



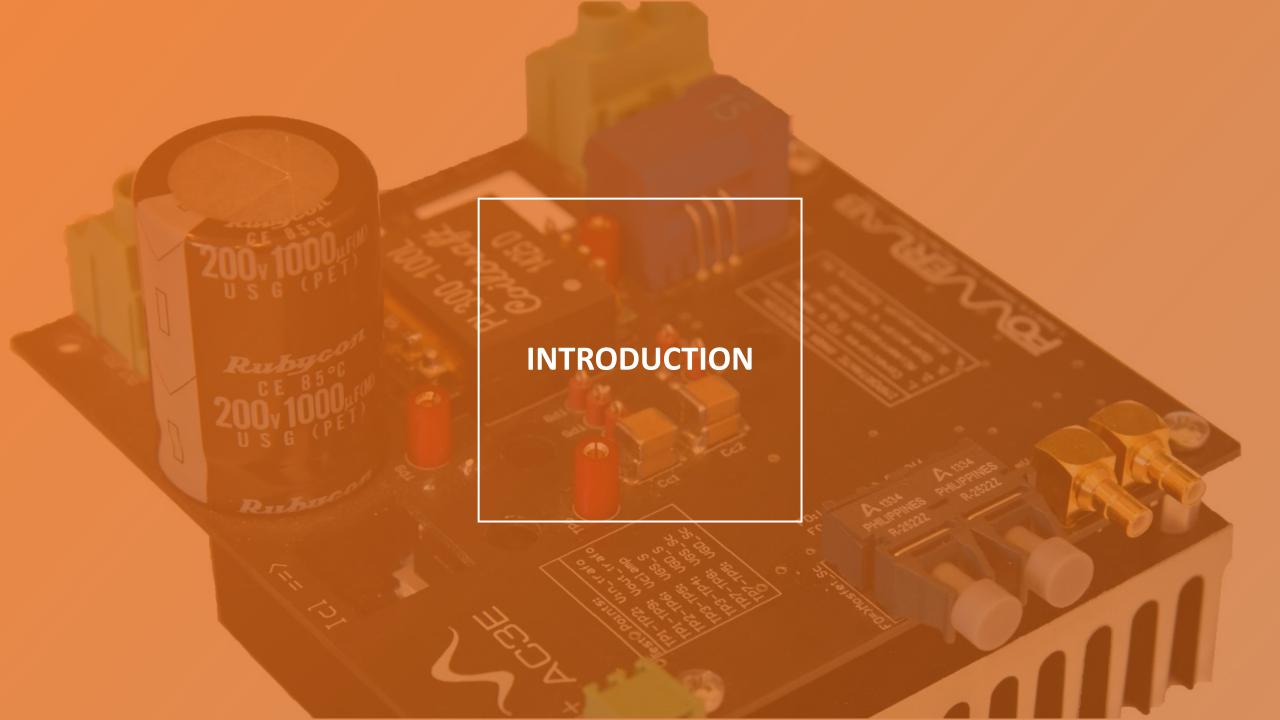
Introduction to the Opensource Analog Design Flow

Jorge Marin, Postdoctoral Researcher AC3E-UTFSM, Valparaiso, Chile

UNIC-CASS 2023

jorge.marinn@usm.cl



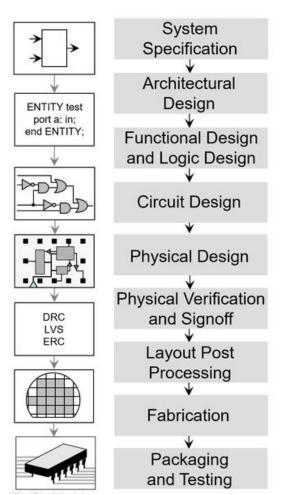


Analog vs digital design flows

ACSE Advanced Center for Liectrical and Electronic Engineering

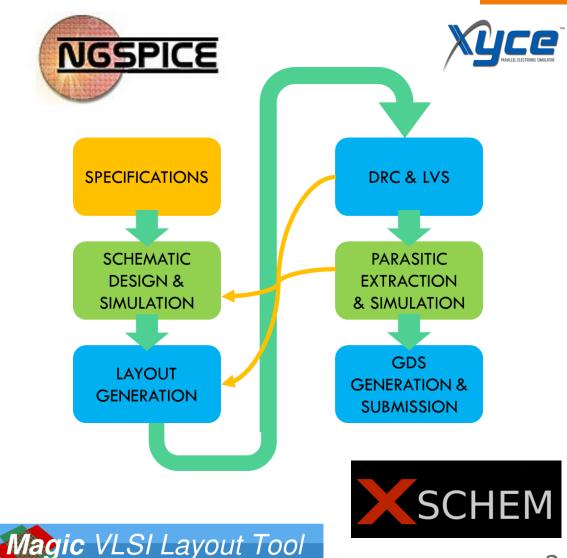
OPENSOURCE EDA TOOLS

DIGITAL FLOW



ш **U**

Current distribution version 8.2





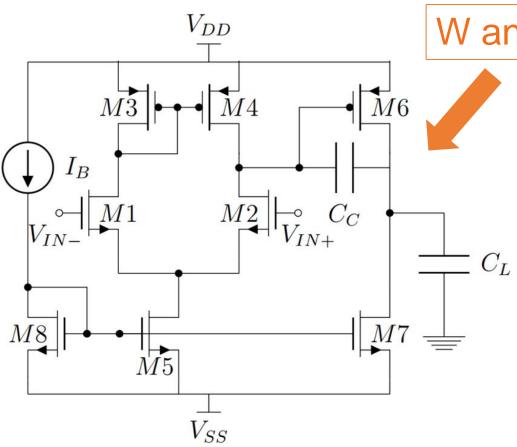






Tradeoffs in analog design





W and L for each transistor M1 to M8?

Power Gain

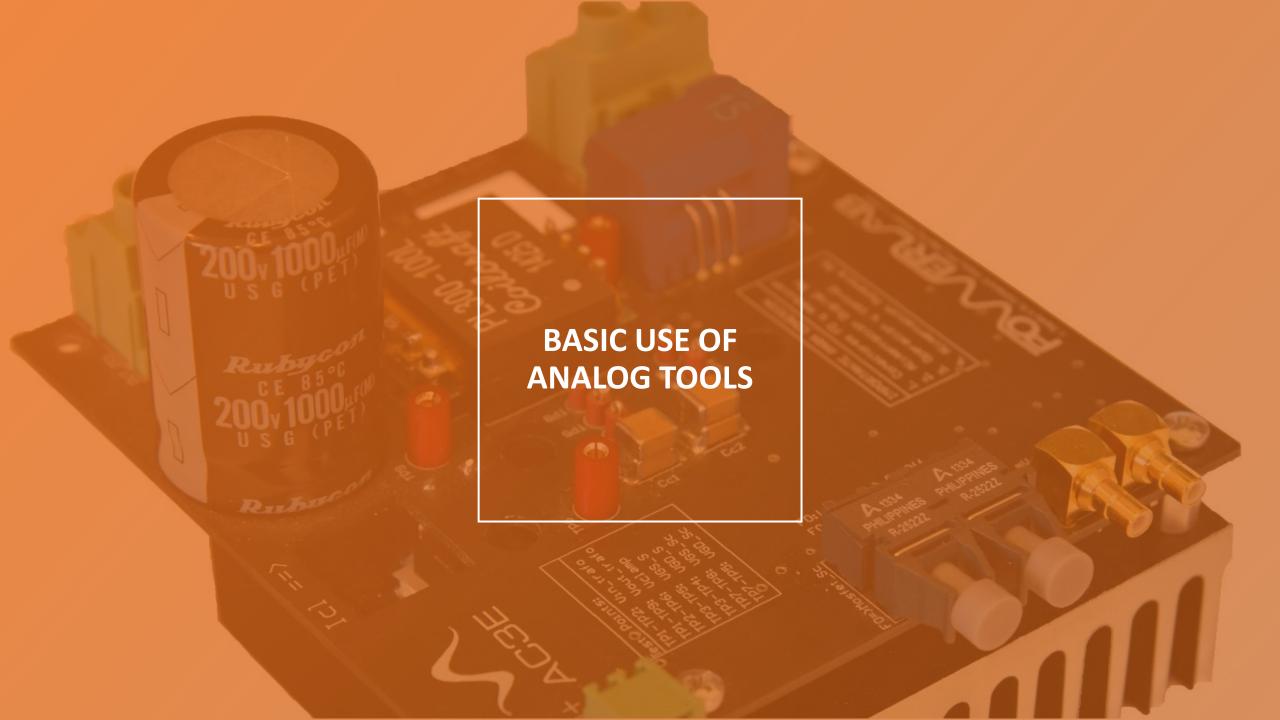
VO Impedance Speed Voltage swing

[B. Razavi]

Analog block example: Miller OTA



Performance is limited by the tradeoff in target specifications



Analog design flow environment setup



How to get started with design and simulation?

- Environment setup
 - Option 1: Analog Mixed Signal Design using docker image & remote desktop
 - e.g. IIC-OSIC-TOOLS docker
 - Clone at: https://github.com/iic-jku/iic-osic-tools
 - Follow detailed instructions
 - Option 2: Analog Mixed Signal Design tools on Linux or WSL using Conda



See the environment setup material in the UNIC-CASS page: https://unic-cass.github.io/02-env-setup.html

Schematic design and simulation



- Relevant tools
 - Xschem -> schematic entry and netlist generation
 - Ngspice

 simulation based on netlist generated by Xschem
- Visualization
 - Ngspice window → quick checks
 - GAW → integrated in Xschem
 - External viewer through raw data (e.g. Python script)



Simulation types



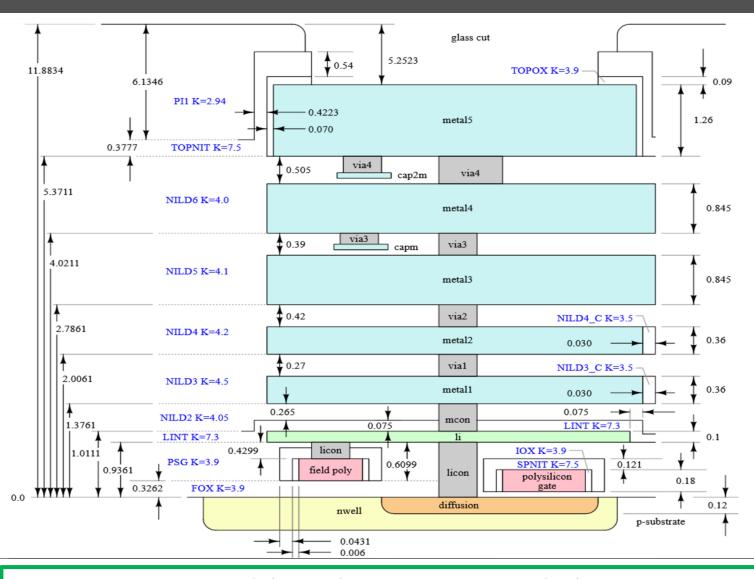
- Simulation types
 - DC → operating point
 - AC → frequency sweep
 - Transient \rightarrow time-domain behavior
 - Noise \rightarrow simulation of device intrinsic noise
 - And others...

Ngspice manual will become your best friend! https://ngspice.sourceforge.io/docs/ngspice-manual.pdf



Available devices in SKY130





SPICE models inside IIC-OSIC-TOOLS docker:

.lib /foss/pdks/sky130A/libs.tech/ngspice/sky130.lib.spice tt

Device Details

1.8V NMOS FET

Spice Model Information

- Cell Name: sky130_fd_pr__nfet_01v8
- Model Name: sky130_fd_pr__nfet_01v8

Operating Voltages where SPICE models are valid

- $V_{DS} = 0$ to 1.95V
- ullet $V_{GS}=0$ to 1.95V
- $\bullet~V_{BS}=+0.3$ to -1.95V

Explore

<u>www.skywater-</u>

<u>pdk.readthedocs.io</u>

for device info

Example



Single-transistor DC simulation

For more detailed explanations, explore:

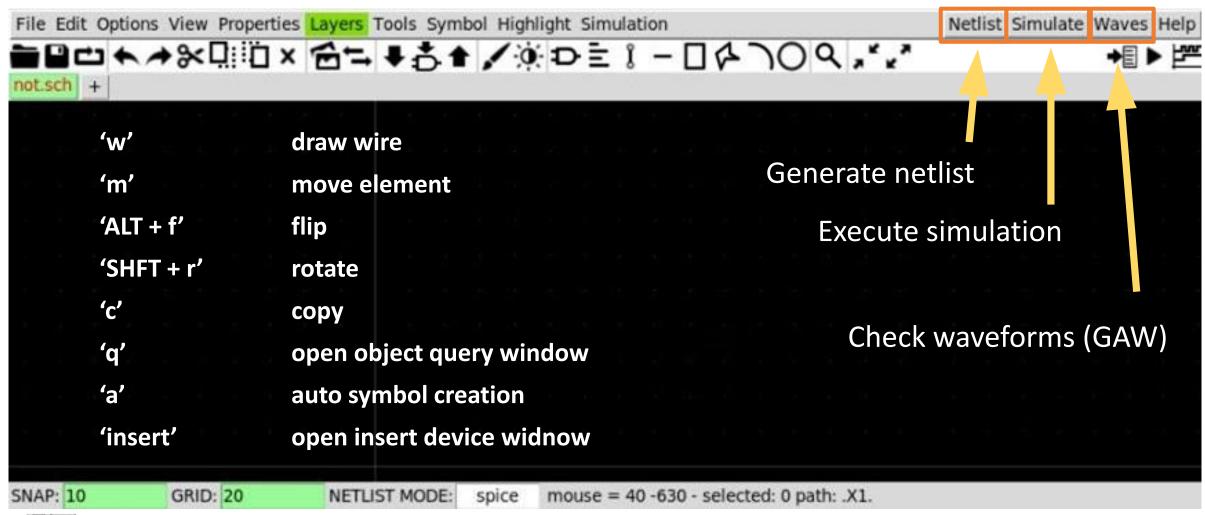
https://unic-cass.github.io/03-analog-design-flow.html#3-analog-design-flow-with-opensource-tools



Xschem GUI



www.xschem.sourceforge.io



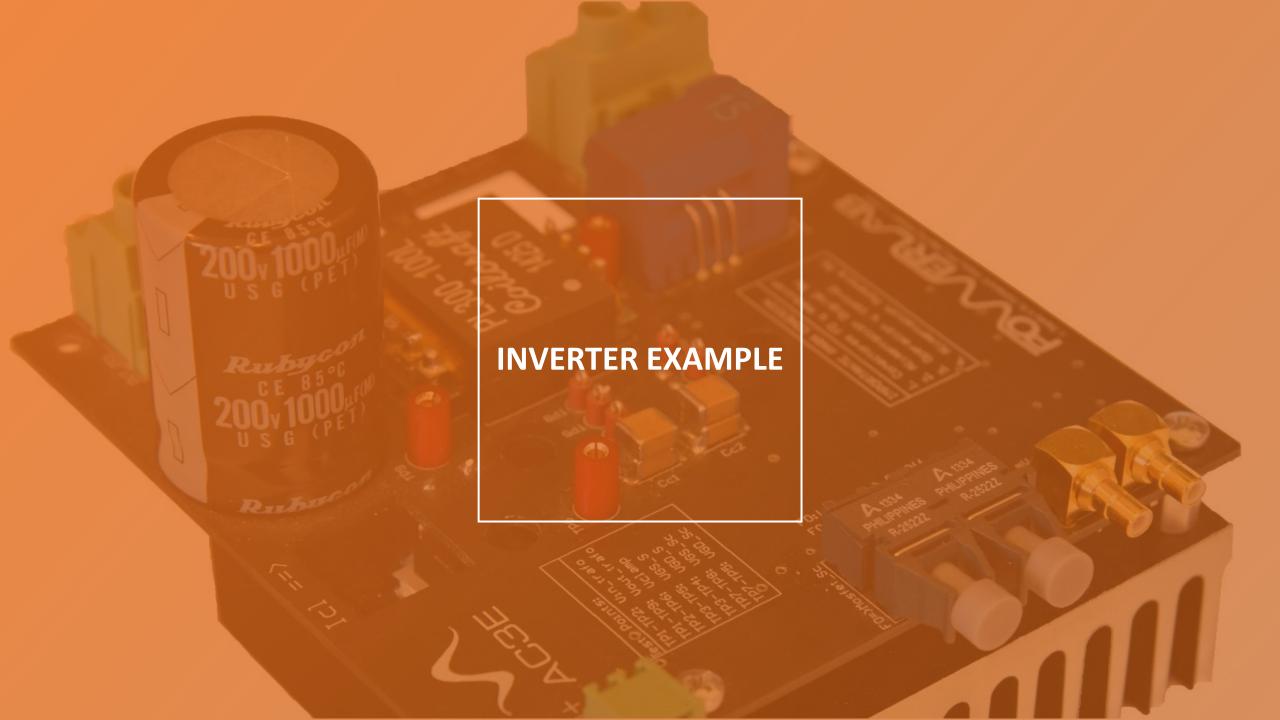
Simulation scope



- Nominal
 - Ideal simulation without considering many fabrication effects
- P(VT) corners
 - Considers global process variation (P) and environment (V, T)

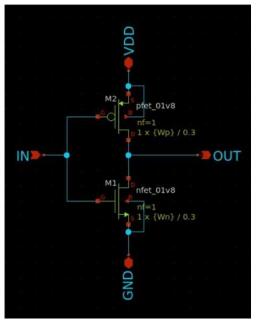
.lib /foss/pdks/sky130A/libs.tech/ngspice/sky130.lib.spice tt

- Mismatch
 - Considers local statistical variation among devices
 - → See: section 7.7 in https://unic-cass.github.io/07-designexamples.html
- Parasitic extraction/ post layout simulations
 Components associated to extrinsic structures (metallization)

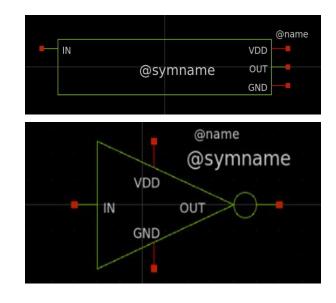


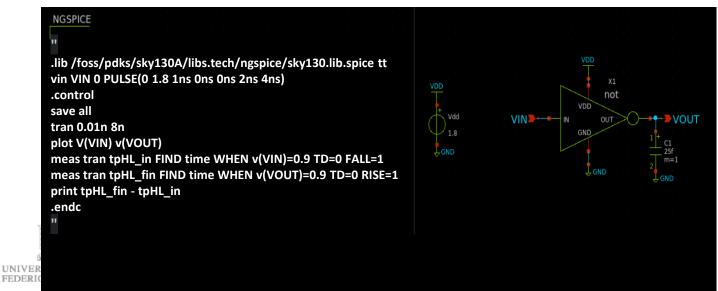
Simple inverter example (to be used in Layout later)

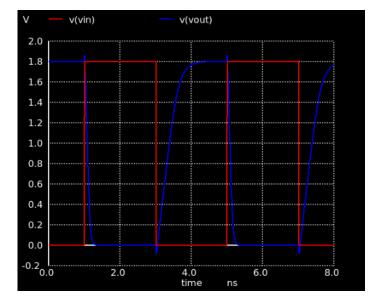


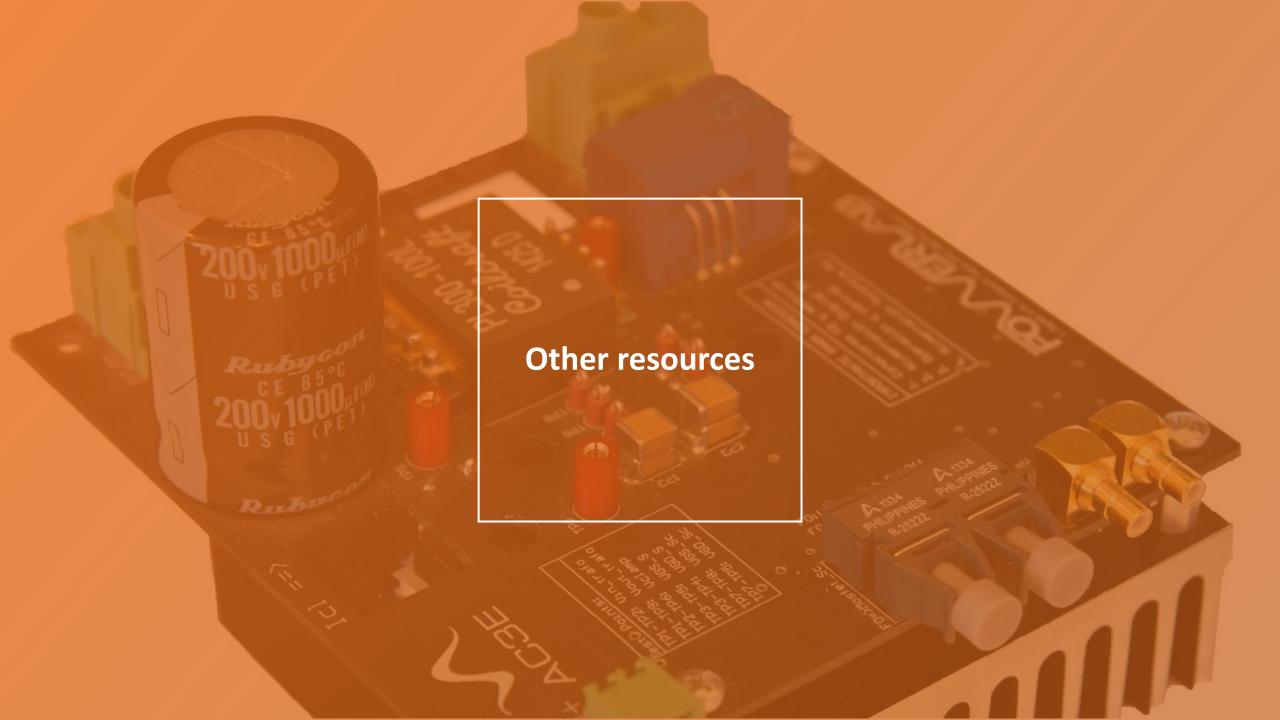












Simple inverter tutorial



Analog flow tutorial for main flow (in Spanish with subtitles)



Link: https://www.youtube.com/watch?v=6NEOxb7CF6A



Opensource designs \rightarrow re-use and get inspired

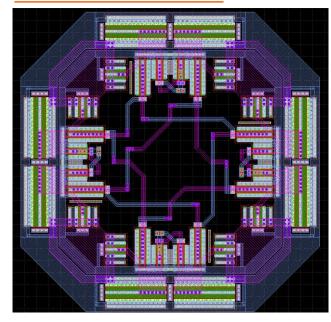


Visit: https://platform.efabless.com/projects/public

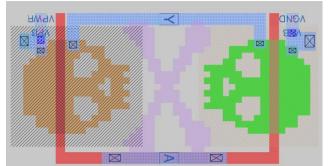
SAR ADC



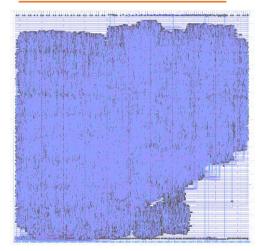
Satellite transceiver



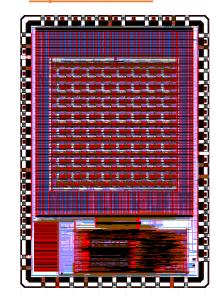
Logic inverter



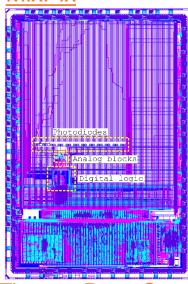
Sudoku Accelerator



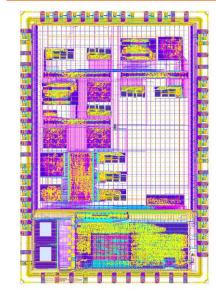
OpenFPGA



MixPix



Time to Data Converter



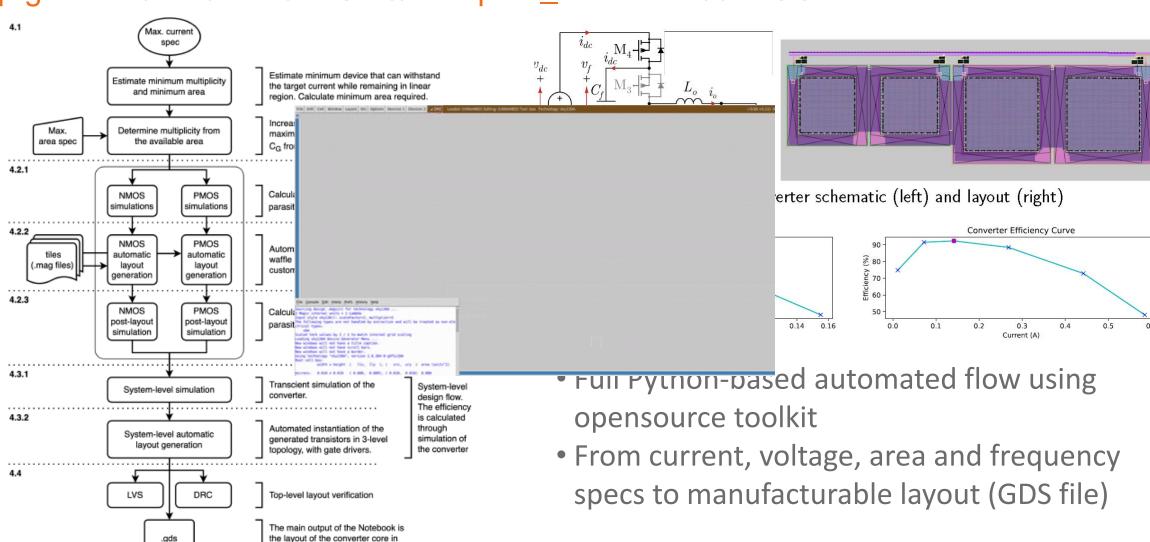


Code-a-chip automated power IC specs-to-GDS flow



https://github.com/sscs-ose/sscs-ose-code-a-chip.github.io/tree/main/VLSI23/accepted_notebooks/3LFCC

GDSII format.





Thank you



AC3E

Advanced Center for Electrical and Electronic Engineering



www.ac3e.usm.cl