

### SAN JOSE STATE UNIVERSITY Electrical Engineering Department

# Bottomup IC designflow using CDS tools

# A tutorial guide for using CDS tools for IC design

David W. Parent Assistant Professor Electrical Engineering, SJSU One Washington Square San Jose, CA 95192-0084 Phone 408.924.3963 • Fax 408.924.2925

### Table of Contents

CHAPTER 1: DESIGN FLOW OVERVIEW	6
Specification:	7
Initial Design (Chanter 2):	8
Schematic Capture (Chanter 3)	8
L avoit (Chapter 3)	8
Design Rule Checking (Chapter 3)	8
Design Rule Checking (Chapter 4)	8
Lavactori (Chapter 4)	0 Q
Layour versus Schematic (Chapter 4)	0
Fost Extraction Simulation (Chapter 4)	9
Fabrication and Test	9
CHARTER 2. INITIAL DESIGN	40
CHAFTER Z. INTHAL DESIGN	10
rechnology Choice:	. 10
Logic Minimization:	. 10
Transistor Sizing:	. 10
Power Estimation:	. 14
Summary:	. 15
CHAPTER 3: GETTING STARTED WITH SCHEMATIC CAPTURE AND SPICE SIMULATION	16
Setting up your account:	. 16
Remote access:	. 16
Starting a terminal:	. 17
Design entry through schematic capture:	. 19
Compiling a new project directory:	. 20
Creating a schematic view:	. 21
Creating a symbol view:	. 32
Creating a test bench:	. 35
Simulation in Spectre Spice using the Affirma environment:	. 40
Transient Analysis:	. 41
DC Analysis:	. 42
Adding Wire Names:	. 44
Choosing which nets to plot.	. 45
Simulation Problems:	. 47
Measuring Propagation Delays and Vinveru	48
Measuring Power	.50
Saving your State	53
Summary	54
Summary	
	55
Cell Design Tutorial	55
Cen Design Futuria	57
Floor Planing:	57
Sizing the VDD and Ground Lines for Electromigration Eailure Prevention	58

Next, we will add the ntaps and ptaps:	
Viewing cells as an outline only.	
To add the PMOS and NMOS Transistors:	71
Adding the Input Layer:	
Wiring up the Transistors with Metal1:	
Connecting the input to all the gates:	
Adding Pins:	
Design Rule Checking (DRC)	
Summary:	
Circuit Extraction:	
Layout versus Schematic (LVS)	
Summary:	
Post Extraction Simulation	
Summary:	
•	

APPENDIX A: FURTHER WORK AND AC ANALYSIS	94
Further Work:	94
AC Analysis:	96

LIST OF FIGURES:	
Figure 1.: Starting a Terminal	17
Figure 2: Terminal	17
Figure 3: Making a directory and listing the contents of a directory	18
Figure 4: Changing your present working directory.	10 18
Figure 5. Starting CDS tools (CD-1C Front to Dack)	10
Figure 7: Library Manager	19
Figure 8: Creating a Library	20
Figure 9: CIW messages while creating a library	21
Figure 10: Updated library manager showing AMI16 project directory	21
Figure 11: Creating a new cell view	22
Figure 12: The schematic capture tool.	23
Figure 15: Adding an instance	24 91
Figure 14. Drowsing for components	24
Figure 16: Getting vdd and gnd	25
Figure 17: Getting vdd.	26
Figure 18: Stamping down vdd	26
Figure 19: Getting an nmos4 device W <sub>N</sub> =52.2µm	27
Figure 20: Getting a pmos4 device W <sub>P</sub> =93.92µm	28
Figure 21: Major symbols of the CMOS inverter	29
Figure 22: Wiring a CMOS inverter	30
Figure 23: Add pin pop-up.	31
Figure 24: Changing direction on the Y pin.	31 29
Figure 25. Completed schenaut	32 33
Figure 27: Symbol editor	33
Figure 28. Importing a symbol.	34
Figure 29: The inverter's symbol	35
Figure 30: Creating the inverter test bench	36
Figure 31: Adding the inverter symbol to the test bench	36
Figure 32: Adding the power supply (V <sub>DD</sub> from specification).	37
Figure 33: Adding a load capacitor (5001 F from specification).	38 20
Figure 54: Adding a test vector (From specification).	აი
Figure 35: The completed test banch	30
Figure 35: The completed test bench	39 40
Figure 35: The completed test bench Figure 36: Affrima Analog Environemt Figure 37: Setting up the simulator to use	39 40 41
Figure 35: The completed test bench Figure 36: Affrima Analog Environemt Figure 37: Setting up the simulator to use Figure 38: Setting up the model path	39 40 41 41
Figure 35: The completed test bench Figure 36: Affrima Analog Environemt Figure 37: Setting up the simulator to use Figure 38: Setting up the model path Figure 39: Setting up the analysis	39 40 41 41 42
Figure 35: The completed test bench	39 40 41 41 42 43
Figure 35: The completed test bench Figure 36: Affrima Analog Environemt Figure 37: Setting up the simulator to use Figure 38: Setting up the model path Figure 39: Setting up the analysis. Figure 40: Setting up a DC analysis. Figure 41: Choosing to sweep DC voltage	39 40 41 41 42 43 43
Figure 35: The completed test bench Figure 36: Affrima Analog Environemt Figure 37: Setting up the simulator to use Figure 38: Setting up the model path Figure 39: Setting up the analysis Figure 40: Setting up a DC analysis. Figure 41: Choosing to sweep DC voltage Figure 42: Completed DC Analysis Setup Figure 42: Adding Wire Narware	39 40 41 41 42 43 43 44
Figure 35: The completed test bench Figure 36: Affrima Analog Environemt Figure 37: Setting up the simulator to use Figure 38: Setting up the model path Figure 39: Setting up the analysis Figure 40: Setting up a DC analysis Figure 41: Choosing to sweep DC voltage Figure 42: Completed DC Analysis Setup Figure 43: Adding Wire Names Figure 44: Schematic with Net Names	39 40 41 41 42 43 43 44 44 44
Figure 35: The completed test bench	39 40 41 41 43 43 43 44 44 45 46
Figure 35: The completed test bench	39 40 41 41 42 43 43 43 44 44 45 46 46
Figure 35: The completed test bench	39 40 41 41 42 43 43 43 44 45 46 46 47
Figure 35: The completed test bench	39 40 41 41 42 43 43 43 43 44 45 46 46 47 48
Figure 35: The completed test bench	39 40 41 41 42 43 43 43 44 45 46 46 46 48 49
Figure 35: The completed test bench	39 40 41 41 41 42 43 43 43 44 44 44 45 46 46 46 47 48 49 50
Figure 35: The completed test bench	39 40 41 41 42 43 44 43 44 45 46 46 47 46 47 48 49 49 49 49 49 49 49 49
Figure 35: The completed test bench	39 40 41 41 42 43 44 43 44 44 45 46 46 46 47 48 49 50 51 52 52 52
Figure 35: The completed test bench	
Figure 35: The completed test bench	$\begin{array}{cccccccccccccccccccccccccccccccccccc$
Figure 35: The completed test bench	$\begin{array}{cccccccccccccccccccccccccccccccccccc$
Figure 35: The completed test bench	$\begin{array}{c} & 39\\ & 40\\ & 41\\ & 41\\ & 41\\ & 42\\ & 43\\ & 43\\ & 43\\ & 43\\ & 43\\ & 44\\ & 44\\ & 44\\ & 44\\ & 44\\ & 45\\ & 66\\ & 51\\ & 52\\ & 53\\ & 55\\ & 55\\ & 56\\ & 56\\ & 56\\ \end{array}$
Figure 35: The completed test bench	$\begin{array}{c} & 39\\ & 40\\ & 41\\ & 41\\ & 41\\ & 42\\ & 43\\ & 43\\ & 43\\ & 43\\ & 43\\ & 44\\ & 44\\ & 44\\ & 44\\ & 45\\ & 46\\ & 46\\ & 47\\ & 48\\ & 49\\ & 50\\ & 51\\ & 52\\ & 53\\ & 55\\ & 55\\ & 55\\ & 56\\ & 56\\ & 57\\ & 7\\ \end{array}$
Figure 35: The completed test bench	$\begin{array}{c} & 39\\ & 40\\ & 41\\ & 41\\ & 42\\ & 43\\ & 43\\ & 43\\ & 43\\ & 44\\ & 44\\ & 44\\ & 45\\ & 46\\ & 46\\ & 46\\ & 46\\ & 46\\ & 46\\ & 50\\ & 51\\ & 52\\ & 52\\ & 52\\ & 53\\ & 56\\ & 56\\ & 56\\ & 57\\ & 61\\ & 1\\ & 62\\ $
Figure 35: The completed test bench         Figure 36: Affrima Analog Environemt.         Figure 37: Setting up the simulator to use.         Figure 38: Setting up the model path.         Figure 39: Setting up the analysis.         Figure 40: Setting up a DC analysis.         Figure 41: Choosing to sweep DC voltage.         Figure 42: Completed DC Analysis Setup.         Figure 43: Adding Wire Names.         Figure 44: Schematic with Net Names.         Figure 45: Schematic with Nets selected for Plotting.         Figure 47: Plots of Transient and DC Analyses.         Figure 48: Adding a Horizontal Marker.         Figure 49: Results from a Horizontal Marker.         Figure 50: Final Results.         Figure 51: Adding a Power Meter.         Figure 52: Schematic with Power Meter Added.         Figure 53: Saving the State.         Figure 54: Waveform with Power (19.78mW).         Figure 55: Saving the State.         Figure 56: Documentation Browser         Figure 57: Cell design tutorial.         Figure 59: Creating a layout view of the inverter         Figure 59: Creating a layout view of the inverter         Figure 50: Layout Editor.	$\begin{array}{c} & 39\\ & 40\\ & 41\\ & 41\\ & 42\\ & 43\\ & 43\\ & 43\\ & 43\\ & 44\\ & 44\\ & 44\\ & 45\\ & 46\\ & 46\\ & 46\\ & 46\\ & 46\\ & 50\\ & 51\\ & 52\\ & 52\\ & 53\\ & 56\\ & 57\\ & 61\\ & 62\\$
Figure 35: The completed test bench         Figure 36: Affrima Analog Environemt         Figure 37: Setting up the simulator to use         Figure 38: Setting up the model path         Figure 39: Setting up the model path         Figure 40: Setting up a DC analysis         Figure 41: Choosing to sweep DC voltage         Figure 42: Completed DC Analysis Setup         Figure 42: Completed DC Analysis Setup         Figure 43: Adding Wire Names         Figure 45: Schematic with Net Names         Figure 45: Schematic with Net selected for Plotting         Figure 46: Selecting which vectors to plot         Figure 47: Plots of Transient and DC Analyses         Figure 47: Plots of Transient and DC Analyses         Figure 50: Final Results         Figure 51: Adding a Horizontal Marker.         Figure 52: Schematic with Power Meter Added         Figure 53: Editing the Capacitor's Value.         Figure 54: Waveform with Power Meter Added         Figure 55: Saving the State.         Figure 56: Documentation Browser         Figure 57: Cell design tutorial         Figure 58: Netscape Browser with Tutorial         Figure 58: Netscape Browser with Tutorial         Figure 58: Creating a layout view of the inverter         Figure 60: Layout Editor         Figure 61: LSW	$\begin{array}{c} 39\\ 39\\ 40\\ 41\\ 41\\ 42\\ 43\\ 43\\ 43\\ 44\\ 44\\ 44\\ 45\\ 46\\ 46\\ 46\\ 46\\ 46\\ 50\\ 51\\ 52\\ 52\\ 52\\ 53\\ 56\\ 55\\ 61\\ 56\\ 57\\ 61\\ 62\\ 64\\ 64\\ 64\\ 64\\ 64\\ 64\\ 64\\ 64\\ 64\\ 64$
Figure 35: The completed test bench	$\begin{array}{c} 39\\ 39\\ 40\\ 41\\ 41\\ 42\\ 43\\ 43\\ 43\\ 44\\ 44\\ 44\\ 45\\ 46\\ 46\\ 46\\ 46\\ 50\\ 51\\ 52\\ 52\\ 53\\ 56\\ 55\\ 66\\ 57\\ 61\\ 62\\ 63\\ 64\\ 65\\ \end{array}$
Figure 35: The completed test bench	$\begin{array}{c} 39\\ 39\\ 40\\ 41\\ 41\\ 42\\ 43\\ 43\\ 43\\ 44\\ 44\\ 44\\ 45\\ 46\\ 46\\ 46\\ 46\\ 46\\ 50\\ 51\\ 52\\ 53\\ 53\\ 55\\ 55\\ 55\\ 55\\ 56\\ 56\\ 56\\ 61\\ 62\\ 63\\ 64\\ 65\\ 65\\ 65\\ 65\\ 65\\ 65\\ 65\\ 65\\ 65\\ 65$
Figure 35: The completed test bench         Figure 36: Affrima Analog Environemt         Figure 36: Setting up the simulator to use.         Figure 37: Setting up the model path.         Figure 39: Setting up the model path.         Figure 39: Setting up the analysis.         Figure 40: Setting up a DC analysis.         Figure 41: Choosing to sweep DC voltage.         Figure 42: Completed DC Analysis Setup.         Figure 43: Adding Wire Names.         Figure 44: Schematic with Nett sames.         Figure 45: Schematic with Nett salected for Plotting.         Figure 47: Plots of Transient and DC Analyses.         Figure 47: Plots of Transient and DC Analyses.         Figure 48: Adding a Horizontal Marker.         Figure 50: Final Results.         Figure 51: Chematic with Power Meter.         Figure 52: Schematic with Power Meter Added.         Figure 53: Editing the Capacitor's Value.         Figure 54: Concumentation Browser         Figure 55: Saving the State.         Figure 56: Documentation Browser         Figure 59: Creating a layout view of the inverter         Figure 59: Creating a layout view of the inverter         Figure 61: Layout Editor.         Figure 62: Setting Display Options.         Figure 63: Starting a Path.         Figure 63: Inverter with VDD and Ground Lines. </td <td><math display="block">\begin{array}{c} 39\\ 39\\ 40\\ 41\\ 41\\ 42\\ 43\\ 43\\ 43\\ 44\\ 44\\ 44\\ 45\\ 46\\ 46\\ 46\\ 46\\ 50\\ 51\\ 52\\ 53\\ 56\\ 55\\ 53\\ 56\\ 56\\ 56\\ 66\\ 57\\ 61\\ 62\\ 65\\ 66\\ 66\\ 66\\ 66\\ 66\\ 66\\ 66\\ 66\\ 66</math></td>	$\begin{array}{c} 39\\ 39\\ 40\\ 41\\ 41\\ 42\\ 43\\ 43\\ 43\\ 44\\ 44\\ 44\\ 45\\ 46\\ 46\\ 46\\ 46\\ 50\\ 51\\ 52\\ 53\\ 56\\ 55\\ 53\\ 56\\ 56\\ 56\\ 66\\ 57\\ 61\\ 62\\ 65\\ 66\\ 66\\ 66\\ 66\\ 66\\ 66\\ 66\\ 66\\ 66$
Figure 35: The completed test bench Figure 36: Affrima Analog Environemt. Figure 37: Setting up the model path Figure 38: Setting up the model path Figure 39: Setting up the analysis Figure 41: Choosing to sweep DC voltage Figure 41: Choosing to sweep DC voltage Figure 42: Completed DC Analysis Setup. Figure 43: Adding Wire Names. Figure 44: Schematic with Net Names. Figure 44: Schematic with Net Names. Figure 45: Schematic with Net Names. Figure 46: Selecting which vectors to plot Figure 47: Plots of Transient and DC Analyses. Figure 48: Adding a Horizontal Marker. Figure 49: Results from a Horizontal Marker. Figure 51: Adding a Power Meter. Figure 52: Schematic with Power Meter Added. Figure 53: Selting the State Figure 54: Waveform with Power (19.78mW). Figure 55: Selting the State Figure 57: Cell design tutorial. Figure 57: Cell design tutorial. Figure 67: Cerating a layout view of the inverter Figure 61: LSW Figure 61: LSW Figure 61: LSW Figure 64: Draving the Ground Lines Figure 64: Draving the Ground Lines Figure 66: Create Instance Pop-Up.	$\begin{array}{c} 39\\ 39\\ 40\\ 41\\ 41\\ 42\\ 43\\ 43\\ 43\\ 44\\ 44\\ 44\\ 45\\ 46\\ 46\\ 46\\ 46\\ 46\\ 50\\ 51\\ 52\\ 53\\ 53\\ 53\\ 56\\ 56\\ 57\\ 61\\ 62\\ 63\\ 66\\ 66\\ 66\\ 66\\ 67\\ \end{array}$
Figure 35: The completed test bench         Figure 36: Affrima Analog Environemt         Figure 37: Setting up the simulator to use         Figure 38: Setting up the model path.         Figure 39: Setting up the analysis         Figure 40: Setting up the analysis         Figure 41: Choosing to sweep DC voltage.         Figure 42: Completed DC Analysis Setup.         Figure 42: Completed DC Analysis Setup.         Figure 43: Adding Wire Names.         Figure 44: Schematic with Nett selected for Plotting.         Figure 45: Schematic with Nets selected for Plotting.         Figure 46: Selecting which vectors to plot.         Figure 47: Plots of Transient and DC Analyses.         Figure 47: Plots of Transient and DC Analyses.         Figure 50: Final Results         Figure 51: Adding a Horizontal Marker.         Figure 52: Schematic with Netwer Meter Added.         Figure 53: Saving the Capacitor's Value.         Figure 54: Waveform with Power (19.78mW).         Figure 55: Saving the State.         Figure 56: Documentation Browser         Figure 59: Creating a layout view of the inverter         Figure 61: LSW.         Figure 62: Setting Display Options.         Figure 64: Layout Editor.         Figure 65: Inverter with VDD and Ground Lines.         Figure 64: Drawing tor the nap.	$\begin{array}{c} & 39\\ & 40\\ & 41\\ & 41\\ & 41\\ & 42\\ & 43\\ & 44\\ & 44\\ & 44\\ & 44\\ & 44\\ & 44\\ & 45\\ & 46\\ & 46\\ & 46\\ & 47\\ & 50\\ & 51\\ & 52\\ & 53\\ & 53\\ & 53\\ & 53\\ & 53\\ & 56\\ & 57\\ & 61\\ & 62\\ & 65\\ & 66\\ & 67\\$

Figure 69: Placement of the ntap	. 68
Figure 70: Ptap Pop-up.	. 69
Figure 71: Placement of the ptap	. 69
Figure 72: Setting the Display Levels.	. 70
Figure 73: Viewing cells in Outline Only	. 70
Figure 74: PMOS Transistor with a Multiple of 5	. 71
Figure 75: NMOS Transistor with a Multiple of 5	72
Figure 76: Placement of the NMOS and PMOS Transistors.	. 73
Figure 77: Metal1 to Poly Contact.	. 74
Figure 78: Placement of the Metal1 Poly Contact	. 74
Figure 79: Connect Sources of the PMOS Transistor and the ptap. Figure 79	. 75
Figure 80: Connect Sources of the NMOS Transistors and ptap.	. 76
Figure 81: Connecting Drains to form the Output	. 77
Figure 82: Connecting up the Gates of the Transistors to the Input	. 78
Figure 83: Layout of CMOS Inverter.	. 79
Figure 84: DRC with no Errors	. 79
Figure 85: Pin Creation	. 80
Figure 86: A input pin	. 80
Figure 87: Y Out put pin	. 81
Figure 88: Completed INV Layout	. 81
Figure 89: Running DRC	. 82
Figure 90: No DRČ Errors	. 82
Figure 91: Running the extractor.	. 83
Figure 92: Selecting Extract parasitic capacitances.	. 83
Figure 93: Extractor all set to go	. 84
Figure 94: Extracted inverter with no errors.	. 84
Figure 95: Extracted View of CMOS inverter	. 85
Figure 96: Running an LVS Check	. 87
Figure 97: Successful completion of an LVS check	. 87
Figure 98: Output of LVS Check	. 88
Figure 99: LVS Error Display	. 89
Figure 100: Setting for post extraction simulation	. 90
Figure 101: Inserting the extracted view	. 91
Figure 102: The extracted simulation.	. 92
Figure 103: Propagation Delay from Extracted Circuit	. 92
Figure 104: Power used by Extracted Circuit	. 93
Figure 105: Output of Sine Wave Input	. 95
Figure 106: Setting up vsin for an AC Analysis	. 96
Figure 107: Setting up an AC Analysis from 1Hz to 5GHz.	. 97
Figure 108: Results of a Transient an AC Analysis	. 97

Acknowledgements:

This tutorial is based on the NCSU design kit. For more information, see <u>http://www.ece.ncsu.edu/cadence/CDK.html</u>. This tutorial also follows the design flow used by WPI at <u>http://vlsi.wpi.edu/cadence/</u>.

This tutorial was created through a generous grant from CANENCE. Thanks to Sue King, Denny Schneider, Linda Stone, and Tina Fujijara.

# Chapter

## Chapter 1: Design Flow Overview

This chapter will give a brief overview of the bottom-up design flow we will use to design Digital ICs. This tutorial will present the usage of Cadence Tools along with the design of a .5  $\mu$ m effective channel length CMOS inverter based on the AMI06 process Technology.

This technology can be accessed for classes, senior design projects, and individual research at accredited universities free through MOSIS (<u>www.mosis.org</u>).

We have also developed instruction for getting a design ready for tapeout including such items as proposals, padframes and GDSII conversion.

#### Specification:

The specification will explain in as much detail as possible the functionality of the circuit, the conditions under which it will operate, and design constraints (for example: speed, area, target yield, and power). The technology for this design is AMIS C5N. The completion date is not specified in this example.

Table 1: Inverter Specification.

Attribute	Specification
Functionality (logic) using static CMOS circuits	Y=not(A)*
Speed	$\tau_{\text{PHL}}{=}\tau_{\text{PLH}}{=}.1\text{ns}$ Driving a 500f F load $^{\scriptscriptstyle \dagger}$
Area	Cell Height is mandated to be 20 microns. Cell Width is not constrained.
Yield	Yield needs to be high so a minimum of 2 contacts per S/D is required.
Reliability	Needs to be high so electromigration needs to be minimized.
Noise	Would like equal noise margins so $V_{TH} = V_{DD}/2$ .
Power	Less than 20mW@ 1GHz driving 500f F load, VDD=5V
Frequency	1GHz
Duty Cycle	50%
Test Vector (DC)	DC SWEEP A 0 to VDD, Measure Y out put voltage. Plot VOUT vs. VIN. $V_{TH}$ .
Test Vector (TRAN)	VPULSE with a period of 1ns, trise=.1ns and tfall=.1ns pulse width equal to .5ns.

<sup>\*</sup> For more complex functionality, a truth table would be used, or the functionality might be described in an HDL (hardware description language) such as VHDL or verilog.

 $<sup>^\</sup>dagger$  Propagation delays must always specify a load to be driven since propagation delay is directly proportional to the load capacitance.

#### Initial Design (Chapter 2):

In the initial design phase, you decide which technology to use, the functionality of your circuit, and the area and power budget. At this point, you would do any logic reduction possible using K-Maps or any other logic technique. You would size the widths and lengths of the transistors using analytical equations for width and length of your transistors. You also draw the circuit schematic out on paper. You develop test vectors to prove that your design will work.

#### Schematic Capture (Chapter 3)

You enter in the schematic and symbol for your circuit, and create a test bench for simulation. You simulate the circuit to make sure it functions properly, and it meets the time and power specification. Since analytical equations are not as accurate as spice simulation you might need to run change widths of the PMOS and NMOS transistors to get the time and power specifications to simulate properly.

#### Layout (Chapter 3)

Once the schematic matches your specification in terms of power, timing, and functionality, you draw out how the circuit would look under a microscope. These pictures are used to make the photolithography masks that are used to define your circuit on silicon.

#### Design Rule Checking (Chapter 3)

Once the circuit is laid out, you have to complete a design rule check to make sure that the circuit will not have yield problems when it is fabricated.

#### Circuit Extraction (Chapter 4)

Once you have laid out a circuit you need to extract its electrical properties to make sure that you drew the correct functionality and to estimate the parasitic resistances and capacitances that degrade circuit performance. The extracted view can be sent to the simulator.

#### Layout versus Schematic (Chapter 4)

Once you have an extracted view of your circuit, you need to run a layout versus schematic check. This makes sure that the electrical properties of your schematic match those of your extracted view. This is faster than running all your test vectors on the extracted view.

#### Post Extraction Simulation (Chapter 4)

Once you are sure that the circuits are equivalent, you run the simulation again using the extracted view. This will take into account parasitic capacitors. You might have to change the widths of your transistors slightly to match your specification. The problem at this point is that you have to change the drawing now to change the transistor and then re-extract the circuit to finally meet your spec. Then you have to go back and change your schematic as well. The better job you do at predicting circuit performance at the initial design stage and schematic capture stage, the less rework you have to do at the layout stage.

#### Fabrication and Test

The circuit is sent out for fabrication and comes back. You test it using the test vectors you developed earlier. If the circuit does not meet specification you have to start the process at the beginning using the feedback, you received from the actual devices that were made to fabricate your design.

# Chapter

### Chapter 2: Initial Design

We will choose the technology minimize the logic, and come up with our estimation of the sizes of the transistors in our inverter driver in this section.

#### Technology Choice:

This is not an easy decision to make. The smallest feature sized process may not be the most cost effective for the design you are trying to create. We choose the AMI06 process because circuit design with this technology in class can be fabricated free. We choose the static CMOS for its low power, ease of design, and low noise characteristics.

#### Logic Minimization:

Since this circuit is an inverter, no logic simplification is necessary. More complex design examples exist (or are in the process of being created).

#### Transistor Sizing:

In Table 2 and Table 3, we see the spice parameters for the AMI06 process. We will use a subset of these to design the lengths and widths of our NMOS and PMOS transistors. These are the nominal values and these values can vary by as much as 10% due to process variations. Standard static CMOS is not as prone to fluctuations of these parameters as are analog circuits or Digital circuits that use differential elements. Methodologies for layout to combat these variations will not be covered in this tutorial.

Table 2: Spice Parameters for the NMOS Transistors.

Parameter (unit)	Value
W <sub>N</sub> Minimum (cm)	1.5x10-4
L <sub>N</sub> Minimum (cm)	0.6x10-4
TOX (cm)	.014x10-4
CGDO (F/cm)	1.99x10 <sup>-12</sup>
CGSO (F/cm)	1.99x10 <sup>-12</sup>
CJSW (F/cm)	3.825632x10-12
CJ (F/cm <sup>2</sup> )	4.233802x10 <sup>-8</sup>
VT (V) @ Ln=0.6x10 <sup>-4</sup>	.6
KNP A/V <sup>2</sup>	46.3x10 <sup>-6</sup>
Length of Source or Drain (cm)	1.5x10 <sup>-4</sup>

Table 3: Spice Parameters for the PMOS Transistors.

Parameter (unit)	Value
W <sub>P</sub> Minimum (cm)	1.5x10-4
L <sub>P</sub> Minimum (cm)	0.6x10 <sup>-4</sup>
TOX (cm)	.014x10 <sup>-4</sup>
CGDO (F/cm)	2.4x10 <sup>-12</sup>
CGSO (F/cm)	2.4x10 <sup>-12</sup>
CJSW (F/cm)	3.114708x10-12
CJ (F/cm <sup>2</sup> )	7.273568x10 <sup>-8</sup>
VT (V) @ L <sub>P</sub> =0.6x10 <sup>-4</sup>	82
KNP A/V <sup>2</sup>	30x10-6
Length of Source or Drain (cm)	1.5x10 <sup>-4</sup>

To size the NMOS and PMOS transistor lengths we will choose the minimum length due to that fact that area and power are a concern. If reliability or process variation ( $V_T$  vs. Channel length for instance) were a concern, we could increase the channel length to compensate.

We will use the following equations to size the widths of the NMOS and PMOS

transistors: 
$$W_N := \frac{(C_L + C_{JSWN} \cdot 4 \cdot L_D)}{\left[\frac{\tau_{PHL}}{L_N \cdot A} - (1 + \text{Ratio}) \cdot (C_{JSWN} \cdot 2 + C_{JN} \cdot L_D)\right]}$$
(1)  
 $A := m1 \cdot \tau_{PHL} + b_1$  (2)  
Ratio :=  $m2 \cdot \tau_{PHL} + b_2$  (3)  
 $W_P := \text{Ratio} \cdot W_N$  (4)

Table 4 shows the values and explanation of the variable used in equations 1-4. Equations 1-4 are accurate to with in 12% or less when compared to spectre spice simulation, and have been tested with  $C_L$  ranging from 50f F to 10pF and propagation delays from .1ns to .5ns

To find  $W_N$  and  $W_P$ :

- 1. Solve for A using equation 2.  $A=12300\Omega$  in this example.
- 2. Solve for the Ratio using equation 3. Ratio=1.8 in this example.
- 3. Solve for  $W_N$  using equation 1.  $W_N$ =52.2 $\mu$ m.
- 4. Solve for  $W_P$  using equation 4.  $W_P$ =93.9 $\mu$ m.

**Note 1**: These values are our first estimate. To verify our design we still have to test the inverter with a more accurate model in schematic capture.

**Note 2**: We need to estimate the power of this circuit to see if it will meet our power specification before we begin working with the Cadence tools.

**Note 3**: The parameter A is for a gate length of  $.6\mu m$ . If you change the gate length, then  $V_T$  will change and as a result A and Ratio will not be as accurate.

Table 4: Explanation of terms for CMOS design.

Parameter	Description	Value	Units	
W <sub>N</sub>	Width of the NMOS	Design to Spec	cm	
W <sub>P</sub>	Width of the PMOS	Design to Spec	ст	
		Use Equation (4)		
L <sub>N</sub>	Length of NMOS	.6x10-4	cm	
C <sub>L</sub>	Load Capacitance	500f	F	
C <sub>JSWN</sub>	NMOS sidewall Capacitance	3.825632x10 <sup>-12</sup>	F/cm	
C <sub>JN</sub>	NMOS junction Capacitance of Drain bottom	4.233802x10 <sup>-8</sup>	F/cm <sup>2</sup>	
L <sub>D</sub>	Length of the Drain	1.5x10-4	cm	
TPHL.	Delay measured from $V_{DD}/2$ of input to $V_{DD}/2$ of output during the high to low transition of the output	In this case .1n	S	
А	Fitting parameter used to design $W_{\rm N}$	Use equation (2)	Ω	
Ratio	Used to scale W <sub>P</sub> to give symmetric propagation delays	Use equation (3)	unit less	
mı	Fitting parameter (slope) to find A	-5x10 <sup>12</sup>	Ω/s	
m <sub>2</sub>	Fitting parameter (slope) to find Ratio	-250x1 <sup>06</sup>	Hz	
b <sub>1</sub>	Fitting parameter (y intercept) to find A	12800	Ω	
b <sub>2</sub>	Fitting parameter (y intercept) to find Ratio	1.825	unit less	

#### Power Estimation:

We will use the following equations to estimate the power used by our circuit:

$$C_{\text{DTOTAL}} := C_{\text{JSWN}} \cdot 2 \cdot \left[ W_{\text{N}} \cdot (1 + \text{Ratio}) + 2 \cdot L_{\text{D}} \right] + C_{\text{JN}} \cdot W_{\text{N}} \cdot L_{\text{D}} \cdot (1 + \text{Ratio}) (5)$$

$$C_{\text{TOTAL}} := C_{\text{L}} + C_{\text{DTOTAL}} (6)$$
Power :=  $\alpha \cdot f \cdot C_{\text{TOTAL}} \cdot V_{\text{DD}}^{2}(7)$ 

Table 5 gives the details of parameters used in equations 5-7.

Power =  $17.67 \times 10^{0}$  m w in this example, so we are with in specification and may proceed with the rest of the design.

Table 5: Explanation of terms for CMOS design.

Parameter	Description	Value	Units
C <sub>DTOTAL</sub>	Total NMOS and PMOS drain capacitance	Use equation	F
CL	Load Capacitance	500f	F
C <sub>TOTAL</sub>	Total Capacitance the inverter must drive	Use equation	F
f	Frequency	1G	Hz
α	Activity Factor (A clock has an activity factor of 1)	1	unit less
V <sub>DD</sub>	Supply Voltage	5	V
W <sub>N</sub>	Width of the NMOS	$52.2 \times 10^{-4}$	cm
WP	Width of the PMOS	93.9x10 <sup>-4</sup>	cm
L <sub>N</sub>	Length of NMOS	.6x10 <sup>-4</sup>	cm
C <sub>JSWN</sub>	NMOS sidewall Capacitance	3.825632x10 <sup>-12</sup>	F/cm
C <sub>JN</sub>	NMOS junction Capacitance of Drain bottom	4.233802x10 <sup>-8</sup>	F/cm <sup>2</sup>
L <sub>D</sub>	Length of the Drain	1.5x10-4	cm

#### Summary:

We have introduced some equations for sizing the widths and lengths of the inverter example as well as estimating power consumption. We have come up with an initial design and verified that our circuit will meet the timing and power portions of our specification. Next, we will verify the timing and power consumption of our circuit using the schematic capture, and spectre spice simulation tools.

# Chapter

## Chapter 3: Getting started with Schematic Capture and Spice Simulation

Now that we have estimated  $W_N$ ,  $W_P$  and power of our inverter we need to verify the design using a more sophisticated model.

This Chapter will go over:

- 1. Setting up your account
  - a. Design Entry through schematic capture (input  $W_{\rm \scriptscriptstyle N}$  and  $W_{\rm \scriptscriptstyle P})$
  - b. Create INV schematic capture
  - c. Create INV symbol
  - d. Create INV Test bench (use our test vectors indicated in the specification)
- 2. Simulation using CDS's Spectre (Bsim3 model simulator, different underlying algorithms than spice or hspice)

#### Setting up your account:

Your account should have the all paths set to run the software. You need only to log in and start a terminal. There should be a terminal icon on your Common Desktop Environment. Just double click on it and a command window will appear (Figure 2). You type commands in this window just like an MSDOS command line except that the commands are different.

#### Remote access:

Instructions on remote access can be found at: http://www.engr.sjsu.edu/~dparent/ICGROUP/UNIX.pdf

#### Starting a terminal:

To start a terminal right click in the middle of the screen of the CDE. While holding the right mouse button down move, the mouse down until tools is highlighted, and then move the mouse over to tools and highlight terminal. (This should look like Figure 1) Let up the right mouse button and a terminal should start.

Mr.	Mr.	Mr.	Mr.	SM/	Mr.	SH-	Mr.	Mr.	Mr.	SH-	
Solaris											Sc
Mr.	Mr.	M	Mr.	Mr.	SH-	SH-	>1/-	Mr.	M	Mr.	
SOLARIS"											Sc
Mr.	Mr.	SM-	Mr.	SM/	SM/	Mr.	Mr.	Mr.	M	Mr.	1
Solaris											Sc
Mr.	JH-	Mr.	JM/	Mr.	SH-	SH-	>1/-	Mr.	Mr.	Mr.	
SOLARIS"											Sc
Mr.	Mr.	Mr.	Mr.	Mr.	SH-	SH-	SH-	Mr.	Mr.	M.	2
SOLARIS"											Sc
JH-	Mr.	M	M	Mr.	Workspace	Menu	JM/	Mr.	M	Mr.	
SOLARIS"				SOLARIE	Applications Cards	≥ RIS					Sc
JM/	M	M	M	JM_	Files Folders		Mr.	M	M	Mr.	
Solaris"				Solarie	Help Hosts	RIS					Sc
Mr.	M	M	Mr.	JH/	Links Mail		2M	Mr.	M	Mr.	
SOLARIS"				Solarie	Tools		Tools	ARIS			Sc
JH-	Mr.	Mr	Mr.	JM-	Add Item to	Menu SCI	imintool reate Action	4	M	Mr.	
SOLARIS"				SOLARIE	ELOCK Display	ienu 🖆 Di V 🖉 Hi	esktop Control otkey Editor	s			Sc
JH-	Mr.	M	Mr.	All C	🕏 Suspend Syste ब्व Log out	em 🔫 Po Ali Pi	ower Manager Ant Manager	4	M	M	
SOLARIS"					SOLARIS"	SOLA	inter Administ	rator ARIS			Sc
July -	M-						nart Card			1/2	
SOLARIS"	SOLA	Jun 21	<b>F</b>		Four	i innee 🔍 i e	atch Errors		?	ARIS'	Sc
114		-					1880			/	

Figure 1.: Starting a Terminal

🗙 xterm				_ 🗆 ×
Sun Microsystems cadence1% ∎	Inc.	SunOS 5,8	Generic February 2000	

Figure 2: Terminal

After you have started the csh, you need to create a project directory. To do this, type the command **mkdir cell** at the command line. This command makes a directory cell in your home directory. **You only have to create this directory once!** Type the command **ls**. You should get

the result cell as in Figure 3. The command ls lists out the content of your present working directory.

🗙 xterm			- 🗆 🗵
Sun Microsystems Inc. cadence1% mkdir cell cadence1% ls cell local. cadence1%	SunOS 5,8 cshrc.org	Generic February 2000	

Figure 3: Making a directory and listing the contents of a directory.

In order to start CDS tools so that your project files are available you need to change into your project directory before starting the tools. Type in the command **cd cell** at the command line. This changes your present working directory to cell. If you type the command **ls**, you should see no files (Figure 4).

🗙 xterm	
cadence1% cadence1% cadence1% cadence1% cd cell cadence1% ls cadence1%	

Figure 4: Changing your present working directory.

To start CDS tools type in the command **icfb &**. You should see messages similar to Figure 5. After some time, the CIW (Command Interface Window) will pop up (Figure 6). Once the CIW come up you will not need to use the command line.

🗙 xterm	
cadence1% cadence1% cadence1% icfb & [1] 723 cadence1% sh: /usr/bin/%11/%lsfonts: not found	

Figure 5: Starting CDS tools (icfb-IC Front to Back)

🗙 icfb - Log: /home/eecad40/CD5.log							
File Tools Options	Technology File	Help	1				
COPYRIGHT © 1992-2000 © 1992-2000 This Cadence Design 9	CADENCE DESIGN SYSTEMS INC. ALL RIGHTS RESERVED. UNIX SYSTEMS Laboratories INC., Reproduced with permission. Systems program and online documentation are						
Loading NCSU SKILL :	coutines]						
mouse L:	M: R:						
>							

Figure 6: CIW

#### Design entry through schematic capture:

This is the beginning of the design flow as far as using CDS tools is concerned. Prior to starting the tools you need to have define your logic function, decided on an implementation, reduced the number of gates, and done some initial sizing of the lengths and widths of your transistors.

Now that you have created your project directory, whenever you want to work on the design in that directory you just have to:

- 1. Log in
- 2. Start a terminal
- 3. Type in the command, **cd cell** (or the name you selected for your project)
- 4. Type in the command, **icfb &**.

We will be using the NCSU design kit, which automatically starts the library manager (Figure 7). You should see three NCSU libraries and a library named basic and cdsDefTechLib. There is also a sample directory called ANALOG and PADFRAME.

🗽 Library Manager: WorkArea: /he	ome/eecad40/cell	
<u>File E</u> dit <u>Vi</u> ew <u>D</u> esign Mai	nager	<u>H</u> elp
☐ Show Categories	w Files	
- Library		- View
Ĭ.	Ľ	Ĭ
ANALOG NCSU_Analog_Parts NCSU_Digital_Parts NCSU_Sheets_8ths PADFRAME basic cdsDefTechLib		
- Messages		<u>ا</u>
Loading NCSU Library Manage	er customizationsdone.	

Figure 7: Library Manager

#### Compiling a new project directory:

We need to create a project library and compile a technology library to it. The tech library contains all the process specific information needed to design an IC using a particular fabrication house's process. We will be using AMI 0.6u C5N (3M, 2P, High Res) process.

To compile your new project library:

- 1. Goto the library manager popup and goto: file... New... library. A pop-up like Figure 8 should appear.
- 2. Fill out the pop up exactly according to Figure 8. Make sure to click on compile tech library.
- 3. Click OK. You should see messages in the CIW similar to the ones in Figure 9.
- 4. Your library manager should now show your new project directory as in Figure 10. Make sure that your library has all the components listed in column 2 for Figure 10.

Create Library	×
OK Cancel Apply	Help
Library	F
ັນນາ04	¬
Name: Jan 100	
Path:	
	-
If this library will not contain physical design (i.e., layout) data you do not need a tech library	- II
Otherwise, you must either attach to an existing tech library or compile one.	
Choose option:	
✓ No tech library needed	
Attach to existing tech library AMI 0.60u C5N (3M, 2P, high-res)	d
Compile tech library	-
Misc. /	_
I/O Pad Type: 🔶 Perimeter 🗸 Area an ay	7
igure 8: Creating a Library	
Make sure you set the tech library to A	<b>MI0.6</b>
AMI16u. You will have to redo your	t <b>utoria</b>
incorrectly!	

🗙 icfb - Log: /home/e	ecad40/CD5.log		
File Tools Options	Technology File		Help 1
prog((libID libNam createLibCB(formSt hiFormDone(NCSU_cr	e path bag tech ruct@0x295c348) eateLibForm)	<pre>(libName = ((createLibForm-&gt;lib))</pre>	Name)->value)) (path = car(pa:
Loading NCSU SKIL	L routines		
mouse L:	:	4: F	R:
>			

Figure 9: CIW messages while creating a library

🗱 Library Manager: WorkArea: /1	home/cecad40/cell	크미즈
File Edit View Design Ma	anager	Help
_ Show Categories Sho	w files	
Library	Cell	View
<b>Т</b> АХІ D6	I	1
MATIO6 MAIO0 MOSU_Aralog_Parts MOSU_Ingital_Parts MOSU_ideets_Oths PADYAMU basic cdaDefTechLib	N1_KLDC         A1           N1_P         N1           N2_N1         N3           N3_N2         N1           N3_N2         N1           N1_P         N1           Sotive         elsc           n1_n         n1_n           n1_p         n1_poly           n3_w2         nd_w1           n4_w1         N1           n4_w1         N1           n4_w1         N1           n4_w1         N1           n4_w1         N1           n4_w2         N2           n4_w2         N2           netal1         netal2           netal3         //	
Messages		
Loading NCSU Library Manag J.	er customizationsdome.	
C)		12

Figure 10: Updated library manager showing AMI16 project directory.

#### Creating a schematic view:

After creating the library AMI06, we can now start adding our circuit views. Each circuit will have different views. For example, our inverter will have a schematic, symbol, layout, and extracted views. There are more views than these, but these are the only ones we will use in this course.

- 1. Schematic view: The circuit is described by electronic symbols connected by wires. A spice run deck can be compiled directly from this view. For example, an inverter would have two transistors, a power supply, ground, and import and output ports.
- 2. Symbol view: The circuit is described by one symbol. For example, an inverter would look like the Boolean logic symbol for an inverter.
- 3. Layout: The circuit is drawn, as it would appear if you looked at in under a microscope. Each layer corresponds to a process step.
- 4. Extracted: CDS tools can extract a schematic from the drawn layers. This is helpful to make sure you drew the circuit so that it will perform like the schematic. The extracted

view will include parasitic capacitances and resistances from the drawn layers that the schematic view cannot generate.

To create a schematic view:

- 1. Goto the library manager popup and goto: file... New... cell view. A pop-up like Figure 10 should appear.
- 2. Fill out the pop up exactly according to Figure 11.
- 3. Click OK. The schematic entry tool should appear (Figure 12).

🔀 Create	New Fil	e			×			
ок	Canc	el	Defaults		Help			
Library N	lame	_	AMIO	6	_			
Cell Nam	e	Ι	NV					
View Nar	ne	schematic						
Tool Composer-Schematic _								
Library p	ath file							
/home/e	ecad40	/ce	ell/cds.l	ibį́				

Figure 11: Creating a new cell view

Cmd	:				Se	1:	0																						i																							
Fools	Desi	gn	w	ind	low	1	E	lit		Ad	d	C	Cheo		Chec		<b>heck</b>		Chec		Chec		Che		Che		Che		heck		:k		:k		ck		ck		eck		Sheet		et	Options		ptions		s NCSI		CSU		
-2																																																				
⊻	• • •	• •		•	• •			•	•	•		•	•	•	•	•	•				•	•	•		•	•		•	·																							
			1	2				÷	Ì.	2	2	2				Ì.	1		2						÷	1	2	2																								
Ŷ. I																													·																							
	• • •	• •		·	• •			·	•		•	•				•	·		•		•					·	•		·																							
32			1					÷	Ì	2	2	2				Ì	1								÷	1		2	1																							
÷																													·																							
a 19	• • •	• •		•	• •	-		·	•	•			-			•		-		-				-				•	·																							
24				2				÷		2	2	2			2		1				2				2	1		2																								
																													·																							
	• • •	• •		•	• •			•	•							•		-		-		•		-					·																							
~			1	2				÷		2	2	2			2		2								2	2	2	2	1																							
		• •			• •																								·																							
YU -		•			• •				•							•		-		-				-					1																							
3								÷									÷									÷																										
///		• •			• •																								·																							
/		•			• •				•							•		-		-				-					1																							
æ.								÷									÷									÷			1																							
<u>)</u>		• •			• •																								·																							
			1	:				1	:	1	:	:	:	:		:	1				1	:	÷		÷	1		1	1																							
858									Ì							Ì						Ì	Ì																													
翻		• •			• •																								·																							
-, 1		: :	1	1			1	÷	Ì	1	1	2	2	2	2	Ì	÷	2	1	2	1	Ì	Ì	1	÷	÷	1	1	<u>;</u> ],																							
- ( I,																																																				
	<u> </u>					_			_		,		_	_	_	_		_		P		_	_	_	_				***																							
ml	mouse	Ц:								N	11									R	11																															

Figure 12: The schematic capture tool.

The schematic capture tool will be used to draw the schematic representation of your inverter.

To draw the inverter you need to add a pmos4 transistor, nmos4 transistor, ground connection, power supply connection, input pin, output pin, and wire it together. Use the lengths and widths calculated in Chapter 2.

To add items to your schematic goto add... instance in the composer tool, or press the letter i. The add instance pop-up appears like in Figure 13.

Click on browse to graphically get components. A pop-up like Figure 14 should appear.

🔀 Add In	stance	×
Hide	Cancel Defaults	Help
Library		Browse
Cell	¥	
View	symbol	
Names	Y	
Array	Rows 1 Columns	1.
Rotat	e Sideways L	lpside Down

Figure 13: Adding an instance

🗙 Compone	nt Browser	_	미지
Commands	;	Help	3
Library Flatten	AMIO	16	
Filter	۲		1
Uncate layout sym_cc sym_pi	egorized :_macros untacts .ns		-

Figure 14: Browsing for components

To get the parts we need change the library according to Figure 14.

Component Browser
Commands Help 3
Library NCSU_Analog_Parts 🔟
Flatten 🔟
Filter *
Uncategorized
Current_Sources
H_Spice_Only
Microwave_Parts
N_Transistors
P_Transistors
Parasitic_Devices R L C
*====  /
۷/ ×

Figure 15: Getting parts

Click on supply nets and the pop-up should look like Figure 16. The items in supply nets are actually global signals. Global signal are automatically given pins. This makes our symbols cleaner because we only show logic ports.

🗙 Compone	nt Browser	_	
Commands	•	Help	3
Library	NCSU_Anal	log_Pari	<b>is</b>
Flatten	_		
Filter	*		]
powers vcc vcca	Supply		
vccd vdd vdda			
vddd vee veea			
veed vss		-	, ,
Supply	_Nets		

Figure 16: Getting vdd and gnd.

The add instance pop-up should look like Figure 17.

Stamp it down in your schematic like in Figure 18.

🔀 Add In	stance	×
Hide	Cancel Defaults	Help
Library	NCSU_Analog_Parts	Browse
Cell	vdď	
View	symbol <u>i</u>	*
Names	Ĭ	-
Array	Rows 1 Columns	1
Rotat	e Sideways I	Jpside Down

Figure 17: Getting vdd.

Crind	t In	star	IC R	,			1	Se	l: 0																			
0İS	De	sig		w	'ne	law	r	B	at.	ł	<b>V</b> di	d	a	18	:k		8	i e e	t	(	Dp	U	27	5	N	IC:	su	
11													1															
-													- 1															
. I													- 7															
6 I.																												
<i>•</i>																												
21																												
-																												
21																												
6.1																												
2																												
i. I.																												
91																												
-																												
вĿ																												
21																												
-																												
<i>8</i>																												
-																												
١I																												
5																												
-																												
51																												
17 I																												
-																												
3																												
5 II	1			_	-			-	-	_	-	_		_	-	-	_	-	_	-		-	-		-	_	_	2
-1	Lau	ae 1	L :								ю									R								

Figure 18: Stamping down vdd.

Get the gnd symbol and stamp it down in the same manner as you did for vdd.

To add the transistors click on N\_Transistors for the nmos4 and P\_Transistors for the pmos4. Add an nmos4 and a pmos4 transistor in a similar manner. The pop-up for the nmos4 should look like Figure 19, and the pmos4 pop-up should look like Figure 20. **NOTE: If your transistor does not have the widths and lengths like in Figure 20 you did not compile the tech library correctly. You need to start over!** 

#### Use the W<sub>N</sub> and W<sub>P</sub> values from chapter 2!

Stamp them down like Figure 21. It does not have to be exact but neatness will help further.

If you make a mistake and need to get out of add instance mode, press the esc key.

Click on the object you want to delete and press the del key.

🔀 Add In:	stance					X
Hide	Cancel	Defaults				Help
Library Cell	NCSU_An	alog_Part:			Broy	wse
View	, symbolį́				-	
Names	Ĭ				-	
Array	I	Rows 1		Columns	1.	_
Rotat	e	Side	eways		Upside D	own
Model na	976		ami06	N		
Model Ty	/pe		🔶 sys	tem 🗸 use	er	
Multiplier			1			
Fingers			1			
Width (gr	id units)		348			
Width			52.2u	Ц М <u>і́</u>		
Willh (m	isimen)		1.5u	М		
Length ((	grid units)		4			
Length			600n	M		
Length (r	nisiimam)		600n	M		/

If you cannot enter in the width exactly as shown or it does not say ami06N under model name, you have not complied the directory correctly. You will have to delete this directory and start over!

Figure 19: Getting an nmos4 device  $W_N$ =52.2 $\mu$ m.

🔀 Add In:	stance						X
Hide	Cance	l Defa	ults				Help
Library	NCSU_	Analog_	Parts			B	rowse
Cell	pmos4					_	
View	symbol	lį				_	
Names						_	
Array		Rows	1.		Columns	1	
Rotat	e	_	Side	ways		Upside	Down
Model na	me			ami0	6P		
Model Ty	pe			🔶 sy:	stem 🗸 us	er	
Multiplier				1			-
Fingers				1			-
Width (gr	id units)	)		626			-
Width				93.91	u M <u>í</u>		-
Willth (m	isiernem)			1.5u	M		
Length (g	grid unit:	s)		4			7
Length				600n	M		- /

Figure 20: Getting a pmos4 device  $W_P$ =93.92 $\mu$ m.

X Virtu	oso® Sch	ematic Ed	iting: A	MI06	INV sche	matic			$\mathbf{X}$
Cmd	: Move	UEndour	Sel: U	0.1.1	Charak	Oheet	Outions	NCOLL	2
TOOIS	Design	window	Eart	Add	Спеск	Sneet	Options	NCSU	
									H
						 La			
- <b>X</b> . 1					· _ *	I <b>O</b>			
R 1					· · ·				
					_ <mark>_</mark> ₽Ø				
				•		©6P ⊧0.3 011			
						600n			
					<b></b> m:1				
Alitisen:					· · ·				
					<b>_</b> NØ				
1/1				- <b>-</b>		06N 52 20			
<i>.</i>						600n			
$ \land $					<b>—</b> m:1				
Ě.									
	• •				ar	nd .			
解释	• •				. ⊥ <u>,</u>	-			
							· ·		J7
	Monise Lo			M ·			R ·		-
	Point at c	biect to n	nove	11.					-

Figure 21: Major symbols of the CMOS inverter

All that remains to be completed is to wire the connections together and add inport and output pins.

To add a wire, press w. The wire will snap to place at the proper ports of each device.

Wire it up like Figure 22.



Figure 22: Wiring a CMOS inverter.

To add the input and output pins, goto add... pins and a pop up like Figure 22 should appear. Fill it out exactly like Figure 22. The pins names must match the pins names of the symbol view we are going to create later. If they do not match (direction or name), the design, check & save routine will fail.

🗙 Add Pin			×
Hide Cance	Defaults		Help
Pin Names	A A		
Direction	input 💷	Bus Expansio	n 🔶 off 🗸 on
Usage	schematic 🔟	Placement	< single 🗸 multiple
Rotate	Si	deways	Upside Down

Figure 23: Add pin pop-up.

Stamp down the input pin like in Figure 25.

Change the direction of the Y pin in the add pin pop-up to output as in Figure 24.

Stamp it down like in Figure 25.

🗙 Add Pin		×
Hide Cance	Defaults	Help
Pin Names	Я	
Direction	output 💷 🛛 Bus Expa	ansion 🔶 off 👡 on
Usage	schematic 💷 🔷 Placemer	nt 🛛 🔶 single 🗸 multiple
Rotate	Sideways	Upside Down

Figure 24: Changing direction on the Y pin.



Figure 25: Completed schematic.

To connect the substrate of the pmos transistor to vdd, press the esc key to get out of the add pin mode and then press w to enter the wire mode. While in wire mode connect the nmos substrate to ground as well.

**To save and check your schematic for errors, goto... design... check &save.** Any errors will be highlighted in the schematic window. Usually something is not connected, or a port name is wrong. You can exit the schematic capture tool at this time (goto Window .... Close).

#### Creating a symbol view:

We have entered our initial design into a schematic view of the inverter. To use a test bench we must have a symbol view of the inverter. **Do not use this inverter's schematic view as a testbench other wise it will not pass LVS!** 

To create a symbol view:

1. Goto the library manager popup and goto: file... New... cell view. A pop-up like Figure 26 should appear.

2. Fill out the pop up exactly according to Figure 25. Click OK. The symbol editor tool should appear (Figure 27).

XCreate	New File			×
ок	Cancel	Defaults		Help
Library N	lame _	AMIO	6	
Cell Name	e	INA		_
View Nan	ne [	symbol		
Tool	_	Composer	- Symbol	
Library p	ath file			
/home/dj	parent/c	ell/cds.l	ibį́	

Figure 26: Creating a symbol

🗙 Virtu	IOSO®	Syn	nbol	Edit	in;	g: AMJ	(06 ]	INV	syn	nbol							×
Cmd	:			Se	91: 1	0											4
Tools	Desi	gn	Wir	ndov	v	Edit	Ad	d	Che	ck	Ор	tion	s		H	leip	
- <b>S</b>		•	-	•	•	-	•	•	•	-	•	-	•	•	•	•	I
$\mathbb{R}^2$	•	•	•	•		•	•	•		•	•	•	•	•	•	•	
<b>Q</b> <sup>2</sup>																	
				•	•	•		•					•		•	•	
	•		•	•	•		•	•		•	•			•	•	•	_
	nouse	<b>L</b> :	mou	seSi	ub:	Sele	M :					R:					-

Figure 27: Symbol editor.

The symbol editor allows you to draw any kind of shape to represent you logic gate. The important part is to have the input and output pins match whatever it is you are representing. In this case, we need to match the input pin A and output pin Y of the inverter. Rather than

drawing an inverter symbol from scratch, it is much easier to copy one that has already been drawn.

To import a symbol:

- 1. Goto add... import symbol. A pop-up like Figure 28 should appear. Click on browse and select the inverter in the digital library of the NCSU kit. The pop-up should look exactly like Figure 28.
- 2. Stamp it down in the symbol editor like in Figure 29.
- 3. Press the esc key to get out of add symbol mode.
- 4. Press the f key to fit the symbol in your window.
- 5. Goto design... check & save. You should have no errors. If there are errors, the pins must be wrong in your schematic. Go back to your schematic and fix the pin assignments. Check for direction and name. If there is one pin misnamed there will be two errors:
- 1. A pin in the schematic view was not found in the symbol view.
- 2. A pin in the symbolic view was not found in the schematic view.

🗙 Import	: Symbol		×
Hide	Cancel	Defaults	Help
Library N Cell Nam	lame e	NCSU_Digital_Partă	Browse
View Nar	ne	symbol	
Rotat	e	Sideways	Upside Down

Figure 28: Importing a symbol.



Figure 29: The inverter's symbol.

To close the symbol editor, goto window... close.

#### Creating a test bench:

Now that you have created a schematic and symbolic view of your inverter, it is time to create a test bench. This will be a virtual test bench, but it will have to have a power supply, input vectors, and a ground (plus the inverter) to test its transient response and DC response.

To create the inverter test bench:

In the library manager, goto file... new... cell view. A pop-up like Figure 29 should appear. Fill it out exactly like Figure 30 and click on ok. The schematic editor will appear.
🗙 Create New File 🔀 🔀					
ок	Cancel Defaults He				
Library Name			AMIO	6 _	
Cell Name					
View Name schematic			-		
Tool	Tool Composer-Schematic 💷				
Library path file					
/home/dparent/cell/cds.lib					

Figure 30: Creating the inverter test bench.

In the schematic editor press i or goto add... instance. Click on browse, select library AMI06, click on flatten and select your inverter as in Figure 31. Stamp it down in your schematic.

While still in add instance mode add the gnd and vdd global symbols just like you did when creating the schematic for your inverter.

Add the power supply (vdc from voltage sources in NCSU\_Analogparts) according to Figure 32. Make sure set the DC voltage to 5 volts.

🔀 Add In	stance	×
Hide	Cancel Defaults	Help
Library	AMI06	Browse
Cell	INV	-
View	symboli	r
Names	Y 	F
Array	Rows 1. Columns	1
Rotat	e Sideways L	Jpside Down

Figure 31: Adding the inverter symbol to the test bench.

🔀 Add In	stance			×
Hide	Cancel	Defaults		Help
Library	NCSU_An	alog_Partšį́		Browse
Cell	vdc			-
View	symbol			-
Names				-
Array		Rows 1	Columns	<b>,</b> 1
Rotat	e	Sideway	ys	Upside Down
AC magn	itude	Ĭ		
AC phase	)	Ĭ.		
DC volta	ge	5 ¥.		
Noise file	e name	Ι		
Number (	of noise/fr	eq pairs 🛛 🖞		

Figure 32: Adding the power supply (V<sub>DD</sub> from specification).

Add the load capacitor according to Figure 33. Make sure it is 500f F not 500 F! This simulation will not work if the capacitance is too big!

Add the test vector (vpulse) according to Figure 34.

🔀 Add In	stance			×
Hide	Cancel	Defaults		Help
Library	NCSU_An	alog_Partšį́		Browse
Cell	capi			.
View	symbol			.
Names	¥			.
Array	l	Rows 1	Columns	1
Rotat	te	Sideways		Jpside Down
Rotat	te	Sideways	<u> </u>	Jpside Down
Rotat Capacita	nce	Sideways		Jpside Down
Rotat Capacita Initial con	nce ndition	Sideways	Y	Jpside Down
Rotat Capacita Initial con Model na	nce ndition	Sideways	Y Y	Jpside Down
Rotat Capacita Initial col Model na Width	nce ndition ume			Jpside Down
Rotat Capacita Initial con Model na Width Length	nce ndition ume	Sideways	У Т	Jpside Down

Figure 33: Adding a load capacitor (500f F from specification).

🔀 Add Ins	stance				×
Hide	Cancel	Defaults			Help
Library	NCSU_Ar	halog_Part	tš		Browse
Cell	vpulse				_
View	symbol				
Names	j				-
Array		Rows	1	Columns	1
Rotat	e	Sic	leways		Upside Down
AC magni	tude		¥		
AC phase			,ŭ.		
Voltage 1			0 V		[_
Voltage 2			5 V <u>ř</u>		
Delay tin	ie		∐0 s		
Rise time	1		100.0p	S.	
Fall time			100.0p	š	
Pulse wid	lth		500p š		
Period			ln s		

Figure 34: Adding a test vector (From specification).



Figure 35: The completed test bench

Press the esc key to get out of add instance mode. Move the symbols around by pressing the m key for move.

Press the esc key to get out of move mode and wire up the schematic according to Figure 35.

Goto Design... Check & Save to check for errors.

You are now ready to simulate your inverter!

If you receive warnings and errors about pins being in one view and not the other, you have either name differences in the schematic vs. symbol or pin direction differences in schematic vs. symbol.

The symbol should be perfect so check the schematic you did to make sure the pins are labeled correctly and the directions are correct.

# Simulation in Spectre Spice using the Affirma environment:

Now that you have created a schematic and symbolic view of your inverter and created a test bench to test your inverter, it is ready to run the Affirma Analog environment tool. Do not be confused because we are in a digital design class! We are simulating a digital circuit in a physics based environment that solves for voltages and currents over time. This is different that an HDL, which is just doing a high, low, Z or X analysis.

To verify your circuit:

Goto tools... analog environment. A pop-up like Figure 35 should appear. You will need to customize your environment the first time you run Affirma.

🗙 Affirma Analog Circuit Desig	n Environment (1)	
Status: Ready	T=27 C Simulator: spectres	s 7
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	Ł
Library AMIO6	# Type Arguments Enable	JAC ■ TRAN JDC
Cell INV_TB View schematic		
Design Variables	Outputs	ŒŰ,
# Name Value	# Name/Signal/Expr Value Plot Save March	<b>3</b>
		8
		8
>	1	$\sim$

Figure 36: Affrima Analog Environemt.

You will be using the Spectre Spice simulator, which is a spice like simulator, using BSIM3 model decks, but the underlying algorithms are different. These algorithms are transparent to the user.

Goto Setup... Simulator/Directory/Host. A pop-up like Figure 36 should appear. Make sure the Simulator is set to spectreS. Click ok when done.

🗙 Choosing Simulator/Directory/Host Affirma Analog Circuit Design Environmen 🗙				
ок	Cancel	Defaults	Help	
Simulato	r	spectr	es _	
Project D	irectory	~/caden	ce/simulation]	
Host Mod	le	🔶 local 🦂	🗸 remote 🔍 distributed	
Host				
Remote (	)inectory			

Figure 37: Setting up the simulator to use.

Now you need to make sure that Affirma can find the model decks for our process.

Goto Setup... Model path and a pop-up like Figure 37 should appear. Make sure it matches exactly the model path in the pop-up.

🗙 Setti	ng Model	Path Affirma	a Analog Circuit Design Environment (1)	×
ок	Cancel	Defaults App	ply Apply & Run Simulation	Help
Directo	ries	/home/ee /apps/ca	ecad40/cadence/models/spectre adence/local/models/spectre/nom	
New Di	rectory			
		Add Abov	ve Add Below Change Delete	
Corner		session-d	default	
		New Con	ner Copy Corner Delete Corner	

Figure 38: Setting up the model path.

#### Transient Analysis:

The transient analysis will allow us to measure propagation delay times. Our specification calls for 100ps for both propagation delays. To set up a the transient analysis: Goto Analyses... Choose and fill out the pop-up according to Figure 38. In order to measure  $V_{\rm INV}$  (Where  $V_{\rm OUT}=V_{\rm IN}$ ) we need to do a DC analysis as well.  $V_{\rm INV}$  is a measure of the noise performance and the closer  $V_{\rm INV}$  is to  $V_{\rm DD}/2$  the better the noise performance of the inverter.

🗙 Choosing Analyses Affirma Analog Circuit Design Environment 🔀					
OK Cancel Defa	ults Apply		Help		
Analysis 🛛 🔶 tran	√ ac	🗸 sp	🗸 pdisto 🗸 spss		
√ dc	√ xf	<b>√pss</b>	√ noise		
	Transient	Analysis			
Stop Time 2n					
Accuracy Defaults (errpreset) _  conservative _  moderate _  liberal					
Enabled 🔟			Options		

Figure 39: Setting up the analysis

#### DC Analysis:

In order to measure  $V_{\rm INV}$  (Where  $V_{\rm OUT}{=}V_{\rm IN}$ ) we need to do a DC analysis as well.  $V_{\rm INV}$  is a measure of the noise performance and the closer  $V_{\rm INV}$  is to  $V_{\rm DD}/2$  the better the noise performance of the inverter.

Goto Analyses... Choose and fill out the pop-up according to Figure 40.

Click on Select Component and then in the schematic click on the vpulse.

Select VDC when the pop-up like Figure 41 appears.

Select the start and stop voltages to be 0 and 5. Your pop-up should look like Figure 42 Note: When simulating a Schmitt trigger you will have to do this analysis twice 0 to  $V_{DD}$  and  $V_{DD}$  to 0 to get the hysterisis curve.

Choosing Analyses Affirma	Analog Circuit Design Environment 🗙
OK Cancel Defaults App	ly Help
Analysis 🗸 tran 🗸 ac	√sp √pdisto √spss
🔶 dc 🔍 xf	√pss √noise
DC /	Analysis
Save DC Operating Point	1
Sweep Variable _ Temperature Tomponent Parameter _ Model Parameter	Component Name Select Component Parameter Name
Sweep Range Start-Stop Start Center-Span Sweep Type Automatic	š. Stop
Add Specific Points 🔟	
Enabled 🔟	Options

Figure 40: Setting up a DC analysis.

🗙 Select C	omponent Par	ameter X
_ок _а	ancel	Help
do	vdc	"DC voltage"
mag	acm	"AC magnitude"
phase	acp	"AC phase"
val0	v1	"Voltage 1"
val1	<b>v</b> 2	"Voltage 2"
period	per	"Period"
delay	td	"Delay time"
rise	tr	"Rise time"
fall	tf	"Fall time"
width	pw	"Pulse width"
tc1	tc1	"Temperature coefficient
tc2	tc2	"Temperature coefficient
tnom	tnom	"Nominal temperature"
		4
N		

Figure 41: Choosing to sweep DC voltage.

	sing Ana	lyses A	ffirma	Analog Circui	t Desian En	vironment	x
OK	Cancol	Default		1			
	Cancer	Derauru	s whic	hoosing Analys	es Affirma	Analog Circuit	Des
Analy	sis	∕ tran	√ ac	√ sp	🗸 pdist	to 🗸 spss	
	4	) dc	√Xf	√pss	🗸 noise	9	
Save Swee 1 1	DC Ope p Variab Femperat Compone Model Pa	rating Po le ture nt Param rameter	DC A int neter	nalysis _ Component Sel Parameter	Name /V lect Compo	r1	_
Swee Swee Swee Aut Add S	ep Range Start- St Center- S ep Type omatic Specific I	op s pan  Points _	itart	Ŭ	Stop	5	
Enabl	ed 🔳					Options	

Figure 42: Completed DC Analysis Setup.

#### Adding Wire Names:

To easily see which signal is the input and which is the output it is advisable to add wire names to the wires connecting A and Y of the inverter. To add:

Goto Add ... Wire names and fill out the pop-up according to Figure 43.

Stamp down each name as in Figure 44.

🗙 Add Wire Nar	ne		×
Hide Can	cel Defaults		Help
Names	ΑY		
Font Height	0.0625	Bus Expansion 🔶 off 🗸 on	
Font Style	stick 🔟	Placement 🔷 🔶 single 🧹 m	ultiple
Justification	lowerCenter 💷	Purpose 🔷 🔶 label 👡 alia	s
Entry Style	fixed offset 💷	Show Offset Defaults	
Rotate			

Figure 43: Adding Wire Names.



Figure 44: Schematic with Net Names.

## Choosing which nets to plot.

Now you need to choose which vectors you want to plot. Goto Outputs... To be plotted... Select... on schematic. Go back to your inverter test bench schematic and click on the wire going into your inverter and the wire between the output of your inverter and the load capacitor. The lines should change color as you select them. If you click on a pin it will probe the current and the terminal of the device you clicked will have circles drown around them. When you are done it should look like Figure 45.

Press the esc key to get out of selection mode. Your Affirma pop-up should look like Figure 46.



Figure 45: Schematic with Nets selected for Plotting.

🗙 Affirma Analog Circuit Desig	n Environment (1)	
Status: Selecting outputs	to be plotted T=27 C Simulator: spectres	S 7
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	Ł
Library AMI06	# Type Arguments Enable	J AC TRAN
Cell INV_TB View schematic	1 tran 0 2n yes 2 dc 0 5yes	
Design Variables	Outputs	<b>₽</b>
# Name Value	<pre># Name/Signal/Expr Value Plot Save March</pre>	
	1 A yes allv no 2 Y yes allv no	8
		8
> Select on Schematic Outpu	 ts to Be Plotted	$\geq$

Figure 46: Selecting which vectors to plot.

In Affirma, Goto Simulation... Run. The CIW and the Affirma pop-up will show you the process it goes through to simulate your circuit. If all went well, a plot like Figure 47 should appear.

If the simulations worked, go back to the CIW and goto Options... Save defaults and click OK on the pop-up. This will save your simulator environment, but not your plot settings.



Figure 47: Plots of Transient and DC Analyses.

#### Simulation Problems:

Oh no, the simulation did not work!

If you get error messages like "model XXX not found" you did not set the path correctly.

If you get a spice error message, you probably did not set your vpulse, or vdd correctly.

If it takes a long time to run, and or simulator times out, the capacitor could be too big, or the width or length of a MOSFET could have the wrong units (for example, instead of 4 microns it is 4 meters).

If the simulation run but the output does not look like Figure 47, the capacitor could be too big or the DC voltage is not set correctly.

To fix or change properties of the inverter goto Design... Hierarchy... descend edit and click on your inverter. Change the pop-up to schematic and click ok. To edit a mosfet or anything else press q and click on what you want to edit. To return to the top, goto design... hierarchy... return to the top.

#### Measuring Propagation Delays and $V_{INVTH}$ :

It is very hard to read off propagation delays from the plot. To easily read propagation delays:

Make sure the Transient Plot is selected by clicking on the number 1 in the waveform window. In the Waveform Window Goto Markers... Horizontal Markers and fill out the pop-up according to Figure 48. Click on Apply and then Display Intercept Data

🔀 Horiz	zontal Ma	rkers (w	indow:9)	×
ок	Cancel	Apply	Display Intercept	Data Help
Marker	location	s 2.5		
			dd Graphically	Delete All

Figure 48: Adding a Horizontal Marker.

You should see results like in Figure 49.

To find  $\tau_{PLH}$  subtract Curve 1's time form Curve 2's at the first intersection of  $V_{DD}/2$  (149.23p – 50p). We get  $\tau_{PHL}$  to be very close to 100ps.

To find  $\tau_{PLH}$  subtract Curve 1's time form Curve 2's at the second intersection of  $V_{DD}/2$  (748.99p –650p). We get  $\tau_{PLHL}$  to be very close to 100ps.

#### We met timing specification!

🗙 Results Display Window				
File			Help	11
Curve name map: 				Ē
Curve2 - /Y Curve1 - /A				
Curve table:				
	Y value	Curve2	Curve1	
м2	2.5	149.23641223p 748.99182944p 1.148876499n 1.748354198n	50p 650p 1.05n 1.65n	/
1				

Figure 49: Results from a Horizontal Marker.

To check for  $V_{INVTH}$  click on plot 2 DC analysis and press A for a cross hair marker. Place it on the intersection of the two lines and read the value off the bottom as in Figure 50. We get  $V_{INVTH}$  to be 2.4 Volts, which is good enough to met specification.



Figure 50: Final Results.

#### Measuring Power:

We have met our timing specification, but are we within our power budget? We can add a power meter to the schematic (Kang and Leblebici page 245). Too add a power meter to the schematic:

- 1. Press i to bring up the add component pop-up and click on browse. Set the library to NCSU\_Analog\_Parts and then click on Uncategorized. You should see a pop-up like Figure 51. Figure 52.
- 2. Stamp it down in your schematic and wire it up like Figure 52.
- 3. Add a capacitor and ground to the circuit, and wire it up like in Figure 52.
- 4. Edit the capacitor's value to be  $T/V_{DD}$  (1e-9/5, do not use units!). This is important so that the power is scaled properly. If you change  $V_{DD}$  or the period of the clock, you have to edit the capacitor value. (See Figure 53.)
- 5. Add the wire Name P to the net connecting the capacitor and the power meter as in Figure 52.

- 6. Add the net P to the plot list and run the simulation.
- 7. In the waveform window, go o Axes... Strip and you should see results like in Figure 54.
- 8. Add a cross hair marker A and B like in Figure 54. The delta will give the power for 1 clock cycle. Each clock cycle the power that is used will be added to the power already used. (Note even the axis is label in volts, read off the values in Watts.)

We used 19.78mW of power, which means we met specification! (Just barely, due to statistical variation it might be possible for some manufactured circuits to use more power.)

Component Browser
Commands Help 12
Library NCSU_Analog_Parts -
Flatten _
Filter *
(Go up 1 level)
POWER_METER
circuit
h
invertor
nclk
nmoscap.oid
sneeti@sneetUUI
Suri
Uncategorized

Figure 51: Adding a Power Meter.



Figure 52: Schematic with Power Meter Added.

Kedit Object Properties		2
OK Cancel Apply D	efaults Previous Next	Help
Apply To only cur Show syst	rent   instance   em  = user  = CDF	
Browse	Reset Instance Labels Display	
Property	Value	Display
Library Name	NCSU_Analog_Parts	Off
Cell Name	capi	
View Name	symbol	
Instance Name	CI	
	Add Delete Modify	y
CDF Parameter	Value	Display
Capacitance	1e-9/5 ř <u>í</u>	off _

Figure 53: Editing the Capacitor's Value.



Figure 54: Waveform with Power (19.78mW).

#### Saving your State:

To make sure you do not have to enter all the information into Affirma every time you want to run a simulation, save the state. To save the state goto Session... Save the state in the Affirma Window and fill it out according to Figure 55.



Figure 55: Saving the State.

#### Summary:

We have simulated our model with a more accurate tool and have met the logic, timing, noise and power specifications. We have set up a test bench to verify our inverter that will be used again. We are not at the point where we can have our inverter fabricated. We need to create a layout view of our inverter, which is essential, a picture that shows the shape of a CMOS inverter, as it would look like under the microscope. The layout tools is used to create a representation of the circuit that can be used by the fabrication house's mask maker to make the phototypographic plates that will be used in production. In chapter, three we will learn how to use the layout tool by following the Cell Design tutorial developed by Cadence, and then we will create the layout view of our inverter.

Chapter

## Chapter 4: Layout

We will take a break from our inverter and complete the Cell Design Tutorial developed by Cadence so learn how to use the layout tool, perform Design Rule Checking (DRC) and Layout vs. Schematic (LVS). After completing chapter's 1-4 of the Cell design Tutorial we will then layout our inverter.

### Cell Design Tutorial

CDS has developed a cell design tutorial, which takes the user through the layout, DRC, and LVS parts of the design flow. It does not use a library that can be fabricated through MOSIS, but it is very well done. If you complete the cell design tutorial of CDS it will be much easier to complete this tutorial., but you do not have to. Also, the LVS check will not work for this tutorial. It is probably fine just to read the tutorial. The CDS tutorial is started in the following manner:

- 1. Make sure you are using a c-shell (Type csh at terminal)
- 2. Type in the command **cdsdoc &** at the % prompt.
- 3. A pop-up will appear like Figure 56.
- 4. Click in the upper right of the pop-up and set to docs by family.
- 5. Go down in the window, select Virtuoso, and select cell design tutorial (Figure 57.)
- 6. You might have to click several times for it to work. A netscape browser will appear like in Figure 58. All you have to do is follow the instructions.

- Cadence Docume	tation	•
Active Document Hierarchy:	•	
IC4.4.6 /apps/cadence/ic4	46	•
	Docs by Family	-
← ☐ Affirma ← ☐ Assura		
🗢 📑 Cadence		
Correction Correct		
Search All Open	Exit Hel	p

Figure 56: Documentation Browser

-	Cadence Documentation 🗾 🔹 🗌
	Active Document Hierarchy:
	IC4.4.6 /apps/cadence/ic446 🗸
	Docs by Family 🔻
	Cauence oser interface Skill Functions Referen
	🗋 Cadence User Interface SKILL Known Problems 🕅
	🗋 Cadence to Synopsys Interface User Guide
	🗋 Cell Design Tutorial
	🗋 Communicàtions Manager Reference
	🗋 Compatibility Guide 🧮 🗧
	🗋 Composer to Spectre Simulation Solution User
	🗋 ConclCe Help
	🗋 Custom Layout SKILL Functions Reference
	🗋 Design Data Translator's Known Problems and
	🗋 Design Data Translator's Product Notes
	Design Data Translator's Reference
	Design Framework II Help
	Design Framework II Known Problems and Solu
	Design Framework II Product Notes
	Design Framework II SKILL Functions Reference
1	Display Resource Editor Known Brohlems and S
	Search All Open Exit Help
L.	
Γ	Starting viewer

Figure 57: Cell design tutorial



Figure 58: Netscape Browser with Tutorial

Complete chapters 1-4 of the cell design tutorial.

## Preliminary Steps to Layout of the Inverter:

Now that you know how to use the layout tool, run a DRC Check, extract the spice model from your layout and run an LVS check, it is time for you to do it on the inverter design. We have good values for  $W_N$ ,  $L_N$ , and how to connect the transistors, we just do not have a layout view so it can be ultimately fabricated.

We have sized the lengths and the widths of the PMOS and NMOS transistors and we relatively certain that we will meet specification. Now we will create the layout view of the inverter

We need to do some planning before we start the layout. We need to come up with a rough floor plan, make sure our metal1 power line widths are not prone to electromigration failure and make sure that our transistors fit within the cell height.

Floor Planning:

First, we need to make some assumptions about where everything will go. Our specification requires that the cell height needs to be 40 $\mu$ m, but our transistor widths ( $W_N$  and  $W_P$ ) total to over 140 $\mu$ m! The solution to this is to have smaller transistors in parallel, but how many parallel transistors should we use? We cannot use the whole 40 $\mu$ m because we need room for routing the power lines, input lines and minimum separation requirements. We need to make sure the power lines are wide enough to ensure that there will be no electromigration failures. The general floor plan of our inverter will be standard. The top of the circuit will be the  $V_{DD}$  power line. The PMOS transistors will be below the  $V_{DD}$  lines. The poly input lines will be in between the PMOS and NMOS transistors, as well as the metal1 output lines. The ground line will be beneath the NMOS transistors. The ntaps(body connection for the PMOS transistors) and the ptaps (body

connection for the NMOS transistors) will be to the left of the transistors. The input will be to the left, while the output will be to the right. For ease of routing, only metal1 and poly will be used in this inverter. To start putting things down we need to know how wide the metal1  $V_{\rm DD}$  and ground lines should be. Then we can design the number of fingers (parallel transistors) we should use.

Sizing the VDD and Ground Lines for Electromigration Failure Prevention:

The way to keep lines from having electromigration failures is too keep the average current density below a critical value. The design equation for the minimum width of the metal1 due to electromigration effects is:

$$w_{M1} := rac{I_{ave}}{J_C}$$
 (8)

Where:

$$I_{ave} := \frac{1}{12} \cdot \frac{K_{N} \cdot \tau \cdot f_{CLK}}{V_{DD}} \cdot \left(V_{DD} - V_{TN}\right)^{3} (9)$$

and

$$J_{\rm C} := .6 \cdot 10^{-3} \cdot \frac{\rm A}{\mu m} (10).$$

 $K_{\scriptscriptstyle N}$  can be found by first extracting  $K_{\scriptscriptstyle NP}$  from:

$$K_{NP} := \frac{1}{(V_{DD} - V_{TN})} \cdot \frac{1}{A} \cdot \left[ \frac{2 \cdot V_{TN}}{(V_{DD} - V_{TN})} + \ln \left[ 4 \cdot \frac{(V_{DD} - V_{TN})}{V_{DD}} - 1 \right] \right] (11)$$

Then solve for K<sub>N</sub>:

$$\mathbf{K}_{\mathbf{N}} \coloneqq \frac{\mathbf{W}_{\mathbf{N}}}{\mathbf{L}_{\mathbf{N}}} \cdot \mathbf{K}_{\mathbf{NP}} (12).$$

In this case, the minimum width of the metal 1 power lines ( $W_{MI}$ ) are .456µm. This is almost half of the minimum allowable metal1 width of .9µm! We are well within our specification but we will use 1.8µm because other circuits will share these power lines. The parameters are listed in Table 6.

Determining the Number of Parallel Transistors to use:

Now that we have our metal line widths specified that means there is approximately  $4\mu m$  of height that we cannot use. There is an additional  $4\mu m$  of height that is unavailable, due to other spacing requirements. The width of the NMOS transistors is given by:

$$W_{NF} := \frac{H_{CELL} - Spacing}{1 + Ratio} (13)$$

The number of fingers is given by:

$$N_{F} := \frac{W_{N}}{W_{NF}} (14)$$

In our example, this turns out to be 4.576. We need to use an integer number of fingers so we pick  $N_{\rm F}$  to be 5 and then solve for the width of the NMOS parallel transistors to be 10.46 $\mu m$ . Using the ratio of 1.8, we find the width of the PMOS parallel transistors to be 18.83 $\mu m$ . The parameters are listed in Table 6.

#### Summary:

We have designed our metal1 power line widths to be 1.8mm, and the number of parallel transistors to be 5. The width of the NMOS transistors will be 10.46µm while the PMOS transistors will be 18.83µm. Next, we will layout our inverter.

#### Table 6: Parameters Used in Floor planning.

Parameter	Description	Value	Units				
W <sub>N</sub>	Width of the NMOS	52.3	μm				
WP	Width of the PMOS	93.9	μm				
W <sub>NF</sub>	Width of the parallel NMOS transistors.	Use equation 13.	μm				
W <sub>PF</sub>	Width of the parallel NMOS transistors.	Multiply $W_{\rm NF}$ times Ratio	μm				
HCELL	Maximum cell height	40	μm				
Spacing	Height not available for transistor widths.	8 in this example.	μm				
NF	Number of parallel transistors	See example.	integer				
W <sub>M1</sub>	Minimum width of a metall line due to electromigration considerations	Use equation 8	μm				
I <sub>ave</sub>	Average current used in CMOS inverter	Use equation 9	A				
J <sub>C</sub>	Maximum current density allowed before electromigration becomes a problem.	.6	mA/µm				
K <sub>N</sub>	Transconductance of NMOS scaled by W/L	Use equation 12	A/V <sup>2</sup>				
K <sub>NP</sub>	Transconductance of NMOS scaled by W/L	Use equation 11	A/V <sup>2</sup>				
τ	Rise Time and Fall Times	100p	S				
fcik	Clock Frequency	1G	Hz				
V <sub>DD</sub>	Supply Voltage	5	V				
V <sub>TN</sub>	NMOS threshold Voltage	.6	V				
А	Fitting parameter used to design WN	Use equation (2)	Ω				
Ratio	Used to scale WP to give symmetric propagation delays	Use equation (3)	unit less				

## Layout of the Inverter:

Now that we have a preliminary floor plan we are ready to layout the inverter.

Start the tools in your cell directory by typing *icfb* & at the command line if you are not running the tool already.

First we need to create a layout view of our inverter. Goto the library manager and create a new cell view according to Figure 59.

🔀 Create	New File			×
ок	Cancel	Defaults		Help
Library N	ame _	AMIO	6	
Cell Name	•	INV		
View Nan	ne 🏼	layout		_
Tool	_	Virtuo	0\$0	
Library p	ath file			
/home/e	acad40/d	cell/cds.l	ib	

Figure 59: Creating a layout view of the inverter

XVirt	tuoso@ Layout Editing: AMIOS INV layout						×														
X: -	5.05	۲	r: 9.4	5	(F) 8	select: O		dX:		d	Y;		Dis	t:	Qmd	1:					6
Tools	Design	Win	dow	Create	Edit	Vertify	Conn	ectivity	Optic	ons	Route	NC	8V							Hel	ę
۲																					4
Q																					
Q																					
Q																					
Q																					
EQ.				•																	
Щ																					
¥															 		 	 	 		
383 1980																					
•"ل																					
٦,																					
[std]																					
q																					7
11										_	-									>	
	nouse L: > modify	plot						N :							R:						

Figure 60: Layout Editor

Figure 60 shows the layout editor that you used in the cell design tutorial.

🗙 LSW 👘	
Edit	Help
meta	11 dg
AM	4106
🔳 Inst	📕 Pin
pwell	
M nwell	. dg
activ	re dg
xxx nacti	ve dg
m pacti	ve dg
nsele	ct dg
psele	ct dg
poly	dg
elec 💥	dg
metal	.1 dg
metal	.2 dg
Zmetal	.3  dg
CC	dg
via	dg
🎆 via2	dg
glass	dg
highr	es dg
Modre	:  dg
nolpe	dg d
<mark>&gt; pad</mark>	dg
<b>text</b>	dg
res_i	.d [dg]
cap_i	.d dg
<mark>}}dio_</mark> i	.d dg
pwell	. nt
	<u></u>

#### Figure 61: LSW

Notice that the layers that are available in the LSW (Figure 61). This is the AMI06 process.

To set the display so that all the layers will appear, goto Options... Display in the layout editor. Set the pop-up according to Figure 62 and click ok. Make sure the minor, major, x snap spacing and y snap spacing are correct as in Figure 62.

🔀 Display Options	×						
OK Cancel Defaults Apply	Help						
Display Controls	Grid Controls						
_l Nets 📕 Axes	Type 🗸 none 🔶 dots 🗸 lines						
	Minor Spacing						
Array Icons EIP Surround	Major Spacing						
📕 Label Origins 🔄 Pin Names	X Snap Spacing 0.15						
📕 Dynamic Hilight 📕 Dot Pins	Y Snap Spacing 0.15						
Net Expressions Use True BBox							
Show Name Of vinstance * master	Size Style outlined _						
Array Display Display Levels	Snap Modes						
From Q	Create orthogonal 💷						
Source To 20	Edit orthogonal 💷						
◆ Cellview ↓ Library ↓ Tech Library ↓ File //. cdsenv							
Save To Load Fro	Delete From						

Figure 62: Setting Display Options.

To save these setting, in the CIW, goto options... save defaults. Click OK. It will take a few minutes to save the defaults, but it will save time later.

To make sure we fit in our cell height we will draw the VDD and ground lines first.

To draw the Ground line:

- 1. Select metal1 in the LSW.
- 2. Click on the layout window and press p to start a path. A pop-up like Figure 63 should appear. Change the width to 1.8 as in Figure 63.
- 3. Start the path by click the left mouse button at the coordinate x=0, y=.9 (Figure 64).
- 4. End the path by double click at coordinate x=28.05, y=.9 (Figure 64).
- 5. Press the esc key to get out of path mode.

Create	Path				×
Hide	Cancel	Defaults			Help
Mode	~0	Guided 🔶	Manual	Change To Laye	r
Width	1.8	I			dg 🔟
Fixed Wid	th 🔟				
Offset	0 <u>ĭ</u>			Contact Justification	
Justificat	ion <u>ce</u> n	iter 💷 🛛			
End Type	fl	ush 🔟 📔		_ ■ _	
Begin Ext	ension	0			
Gut Exter	nsian	0	Snap Mod	e orthogonal 🚄	
Net Name	•	Y.			
_ As RC	D Object				
ROD	Name	path0			
Rotati	e	_	Sideways		Upside Down

Figure 63: Starting a Path



Figure 64: Drawing the Ground Line.

To draw the VDD line:

- 1. Press c to enter into copy mode.
- 2. Click on the ground line in the upper left corner and copy it to coordinate x=0, y=39.9 as in Figure 65.
- 3. Make sure the cell height is correct by measuring from the top left corner of the VDD line to the bottom left corner of the Ground line. You should get a cell height of 39.9  $\mu$ m.

6.30	Y: 41.70	(P)	) Select: 0	dDC -	6.30	en: 41.70	DISI: 42.173	Crist: Ruler	
Design V	Vindov G	ewter Ed	it Verify	Connectivity	y Optiona	Bouta: NCSU			
J									

Figure 65: Inverter with VDD and Ground Lines.

Next, we will add the ntaps and ptaps:

- 1. Press the esc to get out of copy mode
- 2. Press i to bring up the Create Instance pop-up (Figure 66).

Create	Instance				×
Hide	Cancel	Default	ts		Help
Library	AMI06				Browse
Cell	Ĭ.				-
View	layout				-
Names	Ĭ.				-
Mosaic		Rows	1	Columns	1.
		Delta Y	0	Delta X	0
Magnifica	ation 1		-		
Rotat	e	_	Sideways	<u> </u>	Upside Down

Figure 66: Create Instance Pop-Up.

- 3. Click on Browse and a pop-up like Figure 67 should appear.
- 4. Select ntap like in Figure 67.
- 5. Fill out the pop-up like Figure 68 making sure to select 7 rows of contact. This is done to make sure the NWELL is well contacted. Since it is impossible to route metal1 over this cell it does not matter if we use extra ntaps.
- 6. Stamp down the ntap so that the upper left corner is aligned with the lower left corner of the VDD line as in Figure 69.

🔀 Library Browser - Create Inst	Ince		
_ Show Categories			
Library	Cell	View	
jant 06	jatap	Layout	
ANDIG NUCSU_Analog_Parts NUCSU_Analog_Parts NUCSU_Sheat_Bths PACTRANS TEST Desid edsDefTechLib	nl.elec nl.n nl_p nl_poly n2_mi n3_mB netall ne	A lepout	
Close	Filters		Help

Figure 67: Browsing for the ntap.

- 7. Go back to the Create Instance pop-up and fill it our according to Figure 70.
- 8. Stamp it down like in Figure 71.

Create	Instance				X				
Hide	Cancel	Default	s		Help				
Library	AMI06				Browse				
Cell	ntapį				-				
View	layout	, layoutį							
Names	IĞ				-				
Mosaic		Rows	1	Columns	1				
		Delta Y	12	Delta X	3.6				
Magnifica	ation 1								
Rotat	e	5	Sideways		Upside Down				
Rows of	contacts		Ĭ.						
Columns	of contact	ts	1						

Figure 68: ntap with 7 Rows of Contacts.



Figure 69: Placement of the ntap.

Create	Instance				×
Hide	Cancel	Default	s		Help
Library	AMI06				Browse
Cell	ptapį				F
View	layoutį				·
Names	I4				,
Mosaic		Rows	1.	Columns	1
		Delta Y	6.9	Delta X	2.4
Magnifica	ation 1				
Rotat	e	S	lideways		Jpside Down
Rows of	contacts		4		
Columns	of contac	ts	11		

Figure 70: Ptap Pop-up.



Figure 71: Placement of the ptap.

Viewing cells as an outline only.

Sometimes when doing a lyout you only want to work on certain levels of the design and seeing all the levels at any given time will make the screen refresh too slow. To tunr off the complete drawing of cells:

Go to Options... Display and a pop-up like Figure 72 should appear Set the display levels from 0 to 0 like in Figure 72 and click apply. You layout should look like Figure 73. Go back to the display and change the display levels back to 0 to 20 to finish the rest of the tutorial.

E Display Options	×						
OK Cancel Defaults Apply	Help						
Display Controls	Grid Controls						
_l Nets 📕 Axes	Type 🗸 none 🔶 dots 🗸 lines						
Access Edges Fath Borders	Minor Spacing						
Array icons EIP Surround	Major Spacing						
E Label Origins 🔟 Pin Names	X Snap Spacing 0.19						
Dynamic Hilight Dot Pins     Not Environmentions	Y Snap Spacing 0.19						
Stretch Handles	Filter						
Show Name Of vinstance + master	Size 3 Style outlined						
Full     Display Levels	Snap Modes						
Border Fram 9.	Create orthogonal 💷						
Source	Edit. orthogonal 🖃						
Coliview Ubrary Tech Library File     -/. odsenv							
Save To Load Fro	om Delete From						

Figure 72: Setting the Display Levels.

× 40	exerî Leş	rout Editio	g tutoria	i inv k	iptud:								
- 30 -	J.36	Y: 22.	85	(l)	Select: O	dDC - 7	94.15	ev: -0	.50	Dist: 24	187	Oref:	5
Tools	Design	Vindew	Create	EiR	Verify	Connectivity	<b>Options</b>	Route	NC3V				Holp
4													1
10													
82													
6													
~						10							
9							+						
m						- 10							
~ 1						+							
30,							1						
100						-	1						
24													
1													
<u> </u>													
2011						_							
2													
1522						19	-						
5						- 13							
5						- 15							
(sid)						000000							
n													
	-										_		·
1	wrate 1:					H:					3:		-

Figure 73: Viewing cells in Outline Only.

To add the PMOS and NMOS Transistors:

- 1. For the PMOS transistor, in the Create Instance pop-up, fill it out like Figure 74. Note the width is rounded to 18.9u by the software. Make sure you set the multiple to 5.
- 2. Stamp it down like in Figure 76. (The top left corner of the transistor is at x=3.6 and y=38.10.
- 3. For the NMOS transistor, in the Create Instance pop-up, fill it out like Figure 75.
- 4. Stamp it down like in Figure 76. (The top left corner of the transistor is at x=4.86 and y=13.8.
- 5. Run DRC and fix any errors.

🗙 Create	Instance				×
Hide	Cancel	Default	s		Help
Library	AMIOG				Browse
Cell	pmos				
View	layout				·
Names	IŠ				
Mosaic		Rows	1.	Columns	1
	1	Delta Y	4.8	Delta X	18.8
Magnifica	tion 1				
Rotate	e	S	ideways		Jpside Down
Model na	<i>ne</i>		ami06	P	
Model Ty	pe		🔶 sys	tem 🧹 use	r
Bulk node	e connecti	on	vdd!∐		
Multiplier			5		
Faigers			1		
Width (gri	id units)		126		
Width			18.9u	M	
With (mi	niman)		1.5u	М	
Length (g	rid units)		4		/

Figure 74: PMOS Transistor with a Multiple of 5.
🗙 Create	Instance				×
Hide	Cancel	Defaults			Help
Library	AMIO				Browse
Cell	nmosį				·
View	layout				
Names	IŚ				·
Mosaic	l	Rows	<u>Ľ</u>	Columns	1
	1	Dolta y 🛛	2.4	Delta X	5.2
Magnifica	tion 1				
Rotate	9	Sic	leways	<u> </u>	Jpside Down
Model na	<i>ne</i>		ami06	N	
Model Ty	pe		🔶 sys	tem 🗸 use	r
Bulk node	e connecti	on	gndl		
Multiplier			5		
Faigers			1		
Width (gri	id units)		70 <u>ĭ</u>		
Width			10.5u	M	
Witth (misimum)			1.5u )	М	
Length (g	rid units)		4		
Length			600n 1	M	/

Figure 75: NMOS Transistor with a Multiple of 5.



Figure 76: Placement of the NMOS and PMOS Transistors.

Adding the Input Layer:

In this case, the input will come from a metal1 line. We need to form a metal1 to poly contact to define were this input will be.

- 1. For the metal1 poly contact, in the Create Instance pop-up, fill it out like Figure 77. Note the width is rounded to 18.9u by the software. Make sure you set the rows and columns to 2. This will ensure that even if there is a bad contact the input will still be connected. In addition, the resistance of the contact will be lower than the minimum sized device.
- 2. Stamp it down like in Figure 78. The top left coordinate is: x=.45, y=16.65.
- 3. Run DRC and fix any errors.

Create	Instance				×	
Hide	Cancel	Default	s		Help	
Library	AMI06				Browse	
Cell	m1_poly					
View	layout		-			
Names	IĞ				-	
Mosaic	R	lows	1.	Columns	1.	
	Ð	elta Y	1.2	Delta X	1.2	
Magnification I						
Rotat	e	S	ideways		Upside Down	
Rows of	contacts		2			
Columns	of contact:	S	2			

Figure 77: Metal1 to Poly Contact.



Figure 78: Placement of the Metal1 Poly Contact.

Wiring up the Transistors with Metal1:

- 1. Zoom in on the top of the layout
- 2. Create paths of 1.2µm wide connecting the ntap and every other S/D of the PMOS transistor as in Figure 79. Note: For all layers except metal1 have been turned off.
- 3. Repeat the same process the Sources of the NMOS transistors (Figure 80).
- 4. Repeat the same process to connect the Drains for the out put (Figure 81).
- 5. Run DRC and fix any errors.

🗙 Virts	oso@Lay	out Editin	g: tutoria	d tew la	ryout							
- X: 6	1.85	Y: 41.2	5	(F) 3	elect: 0	dX: 33	5,85	dY: -1	1.10	Dist: 37.429	Omd:	- 4
Tools	Design	Window	Create	Edit	Vertfy	Connectivity	Options	Route	NCSU			Help
<u>ئ</u> ال												7
<u>8</u>												
9												
<u>ul</u> an I												
io,												
Ì												
$\cap$												
833 36.												
r												
Ð,						1						
[1256]							N.					
9	4				L	11						/
1	MOUSE L:					D1:				B.:		

Figure 79: Connect Sources of the PMOS Transistor and the ptap. Figure 79.



Figure 80: Connect Sources of the NMOS Transistors and ptap.



Figure 81: Connecting Drains to form the Output.

Connecting the input to all the gates:

Now we need to wire up the gates with poly so they are all connected.

- 1. Set the display in the LSW to show only poly.
- 2. Connect the gates of the NMOS and PMOS Transistors to each other and the input with one large rectangle as in Figure 82.
- 3. Run DRC and fix any errors.



Figure 82: Connecting up the Gates of the Transistors to the Input.

The finished layout (with out any pin information) should look like Figure 83. Note: Stretch the right ends of the metal1 power lines to line up with the right side of the NWELL in Figure 83.

Run DRC one final time to make sure you have found all the errors. You might not see all the errors in the layout if they are small. Read the report in the CIW to make sure no errors were generated (Figure 84).

Tools     Design     Mudow     Oracle     Edit     Verify     Connectivity     Options     Fourier     MCSU     Height       Image: Second and the

Figure 83: Layout of CMOS Inverter.

🗙 icfb - Log: /home/eecad40/CDS.log	
File Tools Options Technology File	Help 1
CPU TIME = 00:00:00 TOTAL TIME = 00:00:00 ******** Summary of rule violation for cell "INV layout" ********* Total errors found: 0	7
mouse L: Enter Point M: Pop-up Menu R: Select the figure to be stretched:	

Figure 84: DRC with no Errors.

Adding Pins:

In order to simulate our extracted view of your laid out inverter, you need to add four pins, vdd!, gnd! A and Y. The symbol "!" means that the variable name is global.

Goto Greate... Pin and fill out the pop-up according to Figure 85. You must enter the pin names exactly as in the pop-up or they will not match the schematic and symbolic views of the inverter. **Make sure the layer is set to metal!** 

🔀 Create Symbolic	Fin		×
Hide Cancel			Help
Terminal Names	vdd! gnd! :	ম মূ	
🔟 Keep First Nam	• × esot	y 0 (	Phote D
Mode	🗢 sym pin 🦂	v auto pin 🗸 si	sape pin
🔟 Display Pin Nar	ne	Display Pin Na	ume Option
I/O Type	√input.	🗸 output	🗢 inputOutput
	🗸 switch	√jumper	
Pin Type	metafi 🖃		
Pin Width 0. g		Pin Gaugh	D
Access Direction	📕 Тор 📕 В	ottom 🔳 Left	🖬 Right
	🖬 Any 🔄 N	one	

Figure 85: Pin Creation

Stamp down the vdd! pin in the metal power bar on the pmos side of your circuit.

Stamp down the gnd! pin in the metal power bar on the nmos side of your circuit.

In the Create Symbolic Pin pop-up change the I/O type to input like in Figure 86, and stamp it down in the metal1 poly via.

🔀 Create Symbolic Pi	n 🚬
Hide Cancel	Help
Terminal Names	λ Υ
Mode	> Kitoh U Kitok U ♦ sym pin ↓ auto pin ↓ shape pin
_ Display Pin Name	Display Pin Name Option
LO Type	♦ input ~output ~inputOutput
Pin Type I	√ switch √ juniper metall →
Pin Width 0.9	Pin fanglis 0
Access Direction	■ Top  ■ Bottom  ■ Left  ■ Right ■ Any _  None

Figure 86: A input pin.

In the Create Symbolic Pin pop-up change the I/O type to output like in Figure 87 and stamp it down in the metal region connecting the two transistor drains.

The completed layout with pins can be seen in Figure 88.

We are now ready for the final DRC and then we can extract the circuit for LVS and post extraction simulation.

V Freate Symbolic	Pin		×
Hide Cancel			Help
Terminal Names	1 <u>]</u>		
🔟 Keep First Nam	• X Chot	0 81	Those 0
Mode	🔶 sym pin 🗟	vauto pin vsh	rape pin
🔄 Display Pin Nac	10	Display Pin Na	me Option
ИО Туре	√input.	🗢 output	🗸 input Output
	$\sim$ switch	√jumper	
Pin Type	metall 🖃		
Pin Width 0.9		Pin Gaugh	D
Access Direction	🔳 Тор 🔳 В	lottom 🔳 Left	E Right
	🔳 Any 🔄 N	lone	

Figure 87: Y Out put pin.

The completed layout should look like Figure 88. Now you are ready for the final design rule checking.



Figure 88: Completed INV Layout.

#### Design Rule Checking (DRC)

Goto Verify... DRC and a pop-up like Figure 89 should appear. You just have to click ok and it should run with no errors like in Figure 90.

<b>X</b> DRC		×
OK Cancel Defaults	Apply	Help
Checking Method 🔶 flat	whierarchical whier w/o optimization	
Caon8	nate Sel t	y Cursor
Switch Names	Set	Switches
Run-Specific Command File	L	
Inclusion Limit	1000	
Join Nets With Same Name		
Echo Commands		
Rules File	divaDRC.rulį	
Rules Library	AMI06	
Machine	🔶 local 🗸 remote 🛛 Machine 📗	

Figure 89: Running DRC

🗙 ictb - Log: /home/eecad40/CD5Jog		D.	×
File Tools Options Technology File	Help	1	
<pre>seveneese Summary of rule violation for cell "HW leyout" A******* Total mroars found: 0 E08 started at Fri 701 26 14:32:00 2002 Library: MU06 cell: 1HV View: Jayout Bulas come fram Library AMI06. Bulas puth is divaINU:rul. Enclusion: Link is set to 1000. Burning drclayout manlysis "Lat mode"</pre>			7
Full Checking. IND startedFri Jul 26 14:32:00 2002 completedFri Jul 26 14:32:00 2002 GPU TIME = 00:00:00 TOTAL TIME = 00:00 0:00 ********* Summary of rule violation for cell "INV layout" ******** Total errors found: D		_	7
I		_	

Figure 90: No DRC Errors.

If you have errors, fix them according to the AMI06 design rules with the method you learned in the cell design tutorial.

Once the circuit has no DRC errors save your work.

Summary:

We have drawn a layout view of our inverter and made sure that there were no drawing errors. We still need to know if the as drawn circuit will behave like an inverter according to our timing and power specifications. To do this we will extract the circuit and run an LVS check.

#### Circuit Extraction:

Now that your circuit is laid out with no DRC errors, it is time to check if it is an electrical equivalent of your inverter schematic. Also, we need to extract any parasitic capacitors from the layout that might affect our circuit's performance.

Goto Verify... Extract and a pop-up like Figure 91 should appear.

XEstrad	lor					×
ок	Cancel	Defaults	Apply			Help
Extract N	dethod	+ fia	t <sub>V</sub> macr	o cell 🗸 ful	i hier 🗸 incre	emental hier
Join Nets	: with San	ne Rame	_		Echo Come	nandis 🔟
Switch N	ames					Set Switches
Run-Spe	cific Com	nand File	-			-
Inclusion	Linit		1000			
Meyr Nar	nes	Extracted	extract	ted	Excel	excell
<b>Bules</b> File			dival	T. rulį		
Rules Lib	rary		⊨ [and	ađ	_	
Machine			+ local	<b>∼</b> remote	Machine	1

Figure 91: Running the extractor.

Click on the set switches button in the extractor pop-up and a list of choices should appear like in Figure 92.



Figure 92: Selecting Extract parasitic capacitances.

Click on Extract\_parasitic\_caps and press ok.

Your extractor pop-up should look like Figure 93.

XExtractor		×
OK Cancel Defaults	Apply	Help
Extract Method 🛛 🔶 fla	at 🗸 macro cell 🗸 fu	ll hier 🗸 incremental hier
Join Nets With Same Name	_	Echo Commands 📃
Switch Names	Extract_parasiti	c_caps Set Switches
Run-Specific Command File	_	
Inclusion Limit	1000	
View Names Extracted	d extracted	Excell excell
Rules File	divaEXT. rulį	
Rules Library	AMI06	_
Machine	🔶 local 🗸 remote	Machine 📗

Figure 93: Extractor all set to go.

Click ok to run the extractor. The CIW should give no errors like in Figure 94.

🗙 icfb - Log: /home/eecad40/CD5.log	_	미지
File Tools Options Technology File	Help	1
0 pcapacitor ivpcell NCSU Analog Parts parasitics created. 0 pcapacitor ivpcell NCSU Analog Parts parasitics created. *WARNING* Cannot attach default technology library to the design library NCSU Analog Parts. Please attach an existing technology library to the design library NCSU Analog Parts *WARNING* techPcellEvalTrigger: Internal error since tfCnt is equal to 0 2 pcapacitor ivpcell NCSU Analog Parts parasitics created. 0 pcapacitor ivpcell NCSU Analog Parts parasitics created. 1 pcapacitor ivpcell NCSU Analog Parts parasitics created. 1 pcapacitor ivpcell NCSU Analog Parts parasitics created. 2 wWARNING* techPcellEvalTrigger: Internal error since tfCnt is equal to 0 *WARNING* techPcellEvalTrigger: Internal error since tfCnt is equal to 0 *WARNING* techPcellEvalTrigger: Internal error since tfCnt is equal to 0 saving rep AMI06/INV/extracted Extraction startedFri Jul 26 14:38:20 2002	s.	
>		_

Figure 94: Extracted inverter with no errors.

To see the extracted circuit with parasitic capacitors open up the extracted view from the library manager (Figure 95).

X: -8	5.90	Y: 13.	80	(E) 3	Relect: O	dX:		IY:	Dist:	Cmd:	
ools	Design	Window	Create	Edit	Vertfy	Connectivity	Options	Route	NCSU		Help
2											
88											
3						#+13	a P		i di di	Hag Ngan Mite	
2						•	oľa	1 S		Solar Ballin Solar Ballin	
× .											
R.											
n											
-4											
Ц,											
1											
1											
4							15 14	40 10 - R	J IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	3 <b>±</b> 4	
						D				23. am K18H Ur to - 1976) Ar B- Rôtin	
a.								1 -			
1											
Ч											
201								υ÷ι			
										533	
ų,											
1	Number 1:	showCli	ckInfof	1		N: :	ucuaePopl	hΩ		R: definen ()	

Figure 95: Extracted View of CMOS inverter.

#### Layout versus Schematic (LVS)

An LVS check makes sure that the circuit you laid out is equivalent to the one you entered into your schematic.

To run an LVS check, Goto Verify... LVS and a pop-up should appear like Figure 96.

Use the Browse button to select which schematic and which extracted view you are going to check for equivalence. You pop-up should be filled out just like Figure 96. Click run to start. This will take several minutes. When the LVS check is successful, a pop-up like Figure 97should appear.

If you have a large circuit to check set the priority to a higher number, so it does not suck up system resources too much. You can leave it 0 (highest priority) for now.

NOTE 1: If you think that having your job running at the highest priority will get your job finished quickly, think again. You could crash the system if too many users use too high a priority on an LVS job!

NOTE 2: Cause for the job to fail are incorrectly entering the data into the LVS pop-up. Sometimes you have to open up your schematic and resave it to get the LVS check to succeed.

### NOTE 3: Just because the LVS job succeeded, does not mean your circuit passed an LVS check!

To make sure your circuit passed the LVS check click on output in the LVS pop-up. Looking at Figure 98, we see that all the nets match and that all the devices match.

If you have errors, it can be hard to figure out what they are from the report to see where the errors are:

- 1. Open up the extracted view of the inverter.
- 2. Click on Error Display in LVS.
- 3. A pop-up like Figure 99 should appear. Use it to locate your errors.

XLV5		<u>_ 0 ×</u>
Commands		Help 9
Run Directory	LVŠ	Browse
Create Netlist	schematic	extracted
Library	AMIOG	AMI06
Cell	), Ina	INV
View	schematid	extracted
	Browse Sel by Cursor	Browse Sel by Cursor
Rules File	divaLVS.rulj́	Browse
Rules Library	AMI06	
LVS Options	Rewiring	_ Device Fixing
	⊥ Create Cross Reference	🗉 📕 Terminals
Correspondence	Filelvs_corr_file	Create
Priority 🗓	Run local 💷 👔	
Run	Output Error Display	Monitor Info
Backannotate	Parasitic Probe Build	Analog Build Mixed

Figure 96: Running an LVS Check

XA	nalysis Job Succeeded	×
а а	Job '/home/eecad40/cell/LVS' that was started at 'Jul 26 $14:54$	1:09 2002′ has succeeded
	OK Cancel	Help

Figure 97: Successful completion of an LVS check.

X/home/eecad40/cell/L¥S/	si.out		_	미지
File			Help	10
@(#)\$CDS: LVS version 4	.4.6 06/01/2001	20:24 (cds11612) \$		75
Like matching is enable Net swapping is enabled Using terminal names as Compiling Diva LVS rule	d. correspondence s	points.		
Net-list summary fo count	r /home/eecad40/	/cell/LVS/layout/netlist		
4	nets			
4	terminals			
5	nmos			
Net-list summary fo count 4 4 1 1	r /home/eecad40/ nets terminals pmos nmos	/cell/LVS/schematic/netlist		
Terminal correspond 1 A 2 Y 3 gnd! 4 vdd!	ence points			
The net-lists match.				
	layout sc	chematic		
	instanc	es		
un-matched	0	0		
rewired	0	0		
nruned	0	0		
active	10	2		
total	10	2		
	nets			
un-matched	0	0		
merged	0	0		
pruned	0	0		
active	4	4		
total	4	4		
	termina	als		
un-matched	0	0		
matched but	0	0		
total	U 4	4		
COCAL	4	7		
I				/
V				

Figure 98: Output of LVS Check.

🔀 L¥S Err	or Display		×				
ок	Cancel Explain	Clear Display Probe Form	Help				
Display	First Next	Prev Last All					
Error Col	Error Color hilite d1 Cycle Colors Auto-Zoom						
All 🔳 None 🔄	Unmatched 🗐 Pruned 📄	nets 🔲 instances 🔎 paramet nets 🔎 instances nets — Net Diepley I	ters <b>E</b> terminals				

Figure 99: LVS Error Display.

Summary:

We have drawn a layout view of the circuit, and made sure that it is electrically equivalent to the schematic we simulated in chapter 3. All that remains is to re-run the simulation with he extracted view of the circuit to make sure we still meet specification.

#### Post Extraction Simulation

In post extraction simulation, you verify that your circuit still meets your specification with the additions of parasitic capacitances. For example, in a schematic two wires that are not connected there is no transfer of AC voltage or current. In a layout where two metal lines are close, but not connected there is a capacitance between the two that will cause cross talk. Post extraction simulation will show these errors.

To perform a post extraction simulation on your inverter:

- 1. Open up your INV\_TB schematic Start the Affirma Analog environment.
- 2. Set up the simulation as before (Figure 100).

🔀 Affirma Analog Circuit Desig	n Environment (2)					
Status: Ready	T=27 C Simulator: spectreS	13				
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help				
Design	Analyses	Ł				
Library AMI06	# Type Arguments Enable	JAC TRAN				
Cell INV_TB View schematic	1 tran 0 2n yes 2 dc 0 5yes	X Y Z				
Design Variables	Outputs	<b>₽</b>				
# Name Value	<pre># Name/Signal/Expr Value Plot Save March</pre>	<u></u>				
	1 A yes allv no	<u> </u>				
	2 Y yes allv no					
		8				
> Results in /home/eecad40/cadence/simulation/INV_TB/spectreS/schematic						

Figure 100: Setting for post extraction simulation.

Goto Setup... Environment. A pop-up like Figure 102should appear.

In the Switch View List insert the view extracted before the spectreS view like in Figure 101. Click ok.

Now run the simulation. Your results should match Figure 102.

Note: If you want to use the schematic view for simulation you have to delete the extracted view from the switch list.

Kenvironment Options	×
OK Cancel Defaults App	Help
Init File	)
Update File	<u>[</u>
Parameter Range Checking File	[
Recover from Checkpoint File	L
Netlist Type	🗸 flat 🚸 hierarchical 🗸 incremental
Switch View List	extracted spectreS spice cmos_sch cmos.sch schematic `
Stop View List	spectreS spice
Instance-Based View Switching	L
Instance View List Table	Į.
Instance Stop List Table	Į.
Print Comments	L
Include/Stimulus File Syntax	🔷 cdsSpice 🗸 spectre
Include File	Į.
Stimulus File	

Figure 101: Inserting the extracted view.

We can see that the  $V_{\mbox{\tiny INTH}}$  values has shifted slightly lower to 2.37 Volts, but it is still within specification.

From Figure 103, we can see that we are within 1% of our timing requirements. This is more than sufficient considering that process parameters can vary as much as 10%.

Re-Run the simulation with the net P selected to check power. We can see in Figure 104 that the power used by this circuit is less than 20mW.



Figure 102: The extracted simulation.

🗙 Results Display Window			_	
File			Help	15
Curve name map:				
Curve2 - /A Curve1 - /Y				
Curve table:				
	Y value	Curve2	Curvel	
м2	2.5	50p 650p 1.05n 1.65n	149.23641223p 748.99182944p 1.148876499n 1.748354198n	

Figure 103: Propagation Delay from Extracted Circuit.



Figure 104: Power used by Extracted Circuit.

#### Summary:

The designed circuit met every specification! If the layout view did not meet specification we would have to resize the transistors in the layout view, run DRC, Run extraction, run LVS and this simulation all over again until we got it right.

- The design flow we used is the most efficient way to meet our specification.
- If we did not use hand calculations, we would have been totally guessing about the lengths and widths of our transistors.
- If we did not use schematic capture and tried to draw the layout view we would have to redraw the circuit every time we wanted to change the size of a transistor.
- With out DRC we would have to see if we did not violate any design rules by eye!
- With out Extraction we would not know what our parasitic capacitances were.

# Appendix

## Appendix A: Further Work and AC Analysis

#### Further Work:

Use the inverter test bench already created:

- 1. Change the load capacitor to 1000f F, and re-run the simulation, the propagation time should increase. (Press q in the schematic window and click on the capacitor)
- 2. Change the vpulse to a vsin with an Amplitude of 2.5 V, Frequency of 1GHz, DC offset voltage of 2.5 V, and a delay time of zero seconds. You should get a similar result as in Figure 105. Questions: (Delete the DC Analysis or you will get a warning.)
- a. How can we simulate a sine wave with a digital circuit?



Figure 105: Output of Sine Wave Input

#### AC Analysis:

This portion does not have anything to do with meeting our specification, but it is useful to help you set up an AC analysis for other circuits. It also demonstrates that the spectre spice environment is an analog simulation environment that can be used for digital circuits.

Start the software and open up the testbench of the inverter created in the previous sections. Delete the vlpulse and add a vsin. Press q, click on the vsin and fill out the pop-up like Figure 106. Setting the AC magnitude to 1 will make it easier to read the gain off of the plot window.

Cedit Object Properties				2
OK Cancel Apply D	efaults Previ	ous Next		Hel
Apply To Only cur	rent 💷 🛛 insta	nce 💷 📔		
Show _  syst	em 🔳 user 🖡	CDF		
Browse	Reset Insta	nce Labels D	Neplay	
Property		Value		Display
Library Name	NCSU_Analog	[Parts		110
Cell Name	veing			
View Name	synboli			off 💷
instance Name	instance Name V결.			
	Add	Delete	Modify	
User Property	Master Va	due l	Local Value	Display
tvsignore	TRUE	1		110
CDF Parameter		Value		Display
AC magnitude	1 9			110
AC phase	jî J.			off 💷
Offset voltage	2. S 🖞			💷 110
Amplitude	2. S ¥			off 💷
Frequency	10 HĘ			off 💷
Delete time	0.4			

Figure 106: Setting up vsin for an AC Analysis.

To set up an AC Analysis, in the Affirma window go to Analysis... Choose, and click on ac and fill the pop-up like in Figure 107.

Run the simulation and the results window would look like Figure 108.

**Question 1:** Why don't the voltage gains from the transient and AC analysis match? You can read off the peak voltages with a crosshair marker in the transient response and you can add a vertical marker at 1 GHz and display the intercepted data to get the AC voltage gain.

**Question 2:** Re-Run the simulation with the amplitude set to 1 V instead of 2.5 V. The simulation results are closer, but why is there still some error?

Choosing Analyse	es Affirma Ar	nalog Circui	t Design Environme	nt 🗙
OK Cancel De	efaults Apply			Help
Analysis 🗸 tr	ran 🔶 ac	√ sp	√pdisto √sp	SS
v di	c √xf	<b>√pss</b>	<b>√</b> noise	
	AC Ana	alysis		
Sweep Variable				
Frequency				
🗸 Temperature	)			
🗸 Component I	Parameter			
Model Param	neter			
Sweep Range				
◆ Start-Stop	Start 1		Ston 5eg	
🗸 Center-Spar	1 1			
Sweep Type				
Automatic 💷				
Add Specific Poin	nts _l			
Enabled 📕			Options	s

Figure 107: Setting up an AC Analysis from 1Hz to 5GHz.



Figure 108: Results of a Transient an AC Analysis.