RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

Exercise:
Getting Started with Cadence

Prof. Dr. P. Fischer

Lehrstuhl für Schaltungstechnik und Simulation
Uni Heidelberg

Logging in...

- Log in on the CIP Pool with your university account
  - Systemanwendungen → MATE terminal
- Start a remote session with the provided user name:
  - `ssh -X name@susnx.ziti.uni-heidelberg.de`

  (accept the RSA fingerprint once with `yes`)
- Change the password with command `passwd`
  - ≥ 8 characters, 3 from 4 types (normal, capital, number, special)

Preparing Cadence...

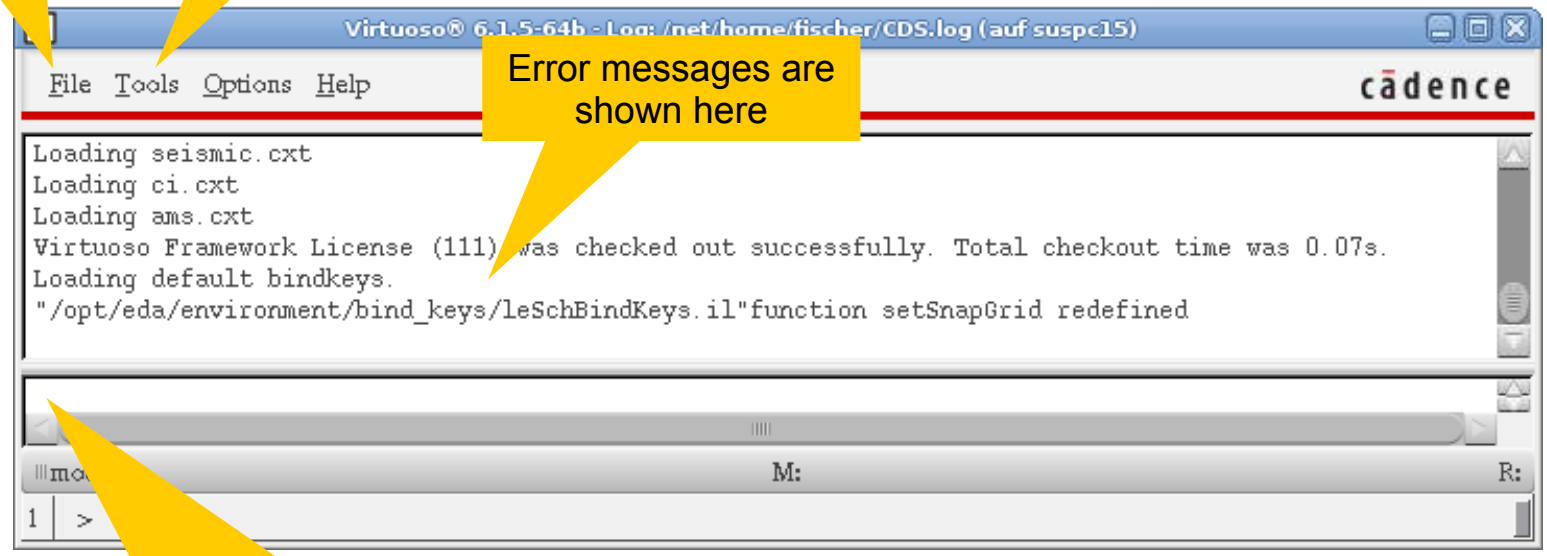
- If you start for the first time:
- Create a new directory '`cadence`' in your home directory
  `mkdir -p ~cadence`
- Copy the files required to run cadence from our 'template' directory into a (newly created) subdirectory `umc`
  `cp -r /shares/designs/UMC/OA/018_1P6M/workdir_template/ ~/cadence/umc`
  (note the 'blank' before '~/cadence')
- Change to the cadence directory and have a look…
  - `cd cadence/umc`
  - `ls -al`
- Your working directory now contains (mainly) a start script `start.sh`, a configuration file `.cdsinit`, a file with library paths `cds.lib`

Starting Cadence

- Start cadence with
  - `./start.sh &`

The usual file menu (with 'exit')

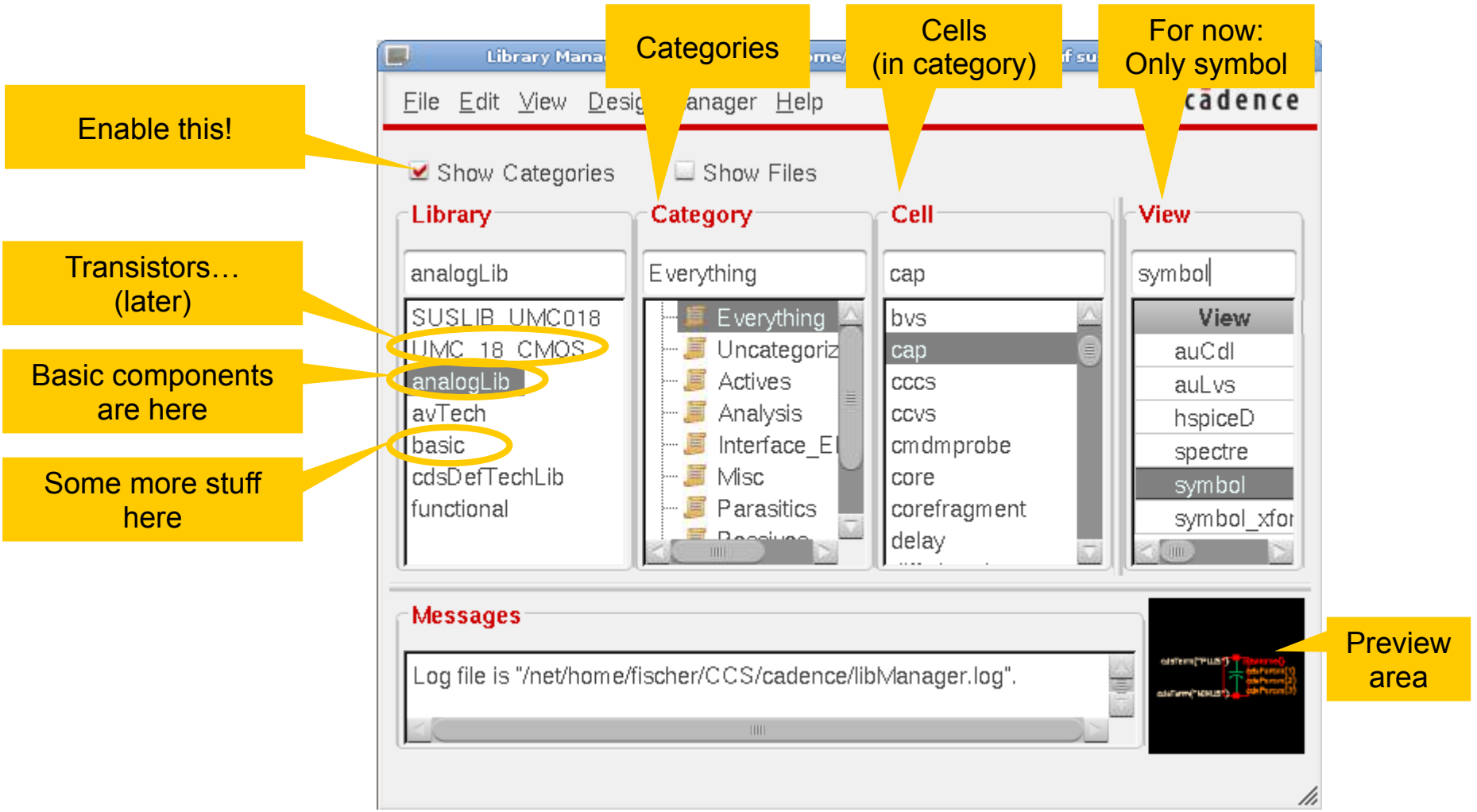
Start more stuff from here



Can type in commands /programs here.
The language 'skill' is very close to LISP
Try **`(plus 3 4)`**
or **`(sqrt 10)`**

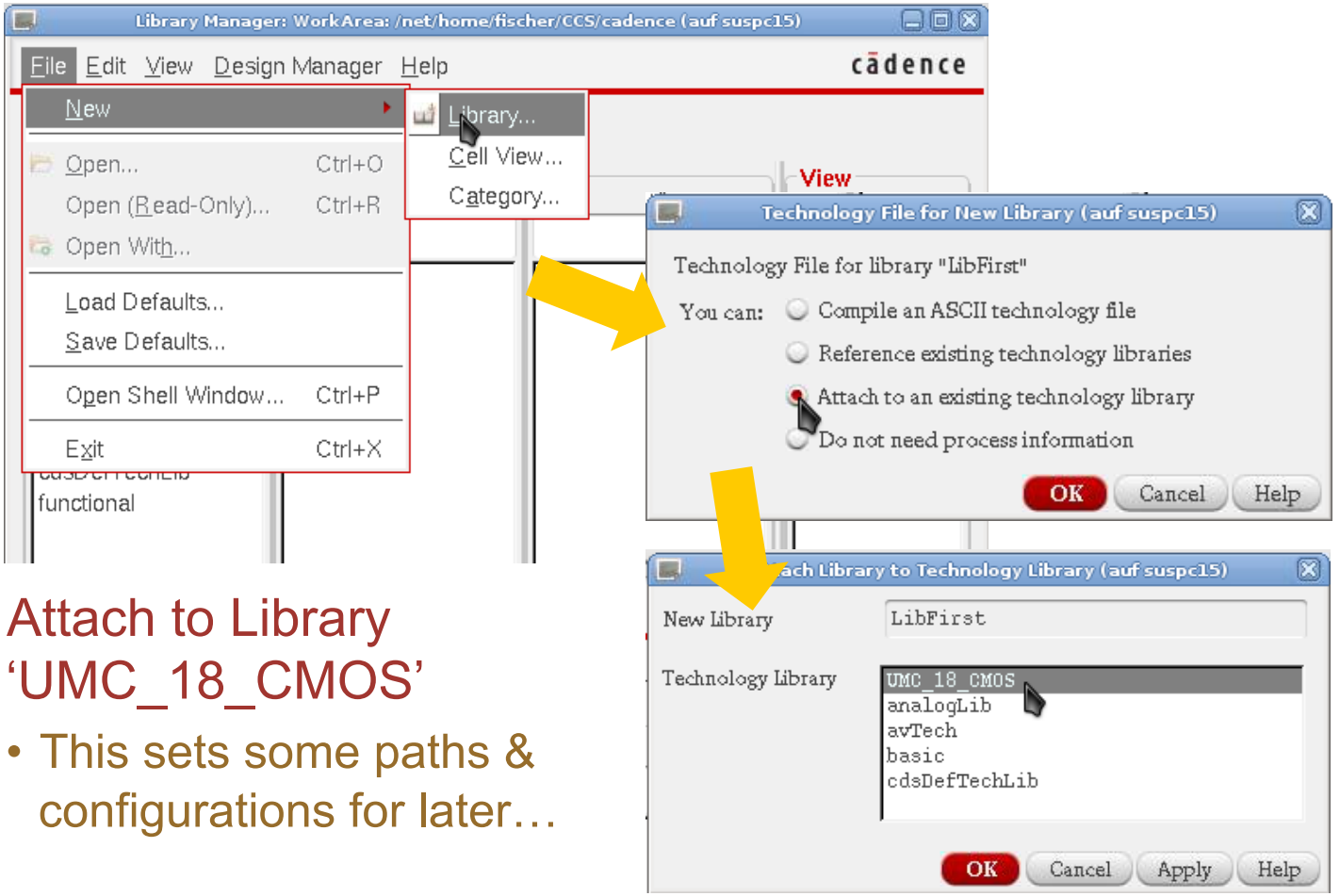
Opening the Library Manager

- Open the library browser under Tools → Library Manager…



Creating a library

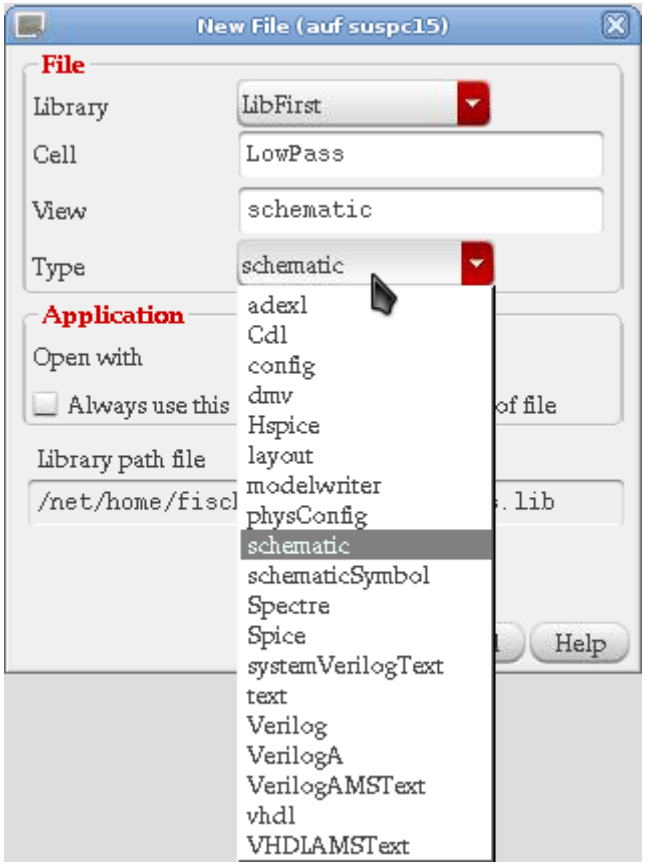
- Create an empty library from the Library Manager under File → New → Library



- Attach to Library 'UMC_18_CMOS'
  - This sets some paths & configurations for later…

Creating a new Schematic

- Select your library
- Create a new schematic with File → New → Cell View…
- Select type ‘schematic’ by selecting from the drop down list
- Give the cell a name
- The schematic editor opens
- Save the cell!

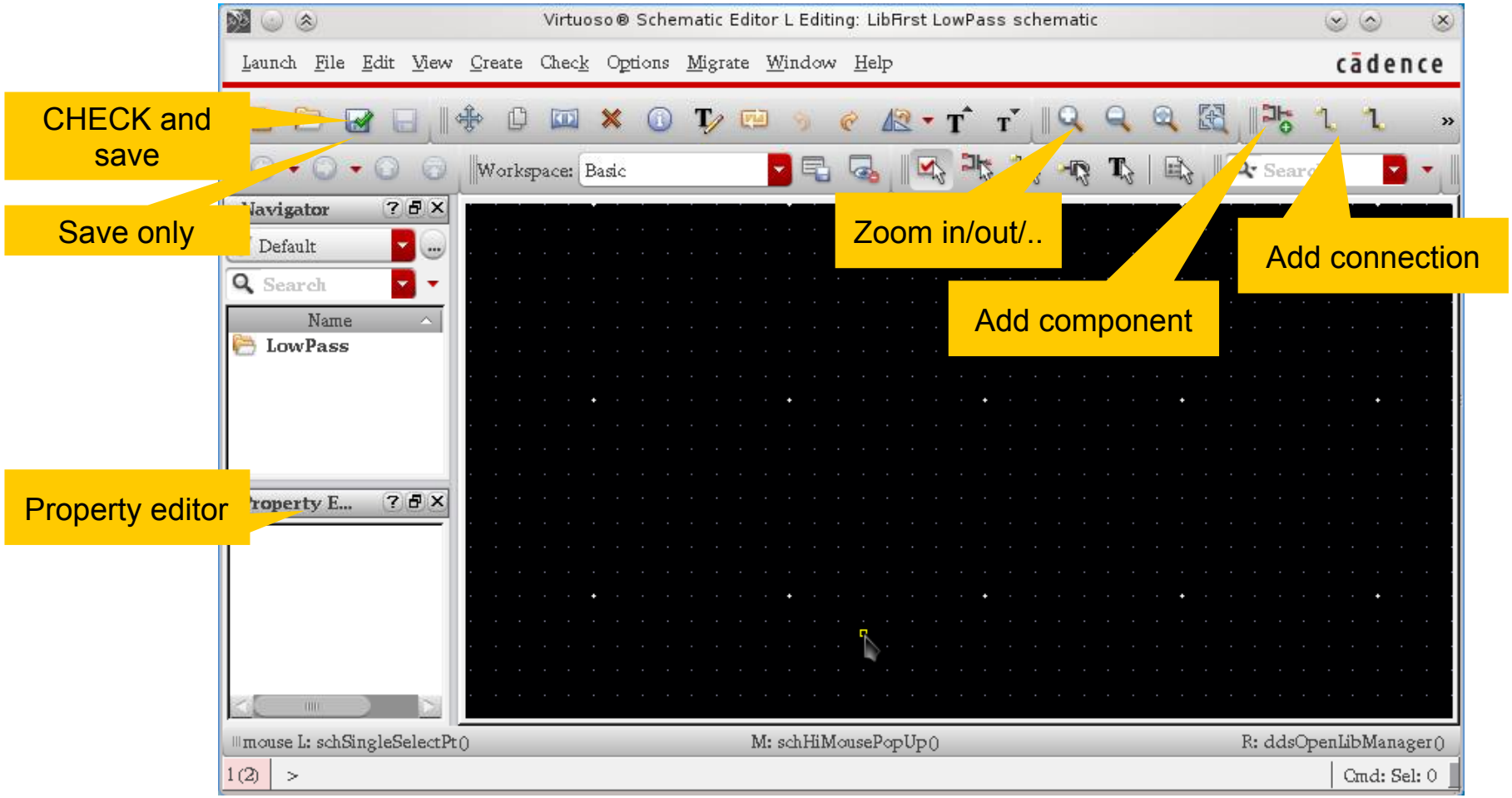


- Check that the cell is now in your library
- If you select the cell, you should see the view ‘schematic’

- (You can create cell categories to sort your stuff with File → New → Category)

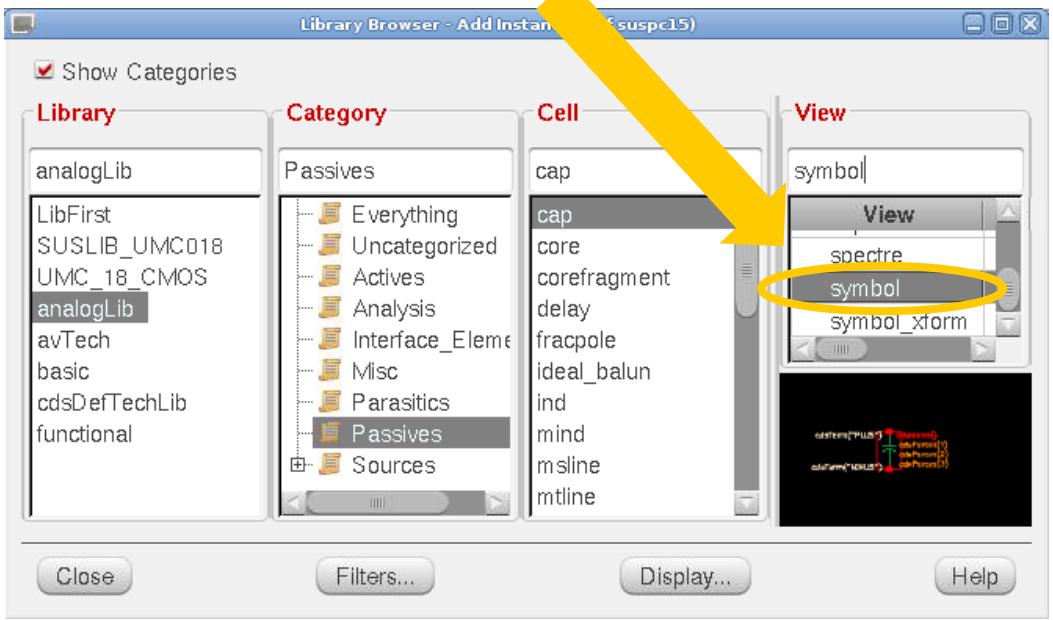
Opening the Schematic

- Double click on the 'schematic' entry (or right click & open)
  - The schematic editor of 'virtuoso' comes up:



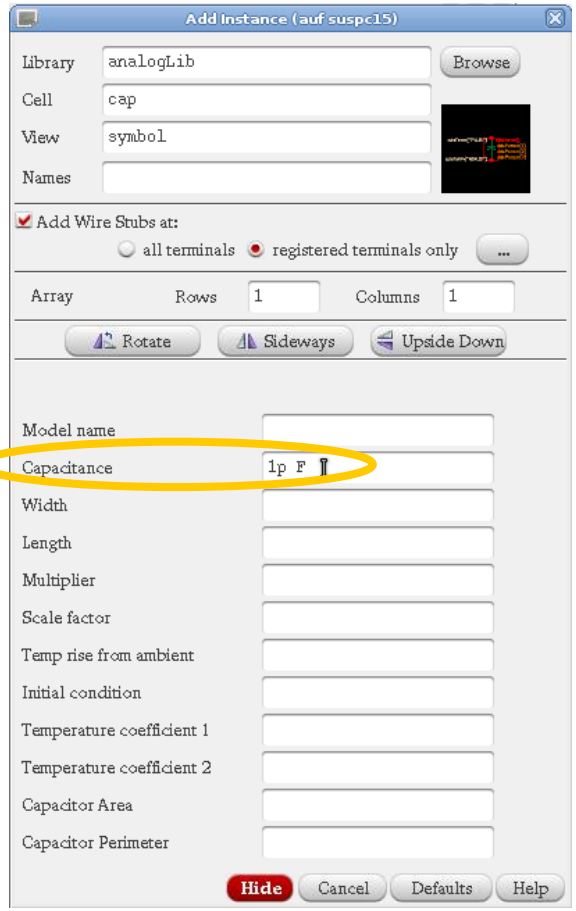
Adding a component (1)

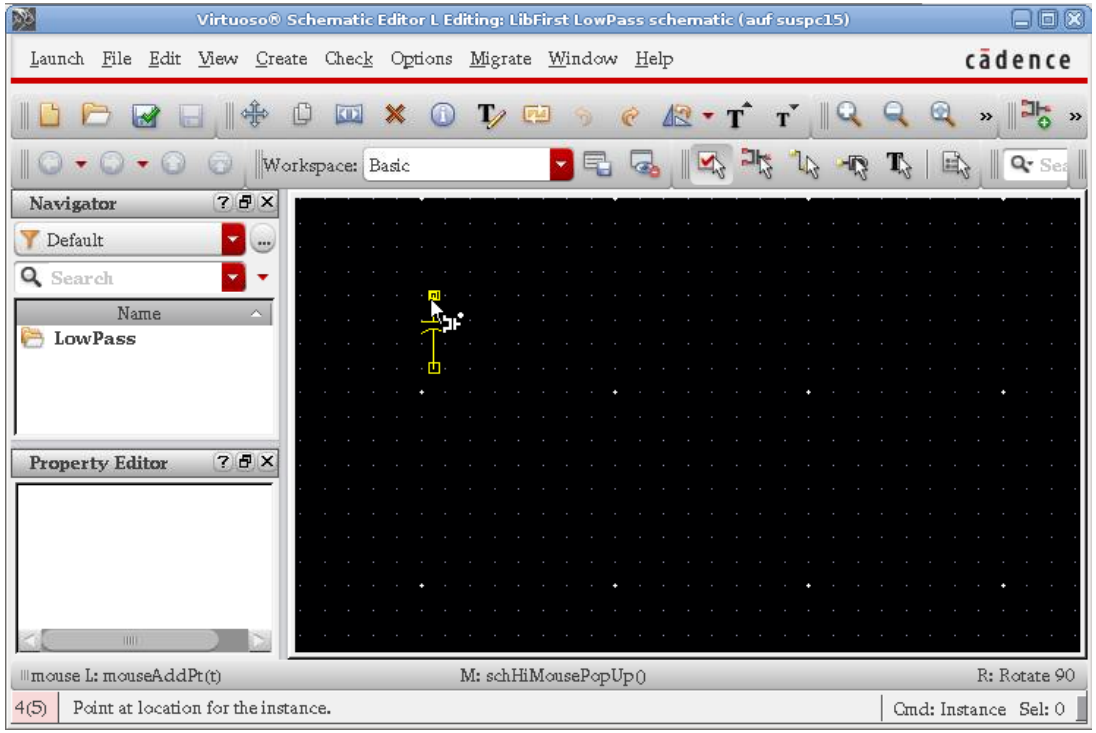
- To add a component ('instance')
  - Press the 'Create Instance' button  or
  - select Create → Instance or
  - press 'i'
- **Browse** to the correct library (for now: analogLib)
- Choose a cell from the library browser.
  - Make sure View 'symbol' is selected!



Adding a component (2)

- Set the parameters (values) of the instance
  - For instance the resistance of a resistor
- Place the instance on the sheet (mouse click)





Press ESCAPE to finish.

Changing Parameters

- To change an instance
  - Press the 'Edit Properties' button  or
  - select Create → Properties → Object or
  - press 'q' or
  - use the Property Editor Panel

- For values, use the suffixes
  - **m** for milli = 10^{-3}
  - u,n,p,f,a for micro = 10^{-6}, nano, pico,…
  - k for Kilo = 10^{3}
  - **M** for Mega = 10^{6} (**m** ↔ **M**!)
  - G for Giga = 10^{9}

- There is no need to supply the unit!
  - It is clear from the context

Executing Commands

- Two possibilities for most commands:

- Execute command once:
  - Select objects(s)
  - Press command key
  - Execute command (once)

- Multiple execution:
  - Press command key → switch to command mode (new cursor)
  - Select objects to execute commands on them
  - Press ESC = escape to end

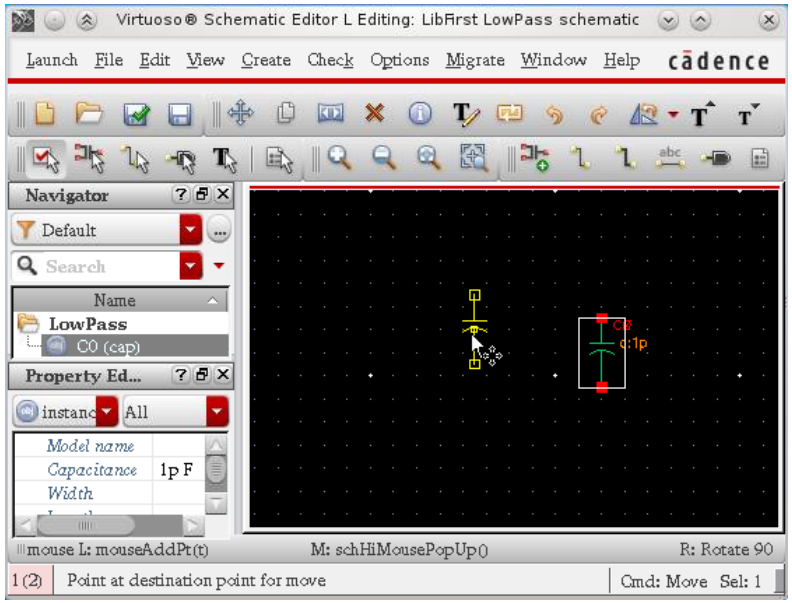
- Example:
  - Select – delete — delete one instance
  - Delete – click – click … - click – escape — delete multiple

Getting more command options

- In general, pressing F3 while executing a command opens a window with more options.
  - rotate, flip
  - allowed routing angles
  - colors
  - …
- Sometimes need to press F3 twice

Moving an Instance

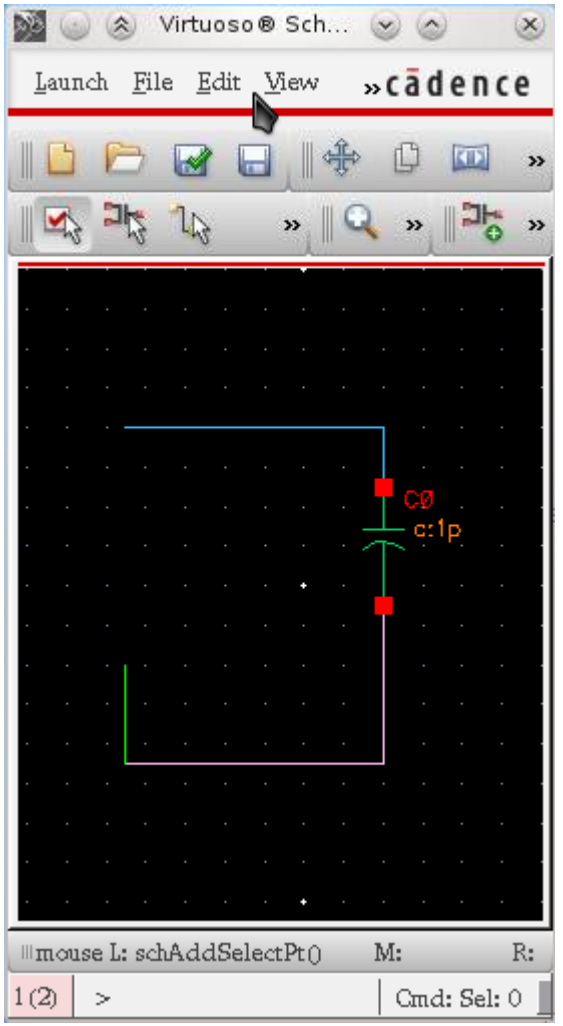
- Select the instance with the mouse
  - leftclick — to select individual instances
  - shift – leftclick — to add instances to selection
  - ctrl – leftclick — to remove instances from selection
  - drag rectangle — select instances in area
- To move
  - Press ‘Move’ button  or
  - select Edit → Move or
  - press ‘m’
- Alternative:
  - First press ‘m’
  - select – move – drop, …ESC
- Alternative:
  - click – drag – drop



- For options (rotate, flip,..): F3 or right mouse

Adding Wires

- Wires connect the pins of instances
- To add a (narrow) wire (‘path’)
  - Select the  button
  - select Create → Wire or
  - press ‘p’
- to change to ‘path mode’
- Connect pins by multiple mouse clicks
- Finish with ESC



- Changing behavior: press F3
  - change angle
  - change color
  - …

Zooming …

- show everything: ‘f’ (fit)
- scroll: arrow keys
- zoom in: ctrl-z or]
- zoom out: shift-z or [
- zoom area: right mouse – drag
- pan selection: tab

- See menu View→ …

Adding net Names

- To identify nets, you can
  - assign names (labels) to nets or
  - connect them to pins.

Nets with the same names (labels) are automatically **connected**!

- To assign a label:
  - Press the  button or
  - Select Create → Wire Name or
  - Press 'l' (label)
- Type in the label name and click on the net
- Continue with further labels
- End with ESC.

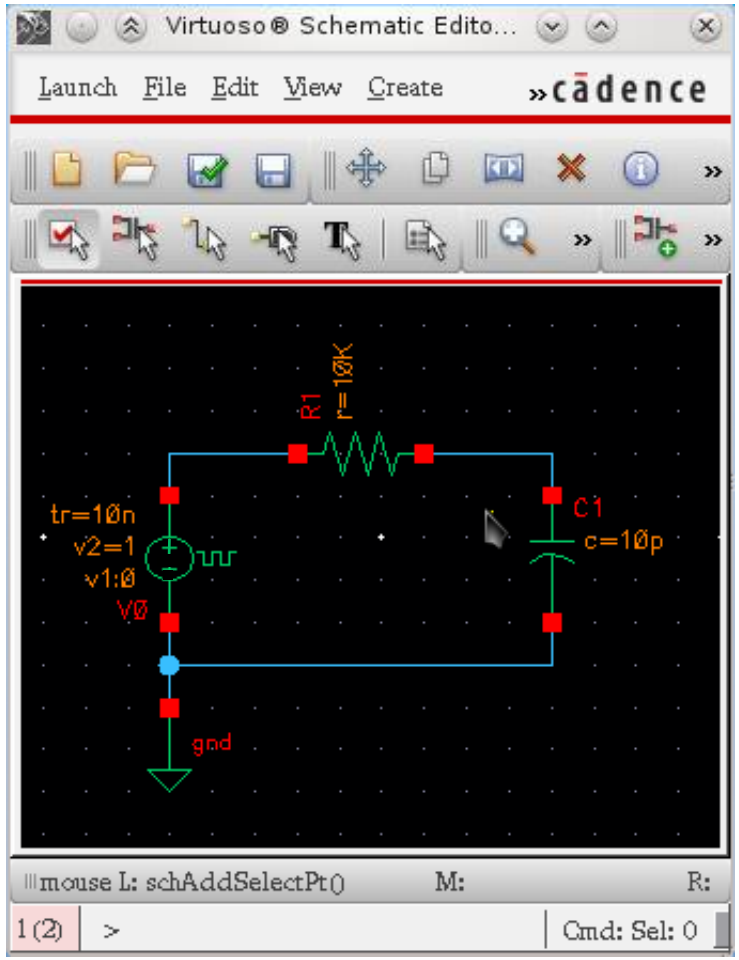
- To add a pin:
  - Press the  button or press ctrl-p or Create → Pin
  - Select input / output and place pin

RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

EXERCISE 1

Exercise 1

- Create a schematic ‘LowPass’
- Draw a Low pass filter with
  - A resistor (‘res’) of 10 kΩ. (name it ‘R1’)
  - A capacitor (‘cap’) of 10 pF (name it ‘C1’)
  - A ground symbol (‘gnd’)
  - A pulse generator (‘vpulse’) which generates rectangular pulses from 0→1V (voltage 1 / voltage 2) at a frequency of 1 MHz with rise / fall times of 10 ns
  - Set 'AC Magnitude' to 1



- Save the design

Exercise 2

- Save the design under the name ‘HighPass’
  - Use File → Save a Copy
- Open the new schematic and modify it to give a High Pass Filter with RC time constant 10µs