Sv	меер	Ē				
C ()esign Va Compone Model Pa	ariable nt Parame rameter	eter F	Parame	Select Cor eter Name	nponent dự	
Swee	ep Range Start-Sto	φ _{et}	art 0	9E		. 11	
$\diamond \mathbf{c}$	Center-S	pan	art	. 24 <u>8</u>	Stop	• <u> 4</u> .	
Swee	ер Туре		⇔s	tep Si	ze		_
Li	near	_	🔶 N	umber	of Steps	3	
Add S	Specific	Points 🗌					
_	od 🔳					Ontions	

2. Open the **Model Library Setup Form** by clicking **Setup -> Model Libraries** in the ADE menu and update the library path of the ADE as shown below.

			spe	ctre0: Model Libi	ary Setup		↑ □ X
ок	Cancel	Defaults	Apply				Help
#Disab	le Model	Library I	file			Section	Biable
/CMC	/kits/cm	osp18/mod	els/spect	tre/cor_mc.scs		mc	Disable
							Discule
							Up .
							Down
Model L	ibrary File					Section (opt.)	
/CMC/k	its/cmosj	p18/model	s/spectr	e∕cor_mc.scsį́		md	
Ad	ld	Delete	Chan	ge Edit File			Browse

That library only includes the models for low-voltage standard-Vth MOSFETs. Note that the section is set to "mc" (Monte Carlo).

Corner Simulations

Corner simulations are an alternative to Monte Carlo analysis. Only manually-chosen combinations of parameters are considered in corner simulations. The default section with typical parameters is called "tt". In addition, there are 4 sections with corner parameters available for transistor libraries: "ss", "ff", "sf" and "fs". The first letter refers to NMOS and the second for PMOS transistors (fast and slow). Just change the section to one of the corners in the Model Libraries window, for example from "mc" to "ff" and run simulations as usual.

For BJT, resistor and capacitor libraries, only "tt" (typical), "ss" (slow) and "ff" (fast) are available.

You'll have to include a separate line in the Model Libraries window for each section selected. For example, one line with "ss_na" for slow low voltage non-adjusted MOSFETs and "ss_bip" for slow bipolar transistor models. The process documentation and the following file can be used to guide your selection:

/CMC/kits/models/icfspectre.init

For example if your design is using both 1.8V and 3V transistors in a Monte Carlo simulation, you have to also include an additional library section in your model library.

/CMC/kits/cmosp18/models/spectre/cor_mc_3v.scs section=mc_3v

Note: in this process, statistical models are only provided for regular MOSFETs (1.8 V and 3.3V). For non-adjusted MOSFETs, BJTs and resistors only corner parameters are provided with this kit. Corner analysis can be performed with those devices.

4. Click OK to continue. Add the following outputs to the analysis: gate voltage, output current and output resistance (you have to figure out how to do that):

			Setting Outputs Virtuoso® An	alog	Design Environment (1)			+ □ ×
ок о	Cancel	Apply							Help
		Se	lected Output		Table	Of Outputs	:		
Name (o	pt.)	Output	Current	#	Name/Signal/Expr	Value	Plot	Save	March
Expressi	on	IDC("/¥	0/minus") <u> </u>	$\frac{1}{2}$	VO/MINUS Output Current	1 0250	yes	yes	no
Calculato Will Be Add	or Dele	Open G Plotte ete Char	et Expression Close d/Evaluated nge Next New Expression	34	Gate Voltage Ro	433.7m 20.68M			

V0/MINUS should be saved (to estimate output resistance).

5. Open the MC's **Analog Statical Analysis Form** by clicking **Tools -> Monte Carlo.**

🔲 Virtuoso® Analog Design Environment (1) (on cmc.lakeheadu.ca 🔺 😑 🔸 🗶								
Status: Ready	T=27 C Simulator: spectro	e 5						
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help						
Design	Analyses Parametric Analysis	s						
Library ct18 Cell test_inverter_3	# Type Arguments Monte Carlo 1 tran 0 1u)						
View schematic	Calculator Outputs							
# Name Value	# Name/Signal/Expr Value P 1 Yout r							
	y y							
Plotting mode: Replace =								

6. Fill in the form as shown below. Remember to check the box to save data between runs to allow family plots.

Ar Ar	nalog Statistical Analysis	↑ _ □ X					
Status: Ready		Simulator: spectre 18					
Session Outputs Simu	llation Results	Help					
	Analysis Setup						
Sampling Method	🔶 Standard 🔷 LHS						
Number of hiss	Ŭ.						
Number of Runs	500]						
Starting Run #	1						
Analysis Variation	Process & Mismatch =						
Swent Parameter None =							
Aunand to Previous Scolar Bata							
Sava Data Datwoon Dune to Allow Family Diate							
Save Process Paramete	ars I						
	Outnuts						
# Name Express	ion/Signal	Data Type Autoplot					
1 S 1 /V0/MTN	US	wave ves					
2 Output IDC("/V	0/MINUS")	scalar yes					
3 Gate_Vo VDC("/n	et8")	scalar yes					
4 Ro (0.75 /	(ymax(IS("/VO/MINUS")) -	scalar yes					
Ĭ.		scalar = no =					
Add Delete Ch	nange Clear Calcu	ator Get Expression					
		• • • • • • • • • • • • • • • • • • • •					

Once you have finished the data entry, click **Simulation** -> **Run**. Initially you can try a smaller number of runs to make sure everything works as intended. Normally 1000 simulations are required to produce meaningful statistical results.

If Analysis Variation is set to "Process Only", parameters change but are the same for all devices in each run. If "Process and Mismatch" is selected, each device is assigned a different set of random parameters.

4. Results

