RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

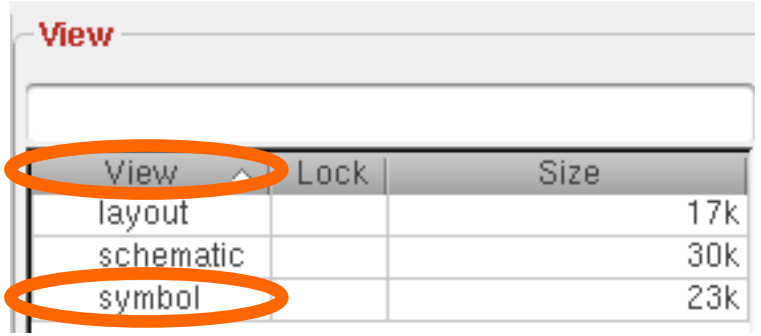
Schematics and Symbols

Prof. Dr. P. Fischer

Lehrstuhl für Schaltungstechnik und Simulation
Uni Heidelberg

What are Symbols ?

- Very often, a circuit (schematic) can be re-used.
- Instead of copying everything, we can 'include' the schematic into another schematic
- In order to identify the nets, we need a ***symbol***
  - This is a new ***view*** type

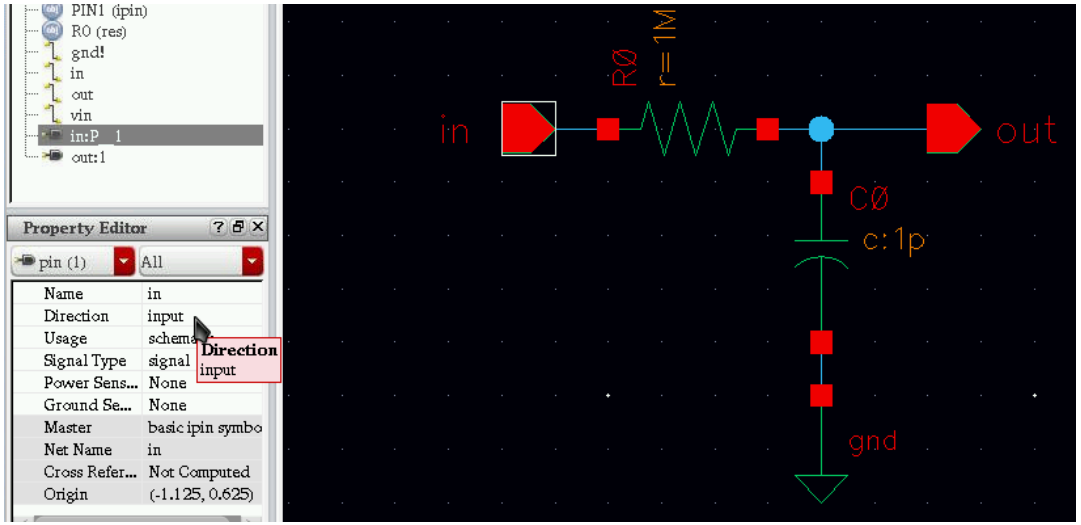


- The nets which are passed to the outside world must be connected to ***pins*** in the schematic.
  - For ***each pin in the schematic*** we also need *a* ***pin in the symbo****l*.

- Pins must have the ***same name*** as the connected net
- They can be ***Input / Output / inputOutput*** (see later)

Preparing the Schematic

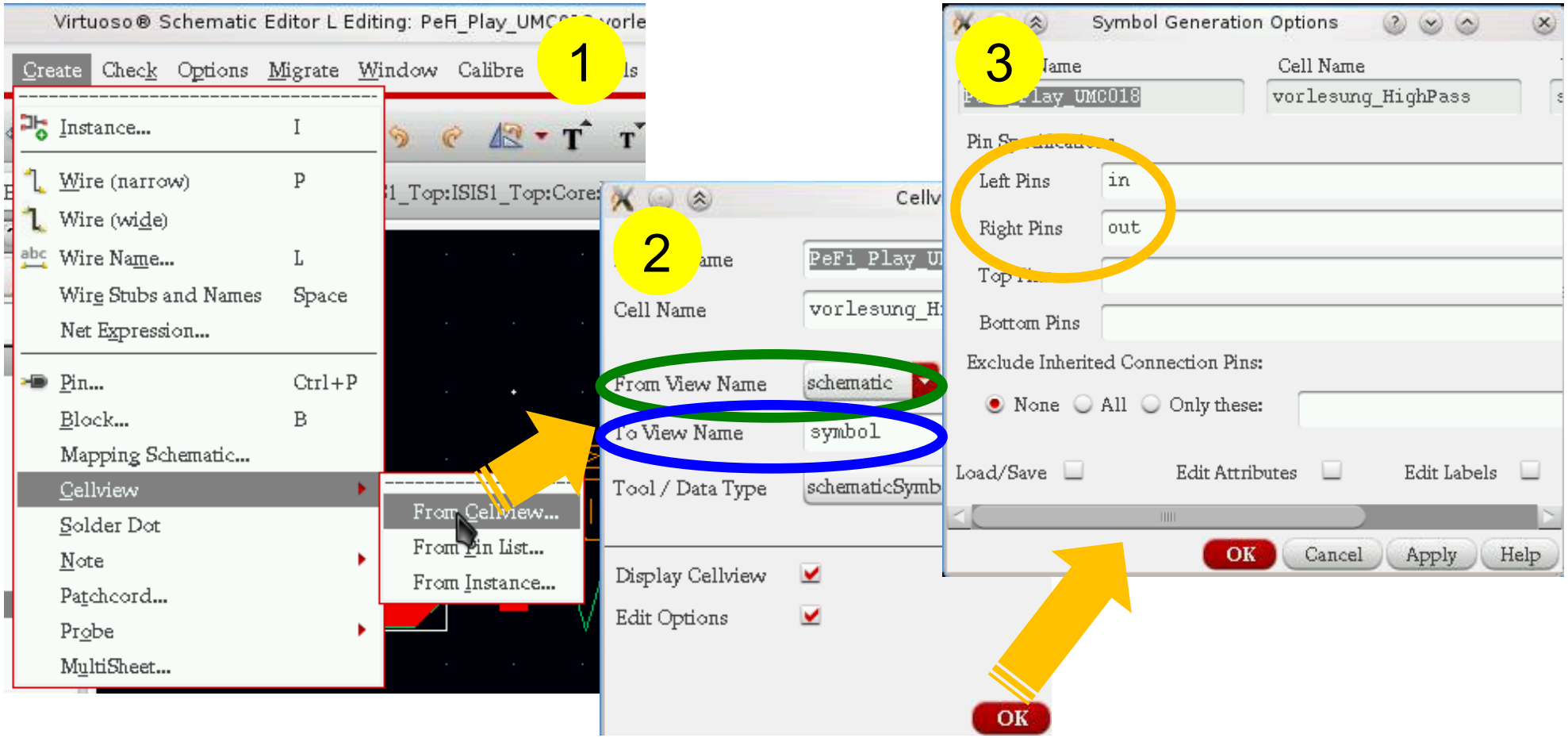
- The easiest way to create a symbol starts from a schematic
- Using Create → Pin (Ctrl-P or button ), create pins for all signals that should be visible 'outside'
  - **outputs** are signals that will drive to other cells
  - **inputs** only receive signals. They **must** be connected later
  - **InputOutput** are most general. Only use if you have to!



- A pin **labels** the net, i.e. a further label is not required
- Better remove all symbols used for simulation (sources..)

Creating a Symbol from the Schematic

1. Select Create → Cellview → From Cellview
2. Check that 'From View' is *schematic* and 'To view' is *symbol*
3. Press ok. In the next window, select the pin locations



Editing the Symbol

- A symbol template is created:

A pin. Here: Input

Name	in
Direction	input

This special label will later be replaced by the name of the master cell

This special label will later be replaced by the name of the INSTANCE

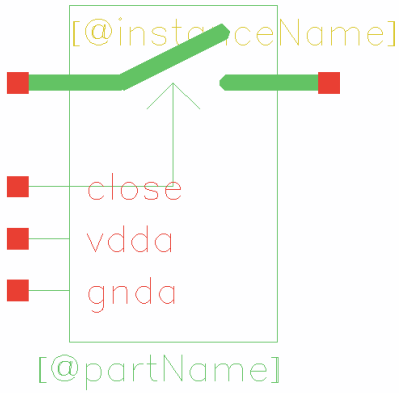
Green lines are only for optics, they have **no function**. You can add more such shapes in the Create → Shapes menu

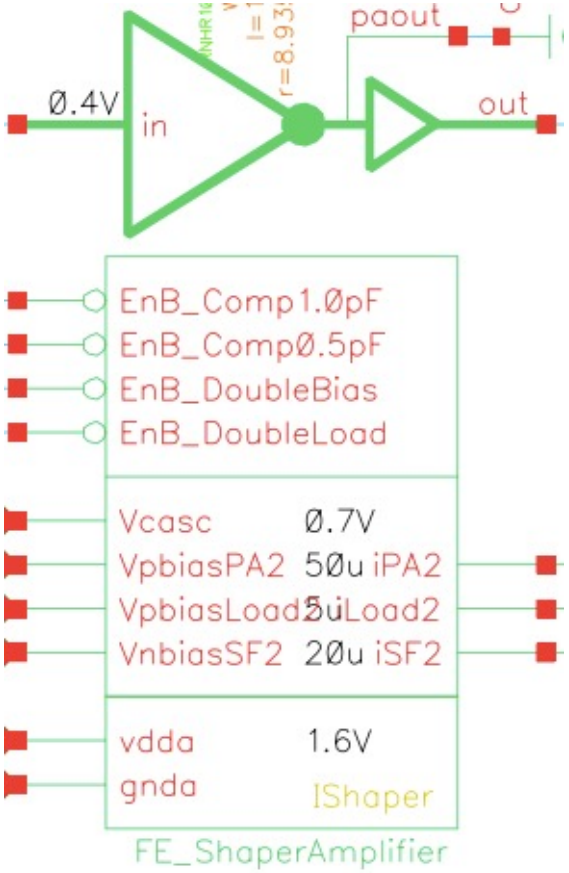
The red BOX is the '**selection box**'. It is the region where you can later select the symbol. (I prefer to keep it small…)

- You can set the origin under Edit → Origin

Make Nice Symbols!

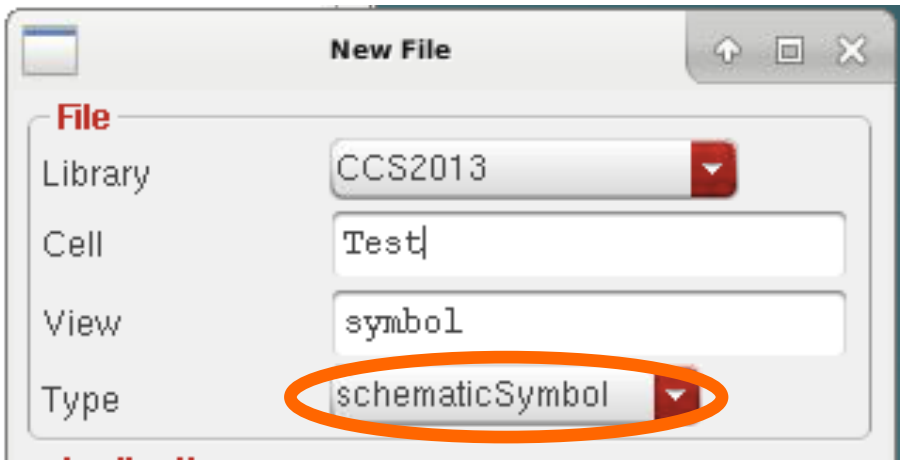
- Your schematics get more readable if the symbols are 'nice':
  - Power (if present at pins) may be grouped at the bottom
  - Group bias signals, use 'good' names
  - Inputs are left / outputs are right
  - Digital signals are grouped
  - Active Low signals have a bullet
  - Clocks have a triangle
  - Add a little drawing of the functionality
    Create→Note→Shape or Create→Shape
  - Add text: Create→Note→Text
  - You may delete trivial labels





Creating a Symbol from Scratch

- You can also create an (empty) new symbol directly from the library browser with File → New → Cell View… with view type ***schematicSymbol***

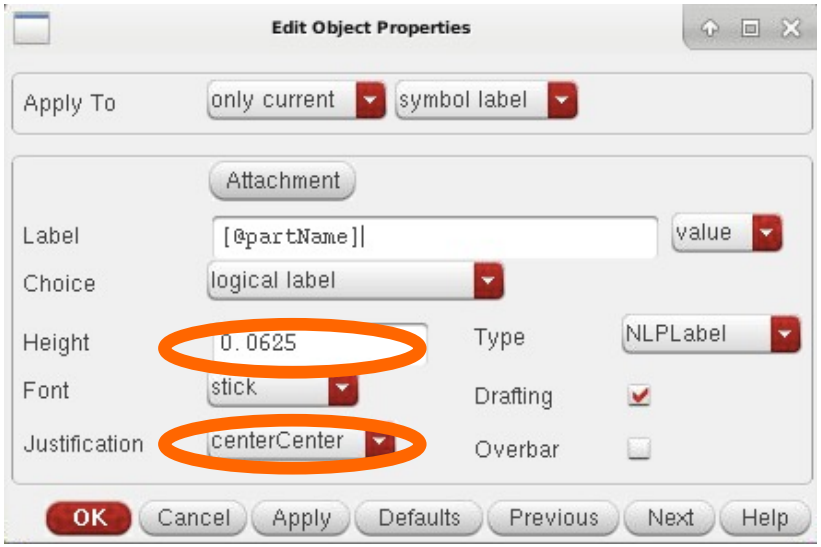


- You must then place all pins, boxes, labels, .. by hand.

Editing a Symbol

- When you (later) add new pins to the schematic, you **also** have to add them to the symbol.
  - Make sure **name** and **type** are the same!
  - Best copy other pins and rename them

- You can move, stretch, … as usual
- You can change the size or 'justification' of the labels

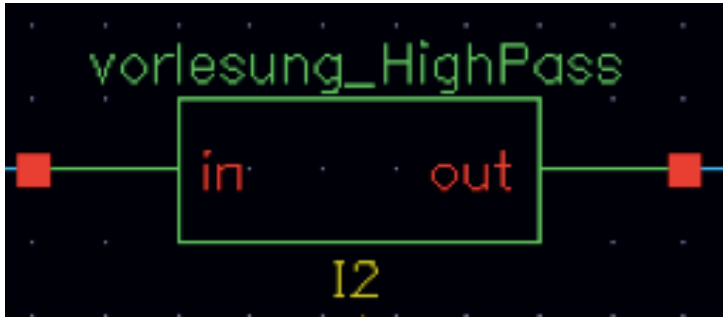


@instanceName and @partName

- Two special labels are created automatically:

do not confuse with @cellName

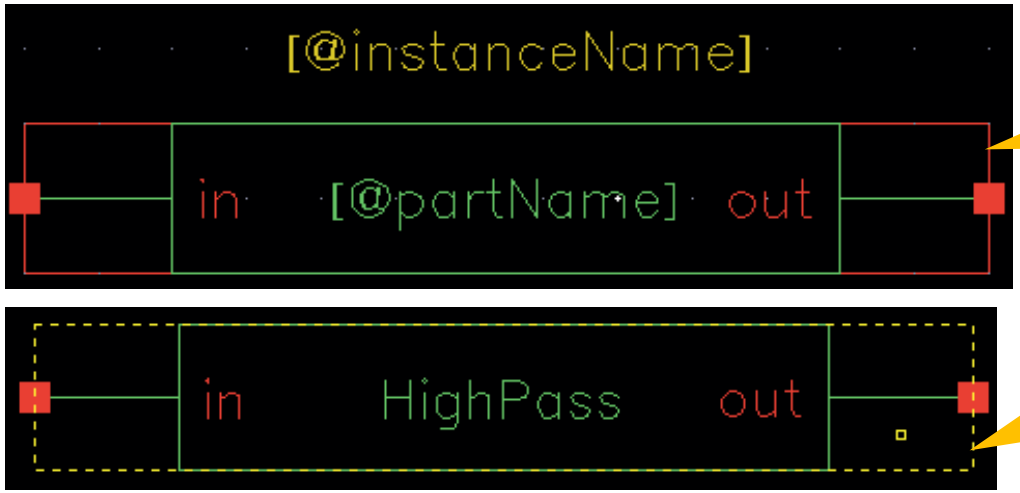
- [@instanceName] will display – surprise! – the name of the instance (of this symbol) that you place in another schematic, i.e. **I2** or, better, **lamp1** or so
- [@partName] displays the (library) name of the cell, i.e. **vorlesung_HighPass** or **NAND2**



- Place them somehow **nicely** (size / alignment / position)

The Selection Box

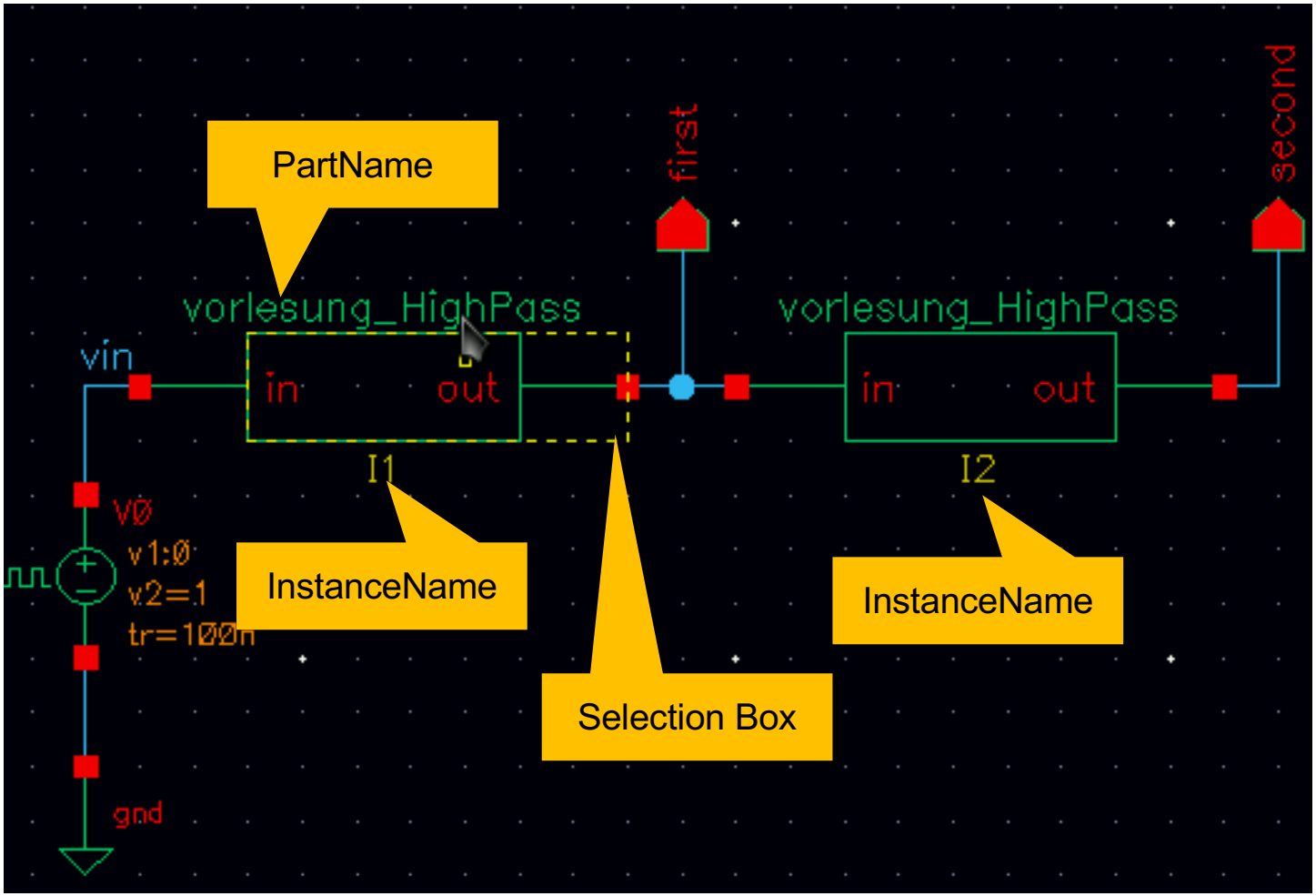
- When created automatically, a (red) *Selection Box* appears
- It marks the area which will be used to 'highlight' / 'select' the instance (in the next hierarchy level):



- The *Selection Box* can be moved / resized
- If lost (or in manually created cells), it can be created by Create → Selection Box
- You cannot route over the Selection Box → keep it small
- If no Selection Box is defined, the maximal symbol size is used.

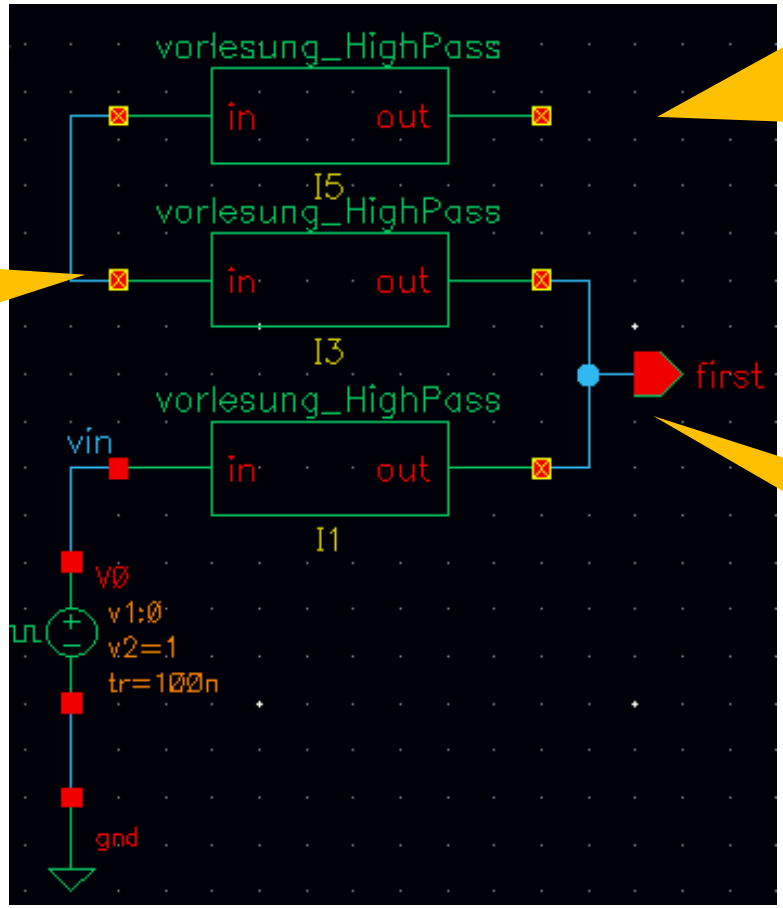
Using the symbol

- In a schematic, you can add your symbol now in the same way as any other instance



Inputs / Outputs / InputsOutputs

- After ‘Check & Save’, warnings may pop up in the CIW:



Unconnected output
→ Warning ‘floating output’

If this missing connection is intended, connect the ‘noConn’ symbol from library ‘basic’

Unconnected inputs
→ Warning ‘floating input’

Shorted output pins
→ Warning ‘shorted output’

- InputOutputs can be connected arbitrarily. Use with caution!
- All schematics should be 'clean', i.e. issue no warnings!

RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

TRAVELING THE HIERARCHY

Traveling in the Hierarchy

- Assume you are in Schematic A which contains an Instance of PartType B
- If you want to modify (the symbol or schematic of) B, you normally have to open that cell from the library browser
- You can better *'dive into'* B by
  - Selecting the instance
  - Edit → Hierarchy → Descend Edit (Shift-X)
  - Select the view
  - Select if you want a new window / new tab / use existing tab

- You then end up in symbol / schematic of B
- When done, return back 'up' with Edit → Hierarchy → Return (Shift-B)
- You can also Descend for Read Only (Ctrl-X) or Edit in Place (x). This Edits B but shows A ! **Powerful** but **dangerous**!

RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

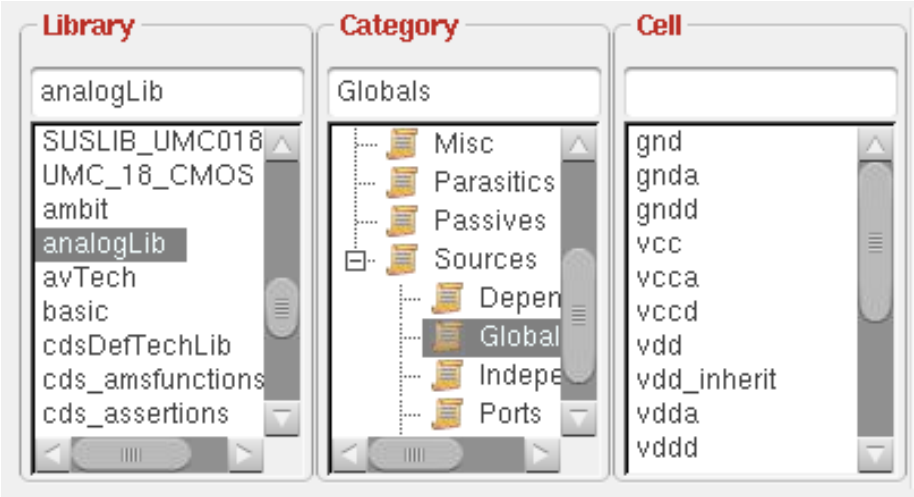
GLOBAL NETS

Global Nets

- A net is normally only known in the corresponding schematic
  - Connecting nets *between* schematics requires *pins*
- This can be tedious for signals which are used very often
  - analogue / digital power / ground
  - substrate potential
- You can use **global** nets, known **everywhere**
  - They are identified by an **exclamation** mark**:** xxx**!**
- Common global nets are
  - gnd! or sub! chip substrate
  - gndd! and vddd! digital ground /supply
  - gnda! and vdda! analogue ground / supply
- Handle them with care, because it is hard to track where they are used...

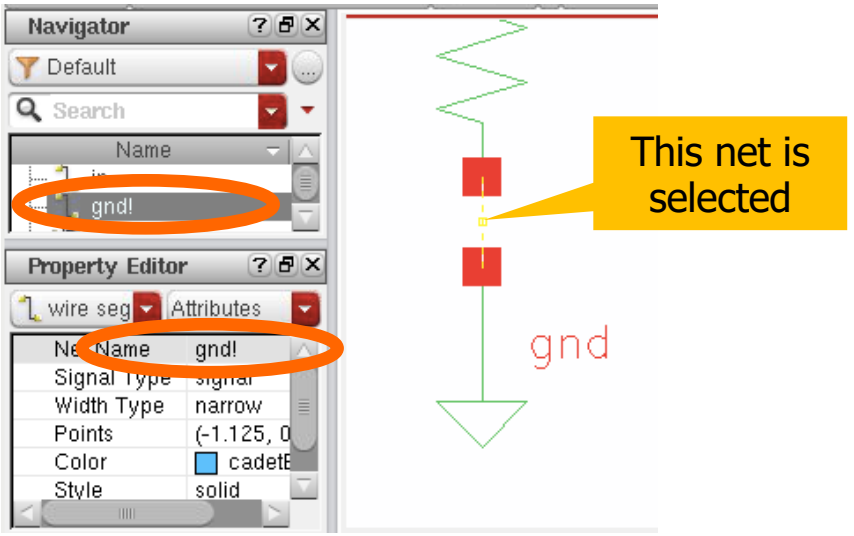
Global Nets

- There are several global 'symbols' in analogLib
  - Under Sources → Global



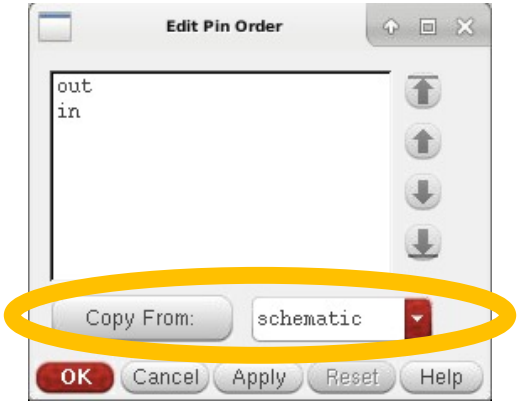
- They connect a net automatically to the corresponding global net

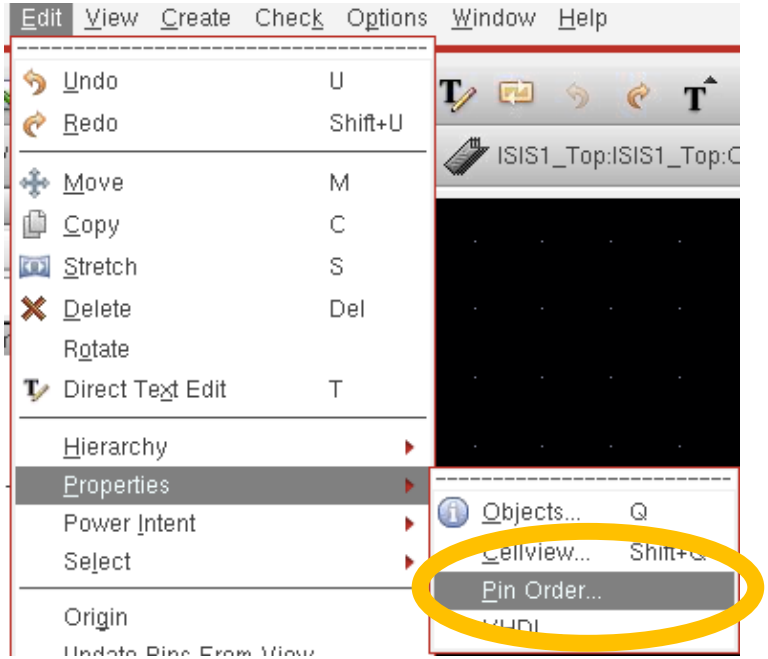
- Therefore:
  Connecting to symbol '**gnd**' is **the same** as labelling a net with '**gnd!**'



Pin Order

- If can happen that the internal order of pins gets messed up
  - You get a warning at Check & Save
  - This happens if you copy pins from other cells, delete pins,..
- To restore correct order, use Edit → Properties → Pin Order
- Best copy the Pin Order from another view:





- In rare cases, you have to regenerate (for instance) the symbol. (There is a step which allows you to just 'repair' the wrong stuff so that you nice drawing is not affected)

RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

BUSSES AND ADVANCED NET NAMING

Advanced Net Names **(Important!)**

- A single 'wire' on the schematic can represent **several** nets, i.e. it can be a 'bus' or bundle of nets.

When a wire has multiple nets assigned:
Imagine the nets **stacked onto each other in the order they are listed**

- Examples:

• Simple wire	in	
• Multiple wires	a,b	separated by **comma**
• Bus	d⟨4:0⟩	5 signals: d⟨4⟩,…,d⟨0⟩
• Bus	x⟨1:5⟩	different index order: x⟨1⟩,…,x⟨5⟩
• Repetition	⟨*3⟩a,⟨*2⟩b	this is equivalent to a,a,a,b,b
• Skip indices	d⟨7:3:**2**⟩	= d⟨7⟩,d⟨5⟩,d⟨3⟩
• Index list	d⟨1:0,3,⟨*2⟩5⟩	= d⟨1⟩,d⟨0⟩,d⟨3⟩,d⟨5⟩,d⟨5⟩
• Grouping	⟨*2⟩(a,b)	= a,b,a,b

- This works for labels and for pins (but better only use busses!)

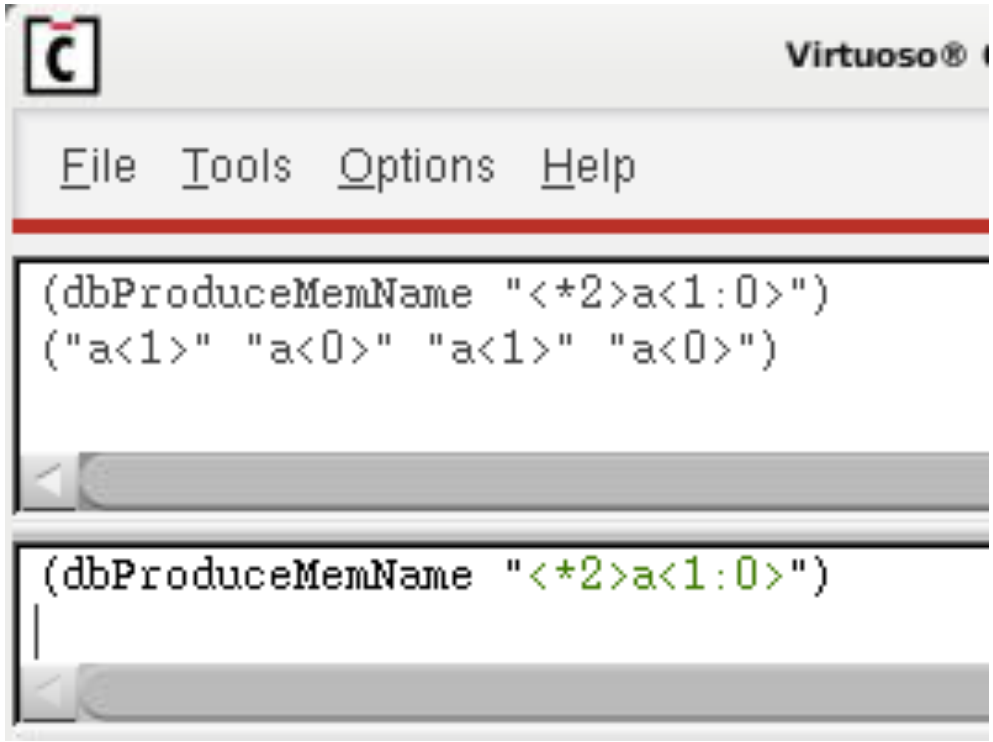
More Complicated Examples

- You often need a 'binary' encoding of signals:
  - ⟨*4⟩(a,b) =
    ⟨*4⟩(⟨*1⟩a, ⟨*1⟩b) = a,b,a,b,a,b,a,b
  - ⟨*2⟩(⟨*2⟩a, ⟨*2⟩b) = a,a,b,b,a,a,b,b
  - ⟨*1⟩(⟨*4⟩a, ⟨*4⟩b) = a,a,a,a,b,b,b,b

Advanced Net Names

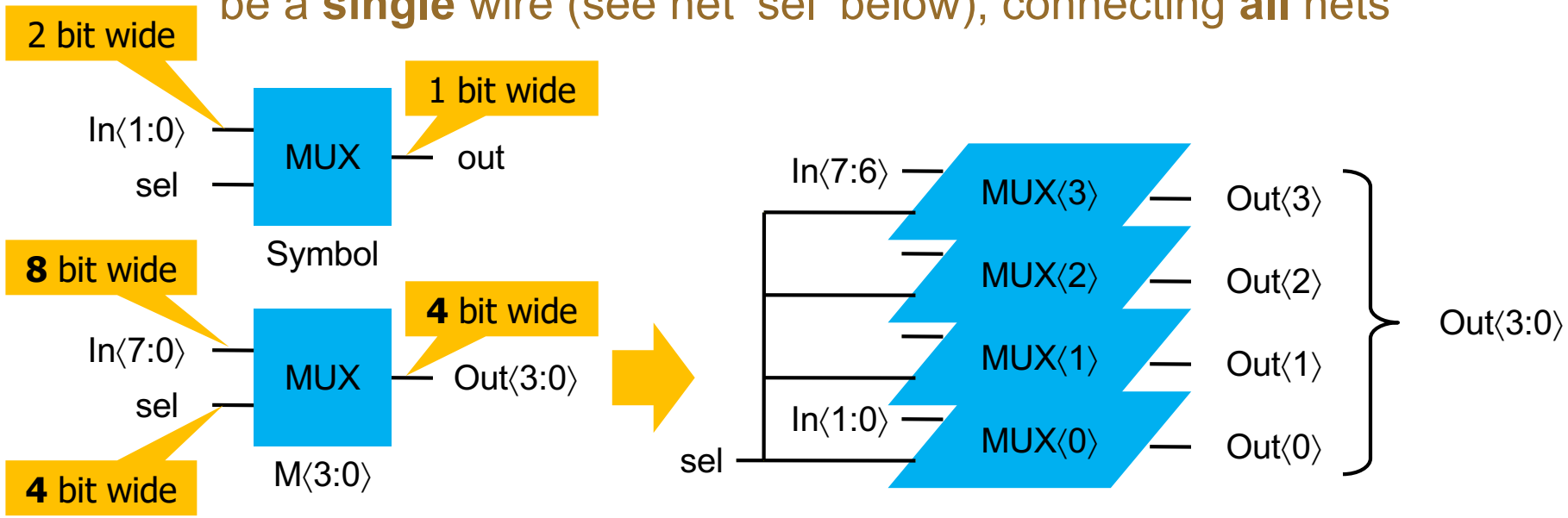
- If you are not certain how a complicated net name expands: Type the expression in the CIW (Command Interpreter Window) using

  (dbProduceMemName "expression")



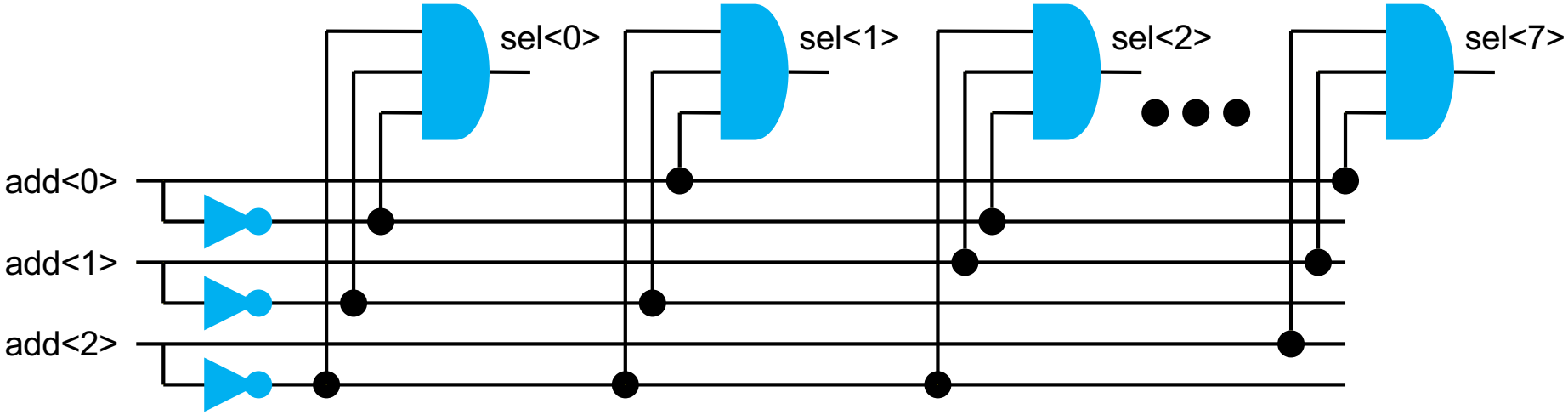
Multiple Symbols (**Very Important!**)

- **Instances** can be indexed as well:
  - An **instance** with name M⟨3:0⟩ contains 4 elements M⟨3⟩…M⟨0⟩
  - They are (again) lying 'on top of each other' (in the order given)
- The instance **pins** are also stacked on top of each other
  - A single pin of N instances becomes a bus which is N nets wide
  - A pin with 2 nets (in⟨1:0⟩) becomes 2N nets wide etc.
  - Connected nets must be have **exact** length OR be a **single** wire (see net 'sel' below), connecting **all** nets

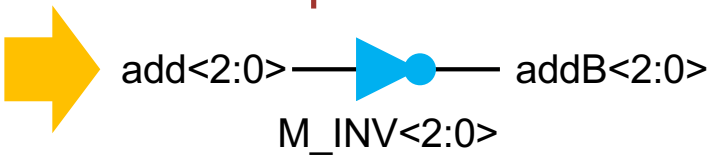


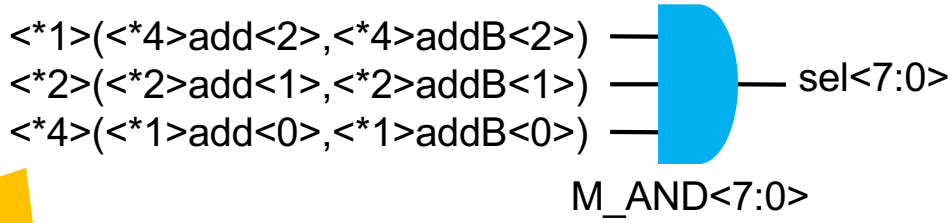
Multiple Symbols: 2nd example

- Here is a 3 Bit address decoder which activates one of 8 output signals sel⟨7:0⟩ as a function of 3 address inputs add⟨2:0⟩:



- Compact:



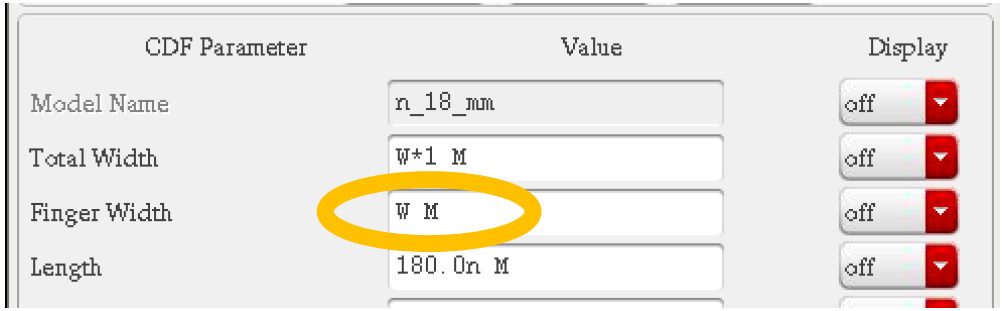


This is: add<0>,addB<0>,add<0>,addB<0>,add<0>,addB<0>,add<0>,addB<0>

COMPONENTS WITH (SYMBOLIC) PARAMETERS = DESIGN VARIABLES

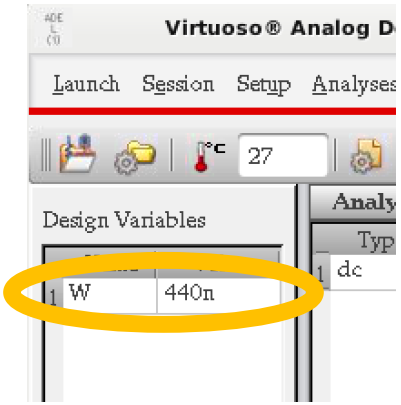
Using **Design Variables**

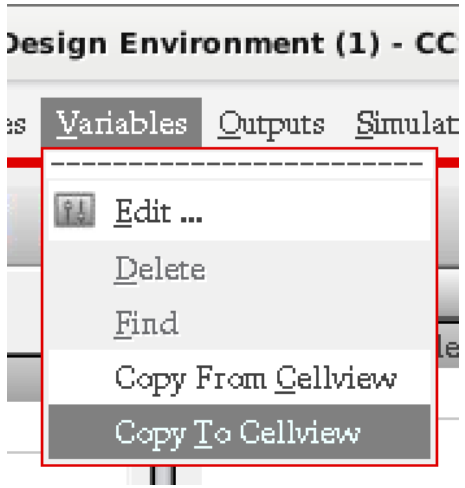
- The parameters of components can be assigned a text value. This shows up in the schematic netlist.




```
// Library name: CCS2017
// Cell name: VLSI_myinv
// View name: schematic
M2 (net12 ctrl 0 0) n_18_mm w=W l=180.0n nf=1 mis_flag=1 \
        ad=((1.4e-07+400.0n)*W*(1-1)/2)+((9e-08+400.0n)*W)/1 \
        as=((1.4e-07+400.0n)*W*(1-1)/2)+((9e-08+400.0n)*W)/1 \
        pd=(2*((1.4e-07+400.0n)+W)*(1-1)/2+2*((9e-08+400.0n)+W))/1 \
        ps=(2*((1.4e-07+400.0n)+W)*(1-1)/2+2*((9e-08+400.0n)+W))/1 \
        m=(1)*(1) mf=(1)*(1)
M0 (out in net12 0) n_18_mm w=440.0n l=180.0n nf=1 mis_flag=1 ad=2.156e-13 \
        as=2.156e-13 pd=1.86u ps=1.86u m=(1)*(1) mf=(1)*(1)
M1 (out in vdd! vdd!) p_18_mm w=440.0n l=180.0n nf=1 mis_flag=1 \
        ad=2.156e-13 as=2.156e-13 pd=1.86u ps=1.86u m=(1)*(1) mf=(1)*(1)
```

- The design variable can then be varied in simulation



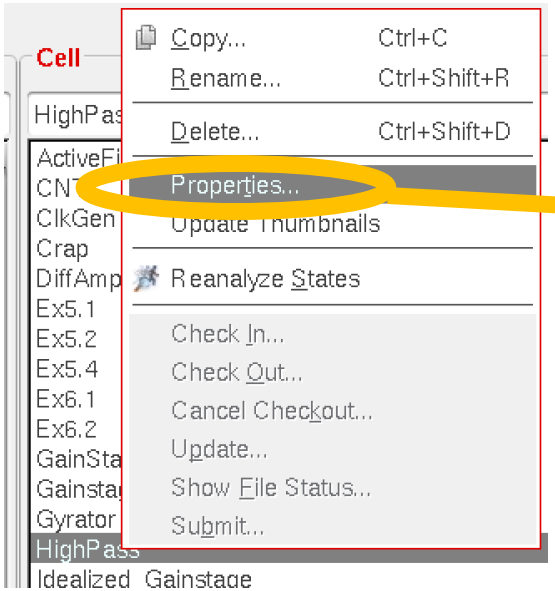


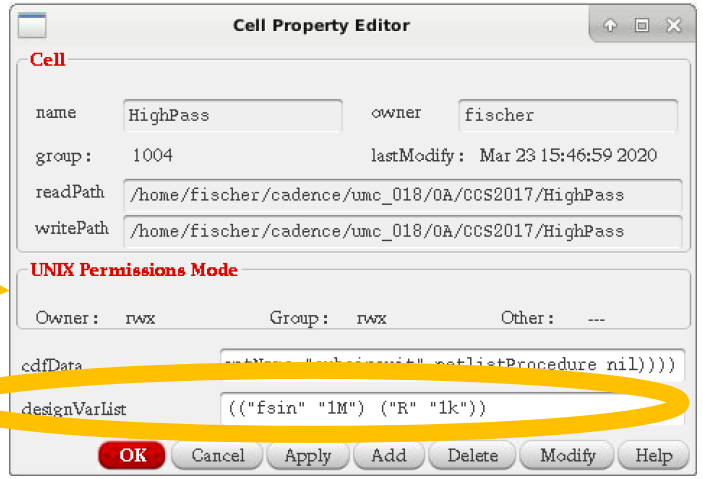
- To preserve the value, you should **write back** the design variable **to the schematic** from the simulation window:
- This is required for later LVS!!!!

```
// Design cell name: VLSI_myinv
// Design view name: schematic
simulator lang=spectre
parameters W=440n
```

Design Variables

- When written back, the design variables are stored in the **cell** (**not** just in the schematic **cell view**) (thanks @ Michael!)
- They can be seen and changed from the library browser in the **cell properties**:






```
(cadr (geGetEditCellView)~>cell~>prop)~>??
(db:0x2619731b cellView nil objType "prop"
                          nil name
    "designVarList"       ject db:0x2619741a range nil
                          
    (("fsin" "1M")
            ("R" "1k")
    ) valueType "ILList" assocTextDisplays
    nil
)
```

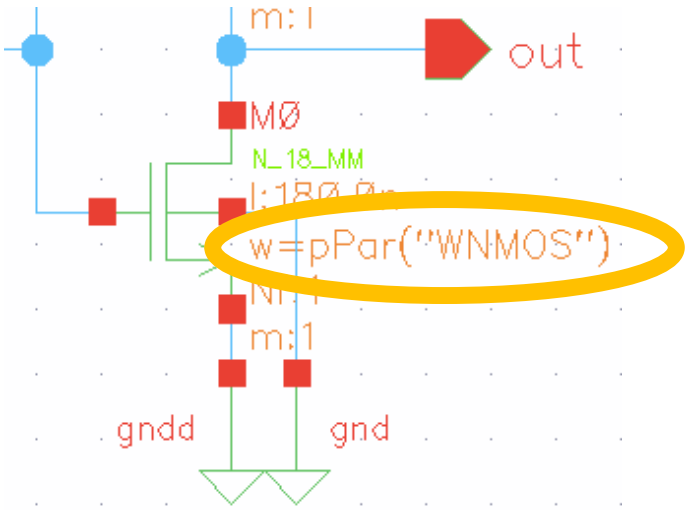
- (Later: Using Skill, search for property 'designVarList' from a schematic: (geGetEditCellView)~>cell~>prop)

SYMBOLS WITH PARAMETERS

Parameterized Symbols (Step 1/3)

- It occurs that you need very similar schematics where only few parameters are changed (often transistor sizes)
  - Example: Inverter with different PMOS widths
  - (Unfortunately, parameters cannot be used everywhere…)
- Instead of creating multiple cells, you can create one cell with a PARAMETER:

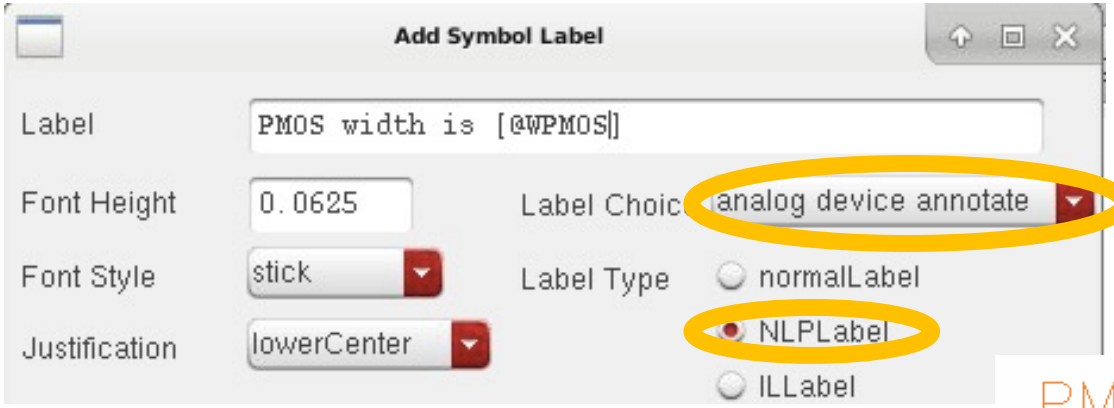
1. In the schematic:
   introduce the parameter with **pPar("pname")** (capital P!)

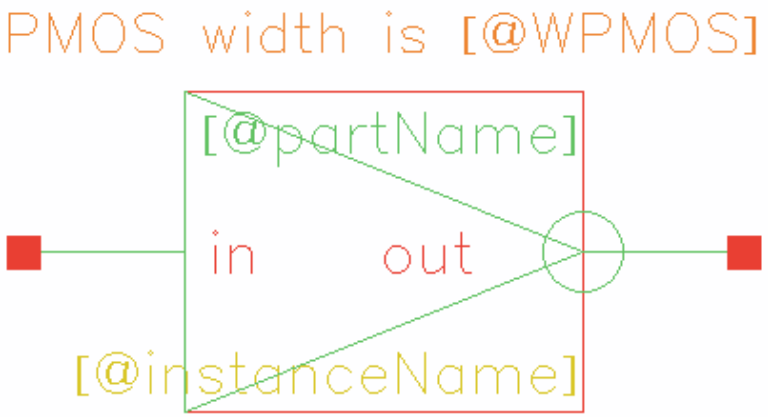


Parameterized Symbols (Step 2/3) – Optional!

2. In the symbol: Add a label
   - Label Choice: *analog device annotate*
   - Label Type: *NLPLabel*

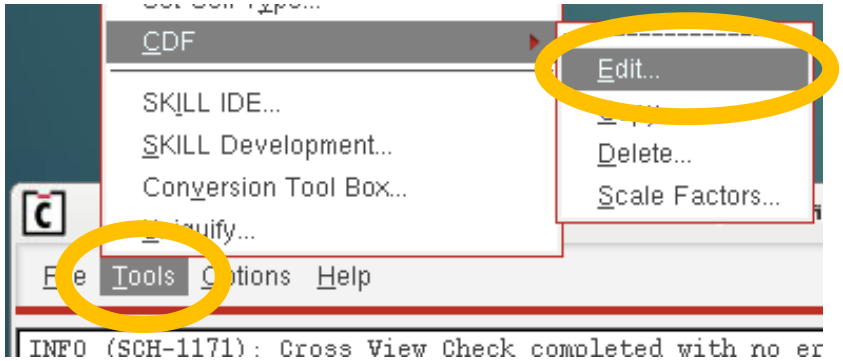
   Add any text, referring to the parameter as [@pname]



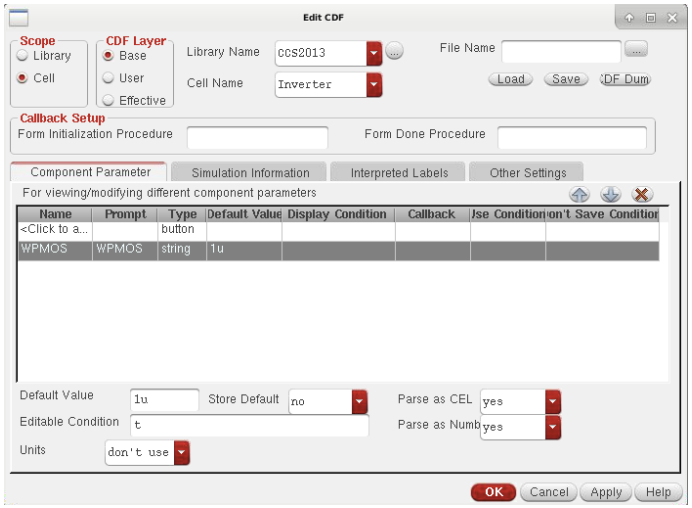


Parameterized Symbols (Step 3/3)

3. Cadence still needs to know about the new parameter:
   - In CIW→Tools→CDF→Edit
   - Choose Scope: Cell
   - Choose CDFLayer: Base
   - Select Cell

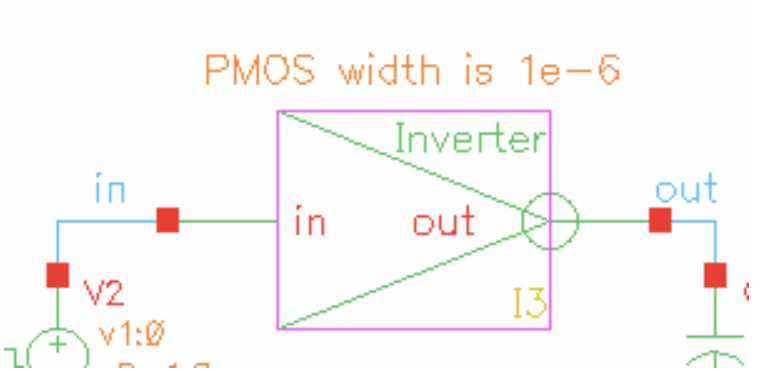


   - Add your *pname* in the form
     - Type: String
     - Set prompt string & default value
     - Store Default: no (=default)
     - Parse as CEL: yes
     - Parse as Number: yes
     - Editable Condition: t (needed ?)
     - Units: don't use (=default)

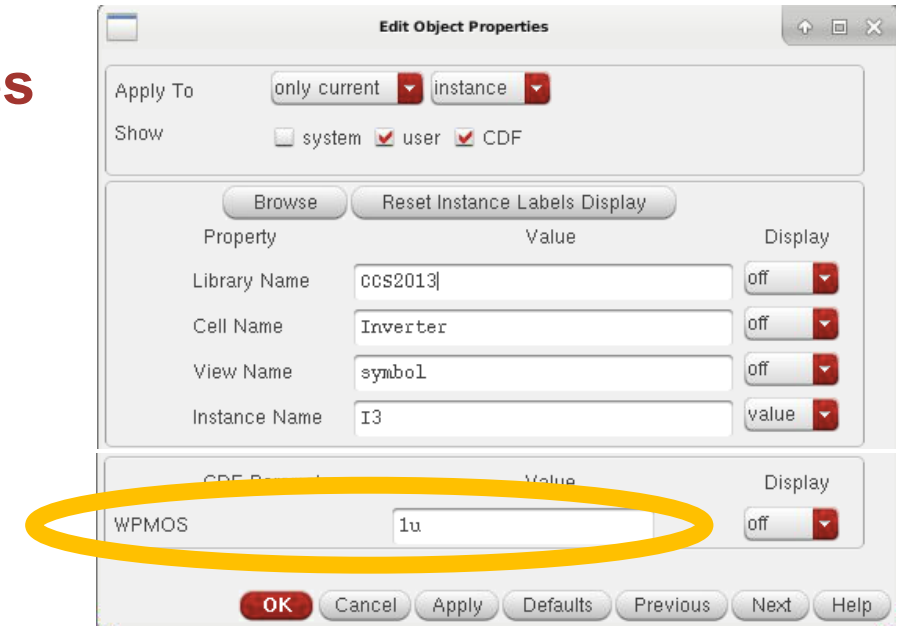


Parameterized Symbols: Instantiation

- The Symbol now shows your text + value
  - You may need to delete and re-instantiate existing symbols..



- You can now change the parameter in the instance **properties**



RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

INHERITED NETS

Inherited Nets

- It is possible to over-write nets in schematics (mostly supplies) from a higher hierarchy level.
- This 'inherited nets' approach is not further described here…

RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

FEATURES OF THE SCHEMATIC EDITOR

Some Features

- The additional display of net names at the pins of transistors is often confusing.
  - You can turn this off under View → Hide Terminal Labels
- If you want to see all places where a net in one schematic connects:
  - Enable Highlighting under View → Net Highlighting
- If you want to follow a signal through the hierarchy:
  - Highlight the net under Create → Probe → Add Net