Exercise:
Getting Started with Cadence
(Installation, Schematic Entry, Simulation)

Prof. Dr. P. Fischer

Lehrstuhl für Schaltungstechnik und Simulation
Uni Heidelberg

Starting Cadence

Remote Login

- Follow the instructions in the additional document to log onto the SuS machine

- In the CIP Pool in OMZ:
  - Login on a machine with your university account
  - Go to https://sus.ziti.uni-heidelberg.de/password/
    to change your password
    $\geq$ 8 characters, 3 from 4 types (normal, capital, number, special)
  - Open (lower left on screen)
    Applications->ziti-Tools->x2Go Circuit Design
  - Use the provided login and the (new) password

Preparing Cadence...

- If you start for the first time:
- Copy the files required to run cadence from our 'template' directory into a (newly created) subdirectory CCS:

```
cp -r /shares/designs/teaching/ccs/workdir_template CCS
```

- Change to the CCS directory and have a look…
  - `cd CCS`
  - `ls -al`

- Your working directory now contains 3 small files:
  - a start script **`start.sh`**
  - a configuration file **`.cdsinit`**
  - a file with library paths **`cds.lib`**

Starting Cadence

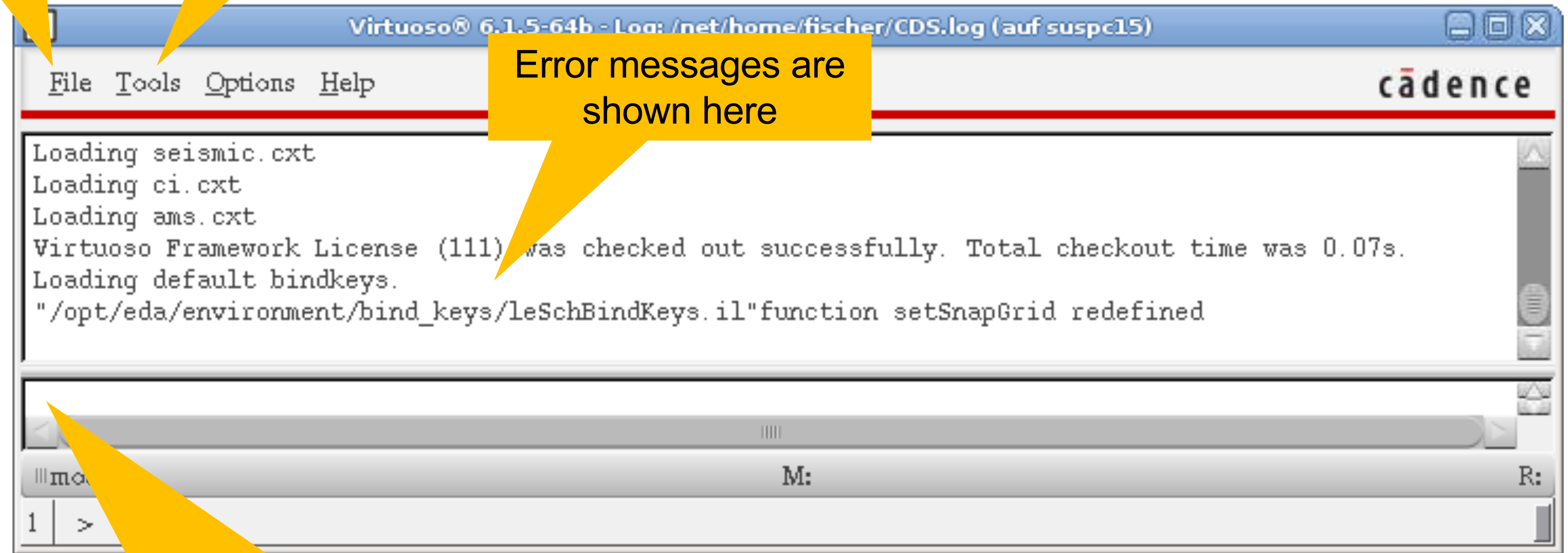
- Start cadence with

```
./start.sh &
```

The usual file menu (with 'exit')

Start more stuff from here

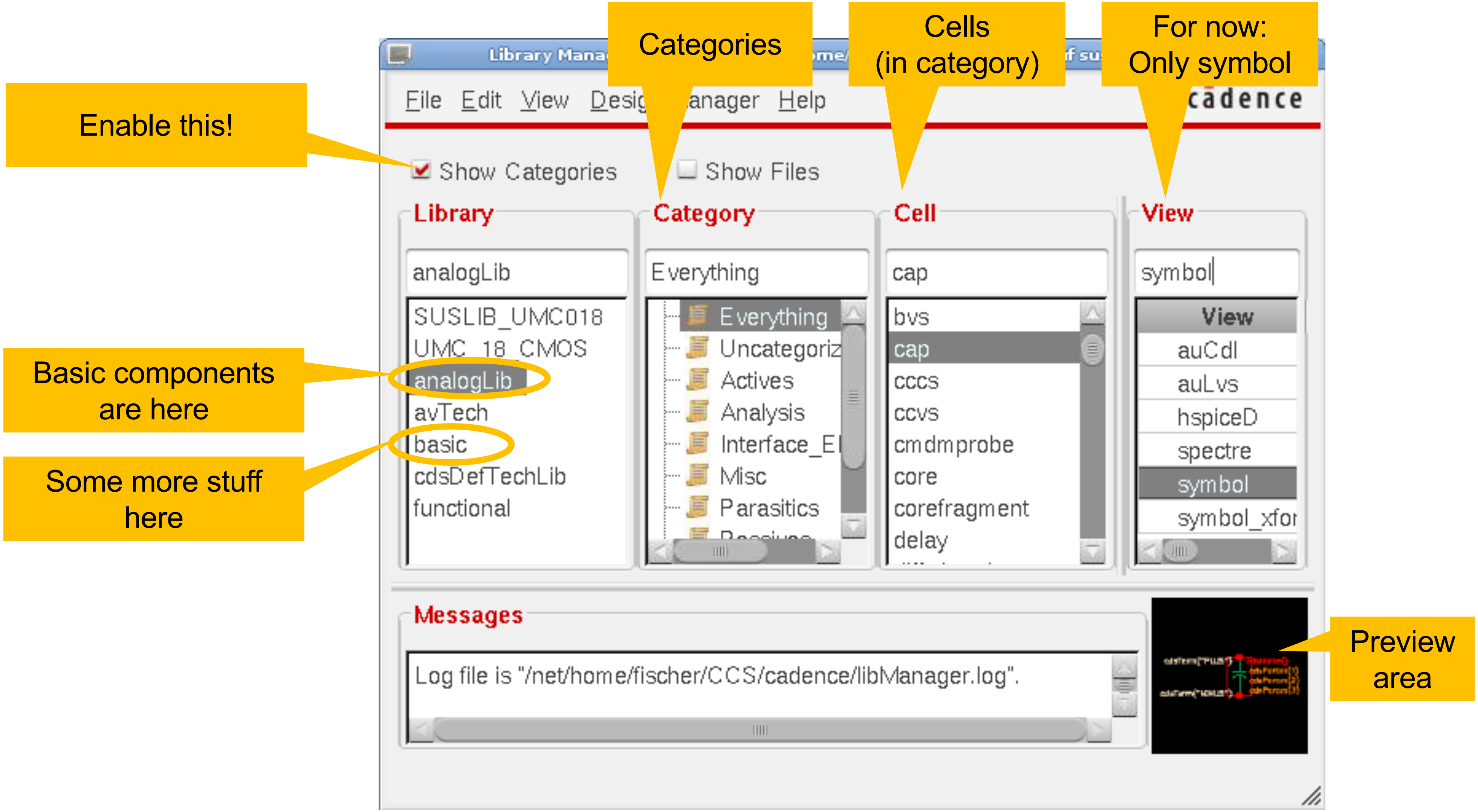
Error messages are shown here



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

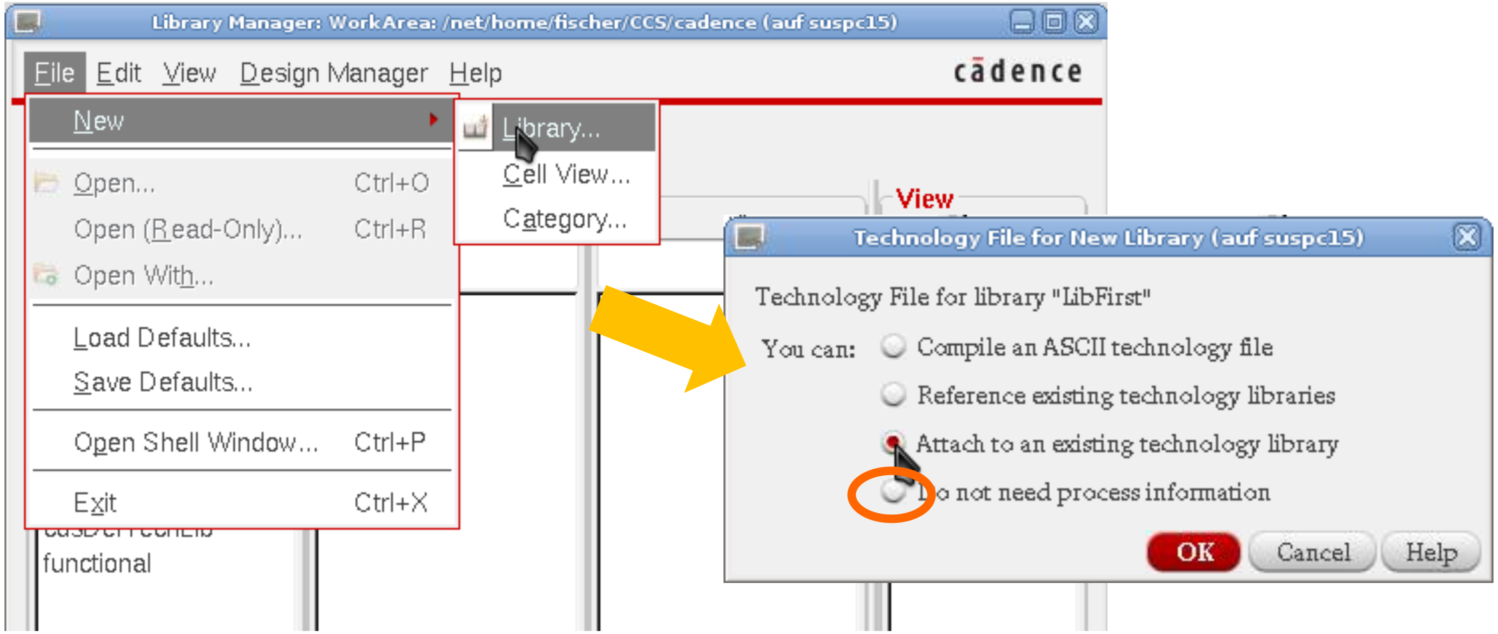
Opening the Library Manager

- Open the library browser under Tools → Library Manager…



Creating a library

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



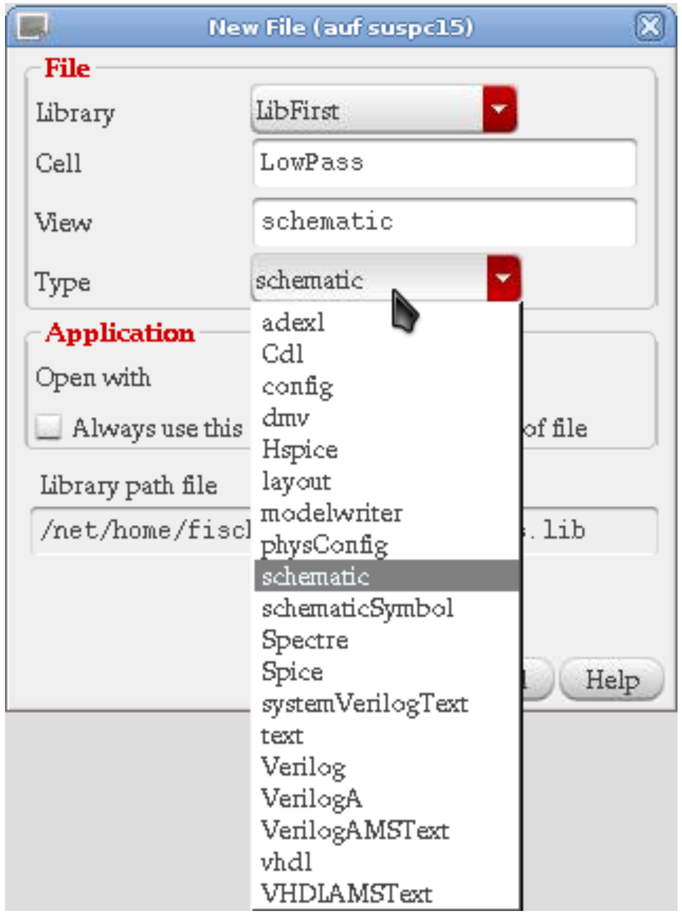
- Choose ‘do not need process information’

- The new library should now be visible in the library browser

ENTERING CIRCUIT SCHEMATICS

Creating a new Schematic

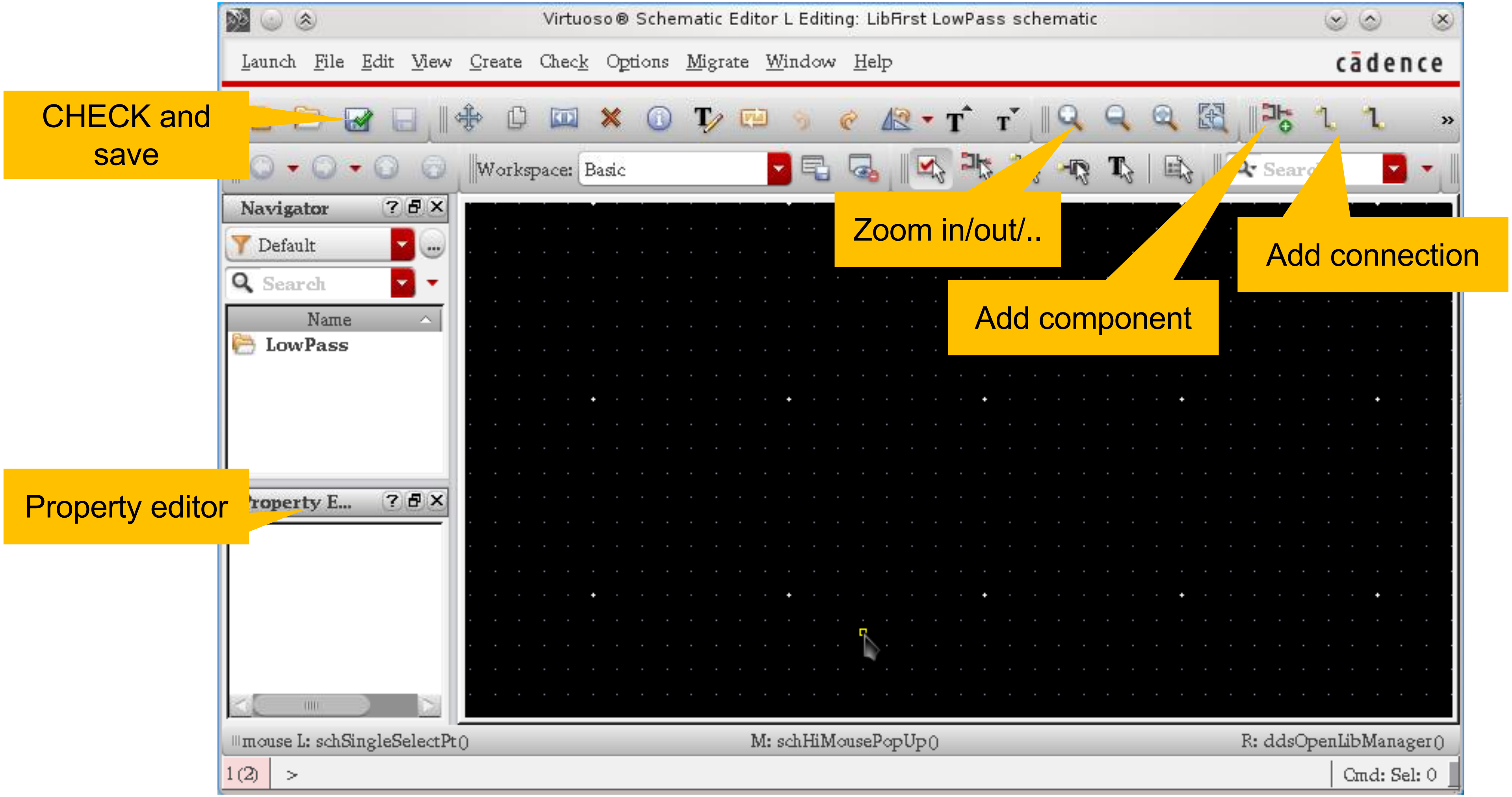
- In the library browser, select your library
- Create a new schematic with File → New → Cell View…
- Select type 'schematic' by selecting from the drop down list
- Give the cell a name
- The schematic editor opens
- Save the cell!



- 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)

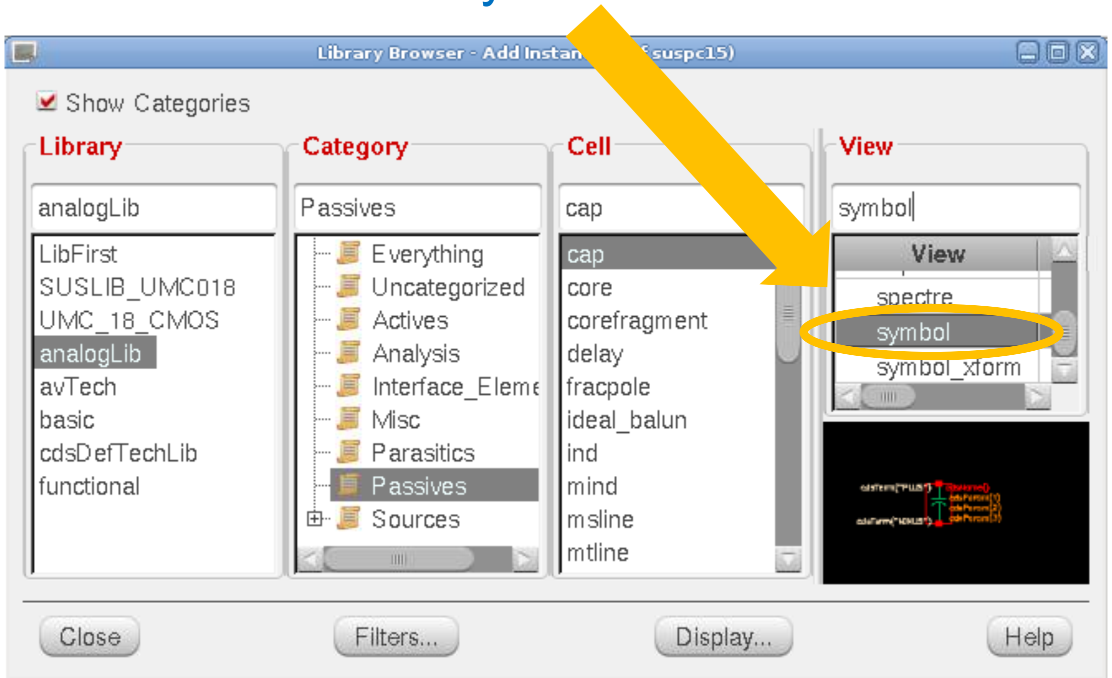
Opening the Schematic

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



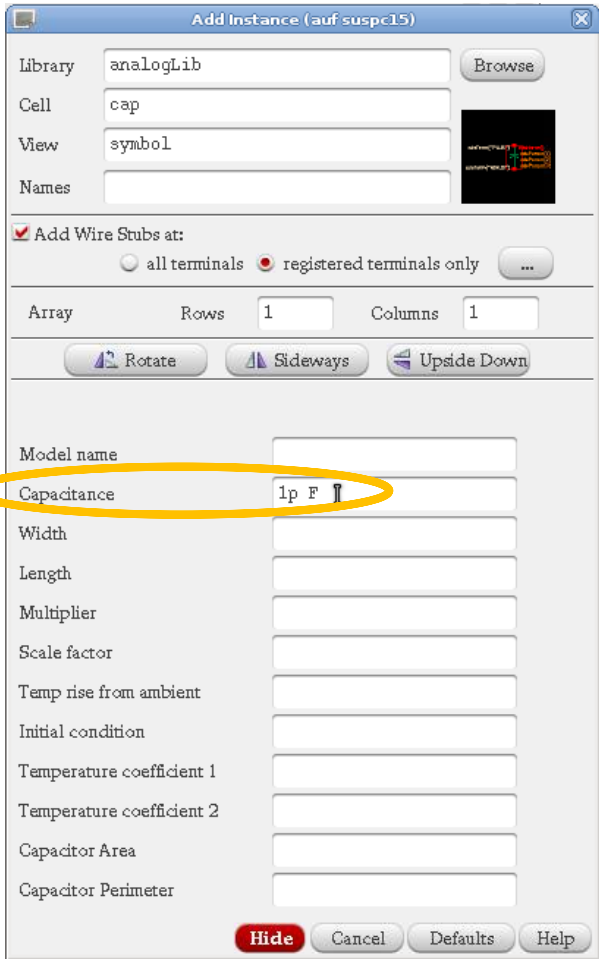
Adding a component (1)

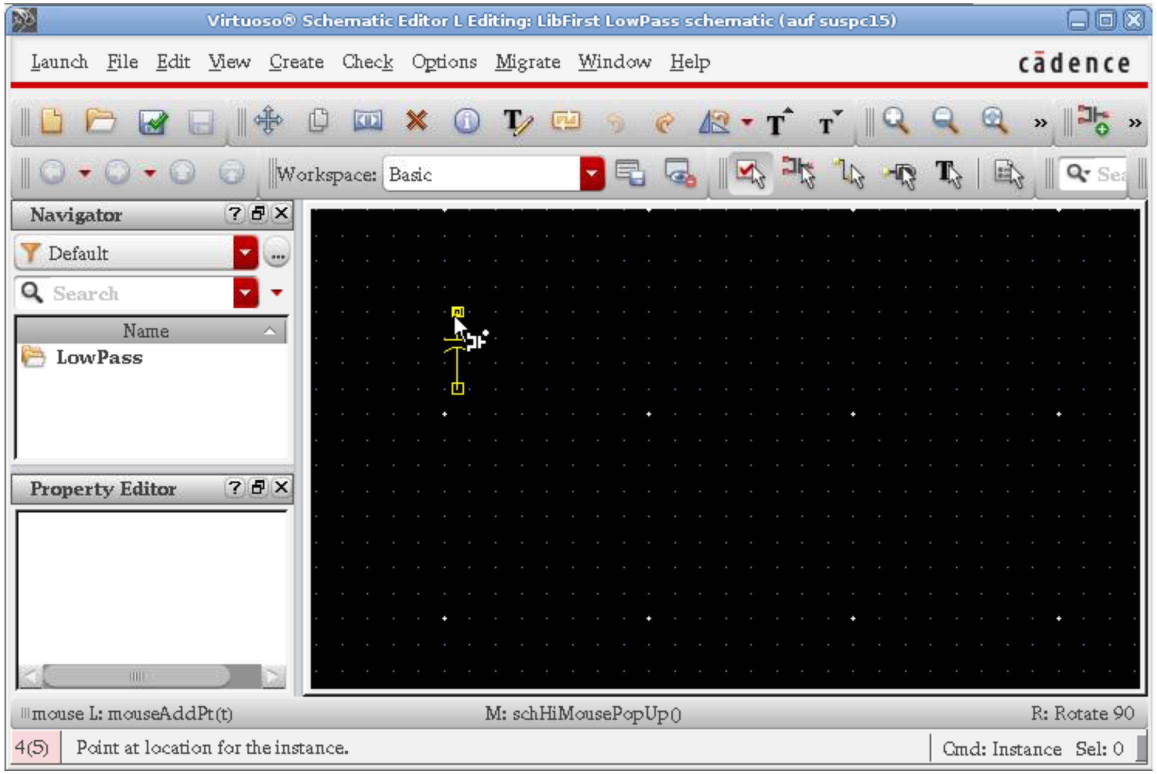
- To add a component (‘instance’)
  - Press the ‘Create Instance’ button  or
  - select Create → Instance or
  - press ‘i’
- **Browse** to the correct library (for now: analogLib)
- Choose a cell from the library browser, for instance ‘cap’
  - Make sure View ‘symbol’ is selected!



Adding a component (2)

- Set the parameters (values) of the instance
  - For instance the capacitance of a capacitor…
- Place the instance on the sheet (mouse click)





Press ESCAPE to finish.

Modifying Parameters

- To modify an existing instance, select it and
  - Press the 'Edit Properties' button or
  - select Edit → Properties → Object or
  - press 'q' or
  - use the Property Editor Panel

- For values, use the suffixes
  - **m** for milli $= 10^{-3}$
  - u for micro $= 10^{-6}$
  - n,p,f,a for nano, pico,…
  - k for kilo $= 10^{3}$
  - **M** for Mega $= 10^{6}$ (**m** ↔ **M**!)
  - G for Giga $= 10^{9}$
- Do **NOT** add a unit (like m**V**)
  - It is added automatically

The Cadence UI: Executing Commands

- There are two possibilities for most commands:

- Execute command once:
  - Select objects (or multiple objects with shift-click)
  - Press command key (for instance ‘c’ for copy)
  - 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

Cadence UI: 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 you need to press F3 twice

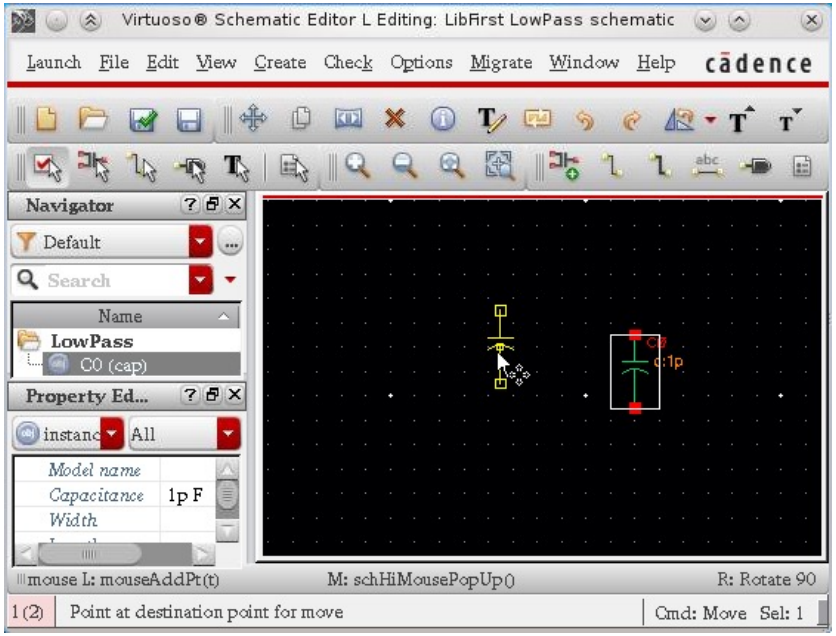
Cadence UI: Zooming …

- 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→ …

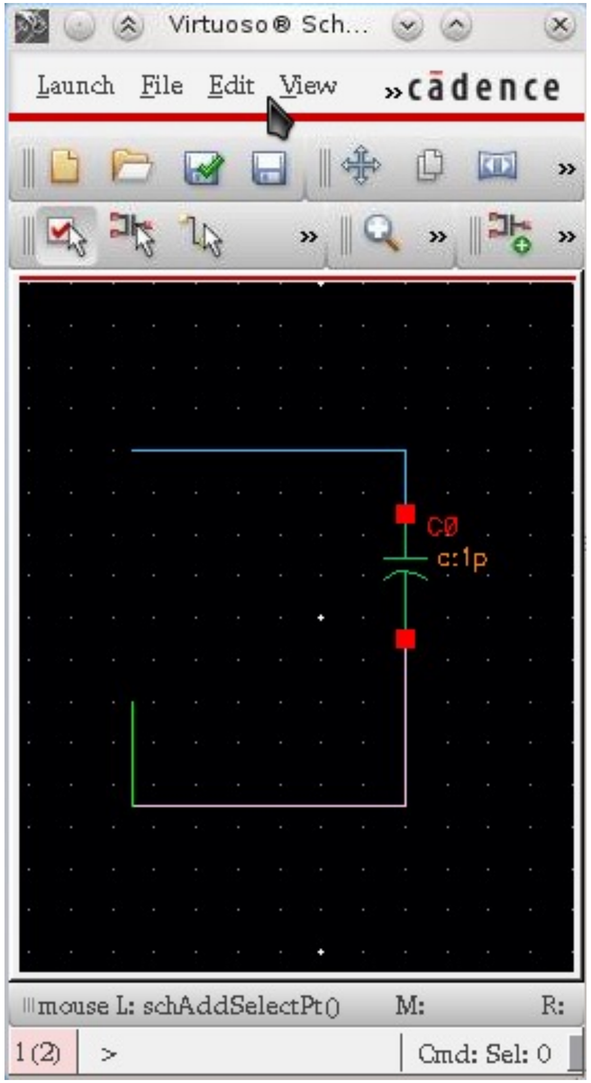
Moving an Instance

- 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

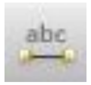


- Changing behavior: press F3
  - change angle
  - change color
  - …

Adding net Names

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

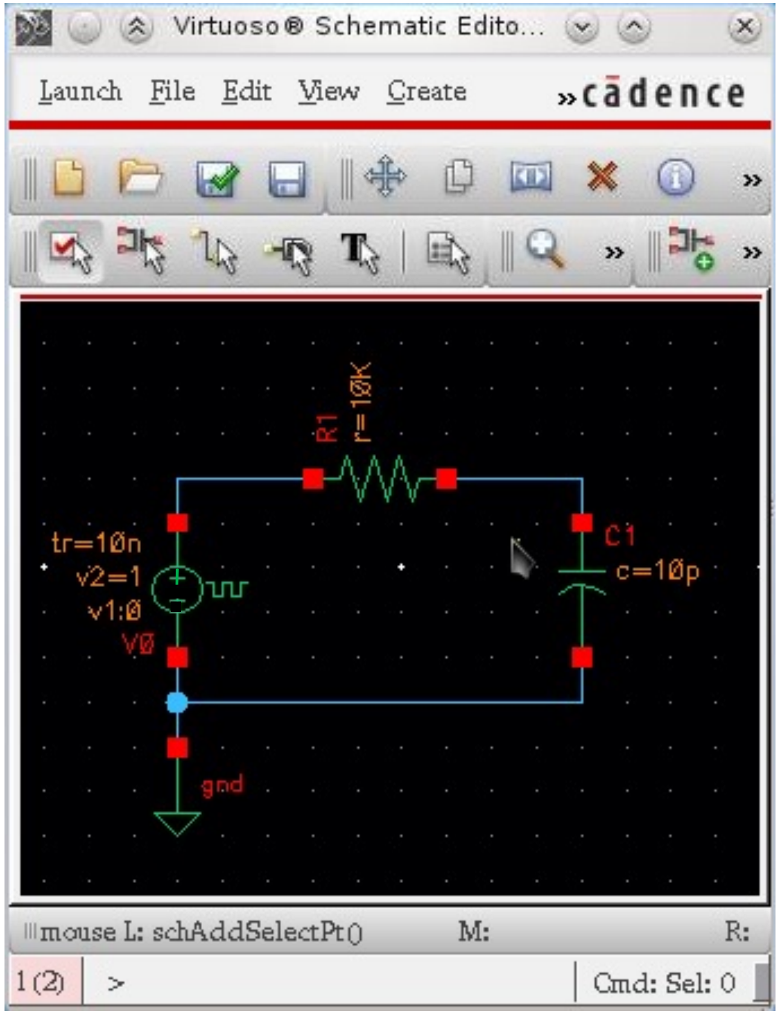
Nets with the same names (labels) are automatically **connected**!

- 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

Example

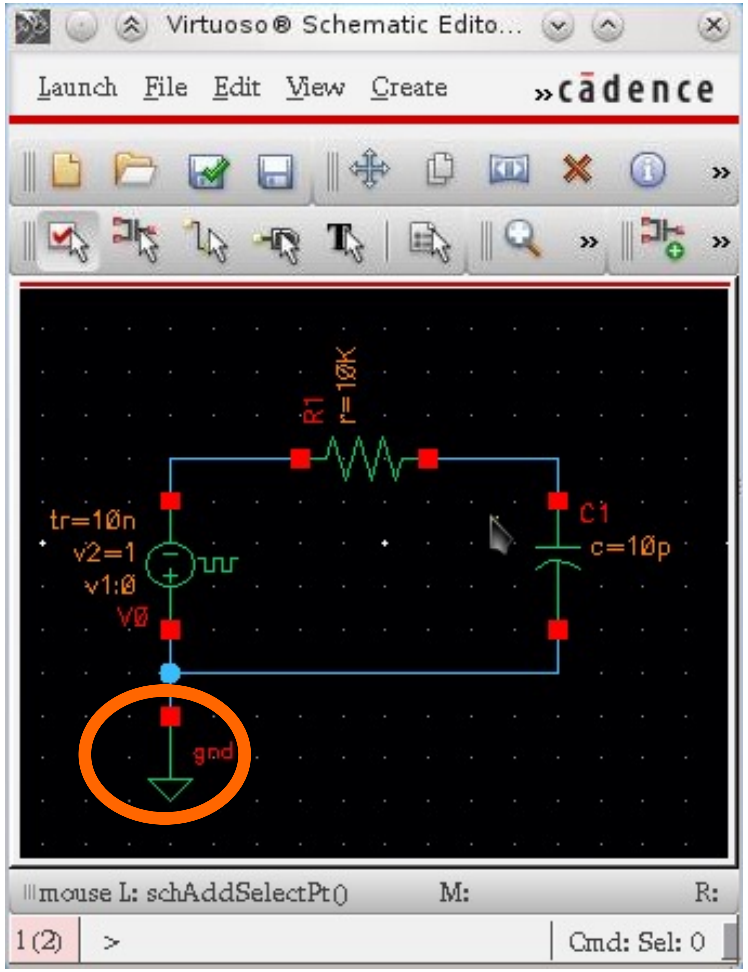
- We create a schematic ‘LowPass’
- We add
  - A resistor (‘res’) of 10 kΩ. (name it ‘R1’)
  - A capacitor (‘cap’) of 10 pF (name it ‘C1’)
  - A ground symbol (‘gnd’)
  - A pulse generator (‘vpulse’) which generates rectangular pulses from 0→1V (voltage 1 / voltage 2) at a frequency of 1 MHz with rise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



SIMULATING A CIRCUIT

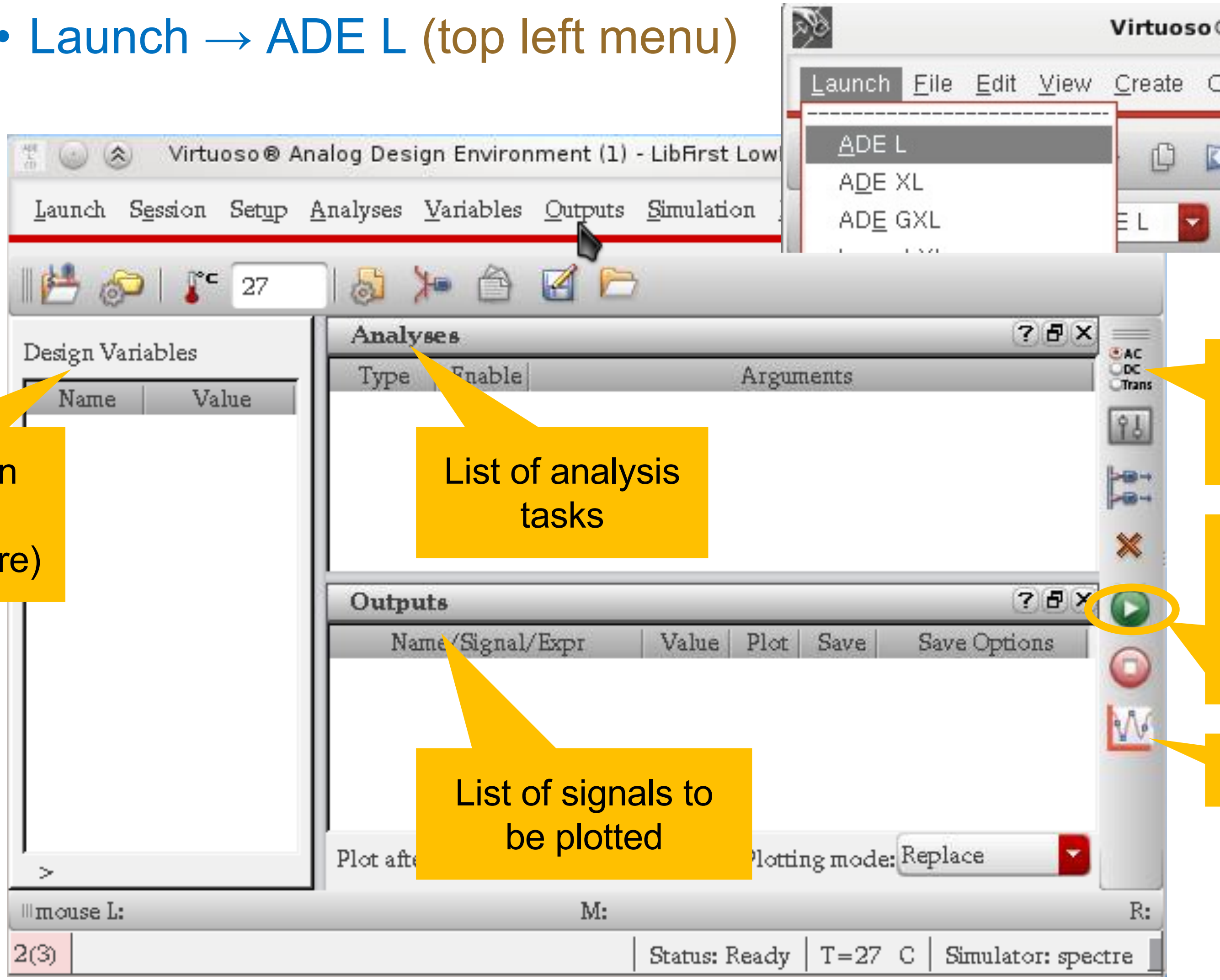
The Ground Net

- For a simulation to work properly, there must be the net ‘gnd!’ in your schematic.
- Best use the ‘gnd’ pin found in analogLib
  - This symbol ‘attaches’ the net name ‘gnd!’ to the net connected to its pin…



Starting the Simulator

- In an open schematic, start the simulator with
  - Launch → ADE L (top left menu)



Select Type of Simulation

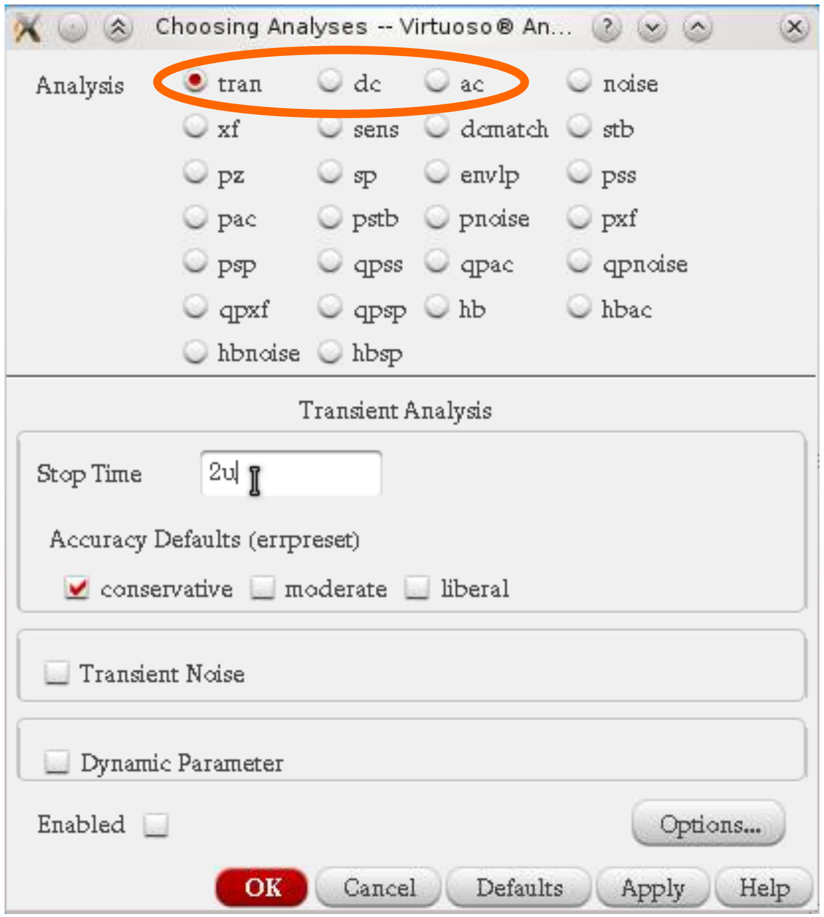
- Open the panel
  - By pressing the  button or

  - In Analyses → Choose Menu

- Choose the analysis you need (we will only use 'tran', 'dc', 'ac')

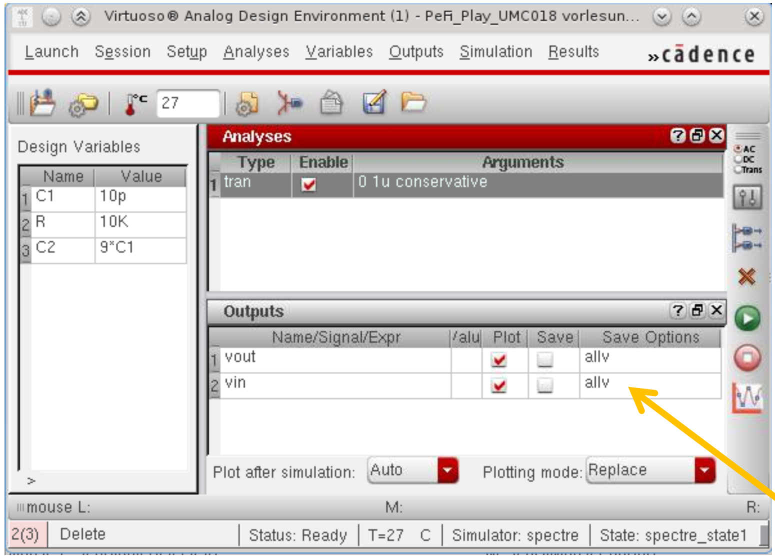
- Provide the parameters required by the analysis

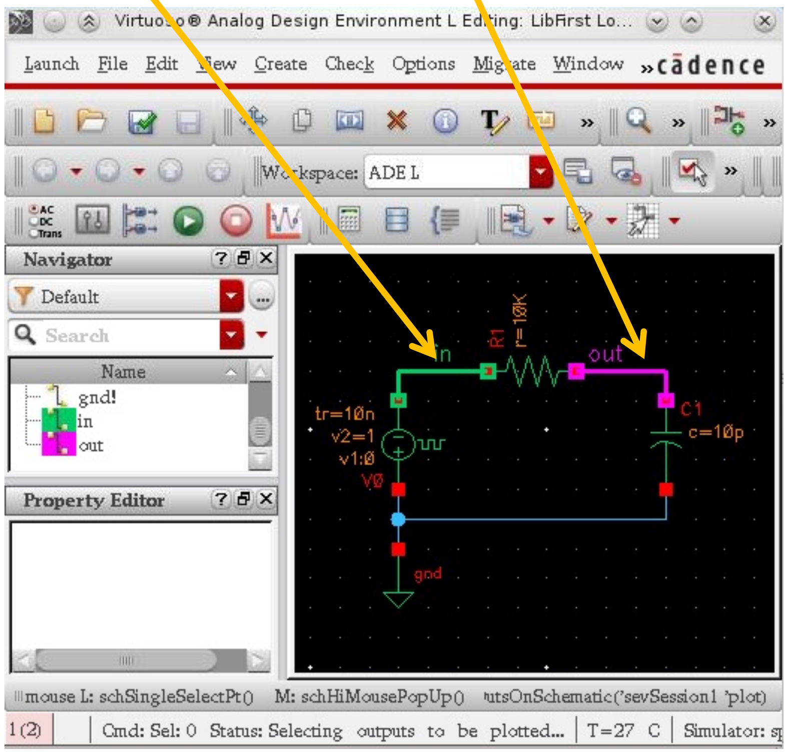
- Press ok



Select Signals to be Plotted

- In simulator window
  - Select Outputs → To be Plotted→ Select on Schematic
- Select the **nets** (they are highlighted with different colors) to show **voltages**
- Select **pins** to show **currents**
- End with ESC (important!)



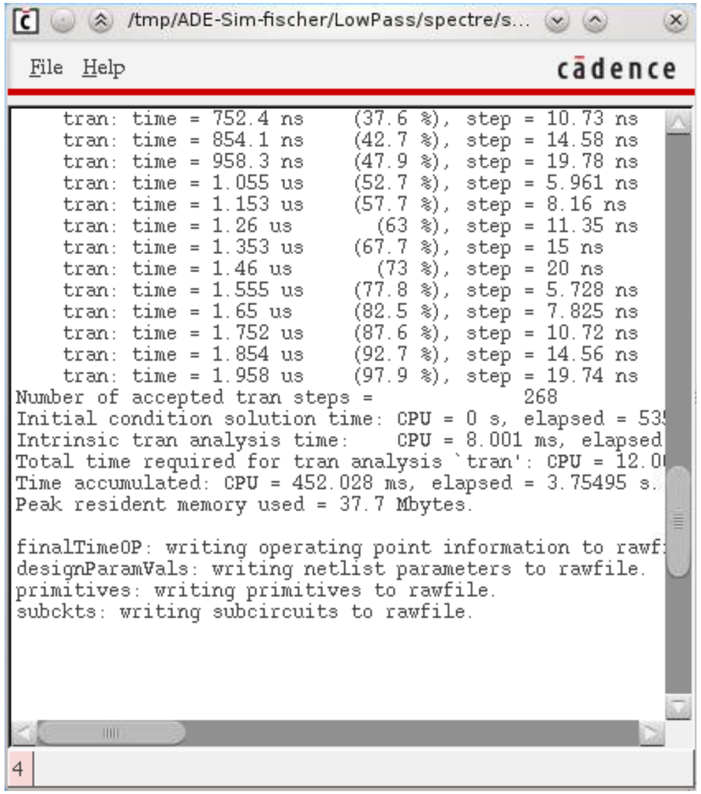


- Signals are listed in the lower right panel of the sim. window

Starting the Simulation

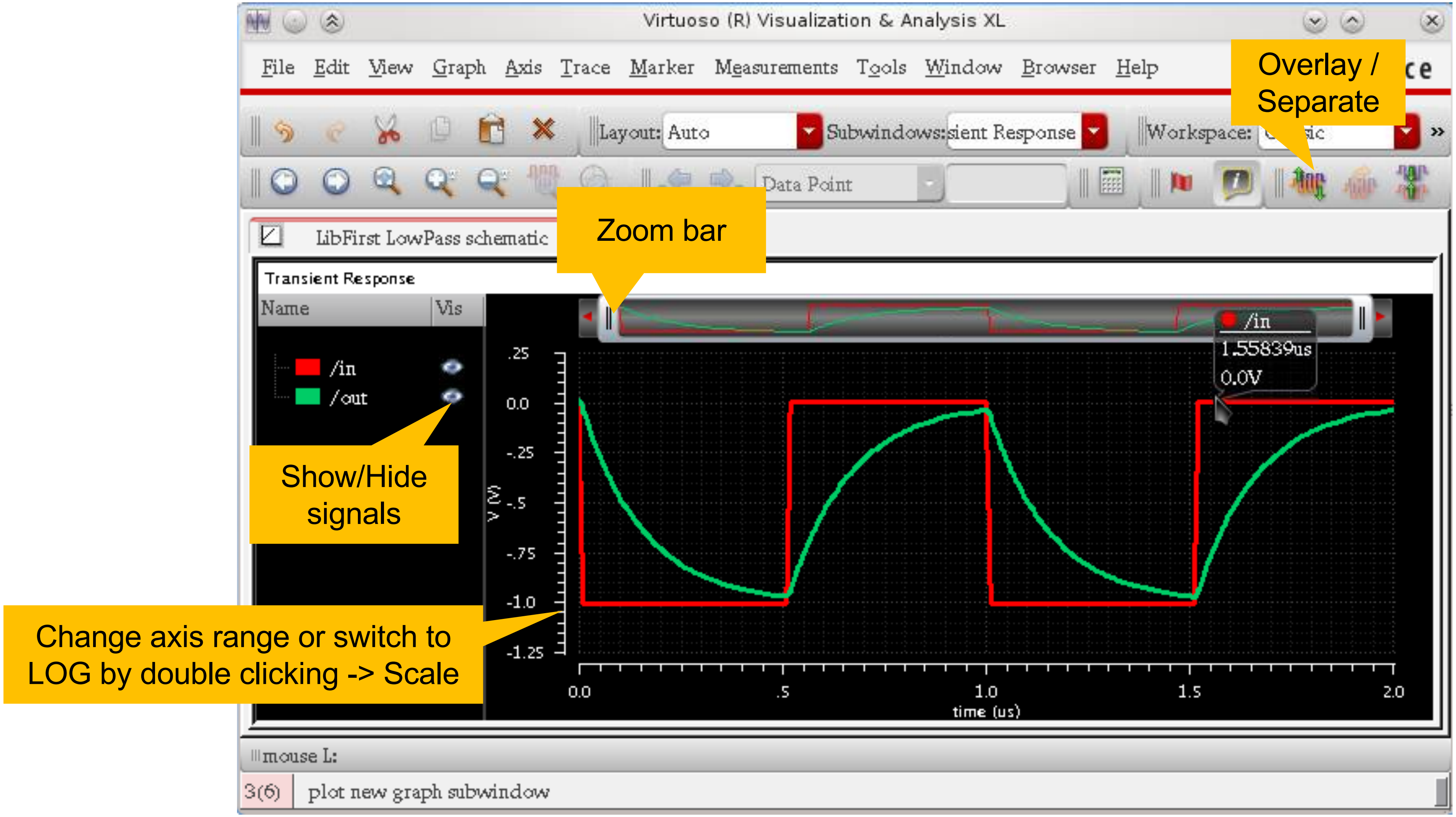
You can disable the automatic display of the log window under Setup → Environment → Automatic output log

- Press or Simulation → Netlist and Run
- A log file shows up
- If your run fails:
  - Check the log file
  - (Re-open it with Simulation → Output Log)
- Some common reasons for failure:
  - Schematic has been changed, but **not** checked & saved (F8)
  - Device parameters (resistor value..) are missing or wrong
  - Design variables (see later) have not been set
  - Circuit has severe errors (shorts..)
  - …
- The waveform viewer should show up



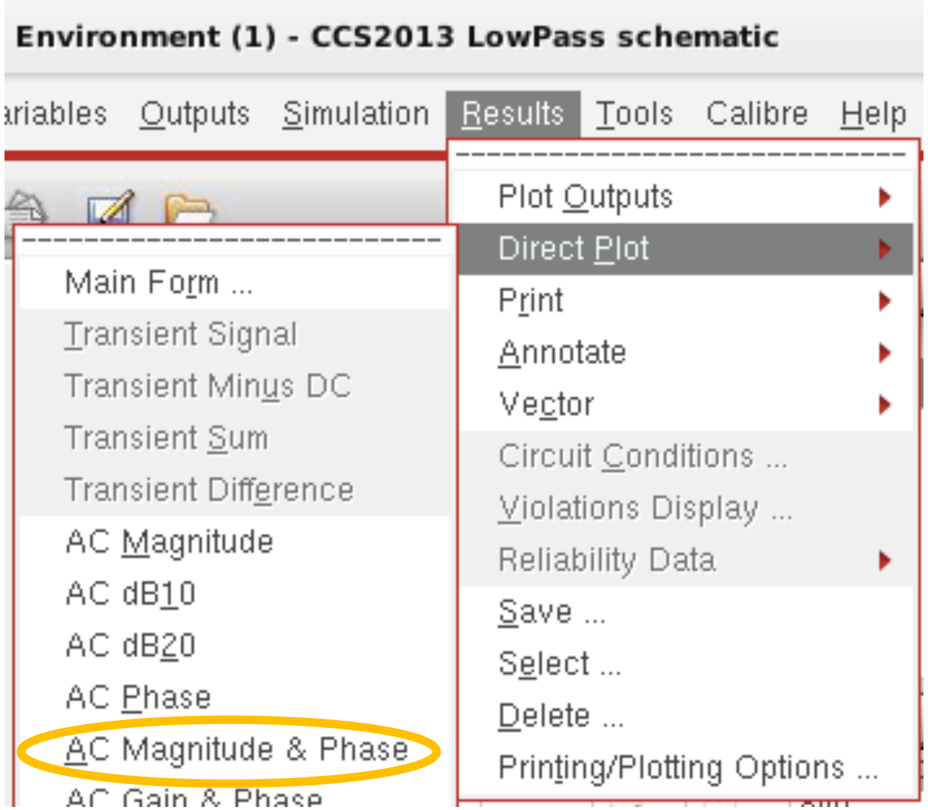
Look at the Results

- The waveform viewer shows all selected signals:



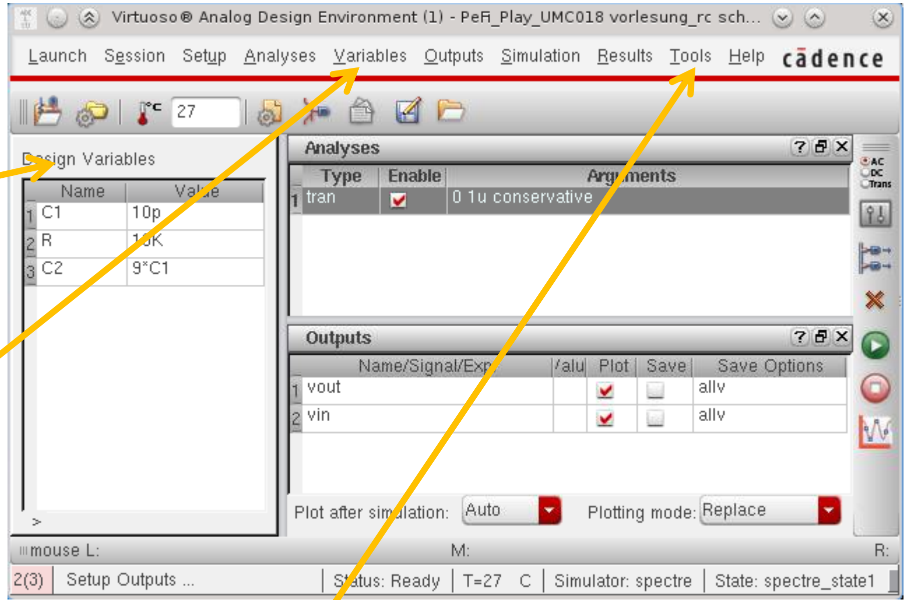
Showing More / Other Signals

- You can also add signals after the simulation using Results → Direct Plot → …
- In this menu, you can select for instance AC Magnitude and Phase
  - As usual, you must then select the net and stop with ESC.



Adding Design Variables

- You can set parameters to symbolic values ('CF', 'FREQ')
  - These 'design variables' do not need to be 'declared'
- You must then
  - Add the 'design variables' by hand in the lower left window or
  - Use the Variables → Copy from CellView command



- You can then change the Design Variables in the simulation window and just re-run the simulation (Simulation→ Run) with **no need** to make a new netlist
- You can also run several simulations with varying values in a Tools → Parametric Analysis

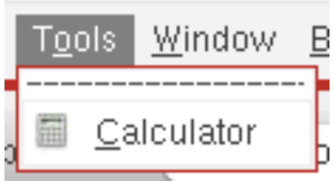
(Copying Design Variables to the Cellview)

- You can copy the design variables and their values to the cell view with Variables → Copy to Cellview
  - This helps you to remember the best values..

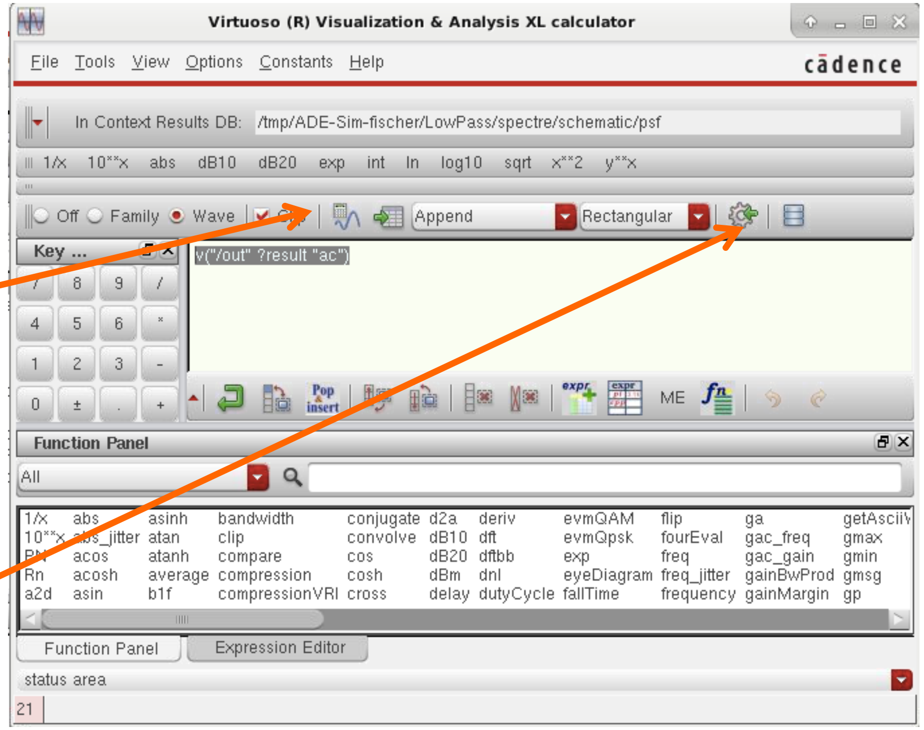
- Caveat:
  - If you delete a variable in a schematic component, so that it is not used any more, it may still be 'saved' in the cell view and simulation will complain.
    In such a case you have to delete the variable in the simulation window and copy the new set to the cellview

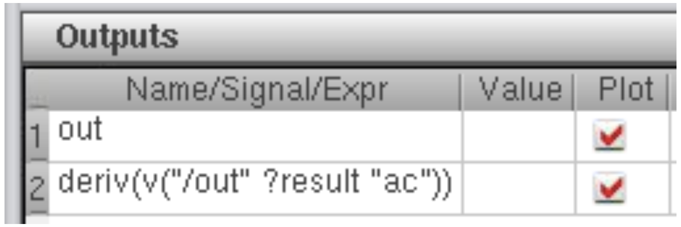
The WaveForm Calculator

- For more complex analysis, you can open the Waveform Calculator under Tools → Calculator
  - Best select the wave you want to analyze first



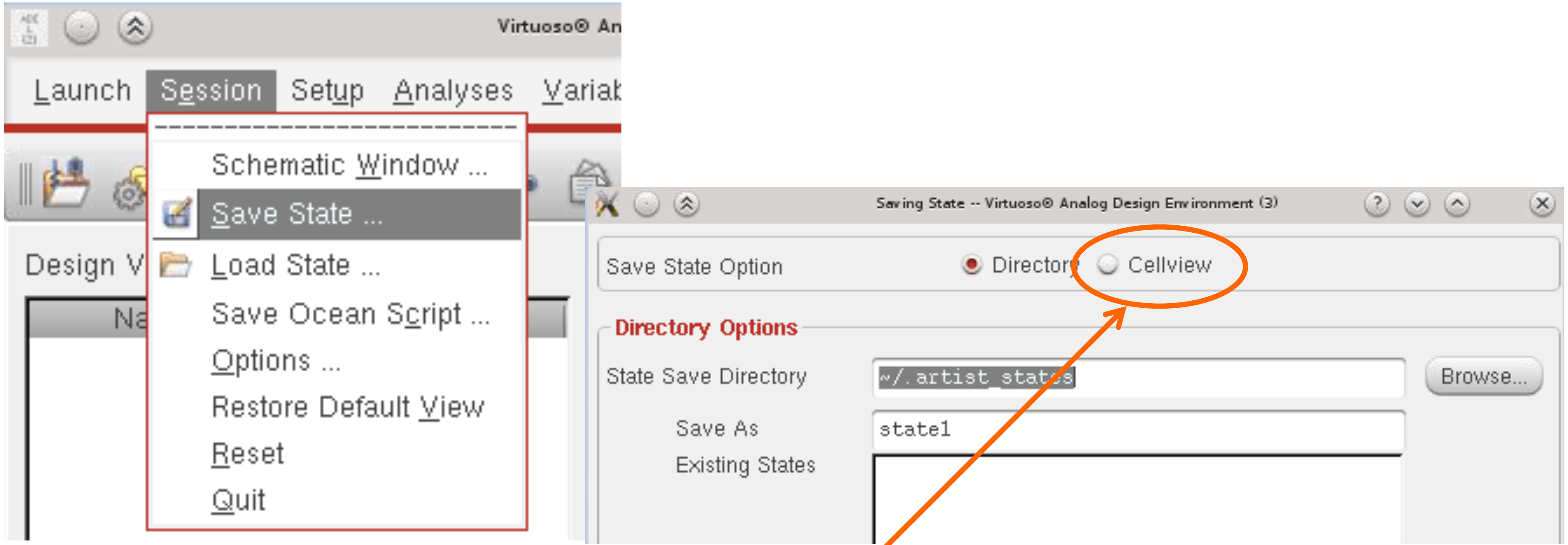
- You can assemble expressions graphically (using RPN)
- Plot the result once  or
- Send the expression to the outputs window  so that it is evaluated every time you run a new simulation





Saving you Simulation Settings

- Before you leave, you can save all settings, results… under Session → Save State



- You can save to a file or to the cellview (view 'spectre_state')
  - Better save to the cellview, so that everything is in the library