

Lab Assignment #4

Copyright©2005 by Gayatri Mehta and Ivan Kourtev
Copyright©2006 by Steven P. Levitan

University of Pittsburgh
Department of Electrical and Computer Engineering
September 22, 2006 (Friday, Week 4)

1 Introduction

The purpose of this lab assignment is to introduce the Micro Magic design tools. This lab is focused on SUE (Schematic User Environment) and MAX (full custom layout editor) tools. A brief description of these tools is as follows:

SUE is more than just a schematic capture tool. It can be used for drawing, viewing, editing schematics, ICONs, etc. It has a built-in netlister to HSPICE, Verilog, IRSIM. It can cross probe directly with MAX, HSPICE, IRSIM and Verilog. It can also run Verilog and IRSIM interactively with logic values displayed on the schematic. The details of this tool and its features can be found in the online documentation.

MAX is the full custom layout editor from Micro Magic Inc. It is not only just a layout editor but it is a complete layout environment. This tool can be used interactively to view and edit hierarchical layout. It provides continuous DRC (Design Rule Check) feedback during layout. More details of the tool and its features can be found in the online documentation.

2 Before you start

1. Make sure you are sourcing the `./CLASS/dot_cshrc` in your own `.cshrc` file
2. Create a `.rhosts` file in your home directory and add yourself to it....
+ username
(that is: plus , space, your login name, return)
3. Create a `.Xdefaults` file in your home directory and add...
Autoraise: false

3 Starting up SUE

- To run the tutorial, you need to make a personal copy of the SUE tutorial directory.
- To install the SUE tutorial, type:

```
mmi_tutorial
```

at the UNIX prompt. Select the appropriate tutorial and then click on “Install Tutorial”. The default installation directory for the tutorial is “<your home directory>/mmi_private/tutorial”. You can also select a different directory to install the tutorial.

- You can view the tutorial in the browser by selecting “View Tutorial in Browser”.

- Before you start the tutorial, “cd” to the directory where the tutorial is installed. If the tutorial has been installed in the default directory, type:

```
cd ~/mmi_private/tutorial/sue
```

- To start SUE type:

```
sue
```

Now you need to follow the instructions given in the tutorial. **Do the following sections from the tutorial:**

Part 1. Getting Started with SUE Design Manager

Part 2. Schematics and Icons

Part 3. Simulating Circuits with SPICE And IRSIM

- Running HSPICE
- Displaying HSPICE Waveforms From Schematics
- Net Names when Netlisting

Note: Skip sections on IRSIM from Part 3.

Part 4. Higher Level Schematics

- Adding Ports
- Naming Multiple Signals
- Making an ICON for our Schematic
- Drawing the ICON
- Adding Text
- Placing our ICON into the schematic
- Modifying the ICON

Note: Skip rest of the sections from Part 4. The main idea of doing Part 4 is to learn how to make an icon that can be used in the testbench.

Part 5. Cross-probing with MAX

4 Starting up MAX

- To run the tutorial, you need to make a personal copy of the MAX tutorial directory.
- To install the MAX tutorial, type:

```
mmi_tutorial
```

at the UNIX prompt. Select the appropriate tutorial and then click on “Install Tutorial”. The default installation directory for the tutorial is “<your home directory>/mmi_private/tutorial”. You can also select a different directory to install the tutorial.

- You can view the tutorial in the browser by selecting “View Tutorial in Browser”.
- Before you start the tutorial, “cd” to the directory where the tutorial is installed. If the tutorial has been installed in the default directory, type:

```
cd ~/mmi_private/tutorial/max
```

- To start MAX type:

```
max
```

Now you need to follow the instructions given in the tutorial. **Do the following sections from the tutorial:**

- Part 1. Introduction
- Part 2. Simple Layout
- Part 3. Bigger Designs and Hierarchy

Note: Skip Part 4 on Max-LS.

5 Assignment (due Friday, September 29, 2006)

1. Complete both SUE and MAX tutorials according to the instructions mentioned above.
2. In addition to the tutorial, design a CMOS inverter. This task involves designing the schematic using SUE, layout using MAX, cross-probe your design from either SUE or MAX and simulate the design using HSPICE. You need to turn in the **printouts** of the **schematic view, layout view** and **simulation waveforms**.

To do Problem 2, create a schematic of a CMOS inverter using NMOS and PMOS transistors in SUE. Add input and output ports to the inverter and make an ICON of the inverter, as described in the tutorial. Now create a new schematic file in SUE and name it as “Inverter_tb.sue”. Add an ICON of the inverter and add a pulse voltage source as shown in the figure below to this file. Now run simulations on the testbench using HSPICE.

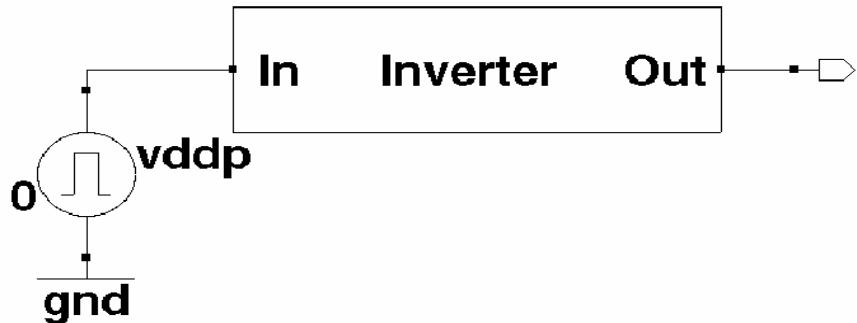


Figure 1. Testbench for the CMOS Inverter

Also draw a layout of an inverter in the MAX. Use the **same filename** in MAX what you have used in SUE for the inverter. For example, Inverter.sue and Inverter.max. Label input and output nodes **but make sure that they are of type input or output (not local)**.

Once you make a layout, you have to cross probe it with the corresponding schematic in SUE. In the pull down “Local” menu in MAX, there is an option called “Spice Convert” that will extract your design.

Note: You need to put .sue files under sue folder and .max files under max folder. When you cross probe your design either from MAX or from SUE, make sure you have schematic and the corresponding layout opened in the SUE and MAX windows respectively. Then do CROSS-PROBING and tell the tool to use the design which is currently opened in the window.

Now in order to run simulations on your design, you need to go back to the SUE window. In the SUE window, you can select from the pull down menu of “Local” whether you want to run simulations on the local schematics or the max extracted netlist. First time, when you ran simulations, they automatically ran on the local schematics. This time, you need to run simulations on the max extracted netlist by selecting that option. **Compare both simulation waveforms and notice if there are any differences in these waveforms.**

Note: You can run simulations only in SUE.