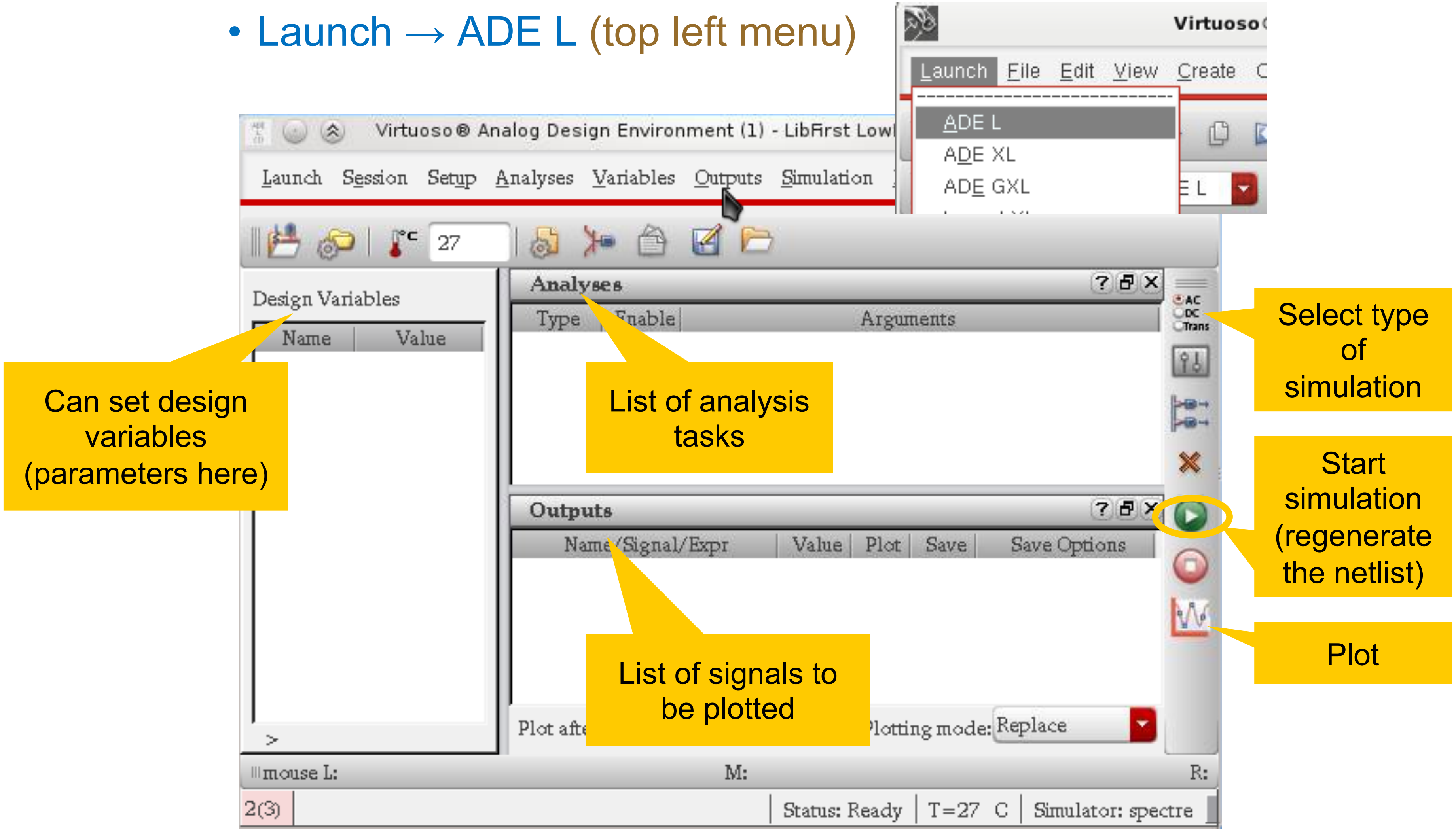
Exercise:
Starting a Simulation

Prof. Dr. P. Fischer

Lehrstuhl für Schaltungstechnik und Simulation
Uni Heidelberg

Starting the Simulator

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



Select Type of Simulation

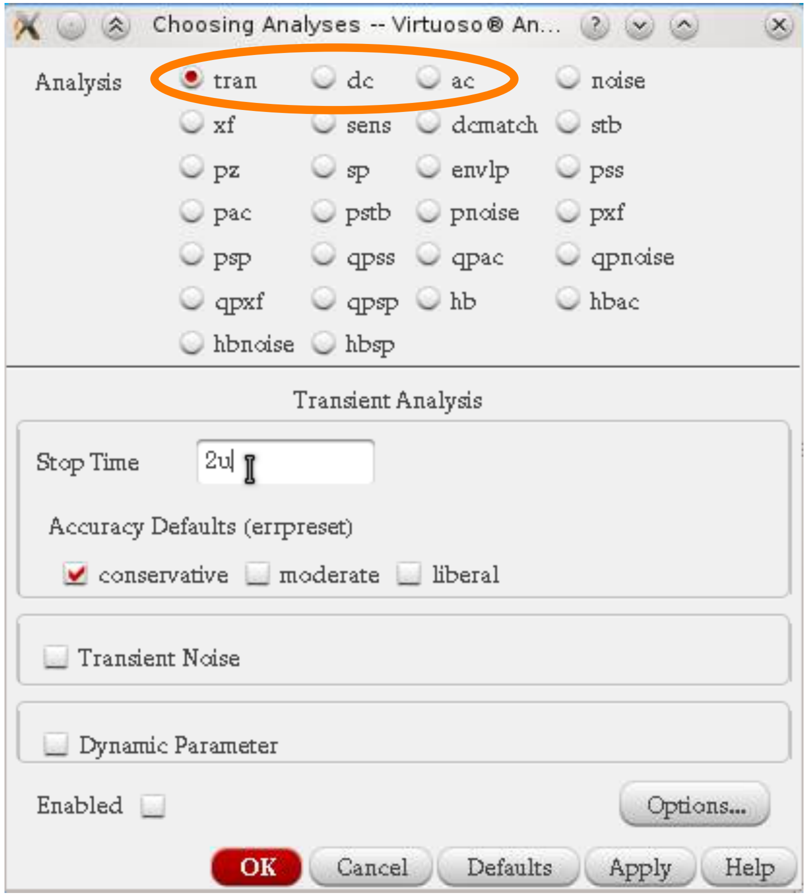
- Open the panel
  - By pressing the 

button or
  - In Analyses → Choose Menu

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

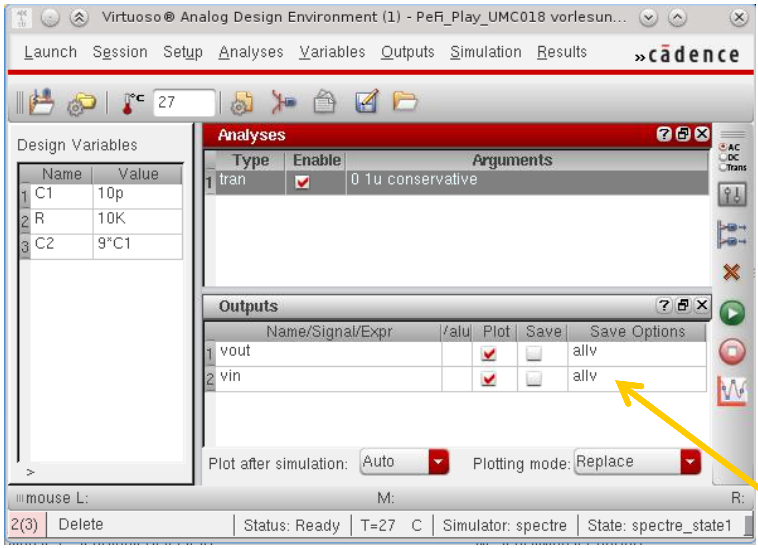
- Provide the parameters required by the analysis

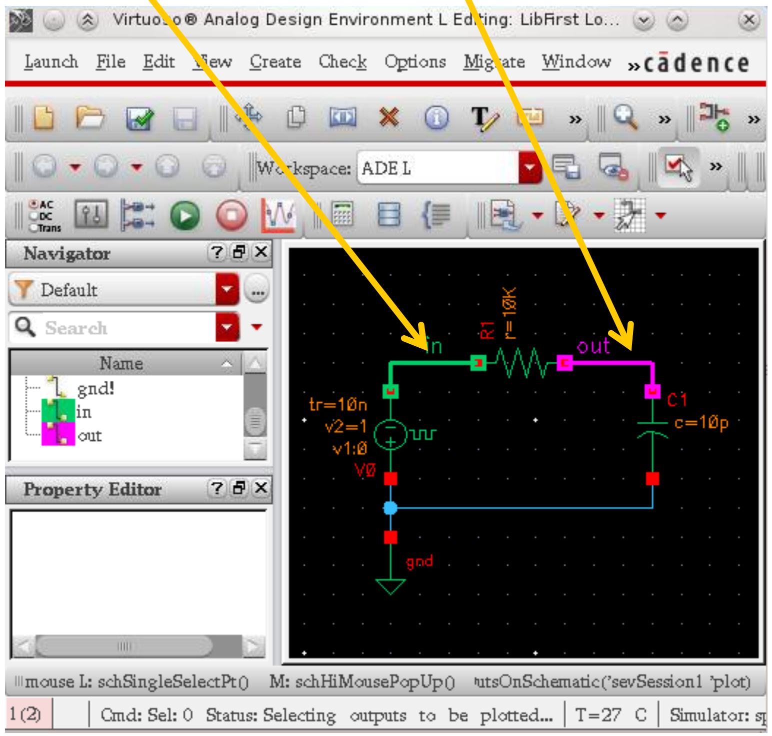
- Press ok



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!)

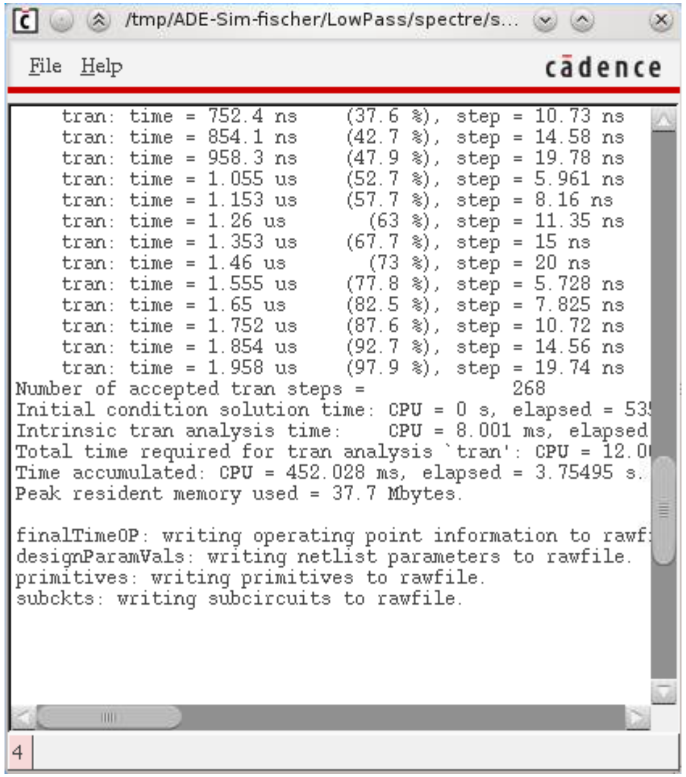




- Signals are listed in the lower right panel of the sim. window

Starting the Simulation

- 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)

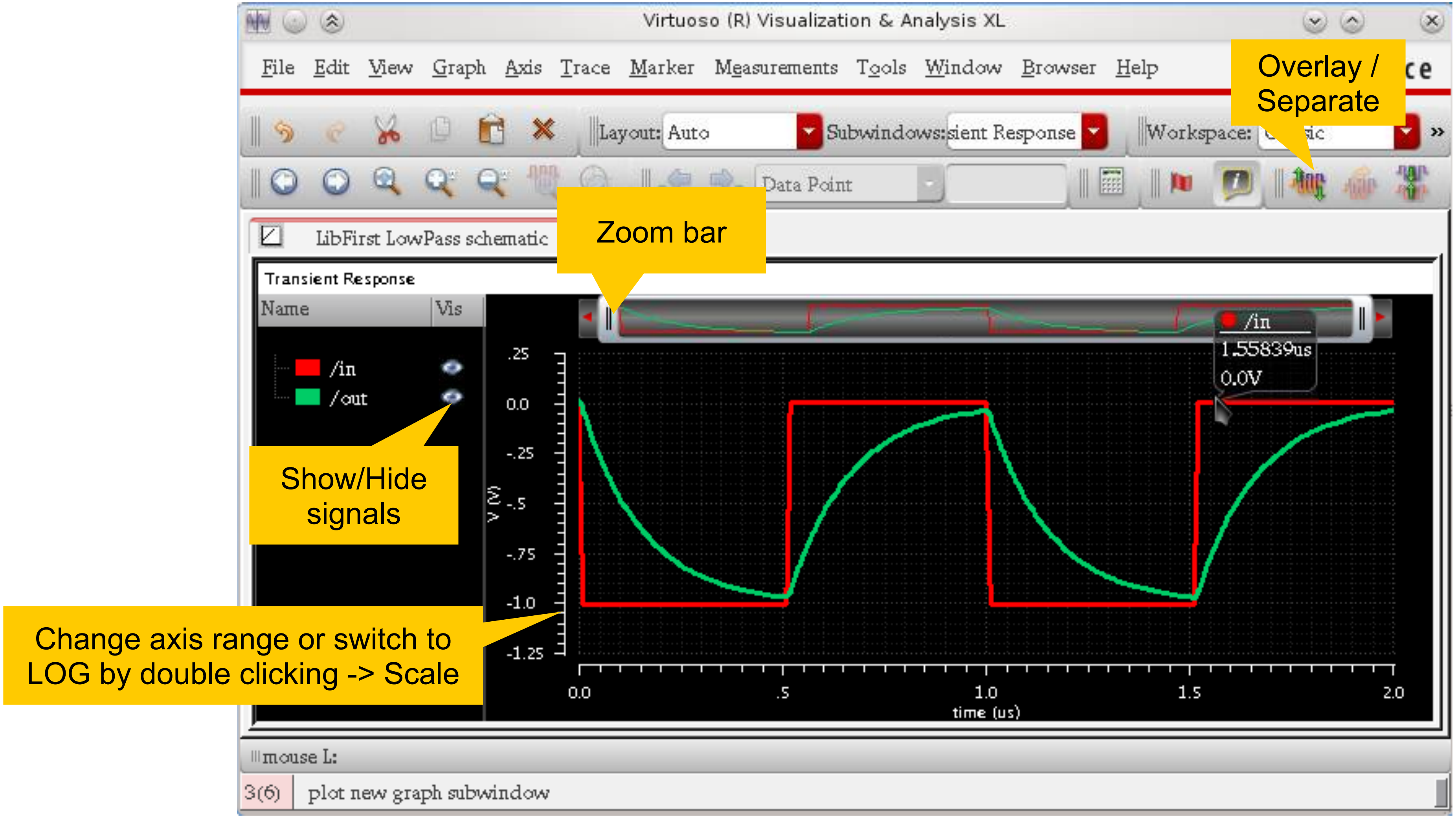


- Some common reasons for failure:
  - Schematic has been changed, but **not** checked & saved (F8)
  - Device parameters (resistor value..) are missing or wrong
  - Design variables (see later) have not been set
  - Circuit has severe errors (shorts..)
  - …

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

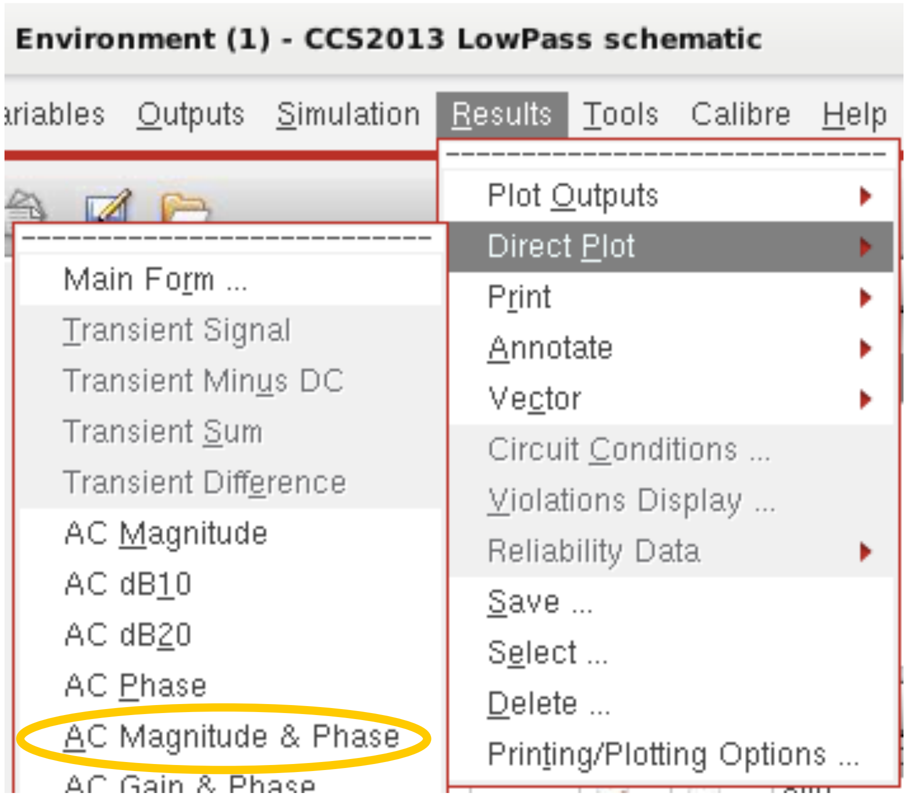
Look at the Results

- The waveform viewer shows all selected signals:



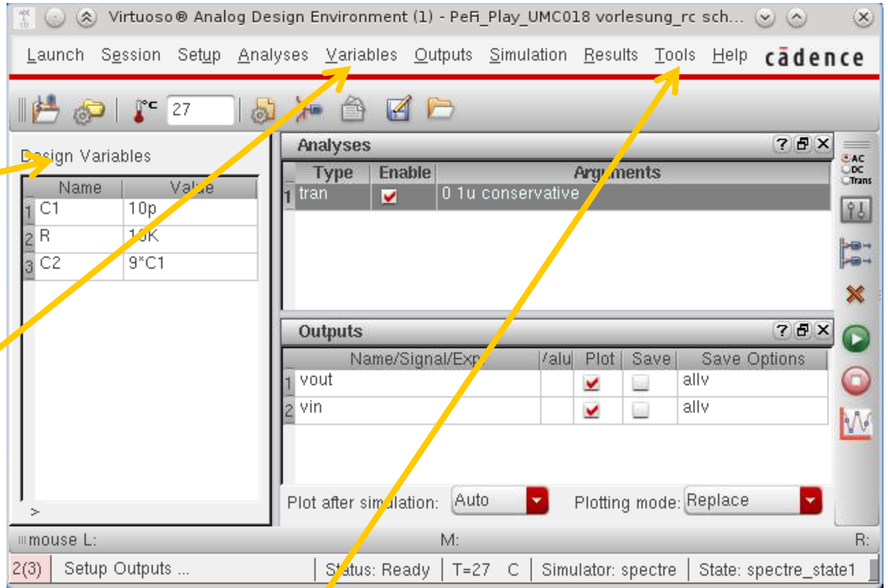
Showing More / Other Signals

- 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

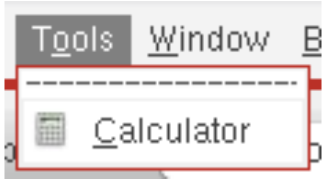
(Copying Design Variables to the Cellview)

- 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..

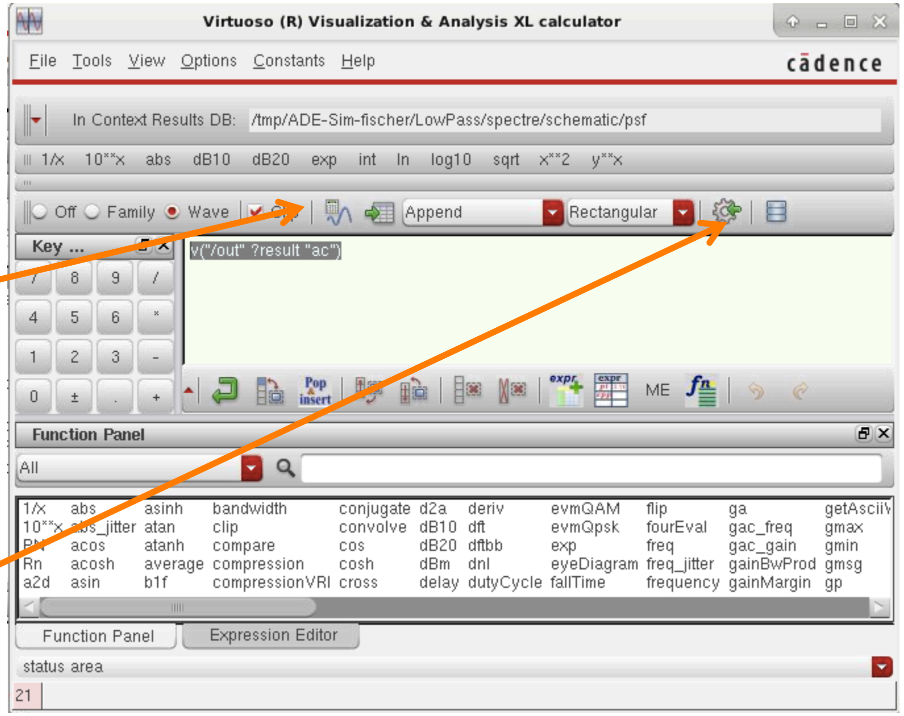
- Caveat:
  - If you delete a variable in a schematic component, so that it is not used any more, it may still be 'saved' in the cell view and simulation will complain.
    In such a case you have to delete the variable in the simulation window and copy the new set to the cellview

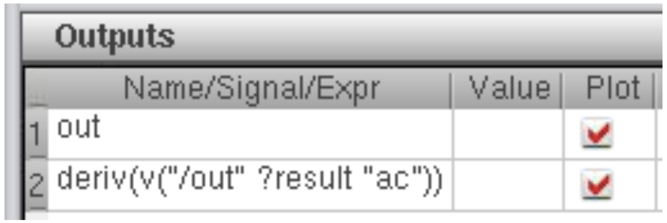
The WaveForm Calculator

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



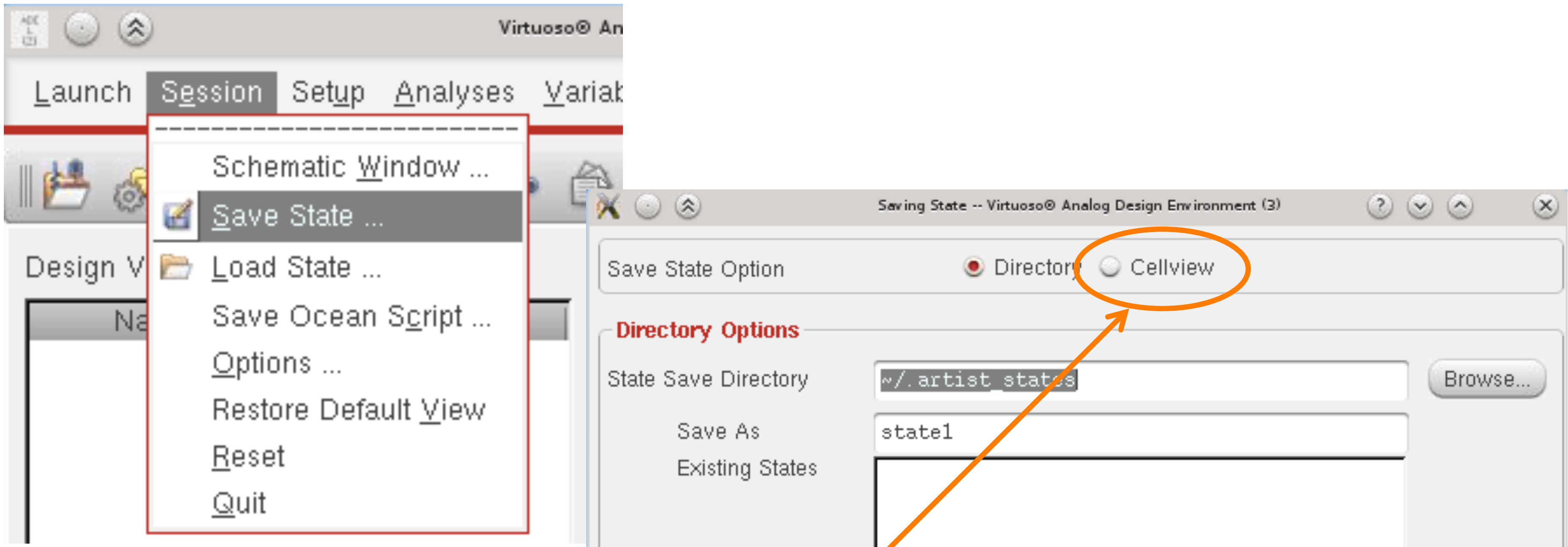
- You can assemble expressions graphically (using RPN)
- Plot the result once  or
- Send the expression to the outputs window  so that it is evaluated every time you run a new simulation





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

EXERCISES

Exercise 1: High Pass – AC analysis

- Use the HighPass circuit from the previous exercise
  - voltage source, ground, R = 1k, C=1n
  - Make sure the voltage source has 'AC Magnitude' set to 1
- Estimate the corner frequency of your circuit
- Chose an AC analysis with frequency span 2-3 orders of magnitude around the corner.
- Plot the Magnitude of the output
- Check that the -3dB point is **exactly** what you calculate!
- Change component values, predict the effect and simulate.
- Make the circuit more complicated (more Rs and Cs)

Exercise 2: High Pass & Rectangular Pulse

- Now use a rectangular pulse generator (vpulse)
  - Chose the frequency much slower than the RC time

  - How does the output waveform look like ?
  - When has the signal decreased to 1/e of the input step?
  - Is this what you expect from the component values?

  - Double the resistor and check what happens!

Exercise 3: High Pass & Sine Input

- Replace the rectangular generator by a sine wave generator (‘vsin’)
  - Set the *delay time* and *offset* to 0, the *amplitude* to 1V
  - Calculate the corner frequency (in Hertz!)
  - Check the output for a frequency ~10 x lower or ~10x higher than the corner
  - What is the output amplitude *exactly* at the corner frequency?
  - What is the phase shift between input and output at the corner frequency?
  - Try to run a parametric analysis, changing the value of the capacitor (or the resistor)