Tips about the project

To run 'magic':

Add environment variable *CAD_HOME* as /apps/magic6_5. Add two new paths, *\$CAD_HOME/bin* and *\$CAD_HOME/lib*, to path definition. For an example of the changes in your *.tcshrc* file look example. Use 'magic' key word to run the program.

To run 'hspice':

Run <u>/usr/local/h97/97/bin/cshrc.meta</u> file using *source* command in the shell. You can add this to your .*tcshrc* file. For an example of the changes in your .*tcshrc* file look example. Use 'hspice' key word to run the program.

Some key point for using 'magic':

The tutorial files are located in the directory, '/apps/magic6_5/lib/magic/tutorial/'. To run 'magic' with a tutorial file, for example first tutorial execute in the shell.

magic /apps/magic6_5/lib/magic/tutorial/tut1

Go over the first four chapters of the magic tutorial. Frequently used commands in the magic: paint, erase, box, select, layers, macro, '.', save, label, getcell, expand, help

To take a print out;

First load the cell in magic. Use 'cif write [file name]' command to generate 'cif' output file. Exit magic. Use *cif2ps* program to convert postscript file. This program will be e-mailed to you. Usage:

cif2ps example.cif > example.ps

To generate a spice file:

First of all, <u>do not forget to put label to the ground, vdd, inputs and output</u> <u>connections</u>. Load the cell in magic. Use 'extract' command to generate circuit extraction file (file with extension 'ext'). Exit magic. Use *ext2spice* program to convert the extraction file to spice file. For example the command '*ext2spice example.ext*' generates '*example.spice*'. This spice file only contains the transistor connections and the capacitances between the connections. So you must write an additional spice file, including the generated spice file, spice model for the transistors, biasing supplies, inputs and the simulation options. Look *example file* below:

* Simulation of cmos inverter .include inverter.spice .include mostr.inc	/	/ Converted spice file Transistor model	
V1 Vdd GND 5 Va in GND pulse(5 0 0 1n 1n 200n	1 400n)		
.tran .1n 400n .option post dcstep=1 option. .op		/	You have to set this
.end			

The transistor model file, 'mostr.inc' will be e-mailed. Download it to your project director and include in the main file as shown above.

Visualization of spice outputs:

A program is called '*awaves*' is used to visualize your simulation and print the results.

Modification in the case you have .tcshrc file:

setenv CAD_HOME /apps/magic6_5

set path=(.	\$CAD_HOME/bin \
	<pre>\$CAD_HOME/lib \)</pre>

source /usr/local/hspice_2000.2/2000.2/bin/cshrc.meta

Modification in the case you have .kshrc file:

export CAD_HOME /apps/magic6_5

set path=(. \$CAD_HOME/bin \ \$CAD_HOME/lib \)

. /usr/local/hspice_2000.2/2000.2/bin/kshrc.meta

For the grading policy visit the course web page.