

RPLY__EX0__SKY130NM

Slideshow PDF

Who

Carsten Wulff

Why

I wanted to create a step by step “tutorial” for the flow. Both to debug the tech setup, and to make it easier for you to learn the tools. I use a simple current mirror as the example.

Changelog/Plan

I’ve tagged the repo at each stage. That means you can checkout the 0.1.0 tag and do things yourself.

Make sure you follow the latest readme though, I’ve fixed bugs in the readme.

When you clone main, you can do

```
pandoc -s -t slidy README.md -o README.html
```

and then open the html file.

To checkout a specific stage, do

```
git checkout 0.1.0
```

Tag	Status	Comment
0.1.0	:white_check_mark:	Fix readme
0.2.0	:white_check_mark:	Made schematic
0.3.0	:white_check_mark:	Typical simulation
0.4.0	:white_check_mark:	Corner simulation
0.5.0	:white_check_mark:	Made layout
0.6.0	:white_check_mark:	DRC/LVS clean
0.7.0	:white_check_mark:	Extracted parasitics
0.8.0	:white_check_mark:	Simulated parasitics
0.9.0	:white_check_mark:	Updated README with simulation results
1.0.0	:white_check_mark:	All done

What

What	Lib/Folder	Cell/Name
Schematic	RPLY_EX0_SKY130NM	RPLY_EX0.sch
Layout	RPLY_EX0_SKY130NM	RPLY_EX0.mag

Signal interface

Signal	Direction	Domain	Description
IBPS_4U	Input	VDD_1V8	Input bias current
IBNS_20U	Output	VDD_1V8	Output current
VSS	Input	Ground	

Key parameters

Parameter	Min	Typ	Max	Unit
Technology		Skywater 130 nm		
AVDD	1.7	1.8	1.9	V
IBPS_20U	16	21	27	uA
Temperature	-40	27	125	C

For details see sim/RPLY_EX0

Getting Started

aicex

This repository was built in aicex, so if you want to try the tutorial from scratch, then you need aicex first.

Initialize new IP

```
cd aicex/ip
cicconf newip ex0
```

Add remote

Create a repository with the same name on your choosen git vendor (for example github)

```
cd rply_ex0_sky130nm
git remote add origin git@github.com:wulffern/rply_ex0_sky130nm.git
git branch -M main
git push -u origin main
```

Fix README

Open README.md in your favorite text editor and make necessary changes

Familiarize yourself with the Makefile and make

```
cd work
make
```

Draw schematic

All commands (except simulation) must be started from work/

```
cd work/
make xview
```

Add Ports

Add IBPS_4U and IBPS_20U ports, the P and N in the name signifies what transistor the current comes from. So IBPS must go into a diode connected NMOS, and N will be our output, and go into a diode connected PMOS somewhere else

Add transistors

Use ‘Shift-I’ to open the library manager. Use the “sky130B/libs.tech/xschem” path. Open the “sky130_fd_pr” library. Find nfet_01v8.sym and place in your schematic.

Select the transistor by clicking on it, press ‘q’ to bring up the properties. Set L=0.36, W=3.6, nf=2 and press OK.

Select the transistor and press ‘c’ to copy it, while dragging, press ‘shift-f’ to flip the transistor so our current mirror looks nice. ‘shift-r’ rotates the transistor, but we don’t want that now.

Press ESC to deselect everything

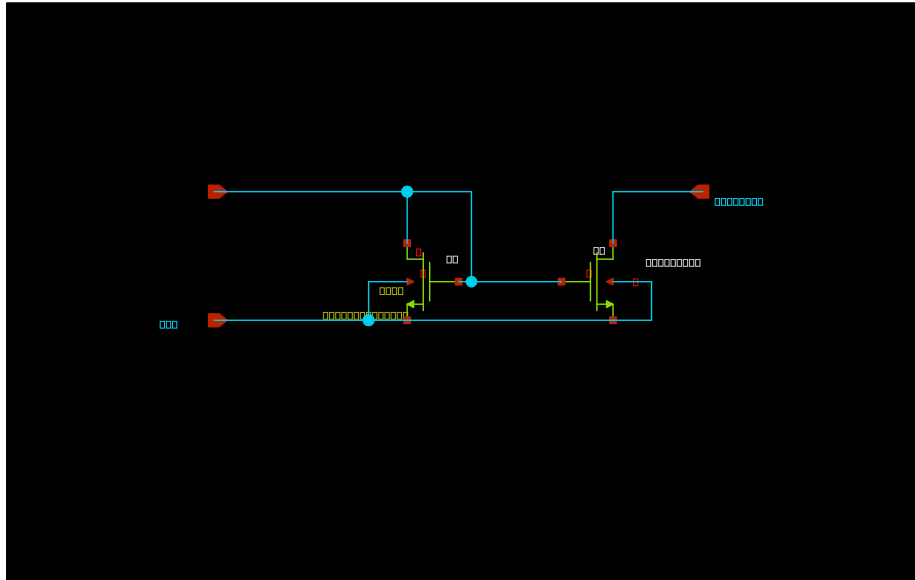
Select ports, and use ‘m’ to move the ports close to the transistors.

Press ‘w’ to route wires.

Use ‘shift-z’ and z, to zoom in and out

Use ‘f’ to zoom full screen

Remember to save the schematic



Netlist schematic

Check that the netlist looks OK

In work/

```
make xsch
```

```
cat xsch/RPLY_EX0.spice
```

Run typical simulation

I've made cicsim that I use to run simulations (ngspice) and extract results

Setup simulation environment

Navigate to the rply_ex0_sky130nm/sim/ directory.

Make a new simulation folder

```
cicsim simcell RPLY_EX0_SKY130NM RPLY_EX0 ../tech/cicsim/simcell_template.yaml
```

I would recommend you have a look at simcell_template.yaml file to understand what happens.

Familiarize yourself with the simulation folder

I've added quite a few options to cicsim, and it might be confusing. For reference, these are what the files are used for

File	Description
Makefile	Simulation commands
cicsim.yaml	Setup for cicsim
summary.yaml	Generate a README with simulation results
tran.meas	Measurement to be done after simulation
tran.py	Optional python script to run for each simulation
tran.spi	Transient testbench
tran.yaml	What measurements to summarize

The default setup should run, so

```
cd RPLY_EX0
make typical
```

Modify default testbench (tran.spi)

Delete the VDD source

Add a current source of 4uA, and a voltage source of 1V to IBNS_20U

```
IBP 0 IBPS_4U dc 4u
V0 IBNS_20U 0 dc 1
```

Add the current in V0 to the plots

Modify measurements (tran.meas)

Add measurement of the current and VGS

```
let ibn = -i(v0)
meas tran ibns_20u find ibn at=5n
meas tran vgs_m1 find v(ibps_4u) at=5n
```

Run simulation

```
make typical
```

and check that the output looks okish.

Often, it's the measurement that I get wrong, so instead of rerunning simulation every time I've added a "--no-run" option to cicsim. For example

```
make typical OPT="--no-run"
```

will skip the simulation, and rerun only the measurement. This is why you should split the testbench and the measurement. Simulations can run for days, but measurement takes seconds.

You should notice that the current is not 20uA. We need to fix the schematic to make that happen. Change the instance name of M2 to “M2[4:0]”, and rerun typical simulation. Remember to save the schematic.

Modify result specification (tran.yaml)

Add the result specifications, for example

```
ibn:
  src:
    - ibns_20u
  name: Output current
  min: -20%
  typ: 20
  max: 20%
  scale: 1e6
  digits: 3
  unit: uA

vgs:
  src:
    - vgs_m1
  name: Gate-Source voltage
  typ: 0.6
  min: 0.3
  max: 0.7
  scale: 1
  digits: 3
  unit: V
```

Re-run the measurement and result generation

```
make typical OPT="--no-run"
```

Open result/tran_Sch_typical.html

Check waveforms

Start Ngspice

```
ngspice
```

Load the results, and view the vgs

```
load output_tran/tran_SchGtAttTtVt.raw
plot v(ibps_4u)
```

```
plot i(v0)
```

Based on the waveform we can see that maybe the voltage and current is not completely settled at 5 ns.

Change the measurement to occur at 9.5ns

Run Corner simulation

All commands should be run in `sim/RPLY_EX0`

Analog circuits must be simulated for all physical conditions, we call them corners. We must check high and low temperature, high and low voltage, all process corners, and device-to-device mismatch.

For the current mirror we don't need to vary voltage, since we don't have a VDD.

Remove Vh and Vl corners (Makefile)

Open Makefile in your favorite text editor.

Change all instances of “Vt,Vl,Vh” and “Vl,Vh” to Vt

Run all corners

To simulate all corners do

```
make typical etc mc
```

where etc is extreme test condition and mc is monte-carlo.

Wait for simulations to complete.

Modify measurements to check settling

Let's say we want to check if the current has settled in our transient. We could extract the current at 9.0 ns and check that it's roughly the same.

Add the following line to `tran.meas`

```
meas tran ibns_20u_9n find ibn at=9n
```

And add the parameter to `tran.yaml`

```
ibn:
  src:
    - ibns_20u
    - ibns_20u_9n
```

Now, as you saw, the simulations take quite a while, so we don't want to rerun that. Instead do

```
make typical etc mc OPT="--no-run"
```

Get creative with python

If you're lazy, like me, then you don't want to spend time checking all the 9.5 ns numbers versus the 9 ns numbers. I'd much rather tell the computer how to do that.

It might be possible to do in ngspice, but sometimes a more complex tool is easier.

Open `tran.py` in your favorite editor, try to read and understand it.

The `name` parameter is the corner currently running, for example `tran_SchGtAmcttTtVt`.

The measured outputs from ngspice will be added to `tran_SchGtAmcttTtVt.yaml`

Delete the "return" line.

Add the following line

```
# Do something to parameters
obj["ibn_settl_err"] = obj["ibns_20u"] - obj["ibns_20u_9n"]
```

Add the error to the result spec `tran.yaml`

```
err:
  src:
    - ibn_settl_err
  name: Current settling error
  typ: 0
  min: -2
  max: 2
  scale: 1e9
  digits: 3
  unit: nA
```

Re-run measurements to check the python code

```
make typical etc mc OPT="--no-run"
```

Generate summary

Run

```
make summary
```

Install pandoc if you don't have it

Run


```
pandoc -s -t slidy README.md -o README.html
```

to generate a HTML slideshow that you can open in browser. Open the HTML file.

Think about the results

From the corner and mismatch simulation, we can observe a few things.

- The typical value is not 20 uA. This is likely because we have a M2 VDS of 1 V, which is not the same as the VDS of M1. As such, the current will not be the same.
- The statistics from 30 corners show that when we add or subtract 3 standard deviation from the mean, the resulting current is outside our specification of $\pm 20\%$. I'll leave it up to you to fix it.

Make layout

Open Magic VLSI

```
magic &
```

Navigate to design directory

```
cd ../design
cd RPLY_EX0_SKY130NM
load RPLY_EX0.mag
```

Now brace yourself, Magic VLSI was created in the 1980's. For it's time it was extremely modern, however, today it seems dated. However, it is free, so we use it.

Magic general

Try google for most questions, and there are youtube videos that give an intro.

- Magic Tutorial 1
- Magic Tutorial 2
- Magic Tutorial 3
- Magic command reference

Default magic start with the BOX tool. Mouse left-click to select bottom corner, left-click to select top corner.

Press "space" to select another tool (WIRING, NETLIST, PICK).

Type "macro help" in the command window to see all shortcuts

Hotkey	Function
v	View all
shift-z	zoom out
z	zoom in
x	look inside box (expand)
shift-x	don't look inside box (unexpand)
u	undo
d	delete
s	select
Shift-Up	Move cell up
Shift-Down	Move cell down
Shift-Left	Move cell left
Shift-Right	Move cell right

Add transistors

In the Window menu, turn grid on, set grid 0.5 um and turn on snap-to grid.

Select “Devices 1 - NMOS”. Match the parameters to schematic (W=3.6, L=0.36, fingers=2)

Unexpand, so it's possible to select the device (shift-x)

Place cursor over the device and select (s)

Move cursor to somewhere else, and copy (c), it will then snap to grid.

Select the old device, and delete (d).

Copy 4 more devices for M2.

Add Ground

In the command window, type

```
see no *
see locali
see m1
```

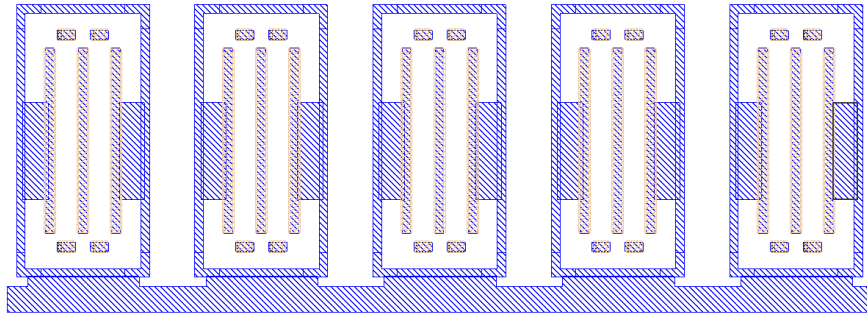
Select a 0.5 um box below the transistors and paint the rectangle (middle click on locali)

Change grid to 0.1 um.

Connect guard rings to ground. Select a smaller box between guardring and the ground rectangle.

Select the rectangle, and copy to the other transistors

Connect the sources to ground.



Route Gates

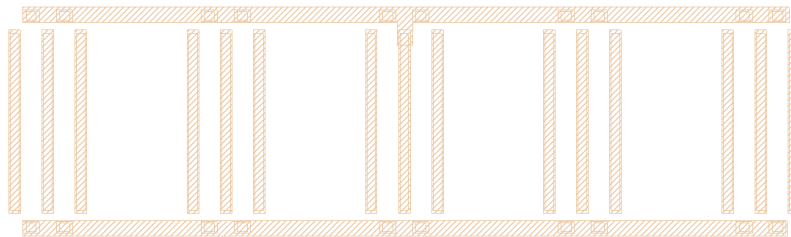
All the gates are connected, so we can enter use the wire mode

`see no locali`

It seems like the device generator adds too small m1 around the gate, so add a rectangle.

Press “space” to enter wire mode. Left click to start a wire, and right click to end the wire.

The drain of M1 transistor needs a connection to from gate to drain. We do that for the middle transistor.



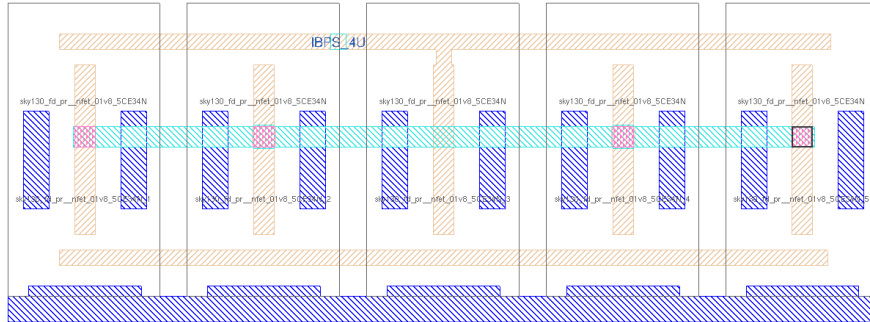
□

Drain of M2

Select a box on the left most transistor drain. Paint m1.

Unexpand all, use the wire tool to draw connections for the drains.

To add vias you can do “shift-left” to move up a metal, and “shift-right” to go down.



Add labels

Select a box on a metal, and use “Edit->Text” to add labels for the ports.

Check layout

The DRC can be seen directly in Magic VLSI as you draw.

To check layout versus schematic navigate to work/ and do

```
make xsch xlv
```

And you should see that it's incorrect. I forgot one transistor of the current mirror, M2 was 5 devices.

Add the fifth transistor and try again. It should still be incorrect.

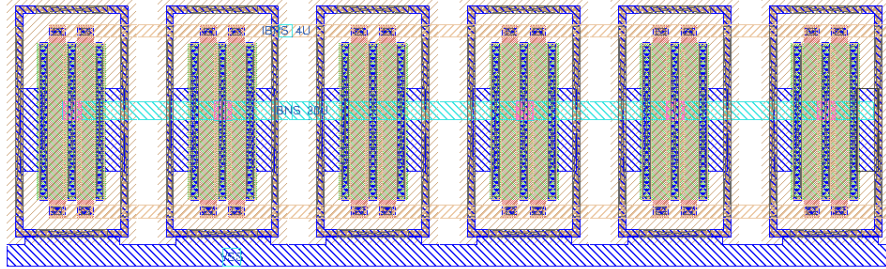
Turns out that the Xschem interpretation of width is different than in Magic VLSI.

In xschem “W=3.6, nf=2” means that the device is actually 3.6 um wide, but has two fingers. In Magic “W=3.6, nf=2” means that the device is 7.2 um wide, and has fingers of 3.6 um.

The easiest way to fix it is to modify the schematic to match the layout.

Open the schematic, select M1, press q, and change “W=7.2”. Do the same for M2.

Now the layout should match the schematic.



Extract parasitics

With the layout complete, we can extract parasitic capacitance.

```
make lpe
```

Check the generated netlist

```
cat lpe/RPLY_EX0_lpe.spi
```

Simulate parasitics

Navigate to `sim/RPLY_EX0`. We now want to simulate the layout.

The default `tran.spi` should already have support for that.

Open the Makefile, and change

```
VIEW=Sch
```

to

```
VIEW=Lay
```

Typical simulation

Run

```
make typical
```

The simulation might not look right.

Open the `work/lpe/RPLY_EX0_lpe.spi` and `work/xsch/RPLY_EX0.spice` and have a look at the `.subckt` line.

For me, the ports were not the same order, which makes the simulation fail.

To fix it, open `design/RPLY_EX0_SKY130NM/RPLY_EX0.mag` in your favorite text editor. Yes, the layout file is a text file!

Take a look towards the bottom, you'll see

```

flabel metal1 4460 1360 4520 1420 0 FreeSans 320 0 0 0 IBPS_4U
port 1 nsew
flabel local1 4200 300 4280 380 0 FreeSans 320 0 0 0 VSS
port 2 nsew
flabel metal2 4520 980 4600 1060 0 FreeSans 320 0 0 0 IBNS_20U
port 3 nsew

```

Change the numbers so we get the same port order as the schematic

```

flabel metal1 4460 1360 4520 1420 0 FreeSans 320 0 0 0 IBPS_4U
port 2 nsew
flabel local1 4200 300 4280 380 0 FreeSans 320 0 0 0 VSS
port 1 nsew
flabel metal2 4520 980 4600 1060 0 FreeSans 320 0 0 0 IBNS_20U
port 3 nsew

```

Open a new terminal, navigate to work/ and extract the parasitics again

```
make lpe
```

Check the work/lpe/RPLY_EX0_lpe.spi again.

Run typical simulation.

Observe that now the difference between “ibps_20u” and “ibps_20u_9n” is a bit large.

Check the current waveform. Change the transient simulation to run a bit longer, and extract a bit later. 19.5 ns seem to work.

Corners

Navigate to sim/RPLY_EX0. Run all corners again

```
make all
```

Summary

Open summary.yaml and add the layout files.

```

- name: Lay_typ
  src: results/tran_Lay_typical
  method: typical
- name: Lay_etc
  src: results/tran_Lay_etc
  method: minmax
- name: Lay_3std
  src: results/tran_Lay_mc
  method: 3std

```

Open the README.md and have a look at the results.

Gotchas

- If you have not resimulated the schematic, then you're comparing apples to oranges since the schematic had $W=3.6$
- In the extracted layout the ad, as, etc looks funky, I don't understand why they are zero
- One of reasons the simulation is slow is that ngspice needs to load about 50 MB of spice files (the skywater models). They could run faster if we only loaded what is necessary, but that's a bit more work.

Bugs

If you find errors in this “tutorial”, then fork, fix, and send me a PR.