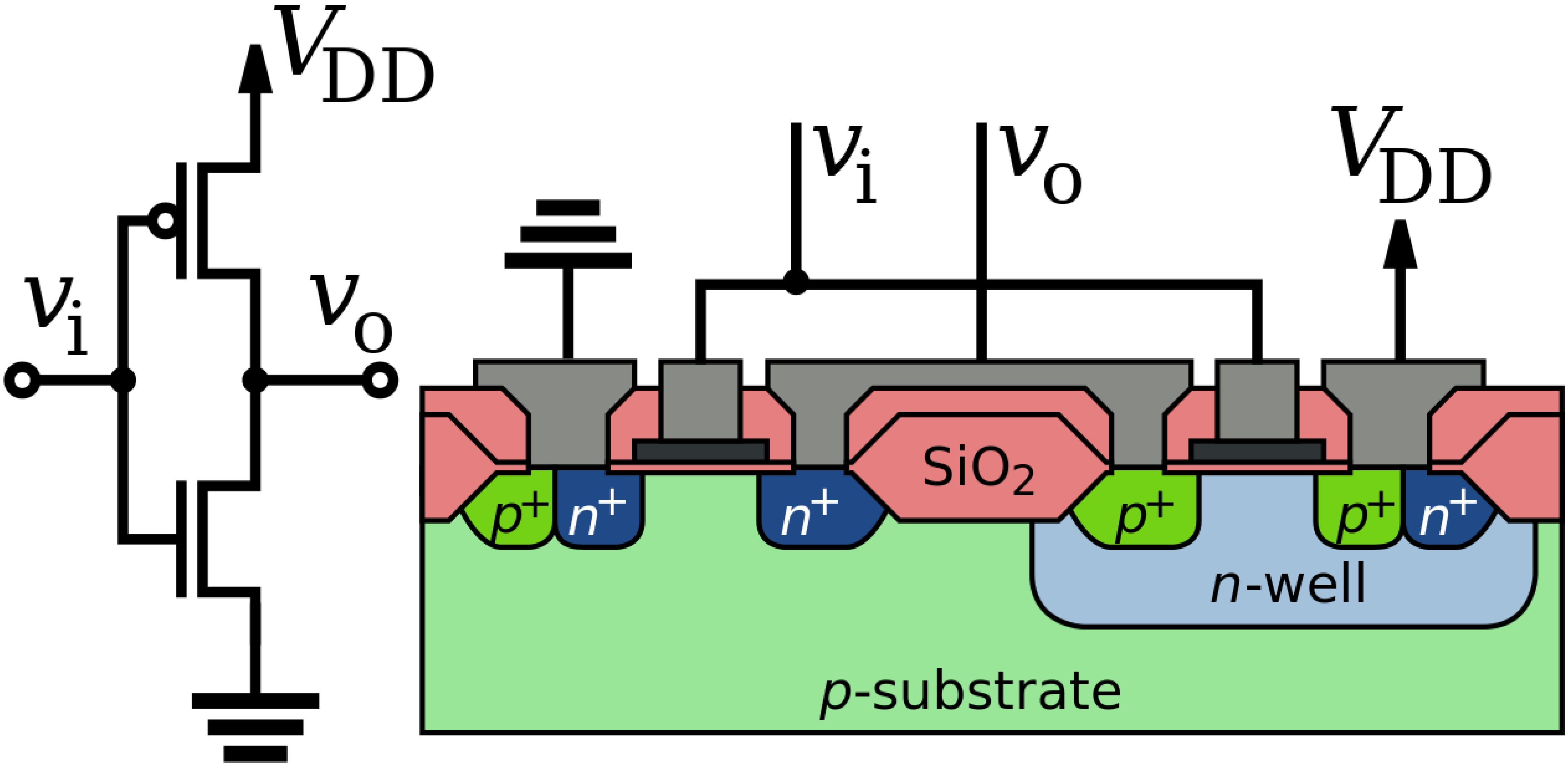
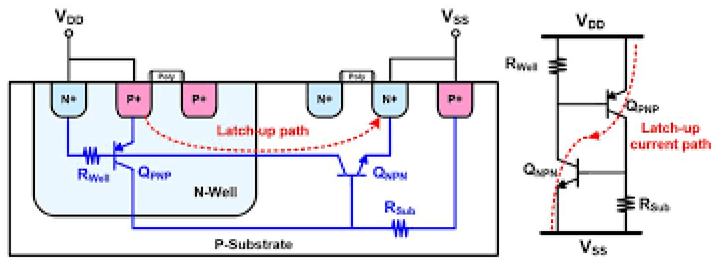
Inverter implementation with open-source tools

CMOS: schematic VS cross-section

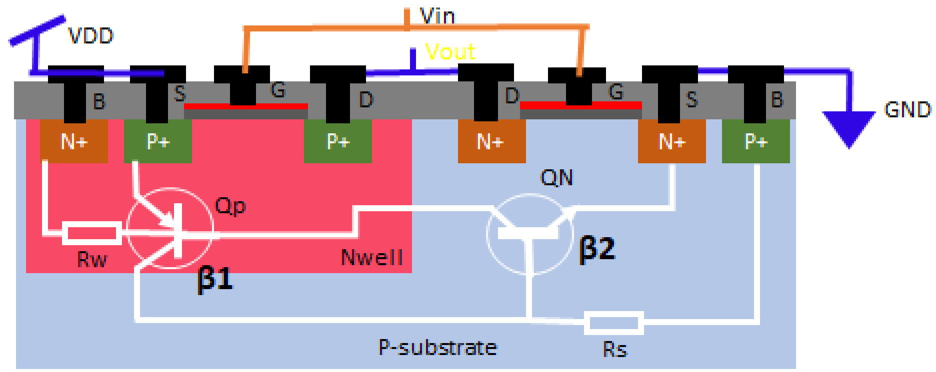




VDD
VSS
N+
P+
Poly
Latch-up path
RWell
QPNP
QNPN
N-Well
RSub
P-Substrate
VDD
RWell
QPNP
Latch-up
current path
QNPN
RSub
VSS

(a) (b)

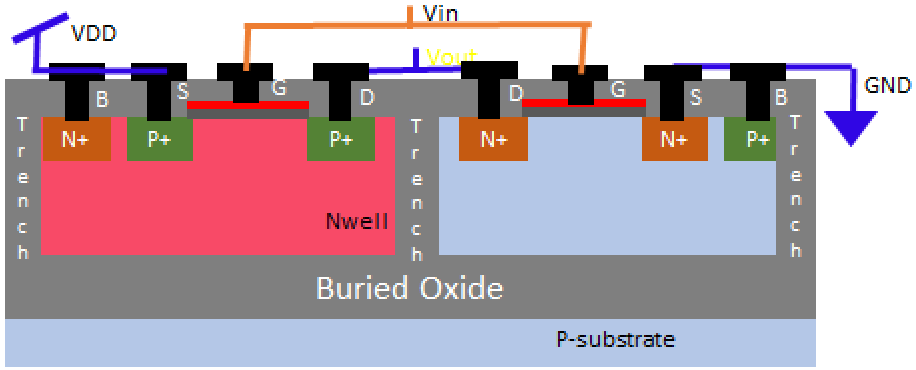
CMOS BULK



- **Proper bias is needed:**
  - →body of the NMOS connected to GND
  - →Nwell of the PMOS connected to VDD

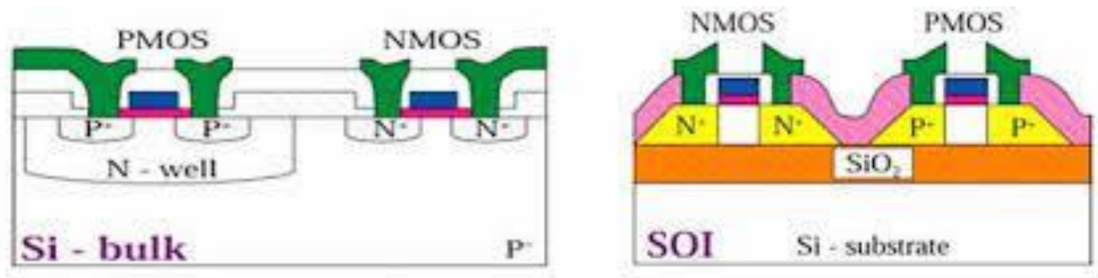
Despite of this, leakage is present

SOI: Trench of Buried Oxide



The additional isolation is aime in reducing the leakage and “isolate” cells from noise/coupling…

CMOS Bulk versus SOI



https://blog.naver.com/PostView.nhn?blogId=ncs1119&logNo=221834592288

CMOS BULK vs CMOS FDSOI

https://www.st.com/content/st_com/en/about/innovation---technology/FD-SOI.html

THE GROWING CHALLENGE OF SEMICONDUCTOR DESIGN LEADERSHIP

https://www.semiconductors.org/the-growing-challenge-of-semiconductor-design-leadership/?utm_content=229882036&utm_medium=social&utm_source=linkedin&hss_channel=lcp-1940570

About FOSSi (Free and Open Source Silicon Foundation)

https://www.eetimes.com/the-democratization-of-chip-design/https://semiengineering.com/toward-democratized-ic-design-and-customized-computing/https://isn.ucsd.edu/courses/beng207/lectures/Tim_Edwards_2021_slides.pdf

Free documentation
https://skywater-pdk.readthedocs.io/en/main/

ABOUT StackMetal Layers
https://skywater-pdk.readthedocs.io/en/main/rules/assumptions.html

http://www.opencircuitdesign.com/
→ Open_PDKs, a PDK installer for open-source EDA tools.
→ --> Magic, the VLSI layout editor, extraction, and DRC tool.
→ --> XCircuit, the circuit drawing and schematic capture tool.
→ --> IRSIM, the switch-level digital circuit simulator.
→ --> Netgen, the circuit netlist comparison (LVS) and netlist conversion tool.
→ --> Qrouter, the over-the-cell (sea-of-gates) detail router.
→ --> Qflow, a complete digital synthesis design flow using open-source software and open-source standard cell --> libraries.
→ --> PCB, the printed circuit board layout editor

To work with open-source tool, an ubuntu/linex terminal is needed

%Settings--> Apps --->Program and Features (under related settings)

%Then go to "Turn Windows features on or off"
%Scroll till Windows Subsystem for Linux. Select is and press ok.
%Restart your PC

%Open Microsoft Store--> App store and search for UBUNTU

%Download the 20.04 version (the newest has bugs….)

%https://sourceforge.net/projects/vcxsrv/files/latest/download
%download VcXsrv

Open Ubuntu

Very good tutorial to be followed:
Running graphical Linux applications on Windows 10 using WSL
https://www.youtube.com/watch?v=F_eQqdFHAOg&t=8s

Magic VLSI Tutorial (part 1), Installation and Technology Files
https://www.youtube.com/watch?v=2LiD-zM6_mw

%go here
http://opencircuitdesign.com/magic/index.html
%Download 2002s, extract the folder and renamed it as current.

%Open the magic system directory. From the ubuntu termina type:
explorer.exe .
%do not forget the dot after the space

mv current/ /usr/lib/x86_64-linux-gnu/magic/sys/
%SUGGESTION: Explore different technology files with different feature size with:
cd /usr/lib/x86_64-linux-gnu/magic/sys/current
%and try different technologies i.e. TSMC18.06.TSMC instead of SCN3ME_SUBM.30 with smaller lamda

%Go back to the home directory
cd ~

%for lamba=0.3
magic –T SCN3ME_SUBM.30

“Playing” with Magic

%Commands for Magic:
paint pdiff
paint ndiff
paint poly
paint metal1

%for connections (vias):
paint pdc
paint ndc
paint polycontact

%To place n-well to VDD connection
paint nwc
%to place body to GND
paint pwc

%For deleting: select the area, click s and then d

%For labeling: Left-click and right-click on the same spot and then write:
label NAME

Save equivalent netlist

File, Save
Then

extract all
%(→convert in ext format)
ext2spice
%(→convert to spice format)

Going back to UBUNTU, respective netlist can be visualized_

vim NOME.ext
%(netlist generica)

vim NOME.ext
%(netlist spice)

Let us use the netlist with (ng)spice

% to install
sudo apt-get install ngspice

% you can run ngspice
ngspice
%to exit
Quit

%create a folder for ngspice
mkdir spice_test

%enter in the folder
cd spice_test

%copy the spice file from Magic into the just-create folder

cp ../inv_magic.spice .
%do not forget the space and the dot to be able to copy

Let us use the graphic (schematic) interface

%xschem installation :-
sudo apt-get install -y xschem