

Magic usage Guidelines

- Starting magic** - magic -T scmos filename (in the working directory)
drawing a layer - point the mouse on a desired layer on the toolbar or an existing layer in the layout and middle click on mouse

Layers on toolbar:

- polysilicon - gate
ndiffusion - nmos source/drain
pdiffusion - pmos source/drain
ndcontact - ndiffusion to metal 1 contact
pdcontact - pdiffusion to metal 1 contact
polycontact - polysilicon to metal 1 contact
m2contact - metal 1 to metal 2 via
m3contact - metal 2 to metal 3 via
psubstratecontact - psubstrate (nmos bulk) to metal 1 contact
nsubstratencontact - nsubstrate (pmos bulk) to metal 1 contact

Keyboard Shortcuts:

- g** - grid on/off
z, Shift+z - zoom in/out
u - undo
c - copy (**point cursor at the desired bottom-left position**)
m - move (**point cursor at the desired bottom-left position**)
r - rotate
v - fit screen
s - selects a rectangle on which the cursor is pointing (double press: shows all the connections; **very handy**)
a - selects all the rectangles in the selected area
i - selects the visible cell
mouse left click - moves the position of selection rectangle
mouse right click - change the shape of selection rectangle
x, Shift+x - show/hide the contents of imported cell
importing a cell - **getcell filename** in magic's terminal/konsole (**cursor should be pointed at the desired bottom-left position**)
label labelname - assigning node names (**left and right click at the same place on a desired layer**)
erase label - erases the label (**select the rectangle with the label**)

Important Points:

- * the whole layout area is a psubstrate (substrate/bulk for nmos)
- * use nwell to cover the whole pmos region
- * place substrate/bulk contacts and connect them to Vdd/Gnd

Design Rule Check:

drc find S.No (of the error) - displays the reason for error

Steps to extract:

- save - saves with .mag extension
- extract - creates a .ext file (contains co-ordinate information)
- ext2spice - converts the .ext file to .spice file (a spice suited netlist is created)
- ext2sim - converts the .ext file to .sim file (For IRSIM usage)

IRSIM usage Guidelines

Starting IRSIM - irsim filename.sim (in the working directory)

Steps:

h nodename1 nodename2 - assigns logic high (1) to those nodes
eg: h a b

l nodename1 nodename2 - assigns logic low (0) to those nodes
eg: l a b

w nodename1 nodename2 - nodes to be observed/plotted
eg: w a b out

s - simulate

d - displays the results of nodes specified in **w command**

analyzer nodename1 nodename2 - plots them in the analyser window
eg: analyzer a b out

Some handy comands:

vector vectormame nodename1 nodename2 - grouping the nodes
eg: vector in a b c

setvector vectorname value1value2 . . . - assigning values to the vector
eg: setvector in 000

Note: Each input set should be simulated individually (timestep b/w two simulations is 10ns by default; can be changed by **stepsize tstep** comand)

eg: setvector in 000
s
setvector in 001
s

Very handy way of assigning value to a vector and simulating

set vlist {000 001 010 011 100 101 110 111}
foreach vec \$vlist {setvector in \$vec ; s}

References:

opencircuitdesign.com - tutorials (magic, irsim etc)
vlsitechnology.org - std. Libraries (examples)