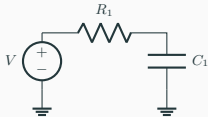


EE5311: Digital IC Design

Tutorial 1

ngspice setup

- To first ensure you have a working setup, try simulating a toy RC circuit:



- Save the equivalent SPICE netlist for the above circuit as `demo.spice`:

```
.title demo
V1 in GND dc 0 PULSE (0 5 1u 1u 1u 1 1)
R1 in out 10k
C1 out GND 1u
.tran 10u 50m
.end
```

- Load the netlist onto ngspice using: `ngspice demo.spice`
- Simulate the transient response of the RC circuit using the command `run` inside the ngspice shell
- Plot the voltage waveforms using: `plot v(in) v(out)`

- This script enables creation of schematic using Sky130 components.
- Create a directory ee5311 and organize all your tutorials in this directory
- For the first tutorial, create a separate directory, navigate into it and invoke xschem:

```
mkdir -p ${HOME}/ee5311/tutorial_1  
cd ${HOME}/ee5311/tutorial_1  
xschem
```

- If you get the error `can't read "PDK_ROOT"`, use the command:
`export PDK_ROOT=/cad/share/pdk`

xschem front screen

xschem - top.sch

File Edit Options View Properties Layers Tools Symbol Highlight Simulation Netlist Simulate Waves Help

top.sch +

Analog primitives validation

Digital standard cells

Verilog-A example

Interesting remote repositories

Git / Google-Skywater PDK links

Git / Google-Skywater RERAM PDK links

Place these useful launchers in your schematic:

PERP, VPP CAP

VARACTORS

MIM

PFET

NFET

RES

DIODE

PNP

NPN

RERAM

ESD

3 PIN PFET

3 PIN NFET

PFET

NFET

These symbols take W as Finger width instead of total width

set ngbehavior=hsa
set ng_nomodcheck

Note: Still untested!

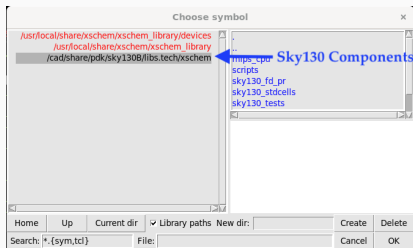
SNAP: 10 GRID: 20 MODE: spice Stretch: 0 NUM LOCK SET mouse = -60 -2230 - selected: 0 path .

Experiment 1

Plot I_{DS} vs V_{GS} for an nMOS $\left(\frac{W}{L} = \frac{0.42\mu\text{m}}{0.15\mu\text{m}}\right)$ with $V_{DS} = V_{GS}$ varied from 0 to V_{DD}

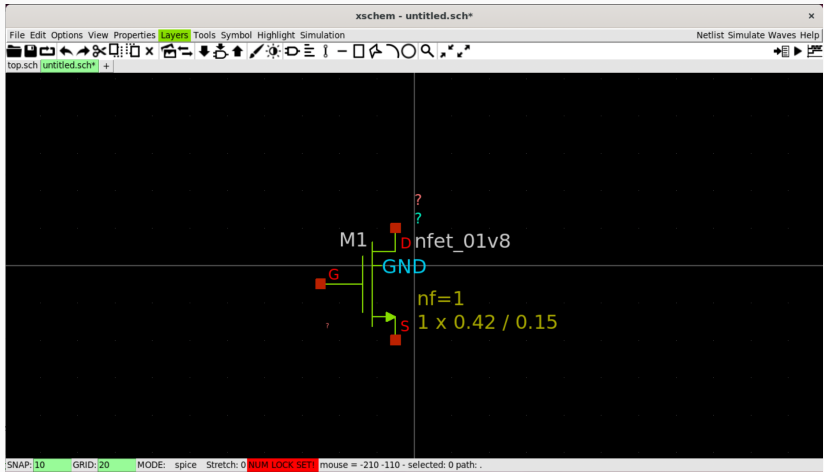
- Create new schematic
- Insert a new instance using the keys: Shift + I
- Choose nMOS instance from:

xschem ► sky130_fd_pr ► nfet3_01v8



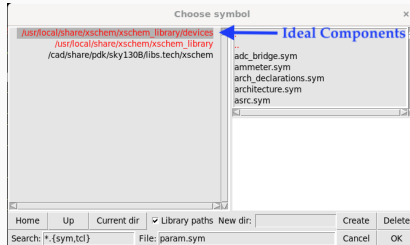
- Change the width of nMOS to $0.42\mu\text{m}$ by selecting it with left mouse click followed by the key q

Experiment 1...



Experiment 1...

- Insert GND pin at the source terminal of the nMOS using :
`xschem_library ► devices ► gnd`



- Insert a pin with label `Vin` at the gate terminal of nMOS using:
`xschem_library ► devices ► lab_pin`
`Alt + R` rotates the label.
- Connect the drain and gate terminals with a wire using the `w` key.
Change direction of wire using `w` or `Space` key.

Experiment 1...

- Create voltage source using:

xschem_library ► devices ► vsource

with name=Vin1, value=1.8V and connect it between Vin and GND

- Insert a SPICE code using:

xschem_library ► devices ► code_shown

with name=sim and value=

```
".control
```

```
set filetype=ascii
```

```
dc Vin1 0 1.8 0.01
```

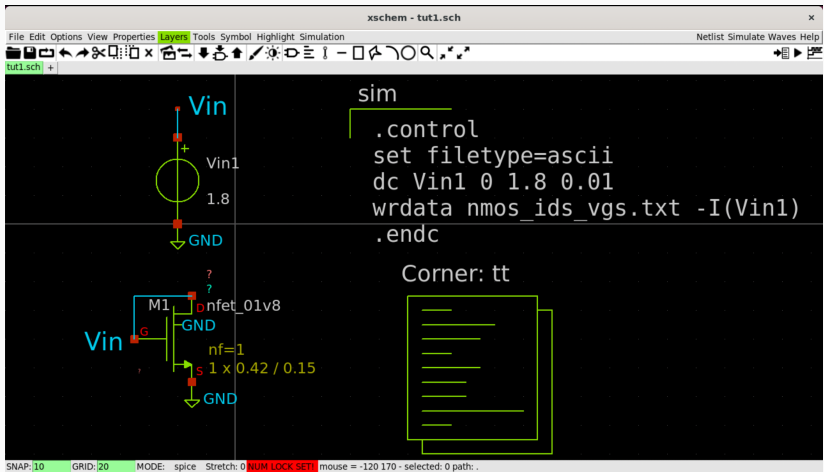
```
wrdata nMOS_ids_vgs.txt -I(Vin1)
```

```
.endc"
```

- Insert the simulation corner using:

xschem ► sky130_fd_pr ► corner

Experiment 1 - complete schematic



The screenshot displays the xschem schematic editor interface. The title bar reads "xschem - tut1.sch". The menu bar includes "File", "Edit", "Options", "View", "Properties", "Layers", "Tools", "Symbol", "Highlight", and "Simulation". The toolbar contains various icons for file operations, editing, and simulation. The main workspace is divided into two panes. The left pane shows a circuit schematic with a voltage source "Vin1" (1.8V) connected to the gate of an nMOSFET "M1". The MOSFET's source is connected to ground, and its drain is connected to a node labeled "GND". The MOSFET model is specified as "nmosfet 01v8" with parameters "nf=1" and "s 1 x 0.42 / 0.15". The right pane shows the simulation code in a text editor, starting with "sim" and ending with ".endc". The code includes ".control", "set filetype=ascii", "dc Vin1 0 1.8 0.01", "wrdata nmos_ids_vgs.txt -I(Vin1)", and ".endc". Below the simulation code, the text "Corner: tt" is displayed. The status bar at the bottom shows "SNAP: 10", "GRID: 20", "MODE: spice", "Stretch: 0", "NUM LOCK SET", and "mouse = -120 170 - selected: 0 path: .".

xschem - tut1.sch

File Edit Options View Properties Layers Tools Symbol Highlight Simulation Netlist Simulate Waves Help

tut1.sch +

Vin

Vin1

1.8

GND

M1

nmosfet 01v8

GND

Vin

nf=1

s 1 x 0.42 / 0.15

GND

sim

.control

set filetype=ascii

dc Vin1 0 1.8 0.01

wrdata nmos_ids_vgs.txt -I(Vin1)

.endc

Corner: tt

SNAP: 10 GRID: 20 MODE: spice Stretch: 0 NUM LOCK SET mouse = -120 170 - selected: 0 path: .

Experiment 1 - xschem I_{DS} vs V_{GS}

- Click Netlist on the top right and then Simulate
- In the ngspice terminal, type: `plot -I(Vin1)`
- The simulated I_{DS} vs V_{GS} data will be saved in: `nmos_ids_vgs.txt`
View the contents of the file using: `gedit nmos_ids_vgs.txt`

