



# **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  
≥ 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

```
Virtuoso@ 6.1.5-64b - Log: /net/home/fischer/CDS.log (auf suspc15)
File Tools Options Help
Loading seismic.cxt
Loading ci.cxt
Loading ams.cxt
Virtuoso Framework License (111) was checked out successfully. Total checkout time was 0.07s.
Loading default bindkeys.
"/opt/eda/environment/bind_keys/leSchBindKeys.il"function setSnapGrid redefined
1 | >
```

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...**

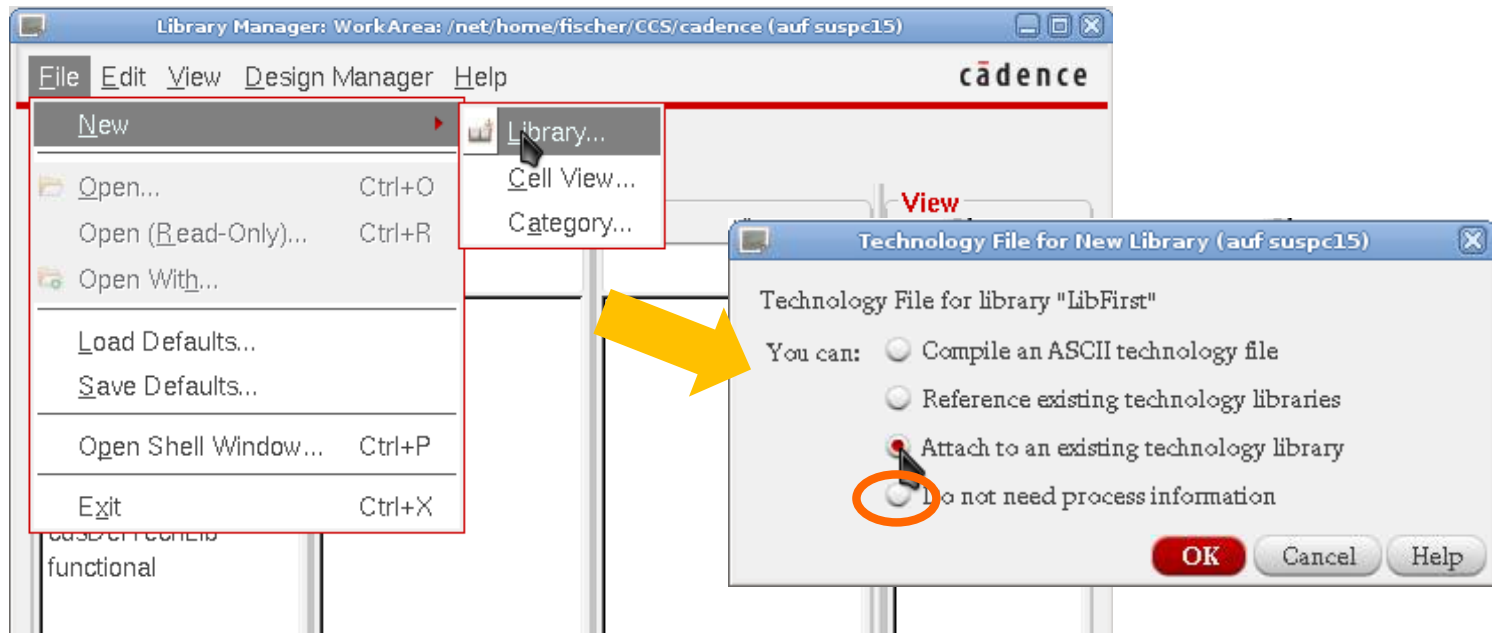
The screenshot shows the Cadence Library Manager window with the following components and annotations:

- Enable this!**: Points to the  **Show Categories** checkbox.
- Categories**: Points to the **Category** pane showing a tree view with 'Everything' selected.
- Cells (in category)**: Points to the **Cell** pane showing a list of components like 'cap', 'bvs', 'cccs', etc.
- For now: Only symbol**: Points to the **View** pane showing a list of views with 'symbol' selected.
- Basic components are here**: Points to the **Library** pane, specifically highlighting 'analogLib' and 'basic'.
- Some more stuff here**: Points to the bottom of the **Library** pane.
- Preview area**: Points to a small preview window in the bottom right corner showing a circuit symbol.



# 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



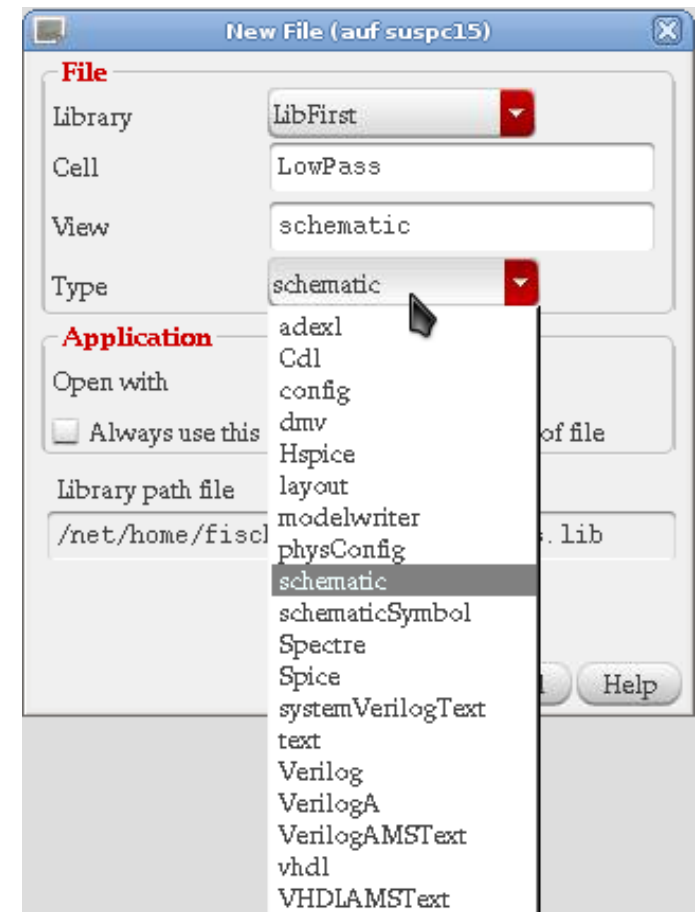
# ENTERING CIRCUIT SCHEMATICS





# Creating a new Schematic

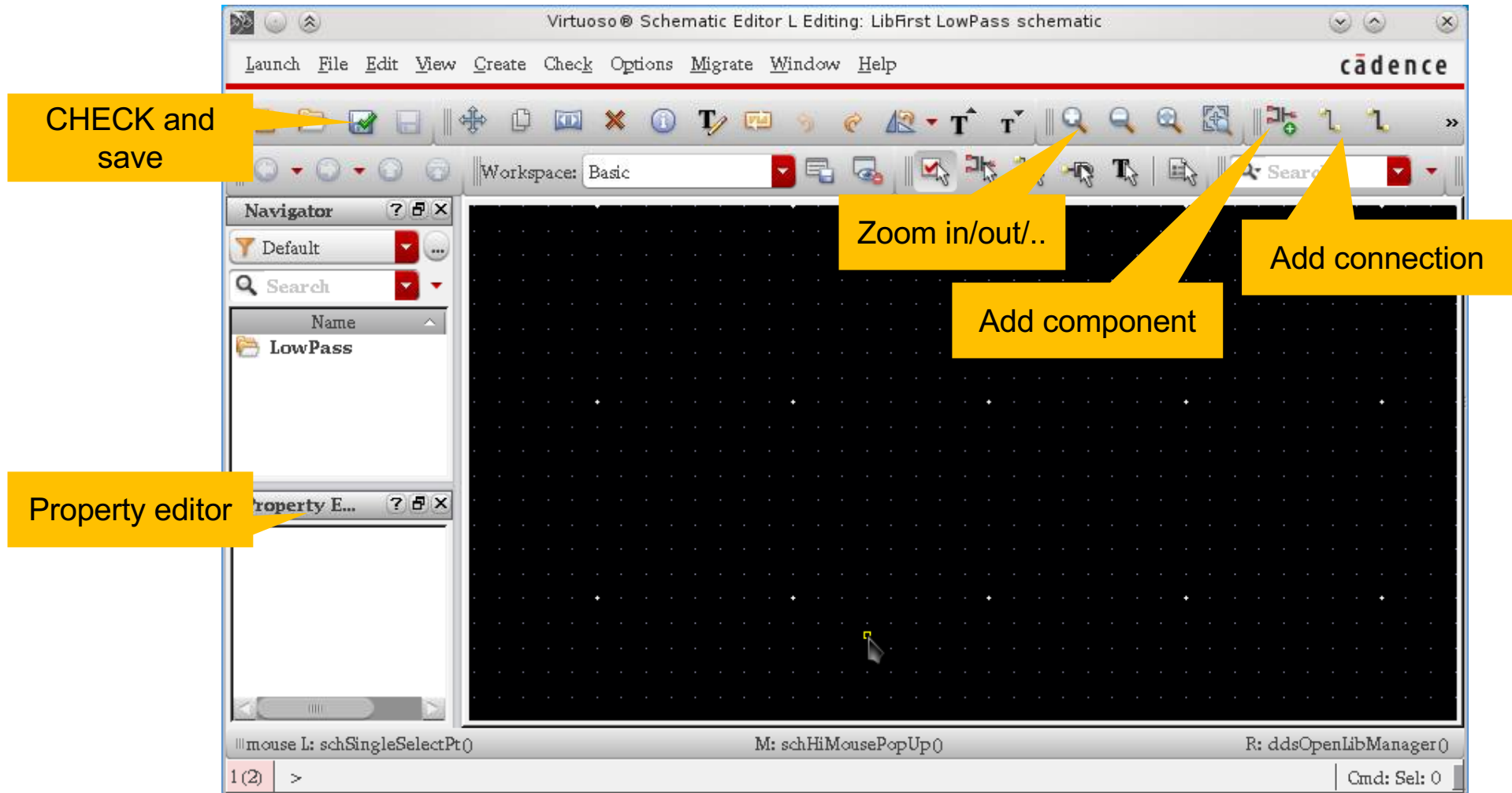
- In the library browser, select your library
  - Create a new schematic with **File → New → Cell View...**
  - Select type 'schematic' by selecting from the drop down list
  - Give the cell a name
  - The schematic editor opens
  - Save the cell!
- 
- Check that the cell is now in your library
  - If 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

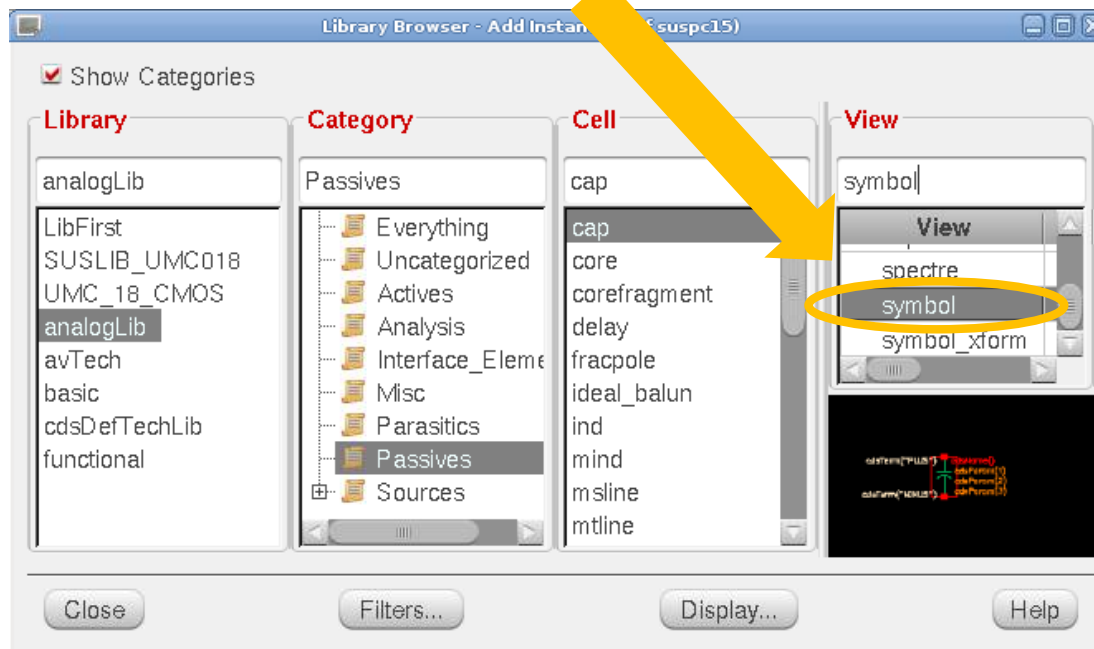
- Double click on the 'schematic' entry (or right click & open)
  - The schematic editor of 'virtuoso' comes up:





# Adding a component (1)

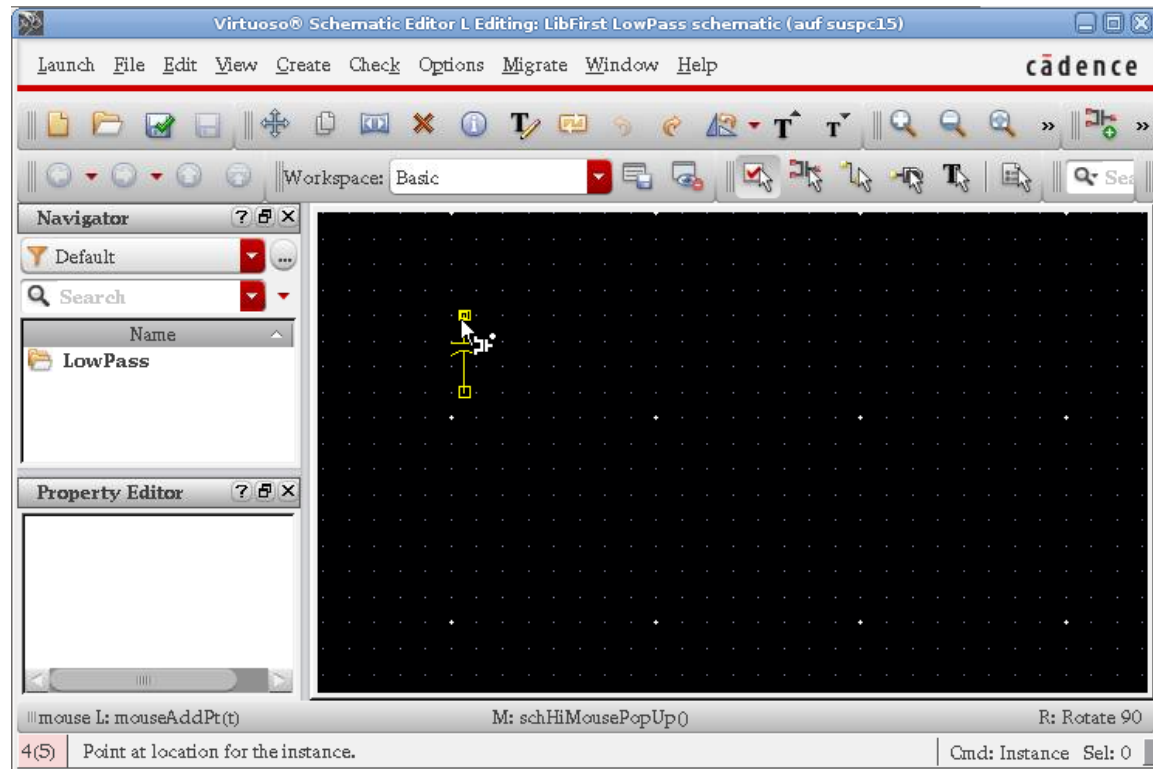
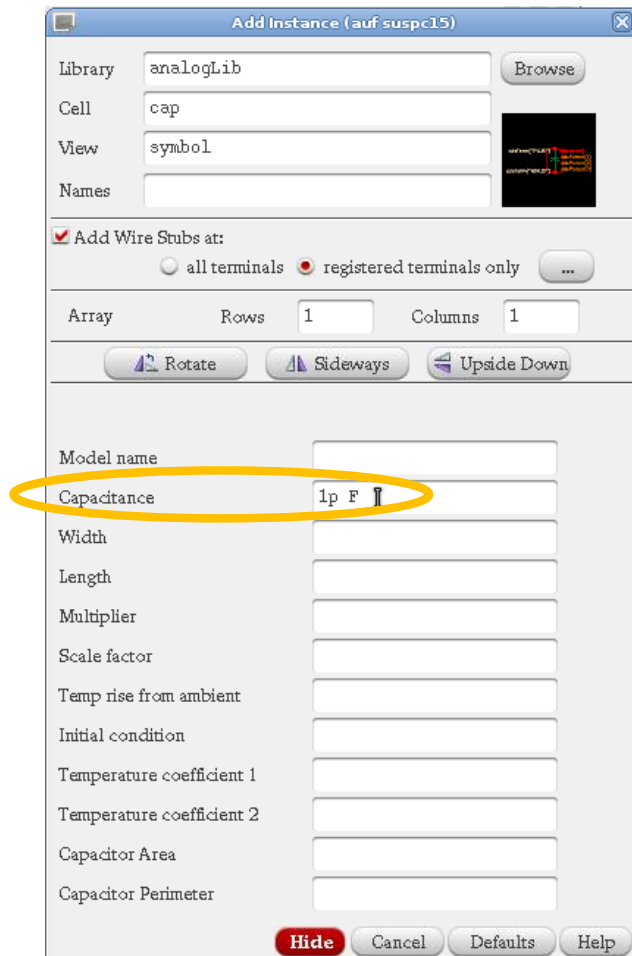
- To add a component ('instance')
  - Press the 'Create Instance' button  or
  - select **Create** → **Instance** or
  - press 'i'
- **Browse** to the correct library (for now: **analogLib**)
- Choose a cell from the library browser, for instance 'cap'
  - Make sure View '**symbol**' is selected!





## 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 **mV**)
  - 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'

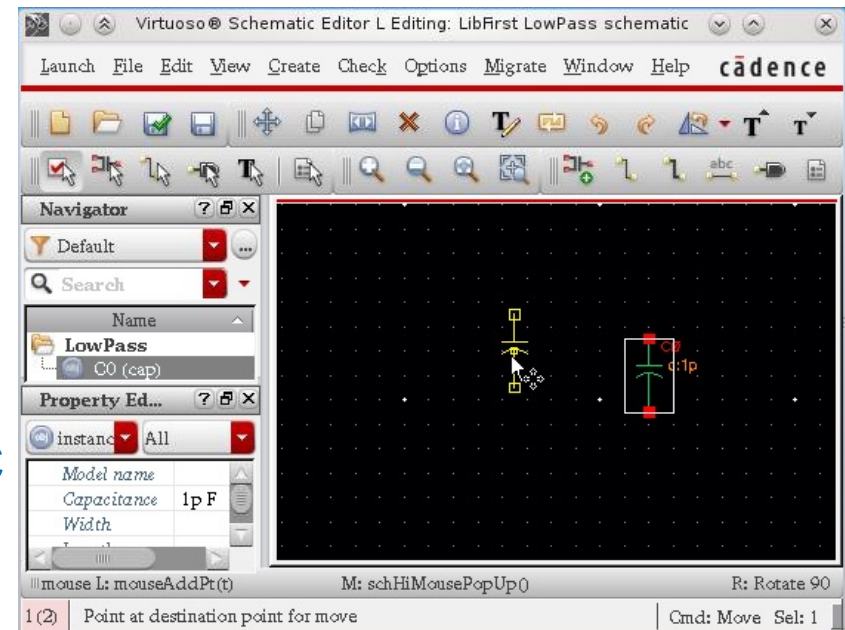
- Alternative:

- First press 'm'
- select – move – drop, ...ESC

- Alternative:


- click – drag – drop

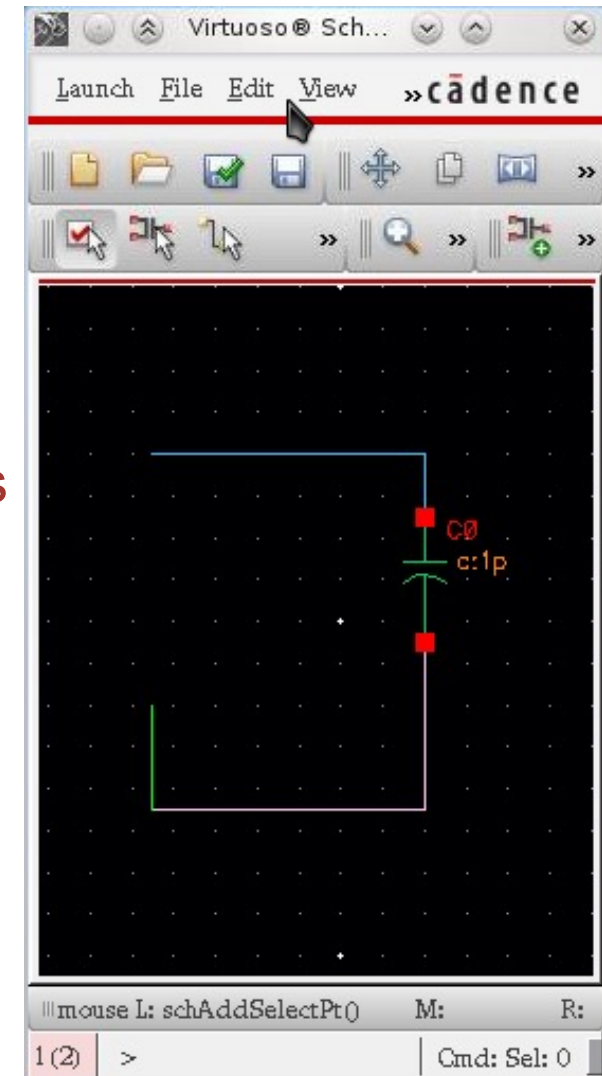
- For options (rotate, flip,...): F3 or right mouse





# 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

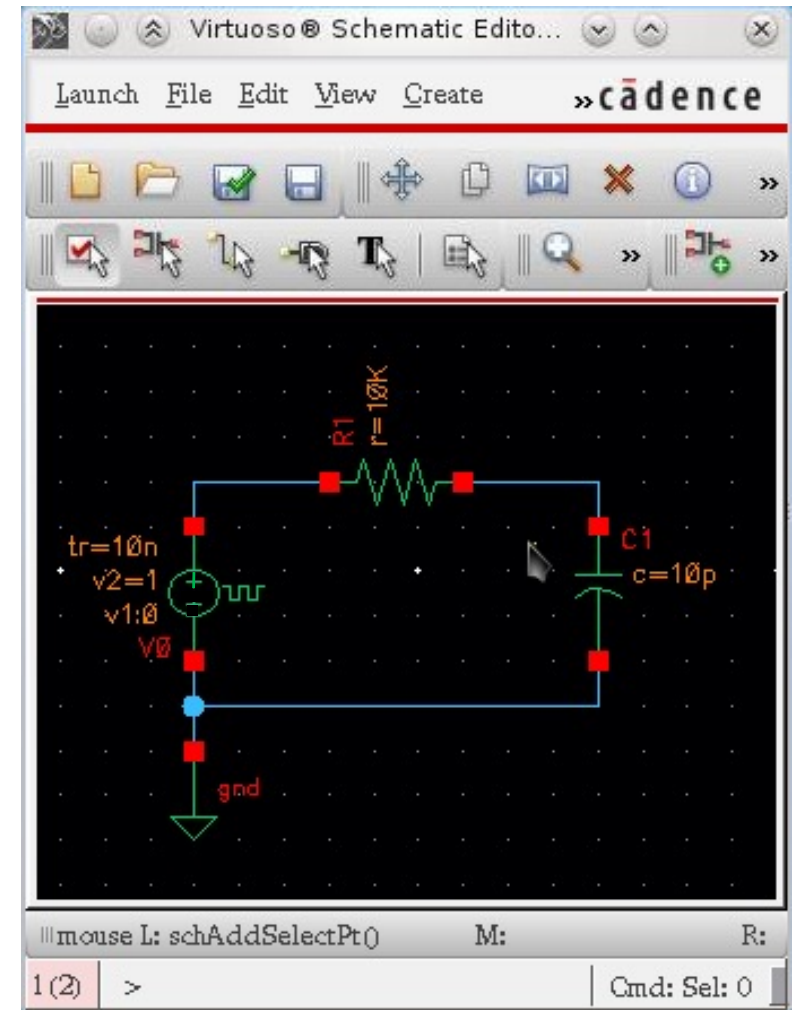
- To identify nets, you can
  - assign names (labels) to nets or
  - connect them to pins.
- To assign a label:
  - Press the  button or
  - Select **Create** → **Wire Name** or
  - Press **'l'** (label)
- Type in the label name and click on the net
- Continue with further labels
- End with **ESC**.
- To add a pin:
  - Press the  button or press **ctrl-p** or **Create** → **Pin**
  - Select **input / output** and place pin

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



# Example

- We create a schematic 'LowPass'
- We add
  - A resistor ('res') of 10 k $\Omega$ . (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 $\rightarrow$ 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



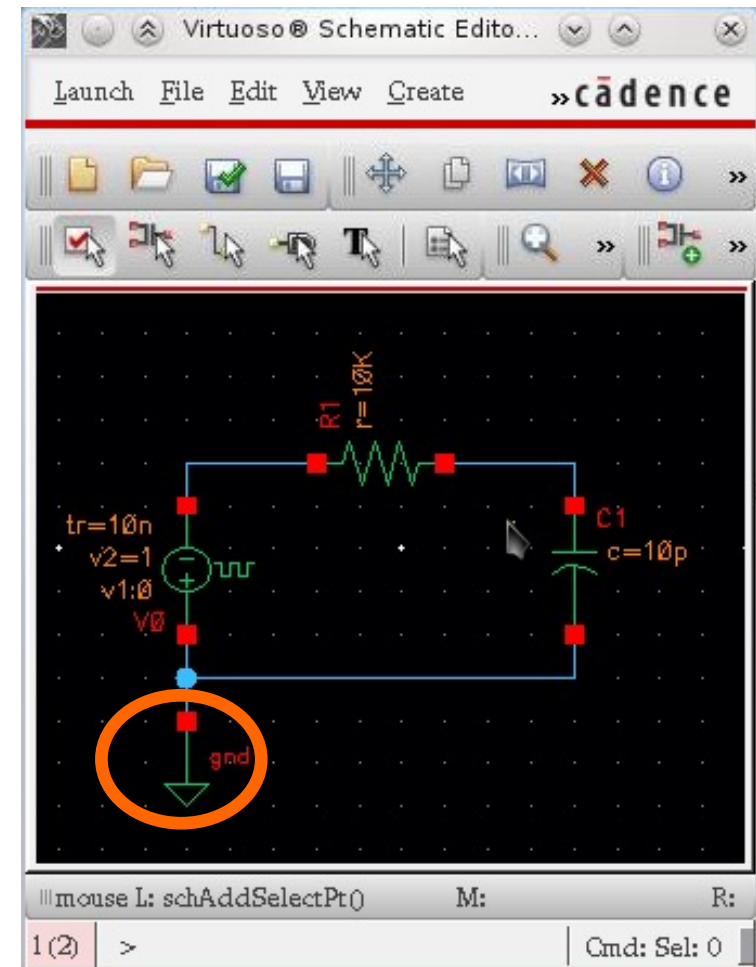


# SIMULATING A CIRCUIT



# 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)


The screenshot shows the Virtuoso Analog Design Environment (ADE) interface. The main menu bar includes 'Launch', 'File', 'Edit', 'View', 'Create', and 'C'. The 'Launch' menu is open, showing options for 'ADE L', 'ADE XL', and 'ADE GXL'. The 'Launch' menu item is highlighted in red. Below the menu bar is a toolbar with various icons. The main workspace is divided into several panels:

- Design Variables:** A table with columns 'Name' and 'Value'. A yellow callout box points to this panel with the text: "Can set design variables (parameters here)".
- Analyses:** A table with columns 'Type', 'Enable', and 'Arguments'. A yellow callout box points to this panel with the text: "List of analysis tasks".
- Outputs:** A table with columns 'Name/Signal/Expr', 'Value', 'Plot', 'Save', and 'Save Options'. A yellow callout box points to this panel with the text: "List of signals to be plotted".
- Simulation Controls:** A vertical toolbar on the right side of the workspace. A yellow callout box points to the 'AC', 'DC', and 'Trans' radio buttons with the text: "Select type of simulation". Another yellow callout box points to the green play button with the text: "Start simulation (regenerate the netlist)". A third yellow callout box points to the plot icon with the text: "Plot".

At the bottom of the interface, the status bar shows "Status: Ready", "T=27 C", and "Simulator: spectre".



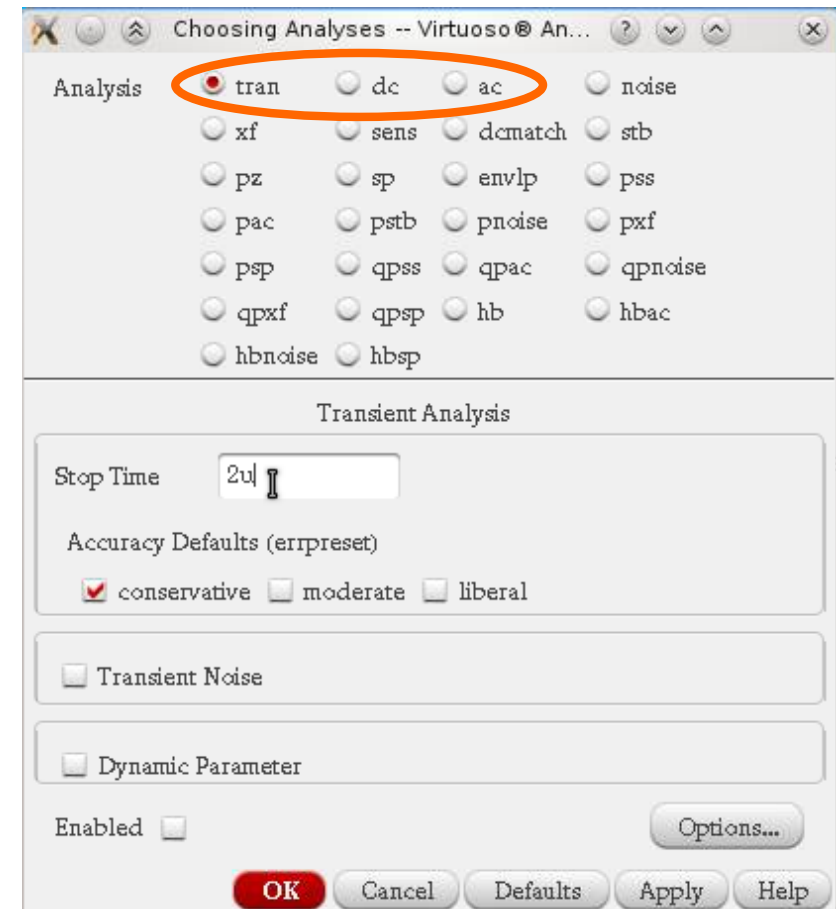
# Select Type of Simulation

- 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**

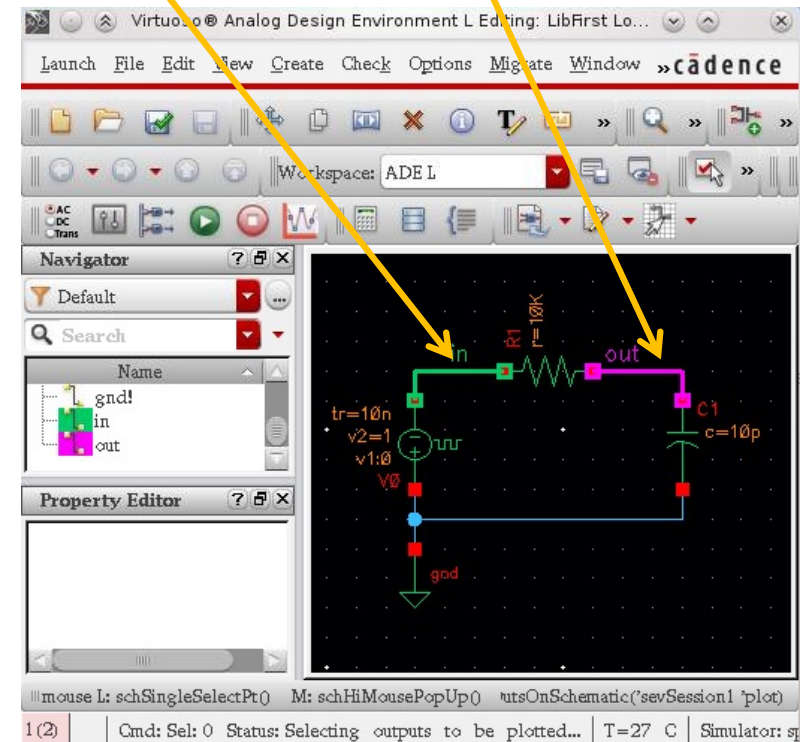
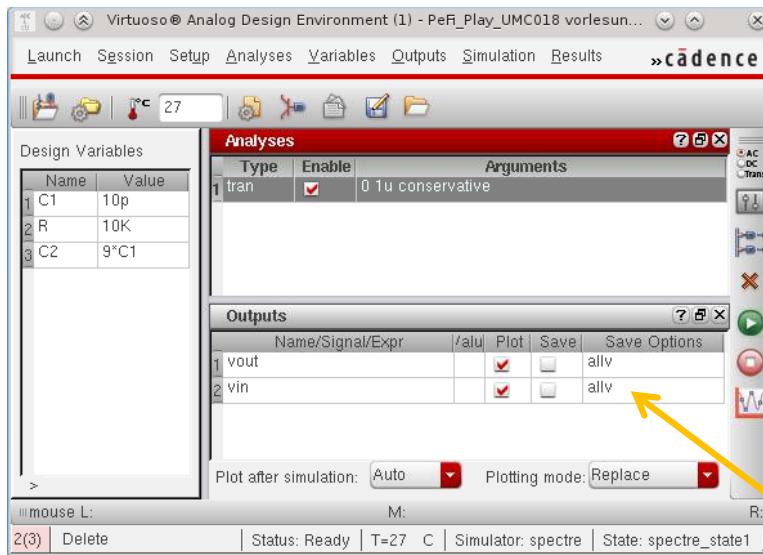






# Select Signals to be Plotted

- 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



```

File  Help  cadence

tran: time = 752.4 ns (37.6 %), step = 10.73 ns
tran: time = 854.1 ns (42.7 %), step = 14.58 ns
tran: time = 958.3 ns (47.9 %), step = 19.78 ns
tran: time = 1.055 us (52.7 %), step = 5.961 ns
tran: time = 1.153 us (57.7 %), step = 8.16 ns
tran: time = 1.26 us (63 %), step = 11.35 ns
tran: time = 1.353 us (67.7 %), step = 15 ns
tran: time = 1.46 us (73 %), step = 20 ns
tran: time = 1.555 us (77.8 %), step = 5.728 ns
tran: time = 1.65 us (82.5 %), step = 7.825 ns
tran: time = 1.752 us (87.6 %), step = 10.72 ns
tran: time = 1.854 us (92.7 %), step = 14.56 ns
tran: time = 1.958 us (97.9 %), step = 19.74 ns
Number of accepted tran steps = 268
Initial condition solution time: CPU = 0 s, elapsed = 53
Intrinsic tran analysis time: CPU = 8.001 ms, elapsed
Total time required for tran analysis `tran': CPU = 12.0
Time accumulated: CPU = 452.028 ms, elapsed = 3.75495 s
Peak resident memory used = 37.7 Mbytes.

finalTimeOP: writing operating point information to rawfile.
designParamVals: writing netlist parameters to rawfile.
primitives: writing primitives to rawfile.
subckts: writing subcircuits to rawfile.
    
```



# Look at the Results

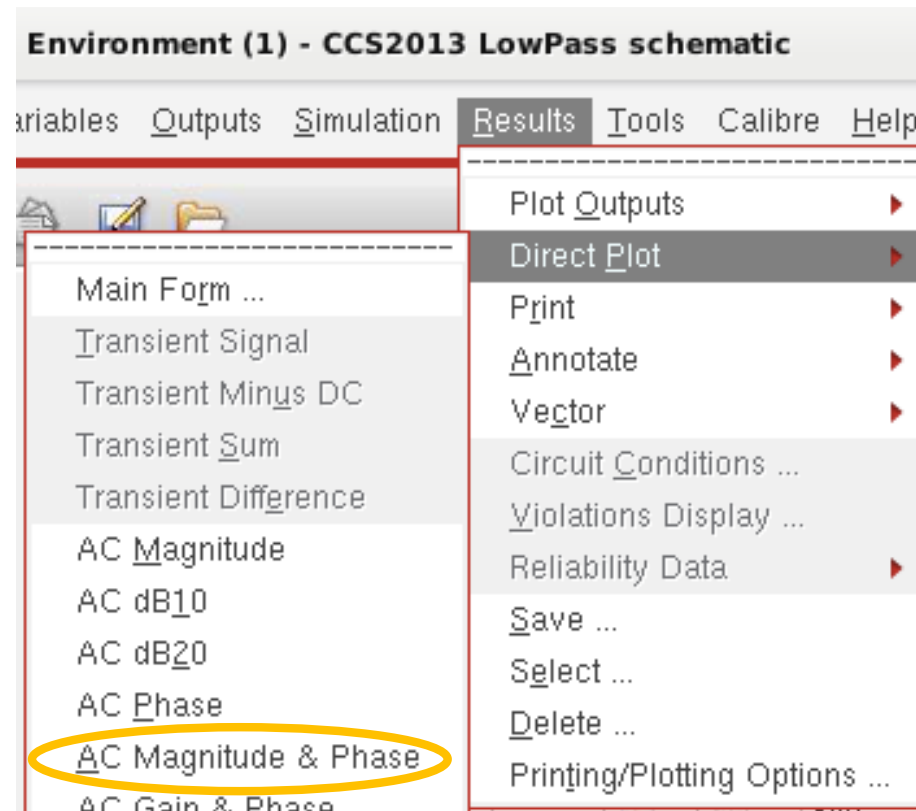
- The waveform viewer shows all selected signals:





# 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**.

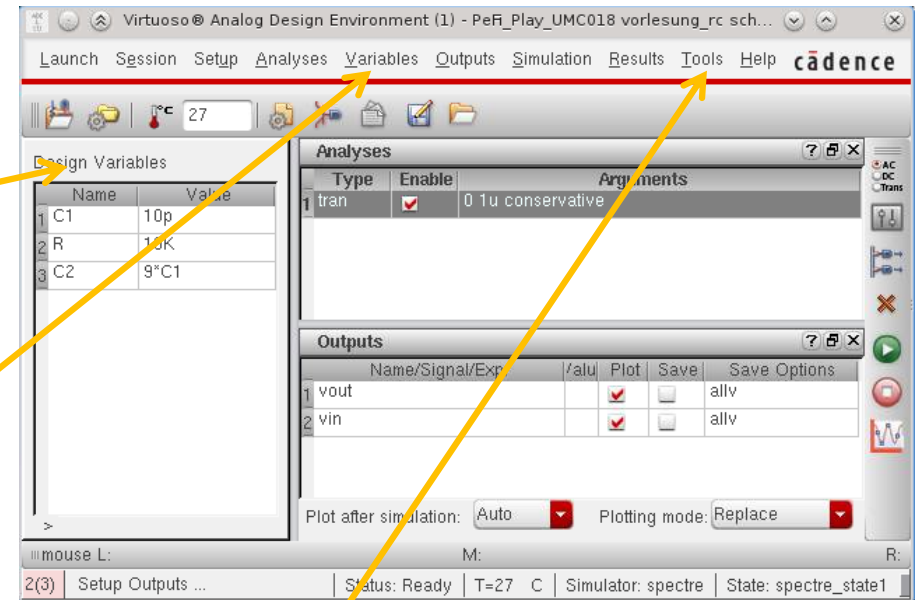




# Adding Design Variables

- 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**



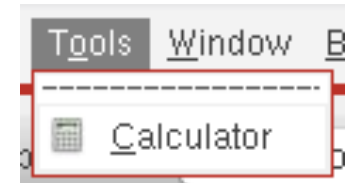
## (Copying Design Variables to the Cellview)

- 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

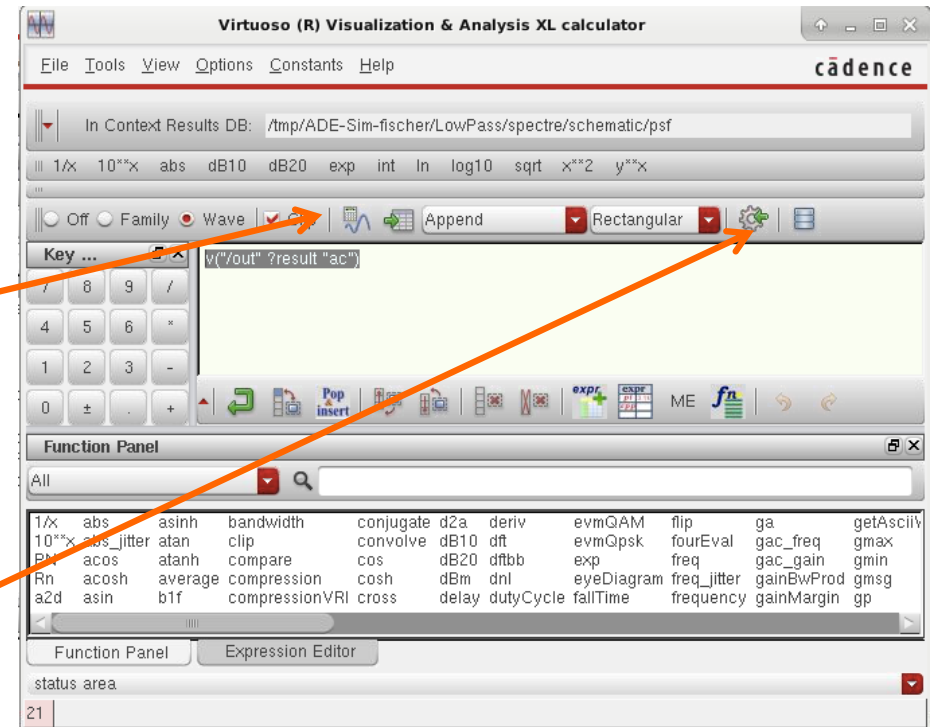


# The WaveForm Calculator

- 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

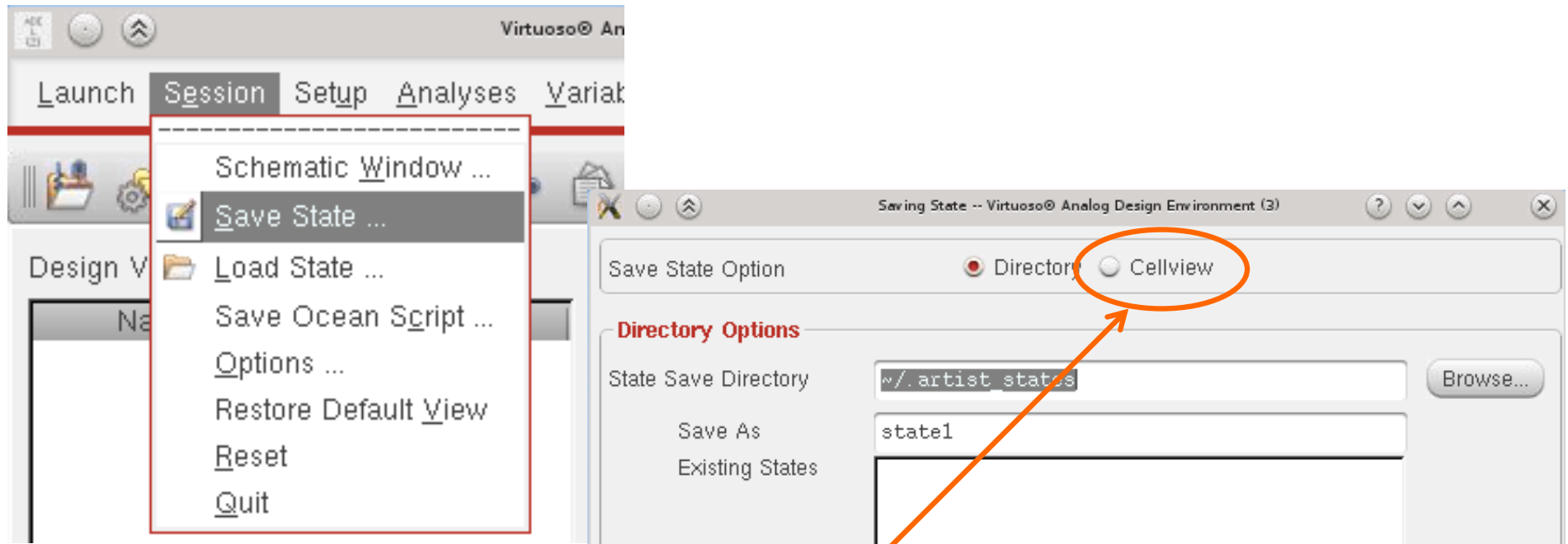


Outputs		
	Name/Signal/Expr	Value Plot
1	out	<input checked="" type="checkbox"/>
2	deriv(v("/out" ?result "ac"))	<input checked="" type="checkbox"/>



# Saving your 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