IBM90nm – FO4 Delay

ECEN689 High Speed I/O Dr Samuel Palermo Ref : Dr Silva [EE474 Lab manuel] http://engineering.tamu.edu/electrical/employee-resources/unixlinux-resources-helpdesk

Starting Cadence for the First Time

-Course Directory : /mnt/lab_files/ECEN689_605

- 1. Make EE689 folder in your home director : mkdir ECEN689
- 2. Go to the EE689 : cd ECEN689
- 3. Copy model(directory), cds.lib(file), and ncsu from course directory

cp -rf /mnt/lab_files/ECEN689_605/model. cp -rf /mnt/lab_files/ECEN689_605/cds.lib. cp -rf /mnt/lab_files/ECEN689_605/ncsu.

4. Run cadence => ./ncsu

[rdliu918]@hera3 ~> (21:00:14 01/24/21) :: cd ECEN720/ [rdliu918]@hera3 ~/ECEN720> (21:01:36 01/24/21) :: cp -rf /mnt/lab_files/ECEN689_605/cds.lib . [rdliu918]@hera3 ~/ECEN720> (21:02:02 01/24/21) :: cp -rf /mnt/lab_files/ECEN689_605/ncsu . [rdliu918]@hera3 ~/ECEN720> (21:02:10 01/24/21) :: cp -rf /mnt/lab_files/ECEN689_605/model . [rdliu918]@hera3 ~/ECEN720> (21:02:17 01/24/21) :: ./ncsu [1] 35942 [rdliu918]@hera3 ~/ECEN720> (21:02:22 01/24/21) ::]

Creating a Library

1. From the CIW select Tools \rightarrow Library Manager to load the Library Manager

Library Manager: WorkArea: /homes/grad/yxs4875/689	L3	
<u>E</u> ile <u>E</u> dit <u>V</u> iew <u>D</u> esign Manager <u>H</u> elp		cādence
Show Categories Show Files Library INV_4 US_8ths andILlb analogLib basic cdsDerTechLib functional rTExamples rtLib	Cell Inv4 Inv4	View schematic View Lock Size Schematic yxs4875@apollo.ece.tamu.edu 30k
Messages		
		/

2. Do not Attach to an existing techfile due to using IBM90nm Model which is not support by cadence.

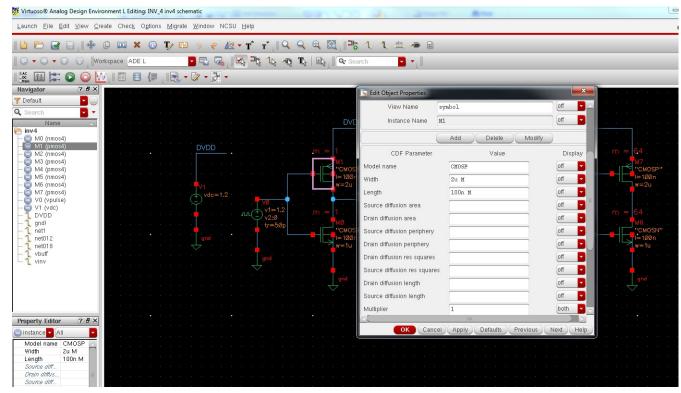
Creating a Schematic

The first circuit we will design is a simple inverter. Select which library you want to put the cell into, in this case "INV4", and then File \rightarrow New \rightarrow Cell. Name your cell inverter. The tool you want to use here is Composer-Schematic

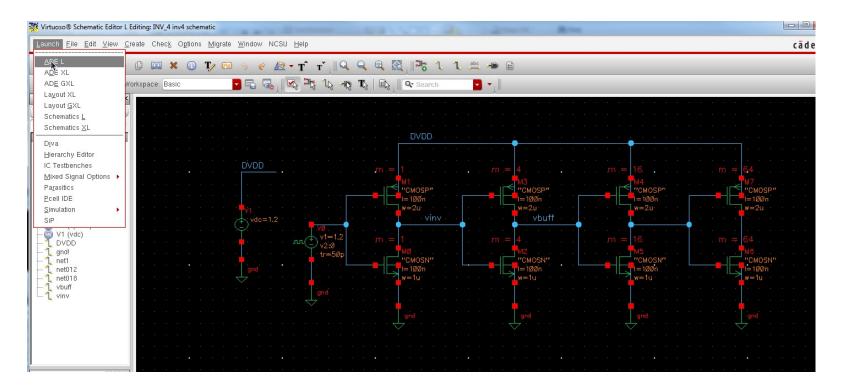
File	
Library	INV_4
Cell	inv4
View	schematic
Туре	schematic 🔽
Application -	
Open with	Schematics L 🧧
🔲 Always us	e this application for this type of file
Library path f	ile
	l/yxs4875/689/cds.lib

After selecting OK, the schematic window opens. We wish to add two transistors so that we can make an inverter. To do this we need to add an instance. You can do this by either clicking Add \rightarrow Instance or by pressing "i" on the keyboard. A window titled "Component Browser" should pop up. Make sure that the library analogLib is selected. Select N_Transistors and then nmos4. Go back to the schematic and select where you would like to add the NMOS transistor. Go back to the Component Browser and select P_transistors and then pmos4. Add this transistor to your schematic. Hit ESC to exit the Add Instance mode. Connect components together using wires. You can select Add \rightarrow Wire or use the "w" hotkey. To change the properties of a device use Edit \rightarrow Properties \rightarrow Objects or use the "q" hotkey.

MINIMUM W/L - 0.12um/0.10um



When finished, your schematic should resemble shown Figure to measure FO4 Delay. Select Design \rightarrow Check and Save to save your schematic and make sure that there are no errors or warnings.



Simulating the Schematic

Start the simulator environment by selecting Launch \rightarrow ADE L

Simulator

5 /	DE L (1) - INV	V4 inv4 schematic@hera3.ece.tamu.edu		– 🗆	×	ADE L (1) - INV4 inv4				- 🗆 X
<u>L</u> aun	ch S <u>e</u> ssion	Setup Analyses Variables Outputs	<u>S</u> imulation <u>R</u> esults <u>T</u> ools	<u>H</u> elp cāde	nce	Launch Session Setu	p <u>A</u> nalyses <u>V</u> ariables	Outputs Simulation Results	<u>T</u> ools <u>H</u> elp	cādence
12	õ 👔	<u>B</u> esign				🎼 🤔 🧊 27	🛛 💩 ⊁ 🖆 I	🗹 🗁		
		🔊 Simulator/Directory/Host	es	? 5	AC DC	Design Variables		Analyses		? 🗗 🗙 🗛 C OC Trans
	Name	High-Performance Simulation	Enable	Arguments	Crans 아니	Name	Value	Type Enable	Arguments	CTrans 약당
		Model Libraries <u> <u> </u> </u>					(D) (1) (1)			
		Stim <u>u</u> li Simulation Files			×		spectre)E L (1)@hera3.ece.tamu.edu	×	*
		<u>E</u> M/IR Analysis				Project Directory		ong/cadence/simulation		
		MATLAB/Simulink	s			Host Mode	🖲 local 🔾 remote 🤇	distributed		? = ×
			me/Signal/Expr Value	Plot Save Save Options	80	Host			Save	Save Options
						Remote Directory	OK	Cancel Defaulte Appl	Halp	
		Plot after	r simulation: Auto	Plotting mode: Replace			Ōĸ	<u>Cancel Defaults Apply</u> Plot after simulation: Auto	Plotting mod	e: Replace
> 			1							
2(4)	Simulator/Dil	rectory/Host	Statu	is: Ready T=27 C Simulator: spe	ctre 📗	2(4) Simulator/Director	yrnost		Status: Ready 1=2	7 C Simulator: spectre 📗

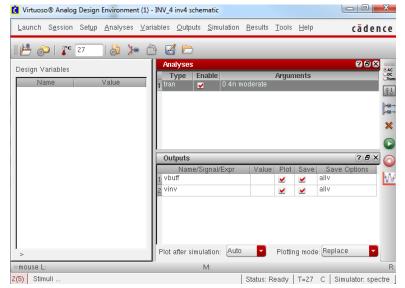
• Change simulator to "spectre"

Select Setup \rightarrow Model path and add CMOSN.m and CMOSP.m

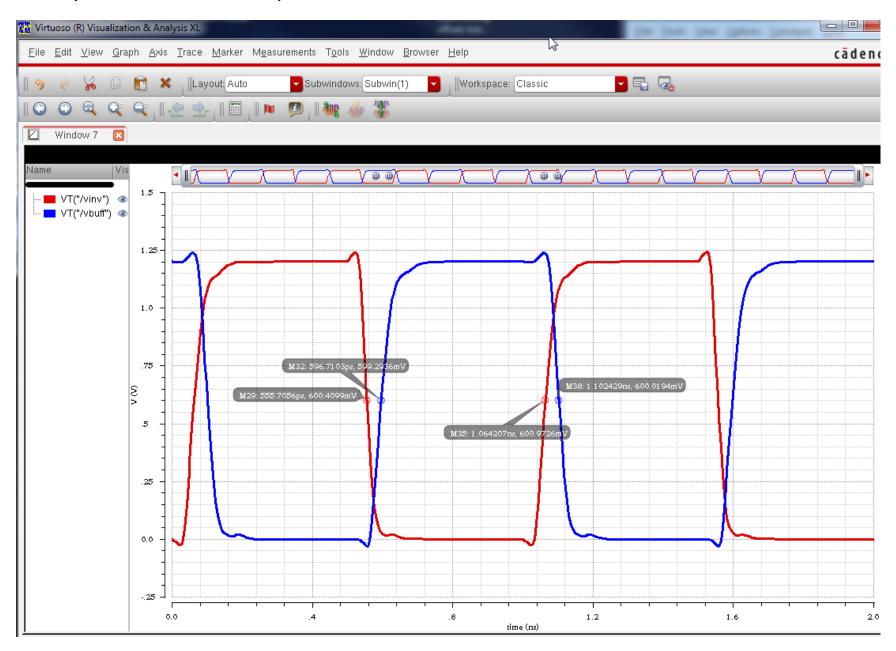
🔀 spectre0: Model Library Setup	A	×
Model File ⊡- Global Model Files 		Section
	ОК	Cancel Apply Help

Next we need to configure the environment to run our first simulation. In the Analog Environment window select Analyses \rightarrow Choose. Select "tran"

Select Simulation \rightarrow Run or click on the green light in the bottom right corner. Once the simulation has completed, we can plot any outputs that we wish. To do this we use the calculator. To access the calculator, select Tools \rightarrow Calculator in the Analog Environment.



Simulation Result FO4 delay for IBM90nm => 40ps



How to use PRBS generator in Cadence

PRBS generator can be found in ahdlLib. It is called rand_bit_stream. Please specify a PRBS generator as shown in Figure 1. Please set seed to 128 for 7 bit PRBS.

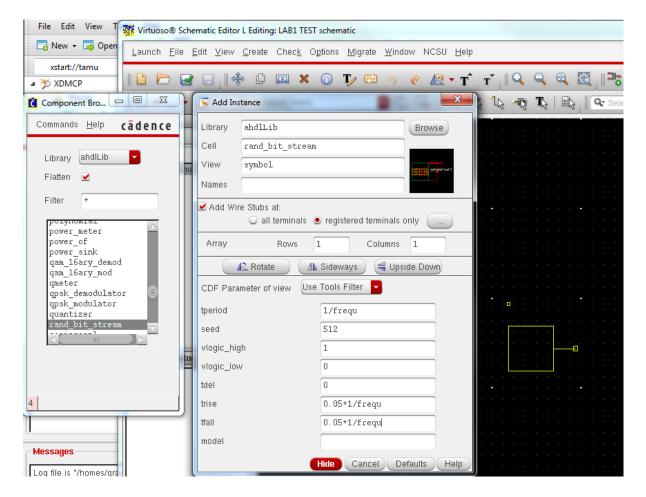
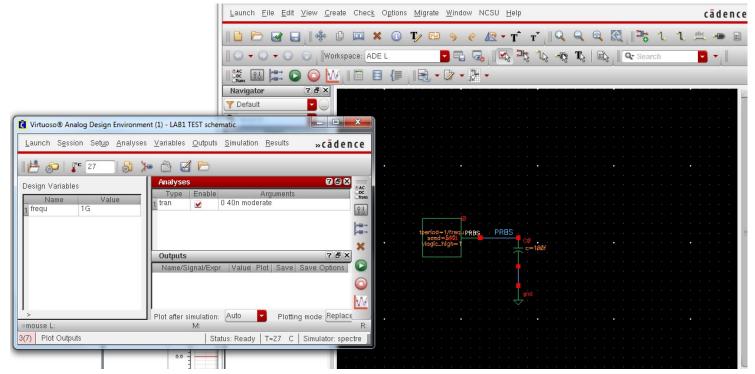


Figure 1 PRBS Generator Property

Cadence Setup



Simulation Result

